Chapter

WaterCAD V8*i* 1

Getting Started in Bentley WaterCAD V8i

Quick Start Lessons

Understanding the Workspace

Creating Models

Using ModelBuilder to Transfer Existing Data

Applying Elevation Data with TRex

Allocating Demands using LoadBuilder

Reducing Model Complexity with Skelebrator

Scenarios and Alternatives

Modeling Capabilities

Calibrating Your Model with Darwin Calibrator

Optimizing Capital Improvement Plans with Darwin Designer

Optimizing Pump Operations

Optimizing Pump Schedules Using Darwin Scheduler

Presenting Your Results

Importing and Exporting Data

Menus

Technical Reference

Technical Information Resources

Element Properties Reference

Glossary

WaterCAD V8*i* 1

Getting Started in Bentley WaterCAD V8*i* 1

What's New in WaterCAD V8*i*? 1

Municipal License Administrator Auto-Configuration 2

Starting Bentley WaterCAD V8*i* 2

Working with WaterCAD V8*i* Files 2

Exiting WaterCAD V8*i* 4

Using Online Help 5

Software Updates via the Web and Bentley SELECT 8

Troubleshooting 8

Checking Your Current Registration Status 9

Application Window Layout 9

- Standard Toolbar 10
- Edit Toolbar 12
- Analysis Toolbar 13
- Scenarios Toolbar 15
- Compute Toolbar 16
- View Toolbar 18
- Help Toolbar 19
- Layout Toolbar 20
- Tools Toolbar 24
- Zoom Toolbar 27
- Customizing WaterCAD V8*i* Toolbars and Buttons 30
- WaterCAD V8*i* Dynamic Manager Display 31

WaterObjects Help for Model Users 36

Quick Start Lessons 43

Building a Network and Performing a Steady-State Analysis 43

Extended Period Simulation 60

Scenario Management 68

Reporting Results 79

Automated Fire Flow Analysis 93

Water Quality Analysis 100

Darwin Designer to Optimize the Setup of a Pipe Network 109

Darwin Designer to Optimize a Pipe Network 121

Scenario Energy Costs 149

Pressure Dependent Demands 157

Criticality and Segmentation 181

Flushing 195

Understanding the Workspace 207

Stand-Alone 207

- The Drawing View 207
  - PANNING 207
  - ZOOMING 208
    - Zoom Dependent Visibility 211
  - DRAWING STYLE 213
- Using Aerial View 214
- Using Background Layers 215
  - IMAGE PROPERTIES 222
  - SHAPEFILE PROPERTIES 224
  - DXF PROPERTIES 225
- Show Flow Arrows (Stand-Alone) 226

MicroStation Environment 226

- Getting Started in the MicroStation environment 227
- The MicroStation Environment Graphical Layout 229
- MicroStation Project Files 231
  - SAVING YOUR PROJECT IN MICROSTATION 231
- Bentley WaterCAD V8*i* Element Properties 232
  - ELEMENT PROPERTIES 232
  - ELEMENT LEVELS DIALOG 233
  - TEXT STYLES 233
- View Associations (MicroStation Only) 233
- Working with Elements 234
  - EDIT ELEMENTS 235
  - DELETING ELEMENTS 235

Modifying Elements 235
Context Menu 235
Working with Elements Using MicroStation Commands 235
Bentley WaterCAD V8*i* Custom MicroStation Entities 236
MicroStation Commands 236
Moving Elements 236
Moving Element Labels 237
Snap Menu 237
Background Files 237
Import Bentley WaterCAD V8*i* 237
Annotation Display 237
Multiple models 238
Native Format Contours 238
Working in AutoCAD 238
The AutoCAD Workspace 240
AutoCAD Integration with WaterCAD V8*i* 240
Getting Started within AutoCAD 241
Menus 241
Drawing Setup 242
Symbol Visibility 242
AutoCAD Project Files 242
Drawing Synchronization 243
Saving the Drawing as Drawing*.dwg 244
Working with Elements Using AutoCAD Commands 244
WaterCAD V8*i* Custom AutoCAD Entities 245
Explode Elements 245
Moving Elements 246
Moving Element Labels 246
Snap Menu 246
Polygon Element Visibility 246
Undo/Redo 246
Contour Labeling 247
248
Google Earth Export 248
Google Earth Export from the MicroStation Platform 249
Google Earth Export from ArcGIS 251
Using a Google Earth View as a Background Layer to Draw a Model 254

Creating Models 259

Starting a Project 259
Bentley WaterCAD V8*i* Projects 260
Database Format Conversion 261
Setting Project Properties 262
Setting Options 263
Options Dialog Box - Global Tab 264

Stored Prompt Responses Dialog Box 268
Options Dialog Box - Project Tab 269
Options Dialog Box - Drawing Tab 271
Options Dialog Box - Units Tab 273
Options Dialog Box - Labeling Tab 276
Options Dialog Box - ProjectWise Tab 277
Working with ProjectWise 278
Setting Up ProjectWise Integration 284
About ProjectWise Geospatial 285
Maintaining Project Geometry 286
Setting the Project Spatial Reference System 286
Interaction with ProjectWise Explorer 287
ProjectWise Cross-Discipline Coordination Services Support 288
Elements and Element Attributes 294
Pipes 295
Minor Losses Dialog Box 297
Minor Loss Coefficients Dialog Box 299
Wave Speed Calculator 301
Junctions 303
Demand Collection Dialog Box 304
Unit Demand Collection Dialog Box 304
Hydrants 305
Hydrant Flow Curve Manager 305
Hydrant Flow Curve Editor 306
Hydrant Lateral Loss 308
Tanks 308
Reservoirs 313
Customer Meter Elements 313
External Customer Meter Data Setup 318
Pumps 319
Pump Definitions Dialog Box 320
Efficiency Points Table 329
Pump Curve Dialog Box 329
Flow-Efficiency Curve Dialog Box 330
Speed-Efficiency Curve Dialog Box 331
Pump and Motor Inertia Calculator 331
Pump Curve Display 332
Pump Curve Combinations 336
Variable Speed Pump Battery 341
Pump Stations 341
Pumps Dialog Box 343
Polygon Vertices Dialog Box 344
SCADA Elements 344
Valves 344
Defining Valve Characteristics 349
Valve Characteristics Dialog Box 350
Valve Characteristic Curve Dialog Box 352

General Note About Loss Coefficients on Valves 352
Modulating Control Valve 353
Spot Elevations 354
Turbines 355
Impulse Turbine 357
Reaction Turbines 358
Modeling Hydraulic Transients in Hydropower Plants 360
Turbine Parameters in HAMMER 364
Turbine Curve Dialog Box 365
Periodic Head-Flow Elements 366
Periodic Head-Flow Pattern Dialog Box 367
Air Valves 367
Determining the Type of Air Valve to Use 370
Air Flow Curves Dialog Box 373
Air Flow-Pressure Curve 374
Hydropneumatic Tanks 375
Initial Conditions Attributes 380
Gas Law vs. Constant Area Approximation 382
Transient Simulation Attributes 382
Tracking the Air-Liquid Interface 386
Variable Elevation Curve Dialog Box 387
Surge Valves 388
Check Valves 389
Rupture Disks 390
Discharge to Atmosphere Elements 390
Orifice Between Pipes Elements 392
Valve with Linear Area Change Elements 393
Surge Tanks 393
Other Tools 398
Border Tool 399
Text Tool 399
Line Tool 400
How The Pressure Engine Loads Bentley HAMMER Elements 401

Adding Elements to Your Model 402

Manipulating Elements 403
Select, Move, and Delete Elements 403
Splitting Pipes 405
Reconnect Pipes 406
Modeling Curved Pipes 406
Polyline Vertices Dialog Box 407
Assign Isolation Valves to Pipes Dialog Box 407
Batch Pipe Split Dialog Box 409
Batch Pipe Split Workflow 410
Batch Morph 411
Merge Nodes in Close Proximity 412

Select Adjacent Links 413

Editing Element Attributes 413
Property Editor 413
LABELING ELEMENTS 416
RELABELING ELEMENTS 416
SET FIELD OPTIONS DIALOG BOX 416
Date/Time Formats 417

Using Named Views 418

Using Selection Sets 420
Selection Sets Manager 421
Group-Level Operations on Selection Sets 426

Using the Network Navigator 427
Using the Duplicate Labels Query 433

Using the Pressure Zone Manager 434
Pressure Zone Export Dialog Box 444
Pressure Zone Flow Balance Tool Dialog Box 445

Using Prototypes 446

Zones 450

Engineering Libraries 452
Transient Valve Curve Editor 456
Transient Pump Curve Editor 457
Transient Turbine Curve Editor 458
Valve Relative Closure Curve Editor 459

Hyperlinks 459

Using Queries 467
Queries Manager 467
QUERY PARAMETERS DIALOG BOX 470
Creating Queries 471
USING THE LIKE OPERATOR 477

User Data Extensions 478
User Data Extensions Dialog Box 480
Sharing User Data Extensions Among Element Types 484
Shared Field Specification Dialog Box 485
Enumeration Editor Dialog Box 486
User Data Extensions Import Dialog Box 487
Formula Dialog Box 487

Property Grid Customizations Manager 489
Customization Editor Dialog Box 490

Tooltip Customization 491
Tooltip Customization Editor 492

i-Models 492

Publishing an i-model 493
Publish to Map Mobile i-model 496
Viewing an i-model 497

Storage Chambers Dialog Box 501

Using ModelBuilder to Transfer Existing Data 507

Preparing to Use ModelBuilder 507

ModelBuilder Connections Manager 510
Specify Datasource Location 514
Microsoft Access Database Engine Version 514

ModelBuilder Wizard 514
Step 1—Specify Data Source 515
Step 2—Specify Spatial Options 517
Step 3 - Specify Element Create/Remove/Update Options 519
Step 4—Additional Options 521
Step 5—Specify Field mappings for each Table/Feature Class 524
Step 6—Build operation Confirmation 528

Reviewing Your Results 529

Multi-select Data Source Types 529

ModelBuilder Warnings and Error Messages 530
ModelBuilder Warnings 530
ModelBuilder Error Messages 531

ESRI ArcGIS Geodatabase Support 532
Geodatabase Features 532
Geometric Networks 533
ArcGIS Geodatabase Features versus ArcGIS Geometric Network 533
Subtypes 534
SDE (Spatial Database Engine) 534

Specifying Network Connectivity in ModelBuilder 534
Sample Spreadsheet Data Source 536

The GIS-ID Property 537
GIS-ID Collection Dialog Box 538

Specifying a SQL WHERE clause in ModelBuilder 539

Modelbuilder Import Procedures 539
Importing Pump Definitions Using ModelBuilder 540
Using ModelBuilder to Import Pump Curves 545
Using ModelBuilder to Import Patterns 549
Using ModelBuilder to Import Time Series Data 553

Oracle as a Data Source for ModelBuilder 559
Oracle/ArcSDE Behavior 560

Applying Elevation Data with TRex 561

The Importance of Accurate Elevation Data 561

Numerical Value of Elevation 562
Accuracy and Precision 563

Obtaining Elevation Data 563

Record Types 565

Calibration Nodes 566

TRex Terrain Extractor 566

TRex Wizard 568
TRex Supported Terrain Models 573

Allocating Demands using LoadBuilder 575

Using GIS for Demand Allocation 575
Allocation 576
Billing Meter Aggregation 578
Distribution 579
Projection 581

Using LoadBuilder to Assign Loading Data 582
LoadBuilder Manager 582
LoadBuilder Wizard 583
LoadBuilder Run Summary 598
Unit Line Method 598

Generating Thiessen Polygons 600
Thiessen Polygon Creator Dialog Box 603
Creating Boundary Polygon Feature Classes 605

Demand Control Center 606
Apply Demand and Pattern to Selection Dialog Box 609

Unit Demands Dialog Box 611

Unit Demand Control Center 614

Pressure Dependent Demands 616
Piecewise Linear Dialog Box 622

Reducing Model Complexity with Skelebrator 623

Skeletonization 624
Skeletonization Example 625

Common Automated Skeletonization Techniques 627
Generic—Data Scrubbing 627
Generic—Branch Trimming 627

Generic—Series Pipe Removal 628

Skeletonization Using Skelebrator 629
Skelebrator—Smart Pipe Removal 629
Skelebrator—Branch Collapsing 630
Skelebrator—Series Pipe Merging 631
Skelebrator—Parallel Pipe Merging 633
Skelebrator—Inline Isolation Valve Replacement 634
Skelebrator—Other Skelebrator Features 635
Skelebrator—Conclusion 636

Using the Skelebrator Software 637
Skeletonizer Manager 638
BATCH RUN 642
PROTECTED ELEMENTS MANAGER 644
Selecting Elements from Skelebrator 644
Manual Skeletonization 647
Branch Collapsing Operations 650
Parallel Pipe Merging Operations 652
Series Pipe Merging Operations 654
Smart Pipe Removal Operations 658
Inline Isolating Valve Replacement 660
Conditions and Tolerances 661
PIPE CONDITIONS AND TOLERANCES 662
JUNCTION CONDITIONS AND TOLERANCES 663
Skelebrator Progress Summary Dialog Box 664

Backing Up Your Model 664
Skeletonization and Scenarios 665
Importing/Exporting Skelebrator Settings 666
Skeletonization and Active Topology 667

Scenarios and Alternatives 669

Understanding Scenarios and Alternatives 669
. . . . . Advantages of Automated Scenario Management 669
. . . . . A History of What-If Analyses 670
Distributed Scenarios 670
Self-Contained Scenarios 671
. . . . . The Scenario Cycle 672
. . . . . Scenario Attributes and Alternatives 673
. . . . . A Familiar Parallel 673
. . . . . Inheritance 674
OVERRIDING INHERITANCE 675
. . . . . DYNAMIC INHERITANCE 675
. . . . . .Local and Inherited Values 676
. . . . . Minimizing Effort through Attribute Inheritance 676

. . . . . .Minimizing Effort through Scenario Inheritance 677

Scenario Example - A Water Distribution System 678
. . . . . .Building the Model (Average Day Conditions) 678
. . . . . Analyzing Different Demands (Maximum Day Conditions) 679
. . . . . .Another Set of Demands (Peak Hour Conditions) 680
. . . . . Correcting an Error 680
. . . . . Analyzing Improvement Suggestions 681
. . . . . .Finalizing the Project 681

. . . . . .Scenarios 682
Scenarios Manager 683
Base and Child Scenarios 685
. . . . . Creating Scenarios 685
. . . . . EDITING SCENARIOS 686
Running Multiple Scenarios at Once (Batch Runs) 687
Batch Run Editor Dialog Box 688

Alternatives 689
Alternatives Manager 690
Alternative Editor Dialog Box 692
. . . . . Base and Child Alternatives 693
. . . . . .Creating Alternatives 694
. . . . . Editing Alternatives 694

Active Topology Alternative 696
Physical Alternative 700
Demand Alternatives 704
Initial Settings Alternative 705
Operational Alternatives 709
Age Alternatives 712
Constituent Alternatives 715
CONSTITUENTS MANAGER DIALOG BOX 719
Trace Alternative 720
Fire Flow Alternative 723
. . . . . FILTER DIALOG BOX 728
Energy Cost Alternative 729
Pressure Dependent Demand Alternative 732
Transient Alternative 735
Failure History Alternative 740
User Data Extensions 741

Scenario Comparison 744
Scenario Comparison Options Dialog Box 747
Scenario Comparison Collection Dialog Box 748

Modeling Capabilities 749

Model and Optimize a Distribution System 750

Steady-State/Extended Period Simulation 751
- Steady-State Simulation 751
- Extended Period Simulation (EPS) 751
  - TIME BROWSER 752
    - Time Browser Options 754
- Steady State Run 756

Calculate Network 757

Global Demand and Roughness Adjustments 758

Check Data/Validate 760

User Notifications 761

Using the Totalizing Flow Meter 764
- Totalizing Flow Meters Manager Dialog 765
- Totalizing Flow Meter Editor Dialog 766

System Head Curves 767
- System Head Curves in Closed Systems 767
- System Head Curves Manager Dialog 768

Post Calculation Processor 770

Flow Emitters 772

Parallel VSPs 773

Fire Flow Analysis 774
- Fire Flow Results 775
- Fire Flow Results Browser 776
- Not Getting Fire Flow at a Junction Node 777

Water Quality Analysis 778
- Age Analysis 779
- Constituent Analysis 780
- Trace Analysis 781
- Modeling for IDSE Compliance 781
- Water Quality Batch Run 790
  - SELECT ALTERNATIVES TO ANALYZE DIALOG BOX 794
  - GRAPH ELEMENT SELECTION DIALOG BOX 795
  - GRAPH VIEWER DIALOG BOX 796
    - Animation Options Dialog Box 800
  - STATISTICS TABLE DIALOG BOX 800
- Multi-Species Water Quality Analysis 801
  - MULTI-SPECIES ANALYSIS SETUP 805
  - MULTI-SPECIES MODEL CONFIGURATION 815

Criticality Analysis 819
- Outage Segments 822

Running Criticality Analysis 823
Understanding shortfalls 823
Criticality Results 824
Segmentation 827
Segmentation Results 832
Outage Segment Results 832

Calculation Options 833
Controlling Results Output 842
Flow Tolerance 844
Determining the Transient Run Duration 845
Convergence Improvements for Control Valves 846
Vapor Pressure 847
Selecting the Transient Friction Method 848
Engine Compatibility Calculation Option 850

Patterns 853
Pattern Manager 854
Pattern Curve Editor 858

Controls 859
Controls Tab 860
Conditions Tab 864
Actions Tab 871
Control Sets Tab 875
CONTROL SETS DIALOG BOX 876
Control Wizard 879

Active Topology 880
Active Topology Selection Dialog Box 881

External Tools 883

SCADAConnect Overview 885
SCADA Element 887
SCADA Signals Setup 889
SCADA SIGNALS - DATABASE 891
SCADA SIGNALS - OPC 897
SCADA SIGNALS - CITECT 900
Viewing SCADA Data in Model 902
TIME TOLERANCE 904
REFRESH AND AUTO REFRESH 905
Importing Initial Conditions with SCADAConnect 905
Loading Darwin Calibrator from SCADA Data 909
Alarms 912
Alerts 915
SCADAConnect Simulator 917
CONTROL OVERRIDES IN SCADACONNECT SIMULATOR 921
Displaying Model Results in SCADA Human Machine Interface (HMI)-Overview 922

SCADACONNECT RESULTS PUBLISHING DIALOG 926
SCADA Log 929

Flushing Simulation 989
Type of Flushing 989
Starting model 989
Specifying hydrant flows 990
Flushing Manager 990
FLUSHING TERMINOLOGY 991
FLUSHING WORK FLOW 993
STARTING FLUSHING MANAGER 994
FLUSHING AREA OPTIONS 995
FLUSHING EVENT CREATION 998
FLUSHING MANAGER TOOLBAR BUTTONS 1000
FLUSHING RESULTS BROWSER 1001
FLUSHING AREA REPORT (FLEX TABLE) 1003
FLUSHING OPTIONS DIALOG 1003
FLUSHING NOTIFICATIONS 1004
FLUSHING OPERATOR'S REPORT 1004

Modeling Tips 1005
Modeling a Hydropneumatic Tank 1006
Modeling a Pumped Groundwater Well 1006
Modeling Parallel Pipes 1007
Modeling Pumps in Parallel and Series 1008
Modeling Hydraulically Close Tanks 1009
Modeling Fire Hydrants 1009
Modeling a Connection to an Existing Water Main 1009
Top Feed/Bottom Gravity Discharge Tank 1011
Estimating Hydrant Discharge Using Flow Emitters 1012
Modeling Variable Speed Pumps 1014
TYPES OF VARIABLE SPEED PUMPS 1015
PATTERN BASED 1015
FIXED HEAD 1015
CONTROLS WITH FIXED HEAD OPERATION 1016
PARALLEL VSPS 1017
VSP CONTROLLED BY DISCHARGE SIDE TANK 1018
VSP CONTROLLED BY SUCTION SIDE TANK 1018
FIXED FLOW VSP 1019
Resolving ‘Unbalanced Network’ Errors 1020

Pipe Renewal Planner 1020

Pipe Break Analysis 1030
Pipe Break Group Dialog Box 1041
PICK A SELECTION SET DIALOG BOX 1043



Calibrating Your Model with Darwin Calibrator 1045

Calibration Studies 1049
- Field Data Snapshots Tab 1050
- Adjustment Groups 1056
  - GROUP GENERATOR DIALOG BOX 1058
- Calibration Criteria 1058
  - CALIBRATION CRITERIA FORMULAE 1059

Optimized Runs 1061
- Roughness Tab 1061
- Demand Tab 1062
- Status Tab 1064
- Field Data Tab 1064
- Options Tab 1064
- Notes Tab 1067

Manual Runs 1067
- Roughness Tab 1067
- Demand Tab 1068
- Status Tab 1069
- Field Data Tab 1069
- Notes Tab 1069

Calibration Solutions 1070
- Correlation Graph Dialog Box 1072
- Calibration Export to Scenario Dialog Box 1073

Importing Field Data into Darwin Calibrator Using ModelBuilder 1074
- Import Snapshots 1074
- Import Observed Target 1075

GA-Optimized Calibration Tips 1077
- Darwin Calibrator Troubleshooting Tips 1079

Optimizing Capital Improvement Plans with Darwin Designer 1083

Darwin Designer 1084

Design Study 1085
- Design Events tab 1089
- Boundary Overrides tab 1093
- Demand Adjustments tab 1096
- Pressure Constraints tab 1098
- Flow Constraints tab 1100
- Design Groups tab and Rehab Groups tab 1102
  - REHABILITATION GROUP DESIGNER DIALOG BOX 1107
- Costs/Properties tab 1107
  - REHABILITATION FUNCTIONS 1113

Design Type tab 1113
Notes Tab 1115
Initialize Table From Selection Set Dialog Box 1115
Load From Model Dialog Box 1115

Optimized Design Run 1116
Design Events tab 1117
Design Groups tab 1117
Rehab Groups tab 1118
Options tab (Optimized Run only) 1118
Notes Tab 1120

Manual Design Run 1120
Compute the Design Run 1121
Report Viewer 1125
Graph Dialog Box 1127
Export to Scenario 1132
Schema Augmentation 1135
Set Field Options 1135
Verification Summary 1136

Manual Cost Estimating 1137
Initiating Costing Runs 1137
Building A Cost Function 1138
Identifying Elements for the Cost Calculation 1139
Calculating Costs 1139

Advanced Darwin Designer Tips 1141

Optimizing Pump Operations 1151

Energy Management and Scenario Energy Cost 1151
Energy Management 1155
Power Meters 1159
Scenario Energy Cost Manager 1160
Energy Pricing Manager 1163
Unit Carbon Emissions Dialog Box 1165
Scenario Energy Cost Analysis Calculations 1166
Energy Cost Results 1166
COMPARING COST RESULTS ACROSS SCENARIOS 1171

Optimizing Pump Schedules Using Darwin Scheduler 1173

Best Practices and Tips 1173

Darwin Scheduler 1178
Scheduler Study 1180
Optimized Run 1190
Solutions 1200

Scheduler Results Plot 1204
Export to Scenario Dialog Box 1205

Darwin Scheduler FAQ 1205

Presenting Your Results 1221

Node Histories Tab 1221

Annotating Your Model 1222
Using Folders in the Element Symbology Manager 1225
Annotation Properties 1228
FREE FORM ANNOTATION DIALOG BOX 1229
SYMBOLOGY DEFINITIONS MANAGER 1230

Color Coding A Model 1231
Color Coding Legends 1235

Contours 1236
Contour Definition 1238
Contour Plot 1240
Contour Browser Dialog Box 1241
Enhanced Pressure Contours 1242

Using Profiles 1242
Profile Setup 1244
Profile Series Options Dialog Box 1248
Profile Viewer 1249

Viewing and Editing Data in FlexTables 1257
FlexTables 1258
Working with FlexTable Folders 1262
FlexTable Dialog Box 1263
Opening FlexTables 1265
Creating a New FlexTable 1265
Deleting FlexTables 1266
Naming and Renaming FlexTables 1266
Editing FlexTables 1267
Sorting and Filtering FlexTable Data 1270
CUSTOM SORT DIALOG BOX 1273
Customizing Your FlexTable 1274
Element Relabeling Dialog 1275
FlexTable Setup Dialog Box 1276
Copying, Exporting, and Printing FlexTable Data 1278
Statistics Dialog Box 1280
Using Sparklines 1280
SPARKLINE SETTINGS 1281

Reporting 1281
Using Standard Reports 1282
REPORTS FOR INDIVIDUAL ELEMENTS 1282

Creating a Scenario Summary Report 1282
Creating a Project Inventory Report 1282
Creating a Pressure Pipe Inventory Report 1282
Report Options 1282
Results Table Dialog Box 1284
Graphs 1285
Graph Manager 1285
Add to Graph Dialog Box 1287
Printing a Graph 1287
Working with Graph Data: Viewing and Copying 1287
Graph Dialog Box 1288
Graph Series Options Dialog Box 1293
Observed Data Dialog Box 1294
Sample Observed Data Source 1295
Chart Options Dialog Box 1297
Chart Options Dialog Box - Chart Tab 1298
Series Tab 1298
Panel Tab 1299
Axes Tab 1302
General Tab 1308
Titles Tab 1309
Walls Tab 1314
Paging Tab 1315
Legend Tab 1316
3D Tab 1322
Chart Options Dialog Box - Series Tab 1323
Format Tab 1323
Point Tab 1324
General Tab 1325
Data Source Tab 1326
Marks Tab 1327
Chart Options Dialog Box - Tools Tab 1331
Chart Options Dialog Box - Export Tab 1332
Chart Options Dialog Box - Print Tab 1334
Border Editor Dialog Box 1335
Gradient Editor Dialog Box 1336
Color Editor Dialog Box 1337
Color Dialog Box 1337
Hatch Brush Editor Dialog Box 1338
Hatch Brush Editor Dialog Box - Solid Tab 1338
Hatch Brush Editor Dialog Box - Hatch Tab 1339
Hatch Brush Editor Dialog Box - Gradient Tab 1339
Hatch Brush Editor Dialog Box - Image Tab 1340
Pointer Dialog Box 1341
Change Series Title Dialog Box 1342
Chart Tools Gallery Dialog Box 1342

Chart Tools Gallery Dialog Box - Series Tab 1342
Chart Tools Gallery Dialog Box - Axis Tab 1346
Chart Tools Gallery Dialog Box - Other Tab 1349
TeeChart Gallery Dialog Box 1354
Series 1354
Functions 1355
Customizing a Graph 1355
Time Series Field Data 1360
Select Associated Modeling Attribute Dialog Box 1363
Calculation Summary 1364
Calculation Summary Graph Series Options Dialog Box 1365
Results Table Dialog Box 1366
Print Preview Window 1366
Print Preparation 1368

Importing and Exporting Data 1371

Moving Data and Images between Model(s) and other Files 1371
Importing a WaterCAD V8*i* Database 1373
Exporting a HAMMER v7 Model 1373
Importing and Exporting EPANET Files 1374
Importing and Exporting Submodel Files 1374
Exporting a Submodel 1375
Exporting a DXF File 1377
File Upgrade Wizard 1378
Export to Shapefile 1378

Menus 1381

File Menu 1381
Edit Menu 1384
Analysis Menu 1384
Components Menu 1386
View Menu 1387
Tools Menu 1390
Report Menu 1393
Help Menu 1393
1394

Technical Reference 1395

Pressure Network Hydraulics 1395
Network Hydraulics Theory 1395
The Energy Principle 1396
The Energy Equation 1397
Hydraulic and Energy Grades 1398
Conservation of Mass and Energy 1398
The Gradient Algorithm 1400
Derivation of the Gradient Algorithm 1400
The Linear System Equation Solver 1403
Pump Theory 1404
Valve Theory 1407
CHECK VALVES (CVS) 1407
FLOW CONTROL VALVES (FCVS) 1408
PRESSURE REDUCING VALVES (PRVS) 1408
PRESSURE SUSTAINING VALVES (PSVS) 1408
PRESSURE BREAKER VALVES (PBVS) 1408
THROTTLE CONTROL VALVES (TCVS) 1408
GENERAL PURPOSE VALVES (GPVS) 1408

Friction and Minor Loss Methods 1409
Chezy's Equation 1409
Colebrook-White Equation 1409
Hazen-Williams Equation 1410
Darcy-Weisbach Equation 1411
Swamee and Jain Equation 1412
Manning's Equation 1412
Minor Losses 1413

1414

Engineer's Reference 1414
Roughness Values—Manning's Equation 1414
Roughness Values—Darcy-Weisbach Equation (Colebrook-White) 1415
Roughness Values—Hazen-Williams Equation 1416
Typical Roughness Values for Pressure Pipes 1417
Fitting Loss Coefficients 1418

Variable Speed Pump Theory 1419
VSP Interactions with Simple and Logical Controls 1422
Performing Advanced Analyses 1423

Hydraulic Equivalency Theory 1423
Principles 1424
HAZEN-WILLIAMS EQUATION 1424
MANNING'S EQUATION 1425
DARCY-WEISBACH EQUATION 1426
CHECK VALVES 1428
MINOR LOSSES 1428

NUMERICAL CHECK 1429

Thiessen Polygon Generation Theory 1430
Naïve Method 1430
Plane Sweep Method 1431

Method for Modeling Pressure Dependent Demand 1432
Use Cases 1433
Supply Level Evaluation 1434
Pressure Dependent Demand 1434
Demand Deficit 1435
Solution Methodology 1436
Modified GGA Solution 1437
Direct GGA Solution 1437

References 1438
1442

Technical Information Resources 1443

docs.bentley.com 1444

Bentley Services 1445

Bentley Discussion Groups 1446

Bentley on the Web 1446

TechNotes/Frequently Asked Questions 1446

BE Magazine 1446

BE Newsletter 1447

Client Server 1447

BE Careers Network 1447

Contact Bentley Systems 1447

Element Properties Reference 1451

Edit Element Properties 1452

Pipe Attributes 1452

Junction Attributes 1458

Hydrant Attributes 1463

Tank Attributes 1467

Reservoir Attributes 1471

Periodic Head-Flow Attributes 1473

Pump Attributes 1475

Pump Station Attributes 1480

Variable Speed Pump Battery Attributes 1481

Turbine Attributes 1486

Valve Attributes 1488

- Pressure Reducing Valve (PRV) Attributes 1489
- Pressure Breaker Valve (PBV) Attributes 1494
- Flow Control Vale (FCV) Attributes 1497
- Throttle Control Valve (TCV) Attributes 1499
- General Purpose Valve (GPV) Attributes 1502

Valve With Linear Area Change Attributes 1504

Check Valve Attributes 1506

Orifice Between Pipes Attributes 1508

Discharge To Atmosphere Attributes 1509

Surge Tank Attributes 1511

Hydropneumatic Tank Attributes 1515

Air Valve Attributes 1518

Surge Valve Attributes 1521

Rupture Disk Attributes 1523

Isolation Valve Attributes 1524

Spot Elevation Attributes 1525

Glossary 1527

Glossary 1527

- A 1527
- B 1527
- C 1528
- D 1529

E 1530
F 1530
G 1531
H 1532
I 1532
L 1533
M 1533
N 1535
O 1535
P 1536
R 1537
S 1537
T 1539
V 1539
W 1540
X 1541

Getting Started in Bentley WaterCAD V8*i*

1

Municipal License Administrator Auto-Configuration

Starting Bentley WaterCAD V8i

Working with WaterCAD V8i Files

Exiting WaterCAD V8i

Using Online Help

Software Updates via the Web and Bentley SELECT

Troubleshooting

Checking Your Current Registration Status

Application Window Layout

What's New in WaterCAD V8*i*?

New and upgraded features in WaterCAD V8*i* SELECTseries 4 include:

- New database file format as .sqlite replacing .mdb
- Sparkline display of EPS results
- Batch morph
- Filtering on property grid
- Numerous other enhancements

Note: WaterCAD V8*i* can open and import files from earlier versions but files created with this version are not backward compatible to earlier versions.

Municipal License Administrator Auto-Configuration

At the conclusion of the installation process, the Municipal License Administrator will be executed, to automatically detect and set the default configuration for your product, if possible. However, if multiple license configurations are detected on the license server, you will need to select which one to use by default, each time the product starts. If this is the case, you will see the following warning: “Multiple license configurations are available for WaterCAD V8*i*...” Simply press OK to clear the Warning dialog, then press Refresh Configurations to display the list of available configurations. Select one and press Make Default, then exit the License Administrator. (You only need to repeat this step if you decide to make a different configuration the default in the future.)

Starting Bentley WaterCAD V8*i*

After you have finished installing WaterCAD V8*i*, restart your system before starting WaterCAD V8*i* for the first time.

To start WaterCAD V8*i*

1. Double-click on the WaterCAD V8*i* icon on your desktop.

   or

2. Click Start > All Programs > Bentley > WaterCAD V8*i* > WaterCAD V8*i*.

Working with WaterCAD V8*i* Files

WaterCAD V8*i* uses an assortment of data, input, and output files. It is important to understand which are essential, which are temporary holding places for results and which must be transmitted when sending a model to another user. In general, the model is contained in a file with the wtg.sqlite extension. This file contains essentially all of the information needed to run the model. This file can be zipped to dramatically reduce its size for moving the file.

The .wtg file and the drawing file (.dwh, dgn, dwg or .sqlite) file contain user supplied data that makes it easier to view the model and should also be zipped and transmitted with the model when moving the model.

Other files found with the model are results files. These can be regenerated by running the model again. In general these are binary files which can only be read by the model. Saving these files makes it easy to look at results without the need to rerun the model. Because they can be easily regenerated, these files can be deleted to save space on the storage media.

When archiving a model at the end of the study, usually only the *.wtg.sqlite, *.wtg files, and the platform specific supporting files (*.dwh, *.dgn, *.dwg or *.sqlite) need to be saved.The file extensions are explained below:

- .bak - backup files of the model files
- .cri - results of criticality analysis
- .dgn - drawing file for MicroStation platform
- .dwg - drawing file for AutoCAD platform
- .dwh - drawing file for stand alone platform
- .sqlite - access database file for ArcGIS platform
- .nrg - results of energy calculations
- .osm - outage segmentation results
- .out - primary output file from hydraulic and water quality analyses
- .out.fl - output file from flushing analysis
- .rpc - report file from hydraulic analysis with user notifications
- .seg - results of segmentation analysis
- wtg.sqlite - main model file
- .wtg - display settings (e.g. color coding, annotation)
- .xml - xml files, generally libraries, window and other settings. Some modules like ModelBuilder also use .xml files to store settings independent of the main model.

Using the Custom Results File Path Option

When the **Specify Custom Results File Path** option (found under Tools > Options > Project Tab) is on for the project, the result files will be stored in the custom path specified when the project is closed. When the project is open, all of the applicable result files (if any) will be moved (not copied) to the temporary directory to be worked on. The result files will then be moved back to the custom directory when the project is closed.

The advantages of this are that moving a file on disk is very quick, as opposed to copying a file, which can be very slow. Also, if you have your project stored on a network drive and you specify a custom results path on your local disk, then you will avoid network transfer times as well. The disadvantages are that, should the program crash or the project somehow doesn't close properly, then the results files will not be moved back and will be lost.

If you then wish to share these results files with another user of the model, you can use the **Copy Results To Project Directory** command (Tools > Database Utilities > Copy Results To Project Directory) to copy the results files to the saved location of the model. The user receiving the files may then use the Update Results From Project Directory command (Tools > Database Utilities > Update Results From Project Directory) to copy the results files from the project directory to their custom results file path.

Drag-and-drop File Open

You can open model files by simply dragging them (from Windows Explorer, for example) into the application window (stand alone version only). You can drag either the .wtg or the .sqlite associated with the model.

You can drag multiple files into the application at once. All files must be of a valid type (.wtg or .sqlite) for this to work.

Exiting WaterCAD V8*i*

To exit WaterCAD V8*i*

1. Click the application window's Close icon.



or
From the File menu, choose Exit.

Note: If you have made changes to the project file without saving, the following dialog box will open. Click Yes to save before exiting, No to exit without saving, or Cancel to stop the operation.

Using Online Help

WaterCAD V8*i* Help menu and Help window are used to access WaterCAD V8*i* extensive online help.

Context-sensitive online help is available. Hypertext links, which appear in color and are underlined when you pass the pointer over them, allow you to move easily between related topics.

Note: Certain Windows DLLs must be present on your computer in order to use Online Help. Make sure you have Microsoft Internet Explorer (Version 5.5 or greater) installed. You do not need to change your default browser as long as Internet Explorer is installed.

To open the Help window

1. From the Help menu, choose WaterCAD V8*i* Help.
   The Help window opens, and the Table of Contents displays.

   The Help window consists of two panes - the navigation pane on the left and the topic pane on the right.

2. To get help on a dialog box control or a selected element:
   Press <F1> and the Help window opens (unless it is already open) and shows the information about the selected element.

Subtopics within a help topic are collapsed by default. While a subtopic is collapsed only its heading is visible. To make visible a subtopic's body text and graphics you must expand the subtopic.

To expand a subtopic

Click the expand (+) icon to the left of the subtopic heading or the heading itself.

To collapse a subtopic

Click the collapse (-) icon to the left of the subtopic heading or the heading itself.

The navigation pane has the following tabs:

- Contents - used for browsing topics.
- Index - index of help content.
- Search - used for full-text searching of the help content.
- Favorites - customizable list of your favorite topics

To browse topics using the Contents tab

1. On the Contents tab, click the folder symbol next to any book folder (such as *Getting Started*, *Using Scenarios and Alternatives*) to expand its contents.
2. Continue expanding folders until you reach the desired topic.
3. Select a topic to display its content in the topic pane.

To display the next or previous topic according to the topic order shown in the Contents tab

To display the next topic, click the right arrow or to display the previous topic, click the left.



To use the index of help content

1. Click the Index tab.
2. In the search field, type the word you are searching for.
   or
   Scroll through the index using the scroll bar to find a specific entry.
3. Select the desired entry and click the Display button.
   or
   Double-click the desired entry.
   The content that the selected index entry is referencing displays in the topic pane.

Note: If you select an entry that has subtopics, a dialog box opens from which you can select the desired subtopic. In this case, select the subtopic and click the Display button.

To search for text in the help content

1. Click the Search tab.
2. In the search field, type the word or phrase for which you are searching.
3. Click the List Topics button.
   Results of the search display in the list box below the search field.
4. Select the desired topic and click the Display button.
   or
   Double-click the desired topic.

Search results vary based on the quality of the search criteria entered in the Search field. The more specific the search criteria, the more narrow the search results. You can improve your search results by improving the search criteria. For example, a *word* is considered to be a group of contiguous alphanumeric characters. A *phrase* is a group of words and their punctuation. A *search string* is a word or phrase on which you search.

A *search string* finds any topic that contains all of the words in the string. You can improve the search by enclosing the *search string* in quotation marks. This type of search finds only topics that contain the exact string in the quotation marks.

To add a help topic to a list of “favorite” help topics

1. In the Contents, Index, or Search tabs, select the desired help topic.
2. Click the Favorites tab.
   The selected help topic automatically displays in the “Current topic” field at the bottom of the tab.
3. Click the Add button.

To display a topic from your Favorites list

1. Click the Favorites tab.
2. In the list box, select the desired topic and click the Display button.
   or
   Double-click the desired topic.
   The selected topic's content displays in the topic pane.

Online help is periodically updated and posted on Bentley's Documentation Web site, http://docs.bentley.com/ for downloading. On this site you can also browse the current help content for this product and other Bentley products.

Software Updates via the Web and Bentley SELECT

Bentley SELECT is the comprehensive delivery and support subscription program that features product updates and upgrades via Web downloads, around-the-clock technical support, exclusive licensing options, discounts on training and consulting services, as well as technical information and support channels. It's easy to stay up-to-date with the latest advances in our software. Software updates can be downloaded from our Web site, and your version of Bentley WaterCAD V8*i* can then be upgraded to the current version quickly and easily. Just click **Check for SELECT Updates** on the toolbar to launch your preferred Web browser and open our Web site. You can also access our KnowledgeBase for answers to your Frequently Asked Questions (FAQs).

Note: Your PC must be connected to the Internet to use the Check for SELECT Updates button.

Troubleshooting

Due to the multitasking capabilities of Windows, you may have applications running in the background that make it difficult for software setup and installations to determine the configuration of your current system.

Try these steps before contacting our technical support staff

1. Shut down and restart your computer.
2. Verify that there are no other programs running. You can see applications currently in use by pressing Ctrl+Shift+Esc in Windows 2000 and Windows XP. Exit any applications that are running.
3. Disable any antivirus software that you are running.

   Caution: After you install Bentley WaterCAD V8*i* , make certain that you restart any antivirus software you have disabled. Failure to restart your antivirus software leaves you exposed to potentially destructive computer viruses.

4. Try running the installation or uninstallation again (without running any other program first).

If these steps fail to successfully install or uninstall the product, contact Technical Support.

Checking Your Current Registration Status

After you have registered the software, you can check your current registration status by opening the About... box from within the software itself.

To view your registration information

1. Select **Help** > **About** Bentley WaterCAD V8*i* .
2. The version and build number for Bentley WaterCAD V8*i* display in the lower-left corner of the About Bentley WaterCAD V8*i* dialog box.

   The current registration status is also displayed, including: user name and company, serial number, license type and check-in status, feature level, expiration date, and SELECT Server information.

Application Window Layout

The WaterCAD V8*i* application window contains toolbars that provide access to frequently used menu commands and are organized by the type of functionality offered.

Standard Toolbar

Edit Toolbar

Analysis Toolbar

Scenarios Toolbar

Compute Toolbar

View Toolbar

Help Toolbar

Layout Toolbar

Tools Toolbar

Zoom Toolbar

Customizing WaterCAD V8i Toolbars and Buttons

WaterCAD V8i Dynamic Manager Display

Standard Toolbar

The Standard toolbar contains controls for opening, closing, saving, and printing WaterCAD V8*i* projects.



The Standard toolbar is arranged as follows:

To	Use
Create a new Bentley WaterCAD V8*i* project. When you select this command, the **Select File to Create** dialog box opens, allowing you to define a name and directory location for the new project.	**New** 
Open an existing Bentley WaterCAD V8*i* project. When this command is initialized, the **Select** Bentley WaterCAD V8*i* **Project to Open** dialog box opens, allowing you to browse to the project to be opened.	**Open** 
Closes the currently open project.	**Close** 
Close all the projects that are opened.	**Close All** 

Save the current project.	**Save** 
Save all the projects that are opened.	**Save All** 
Open the Print Preview window, displaying the current view of the network as it will be printed. Choose **Fit to Page** to print the entire network scaled to fit on a single page or **Scaled** to print the network at the scale defined by the values set in the Drawing tab of the project Options dialog (Tools > Options). If the model is printed to scale, it may contain one or more pages (depending on how large the model is relative to the page size specified in the Page Settings dialog, which is accessed through the Print Preview window).	**Print Preview** 
Print the current view of the network. Choose **Fit to Page** to print the entire network scaled to fit on a single page or **Scaled** to print the network at the scale defined by the values set in the Drawing tab of the project Options dialog (Tools > Options). If the model is printed to scale, it may contain one or more pages (depending on how large the model is relative to the page size specified in the Page Settings dialog, which is accessed through the Print Preview window).	**Print** 

Edit Toolbar

The Edit toolbar contains controls for deleting, finding, undoing, and redoing actions in WaterCAD V8*i*.

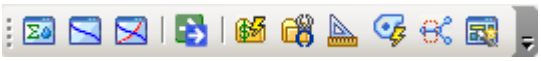


The Edit toolbar is arranged as follows:

To	Use
Cancel your most recent action.	**Undo** 
Redo the last canceled action.	**Redo** 
Delete the currently selected element(s) from the network.	**Delete** 
Removes the highlighting that can be applied using the Network Navigator.	**Clear Highlight** 
Find a specific element by choosing it from a menu containing all elements in the current model.	**Find Element** 

Analysis Toolbar

The Analysis toolbar contains controls for analyzing WaterCAD V8*i* projects.



The Analysis toolbar is arranged as follows:

To	Use
Open the Totalizing Flow Meters dialog box, which allows you to view, edit, and create flow meter definitions.	**Totalizing Flow Meters** 
Open the Hydrant Flow Curves dialog box, which allows you to view, edit, and create hydrant flow definitions.	**Hydrant Flow Curves** 
Open the System Head Curves dialog box, where you can view, edit, and create system head definitions.	**System Head Curves** 
Open the Post Calculation Processor, where you can perform statistical analysis for an element or elements on various results obtained during an extended period simulation calculation.	**Post Calculation Processor** 
Open the Energy Costs dialog box, where you can view, edit, and create energy cost scenarios.	**Energy Costs** 
Open the Darwin Calibrator dialog box, where you can view, edit, and create calibration studies.	**Darwin Calibrator** 

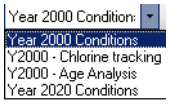
Open the Darwin Designer dialog box, where you can view, edit, and create designer studies.	**Darwin Designer** 
Open the Darwin Scheduler dialog box, where you can view, edit, and create scheduler studies.	**Darwin Scheduler** 
Open the Criticality dialog box, where you can view, edit, and create criticality studies.	**Criticality** 
Open the Pressure Zone dialog box, where you can view, edit, and create pressure zone studies.	**Pressure Zone** 

Scenarios Toolbar

The Scenarios toolbar contains controls for creating scenarios in WaterCAD V8*i* projects.



The Scenarios toolbar is arranged as follows:

To	Use
Change the current scenario.	**Scenario List Box** 
Open the Scenario manager, where you can create, view, and manage project scenarios.	**Scenarios** 
Open the Alternative manager, where you can create, view, and manage project alternatives.	**Alternatives** 
Open the Calculation Options manager, where you can create different profiles for different calculation settings.	**Calculation Options** 

Compute Toolbar

The Compute toolbar contains controls for computing WaterCAD V8*i* projects.

The Compute toolbar contains the following:

To	Use
Run a diagnostic check on the network data to alert you to possible problems that may be encountered during calculation. This is the manual validation command, and it checks for input data errors. It differs in this respect from the automatic validation that WaterCAD V8*i* runs when the compute command is initiated, which checks for network connectivity errors as well as many other things beyond what the manual validation checks.	**Validate** 
Calculate the network. Before calculating, an automatic validation routine is triggered, which checks the model for network connectivity errors and performs other validation.	**Compute** 
Open the Fire Flow Results Browser dialog box.	**Fire Flow Results Browser** 
Open the Flushing Results Browser dialog box.	**Flushing Results Browser** 
Open the Calculation Summary dialog box.	**Calculation Summary** 
Open the User Notifications Manager, allowing you to view warnings and errors uncovered by the validation process. This button does not appear in the toolbar by default but can be added	**User Notifications** 

View Toolbar

The View toolbar contains controls for viewing WaterCAD V8*i* projects.



The View toolbar contains the following:

To	Use
Open the Element Symbology manager, allowing you to create, view, and manage the element symbol settings for the project.	**Element Symbology** 
Open the Background Layers manager, allowing you to create, view, and manage the background layers associated with the project.	**Background Layers** 
Open the Network Navigator dialog box.	**Network Navigator** 
Open the Selection Sets Manager, allowing you to create, view, and modify the selection sets associated with the project.	**Selection Sets** 
Opens the Query Manager.	**Queries** 
Opens the Prototypes Manager.	**Prototypes** 
Open the FlexTables manager, allowing you to create, view, and manage the tabular reports for the project.	**FlexTables** 

Open the Graph manager, allowing you to create, view, and manage the graphs for the project.	**Graphs** 
Open the Profile manager, allowing you to create, view, and manage the profiles for the project.	**Profiles** 
Open the Contour Manager where you can create, view, and manage contours.	**Contours** 
Open the Named Views manager where you can create, view, and manage named views.	**Named Views** 
Open the Aerial View manager where you can zoom to different elements in the project.	**Aerial View** 
Opens the Property Editor.	**Properties** 
Opens the Property Grid Customizations manager.	**Property Grid Customizations** 

Help Toolbar

The Help toolbar provides quick access to the some of the commands that are available in the Help menu.



The Help toolbar contains the following:

To	Use
Open your Web browser to the SELECTservices page on the Bentley Web site.	**Check for SELECT Updates** 
Open the Bentley Institute page on the Bentley Web site.	**Bentley Institute Training** 
Open your Web browser to the SELECTservices page on the Bentley Web site.	**Bentley SELECT Support** 
Opens your web browser to the Bentley.com Web site's main page.	**Bentley.com** 
Opens the Bentley WaterCAD V8*i* online help.	**Help** 

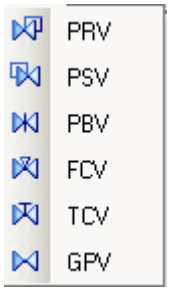
Layout Toolbar

The Layout toolbar is used to lay out a model in the WaterCAD V8*i* drawing pane.



The Layout toolbar contains the following:

To	Use
Change your mouse cursor into a selection tool. The selection tool behavior varies depending on the direction in which the mouse is dragged after defining the first corner of the selection box, as follows: • If the selection is made from left-to-right, all elements that fall completely within the selection box that is defined will be selected. • If the selection is made from right-to-left, all elements that fall completely within the selection box and that cross one or more of the lines of the selection box will be selected.	**Select** 
Change your mouse cursor into a pipe tool.	**Pipe** 
Change your mouse cursor into a junction tool. When this tool is active, click in the drawing pane to place the element.	**Junction** 
Change your mouse cursor into a hydrant tool. When this tool is active, click in the drawing pane to place the element.	**Hydrant** 
Change your mouse cursor into a tank element symbol. When this tool is active, click in the drawing pane to place the element.	**Tank** 
Change your mouse cursor into a reservoir element symbol. When this tool is active, click in the drawing pane to place the element.	**Reservoir** 
Change your mouse cursor into a customer meter element symbol. When this tool is active, click in the drawing pane to place the element.	**Customer Meter** 

Change your mouse cursor into a SCADA element symbol. When this tool is active, click in the drawing pane to place the element.	**SCADA Element** 
Change your mouse cursor into a pump element symbol. Clicking the left mouse button while this tool is active causes a pump element to be placed at the location of the mouse cursor.	**Pump** 
Change your mouse cursor into a pump station element symbol. Clicking the left mouse button while this tool is active causes a pump station element to be placed at the location of the mouse cursor.	**Variable Speed Pump Battery** 
Change your mouse cursor into a valve tool. Click the down arrow to select the type of valve you want to place in your model: • **Pressure Reducing Valve** • **Pressure Sustaining Valve** • **Pressure Breaker Valve** • **Flow Control Valve** • **Throttle Control Valve** • **General Purpose Valve**	**Valves**  
Change your mouse cursor into an isolation valve symbol. When this tool is active, click in the drawing pane to place the element.	**Isolation Valve** 
Change your mouse cursor into a spot elevation symbol. When this tool is active, click in the drawing pane to place the element.	**Spot Elevation** 
Change your mouse cursor into a turbine symbol. When this tool is active, click in the drawing pane to place the element..	**Turbine** 

Change your mouse cursor into a periodic head-flow symbol. When this tool is active, click in the drawing pane to place the element.	**Periodic Head-Flow** 
Change your mouse cursor into an air valve symbol. When this tool is active, click in the drawing pane to place the element.	**Air Valve** 
Change your mouse cursor into a hydropneumatic tank symbol. When this tool is active, click in the drawing pane to place the element.	**Hydropneumatic Tank** 
Change your mouse cursor into a surge valve symbol. When this tool is active, click in the drawing pane to place the element.	**Surge Valve** 
Change your mouse cursor into a check valve symbol. When this tool is active, click in the drawing pane to place the element.	**Check Valve** 
Change your mouse cursor into a rupture disk symbol. When this tool is active, click in the drawing pane to place the element.	**Rupture Disk** 
Change your mouse cursor into a discharge to atmosphere symbol. When this tool is active, click in the drawing pane to place the element.	**Discharge to Atmosphere** 
Change your mouse cursor into an orifice between pipes symbol. When this tool is active, click in the drawing pane to place the element.	**Orifice Between Pipes** 
Change your mouse cursor into a valve with linear area change symbol. When this tool is active, click in the drawing pane to place the element.	**Valve with Linear Area Change** 

Change your mouse cursor into a surge tank symbol. When this tool is active, click in the drawing pane to place the element.	**Surge Tank**
Change your mouse cursor into a border symbol. When the border tool is active, you can draw a simple box in the drawing pane using the mouse. For example, you might want to draw a border around the entire model.	**Border**
Change your mouse cursor into a text symbol. When the text tool is active, you can add simple text to your model. Click anywhere in the drawing pane to display the Text Editor dialog box, where you can enter text to be displayed in your model.	**Text**
Change your mouse cursor into a line symbol. When this tool is active, you can draw lines and polygons in your model using the mouse.	**Line**

Tools Toolbar

The Tools toolbar provides quick access to the same commands that are available in the Tools menu.



The Tools toolbar contains the following:

To	Use
Open a Select dialog to select areas in the drawing.	**Active Topology Selection** 
Open the ModelBuilder Connections Manager, where you can create, edit, and manage ModelBuilder connections to be used in the model-building/model-synchronizing process.	**ModelBuilder** 
Open the TRex wizard where you can select the data source type, set the elevation dataset, choose the model and features.	**Trex** 
Open the SCADAConnect manager where you can add or edit signals.	**SCADAConnect** 
Open the Skelebrator manager to define how to skeletonize your network.	**Skelebrator Skeletonizer** 
Open the LoadBuilder manager where you can create and manage Load Build templates.	**Load Builder** 
Open the Wizard used to create a Thiessen polygon.	**Thiessen Polygon** 
Open the Demand Control Center manager where you can add new demands, delete existing demands, or modify existing demands.	**Demand Control Center** 

Open the Unit Demand Control Center manager where you can add new unit demands, delete existing unit demands, or modify existing unit demands.	**Unit Demand Control Center** 
Opens the Scenario Coparison window, which enables you to compare input values between any two scenarios to identify differences quickly.	**Scenario Comparison** 
Associate external files, such as pictures or movie files, with elements.	**Hyperlinks** 
Open the User Data Extension dialog box, which allows you to add and define custom data fields. For example, you can add new fields such as the pipe installation date.	**User Data Extensions** 
Compact the database, which eliminates the empty data records, thereby defragmenting the datastore and improving the performance of the file.	**Compact Database** 
Synchronize the current model drawing with the project database.	**Synchronize Drawing** 
Ensures consistency between the database and the model by recalculating and updating certain cached information. Normally this operation is not required to be used.	**Update Database Cache** 
This command copies the model result files (if any) from the project directory (the directory where the project .sqlite file is saved) to the working temp location for WaterCAD V8*i* (%temp%\Bentley\HAMMER). This allows you to make a copy of the results that may exist in the model's save directory and replace the current results being worked on with them.	**Update Results from Project Directory** 

This command copies the result files that are currently being used by the model to the project directory (where the project .sqlite is stored).	**Copy Results to Project Directory** 
Open a Batch Assign Isolation Valves window where you can find the nearest pipe for each selected isolation and assign the valve to that pipe.	**Assign Isolation Valves to Pipes** 
Opens the Batch Pipe Split dialog.	**Batch Pipe Split** 
Opens the Batch Morph dialog.	**Batch Morph** 
Open the External Tools dialog box.	**Customize** 
Open the Options dialog box, which allows you to change Global settings, Drawing, Units, Labeling, and ProjectWise.	**Options** 

Zoom Toolbar

The Zoom toolbar provides access to the zooming and panning tools.



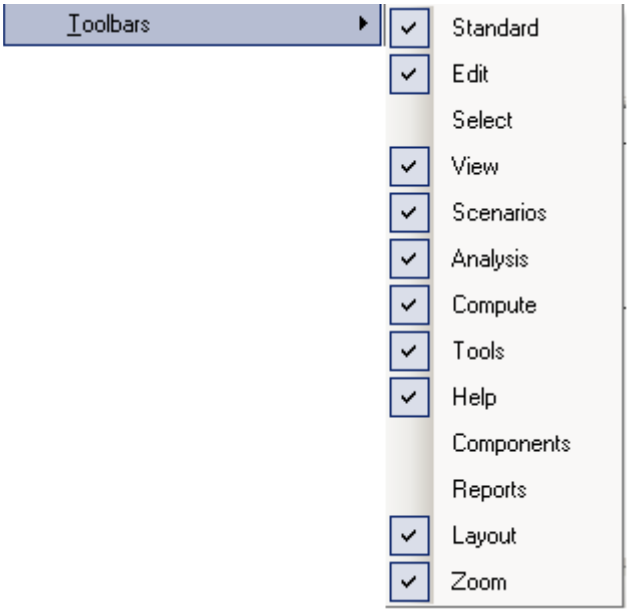
The Zoom toolbar contains the following:

To	Use
Set the view so that the entire model is visible in the drawing pane.	**Zoom Extents** 
Activate the manual zoom tool, where you can specify a portion of the drawing to enlarge.	**Zoom Window** 
Magnify the current view in the drawing pane.	**Zoom In** 
Reduce the current view in the drawing pane.	**Zoom Out** 
Enable the realtime zoom tool, which allows you to zoom in and out by moving the mouse while the left mouse button is depressed.	**Zoom Realtime** 
Open up the Zoom Center dialog box where you can set X and Y coordinates and the percentage of Zoom.	**Zoom Center** 
Enable you to zoom to specific elements in the drawing. You must select the elements to zoom to before you select the tool.	**Zoom Selection** 

Return the zoom level to the most recent previous setting.	**Zoom Previous** 
Reset the zoom level to the setting that was active before a Zoom Previous command was executed. This button also does not appear in the Zoom toolbar by default.	**Zoom Next** 
Activate the Pan tool, which allows you to move the model within the drawing pane. When you select this command, the cursor changes to a hand, indicating that you can click and hold the left mouse button and move the mouse to move the drawing.	**Pan** 
Update the main window view according to the latest information contained in the Bentley WaterCAD V8*i* datastore.	**Refresh Drawing** 

Customizing WaterCAD V8*i* Toolbars and Buttons

Toolbar buttons represent Bentley WaterCAD V8*i* menu commands. Toolbars can be controlled in Bentley WaterCAD V8*i* using View > Toolbars. You can turn toolbars on and off, move the toolbar to a different location in the work space, or you can add and remove buttons from any toolbar.



To turn toolbars on

Click **View** > **Toolbars**, then click in the space to the left of the toolbar you want to turn on.

To turn toolbars off

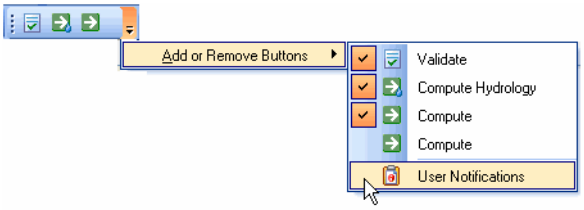
Click **View** > **Toolbars**, then click the check mark next to the toolbar you want to turn off.

To move a toolbar to a different location in the workspace

Move your mouse to the vertical dotted line on the left side of any toolbar, then drag the toolbar to the desired location. If you move a toolbar away from the other toolbar, the toolbar becomes a floating dialog box.

To add or remove a button from a toolbar

1. Click the down arrow on the end of the toolbar you want to customize. A series of submenus appear, allowing you to select or deselect any icon in that toolbar.
2. Click **Add or Remove Buttons** then move the mouse cursor to the right until all of the submenus appear, as shown as follows:



3. Click the space to left of the toolbar button you want to add. A check mark is visible in the submenu and the button opens in the toolbar.

   or

   Click the check mark next to the toolbar button you want to remove. The button will no longer appear in the toolbar.

WaterCAD V8*i* Dynamic Manager Display

Most of the features in Bentley WaterCAD V8*i* is accessed through a system of dynamic windows called managers. For example, the look of the elements is controlled in the Element Symbology manager.

The following table lists all the Bentley WaterCAD V8*i* managers, their toolbar buttons, and keyboard shortcuts.

Toolbar Button	Manager	Keyboard Shortcut
	**Scenarios**—build a model run from alternatives.	<Alt+1>
	**Alternatives**—create and manage alternatives.	<Alt+2>
	**Calculation Options**—set parameters for the numerical engine.	<Alt+3>
	**Totalizing Flow Meters**—create and manage flow meters.	<Alt+4>
	**Hydrant Flow Curves**—create and manage hydrant flow curves.	<Alt+5>
	**System Head Curves**—create and manage system flow curves.	<Alt+6>
	**Element Symbology**—control how elements look and what attributes are displayed.	<Ctrl+1>
	**Background Layers**—control the display of background layers.	<Ctrl+2>
	**Network Navigator**—helps you find nodes in your model.	<Ctrl+3>
	**Selection Sets**—create and manage selection sets.	<Ctrl+4>
	**Queries**—create SQL expressions for use with selection sets and FlexTables.	<Ctrl+5>

Toolbar Button	Manager	Keyboard Shortcut
	**Prototypes**—create and manage prototypes.	<Ctrl+6>
	**FlexTables**—display and edit tables of elements.	<Ctrl+7>
	**Graphs**—create and manage graphs.	<Ctrl+8>
	**Profiles** —draw profiles of parts of your network.	<Ctrl+9>
	**Contours**—create and manage contours.	<Ctrl+0>
	**Properties**—display properties of individual elements or managers.	<F4>
	**Refresh**—Update the main window view according to the latest information contained in the Bentley WaterCAD V8*i* datastore.	<F5>
	**Time Browser**—controls animated displays.	<F7>
	**User Notifications**—presents error and warning messages resulting from a calculation.	<F8>
	**Compute**.	<F9>

When you first start Bentley WaterCAD V8*i* , only two managers are displayed: the Element Symbology and Background Layers managers. This is the default workspace. You can display as many managers as you want and move them to any location in the Bentley WaterCAD V8*i* workspace.

To return to the default workspace

Click **View > Reset Workspace**.

- If you return to the default workspace, the next time you start Bentley WaterCAD V8*i* , you will lose any customizations you might have made to the dynamic manager display.

To open a manager

1. Do one of the following:
   - Select the desired manager from the View menu.
   - Click a manager's button on one of the toolbars.
   - Press the keyboard shortcut for the desired manager.
2. If the manager is not already docked, you can drag it to the top, left- or right-side, or bottom of the WaterCAD V8*i* window to dock it. For more information on docking managers, see Customizing Managers.

Customizing Managers

When you first start Bentley WaterCAD V8*i* , you will see the default workspace in which a limited set of dock-able managers are visible. You can decide which managers will be displayed at any time and where they will be displayed. You can also return to the default workspace any time.

There are four states for each manager:

Floating—A floating manager sits above the Bentley WaterCAD V8*i* workspace like a dialog box. You can drag a floating manager anywhere and continue to work.

You can also:

- Resize a floating manager by dragging its edges.
- Close a floating manager by clicking on the x in the top right-hand corner of the title bar.
- Change the properties of the manager by right-clicking on the title bar.
- Switch between multiple floating managers in the same location by clicking the manager's tab.
- Dock the manager by double-clicking the title bar.

Docked static—A docked static manager attaches to any of the four sides of the Bentley WaterCAD V8*i* window. If you drag a floating manager to any of the four sides of the Bentley WaterCAD V8*i* window, the manager will attach or dock itself to that side of the window. The manager will stay in that location unless you close it or make it dynamic. A vertical pushpin in the manager's title bar indicates its static state; click the pushpin to change the manager's state to dynamic. When the push pin is pointing downward (vertical push pin), the manager is docked.

You can also:

- Close a docked manager by left clicking on the x in the upper right corner of the title bar.
- Change a docked manager into a floating manager by double-clicking the title bar, or by dragging the manager to the desired location (for example, away from the side of the Bentley WaterCAD V8*i* window).
- Change a static docked manager into a dynamically docked manager by clicking the push pin in the title bar.
- Switch between multiple docked managers in the same location by clicking the manager's tab.

Docked dynamic—A docked dynamic manager also docks to any of the four sides of the Bentley WaterCAD V8*i* window, but remains hidden except for a single tab. Show a docked dynamic manager by moving the mouse over the tab, or by clicking the tab. When the manager is showing (not hidden), a horizontal pushpin in its title bar indicates its dynamic state.

You can also:

- Close a docked manager by left-clicking on the x in the upper right corner of the title bar.
- Change a docked dynamic manager into a docked static manager by clicking the push pin (converting it from vertical to horizontal).
- Switch between multiple docked managers in the same location by moving the mouse over the manager's tab or by clicking the manager's tab.

Closed—When a manager is closed, you cannot view it. Close a manager by clicking the x in the right corner of the manager's title bar. Open a manager by selecting the manager from the View menu (for example, View > Element Symbology), or by selecting the button for that manager on the appropriate toolbar.

WaterObjects Help for Model Users

Q. What is WaterObjects?

WaterObjects is a set of application and business logic upon which WaterCAD, WaterGEMS and HAMMER are built. You may think of WaterObjects as the foundation or core workings of the WaterCAD, WaterGEMS and HAMMER applications. Given that WaterObjects is essentially invisible to any user running WaterCAD, WaterGEMS and HAMMER, you might wonder why we decided to give it a special name! The reason is that the application and business logic embodied by WaterObjects is generically re-usable by external parties (and that means you too) in order to create your own custom application extensions or features. So in the most general sense WaterObjects is something that allows 3rd parties to extend the functionality of WaterCAD, WaterGEMS and HAMMER, without having to request the functionality from Bentley and then wait for it to be released in a future version of the software. While the feature is called "WaterObjects", a large majority of the feature is also applicable to Bentley storm and sewer products too. Time you invest in customizing WaterCAD or WaterGEMS for example, will have re-use potential for other Bentley Municipal Products applications.

Q. What can I do with WaterObjects?

As mentioned above WaterObjects provides the ability to write custom features to extend the existing WaterCAD, WaterGEMS and HAMMER functionality. For example, you may have some special calculation and report that you currently create in Excel since your supervisor/client prefers to see it in that format. With WaterObjects you could automate the calculation and generation of the report in Excel. In fact if you need any special additional behavior that you can't do in WaterCAD, WaterGEMS, or HAMMER with the existing functionality (make sure you looked at queries, user data extensions and the post calculation processor features) chances are that you'll able to achieve it with WaterObjects.

Q. What can't I do with WaterObjects?

As mentioned above WaterObjects represents the core workings of WaterCAD, WaterGEMS and HAMMER. As such it includes functionality to be able to read and write model data, to be able to deal with scenarios and alternatives, to be able to run computations and access results. It does not, however, provide ready access to application specific logic at least in a way that can be broken down into its constituent components. This means that you can't use WaterObjects to modify existing calculations (although you could add the calculation of additional results or a completely new computation) and you can't add new menus or buttons to the existing user interface. For example, you couldn't add a new type of graph to the graphing feature or you couldn't add a new right-click menu to the map display.

Q. How do I use WaterObjects?

The answer to this question depends on whether you are a programmer or not. If you are a programmer and are familiar with the terms API, .Net, Interface, Namespace and also with a .Net compliant language such as VB.Net, C#.Net or C++.Net you may be able to pick up WaterObjects pretty quickly, but if you are not a programmer you may need to work with one to do the programming for you.

If you need to hire a programmer (Bentley Professional Services may be able to provide you with one) then you'll need to understand some terminology to allow you to communicate with them more easily.

1. **.NET**: Microsoft's .NET Framework which comprises the Common Language Runtime, CLR, (provides an abstraction layer over the operating system), Base class libraries (pre-built code for low level programming tasks) and development frameworks and technologies (re-usable, customizable solutions for larger programming tasks). The CLR is an implementation of the CLI (Common Language Infrastructure). You or your programmer must write .NET compatible code.
2. **Interface**: A contract in software that defines the nature of the public (or external) makeup of the programming component. The analogy in hardware would be a specific kind of plug (such as DVI video) that can only connect to another plug that supports the same interface. This defines how your custom code interacts with the existing Bentley code.

   An example might be INumericalEngine which defines an interface for dealing with components that support some kind of computational engine or solver.
3. **Classes**: In object oriented programming, a class is a bite sized piece of encapsulated functionality. The class name typically identifies the core function or nature of the class (e.g., PressurePipe might represent a pressure pipe that has a Material property, a Diameter property and so on). An instance of the class represents an actual PressurePipe where as the PressurePipe class is the template or prototype that defines all PressurePipes. If we like we could take out all the uniquely PressurePipe bits of the PressurePipe class and use them to define an IPressurePipe interface.
4. **Namespace**: In .NET this is a way of providing scope to a set of programming objects that all belong in the same collective group.

   For example consider the PressurePipe class from above. Without a namespace we don't know who owns the PressurePipe, but with a namespace such as Bentley.Domain.Water.PressurePipe we know we are talking about a specific kind of PressurePipe. We won't confuse that PressurePipe with HomeHardware.DIY.PressurePipe. We'll also likely find other similar objects in the same location. e.g., Bentley.Domain.Water.PressureValve.

5. **API**: Application Programming Interface. A set of interfaces that provide access to some logical grouping of functionality. WaterObjects is a specific example of an API. You will interact with the WaterObjects API when you write your custom code.
6. **Framework**: In the context of WaterObjects the framework (or the Municipal Development Framework) is itself a sub-set of WaterObjects, providing access to the most generic features such as unit conversions, database access, scenarios and alternatives, graphing, and re-usable user interface components such as tables and lists. An example of a framework component is the FlexGridControl that lives in the Haestad.Framework.Windows.Forms.Syncfusion.Components namespace. This control (or component) is the underlying control for all the tabular based user interfaces in the Bentley Municipal Products applications.
7. **Domain**: A sub-set of the Municipal Development Framework that deals primarily with database operations and core business logic. This logic lives under the Haestad.Domain namespace. Some examples of Haestad.Domain constructs are the IDataSource interface (allowing file open/close access on model files), and the IDomainDataSet interface (allowing access to the model data set and access to managers such as the AlternativeManager (for accessing alternatives), ScenarioManager (for accessing scenarios), the DomainElementManager (for accessing domain elements), and the SupportElementManager (for accessing support elements)).
8. **Domain Element**: An element used for modeling purposes. E.g., a pipe, tank, hydrant, valve etc.
9. **Support Element**: An element used in support of modeling and usually referenced as additional state or information by a domain element. E.g., a pump definition (pump curve and efficiency curve), a valve headloss curve etc.

More information about the technical details of WaterObjects can be found in documentation that accompanies WaterObjects.

Q. How do I get WaterObjects?

WaterObjects is available for licensed users of WaterCAD, WaterGEMS and HAMMER from the Bentley Developer Network, BDN.

http://www.bentley.com/en-US/Corporate/Bentley+Partner+Program/Technology+Partners/Developers.htm

Support for WaterObjects.NET is available through the Bentley Developer Network. See the Member Guide for support options:

http://ftp2.bentley.com/dist/collateral/Web/BPP/BDNMemberGuide.pdf

For more details about getting started with WaterObjects see

http://www.bentley.com/en-US/Products/WaterGEMS/WaterObjects.NET-Bentley.htm

Q. What programming languages can I use with WaterObjects?

WaterObjects is primarily written in Microsoft.NET and therefore requires a .NET compliant language in order to be able to interoperate with WaterObjects. Your choices include:

1. VB.NET (Visual Basic for .NET)
2. C#.NET (Microsoft C#)
3. C++.NET (Microsoft C++)

In addition to these any other CLI (Common Language Infrastructure) language should be able to be used such as:

4. J# (Microsoft J#- A Java like programming language)
5. Fortran.NET
6. #Smalltalk

And many others.

For more potential examples visit http://en.wikipedia.org/wiki/List_of_CLI_languages

It should be noted that internally the Bentley Municipal Products development group predominantly use C# and C++ to develop with WaterObjects. WaterObjects itself is also predominantly written in these two languages. We do not have any direct experience with many of the other possible languages that may be used.

Typically you would choose a language that you or your programmer is most familiar with in order to maximize productivity. If possible, and all other things being equal, you'll find that Bentley will be able to support you more easily if you stick to one of the languages Bentley uses and is familiar with such as VB.NET, C# or C++.Net.

Q. How do write a WaterObjects Program that works in Microsoft Office?

Those familiar with macros and programming Microsoft Office will typically be used to using VBA (Visual Basic for Applications) to customize those applications. Since WaterObjects, however, is a .NET API, it cannot be used with VBA. To solve the problem of Microsoft Office leveraging application logic and APIs written in .NET, Microsoft introduced a technology called VSTO. The latest version of this at the time of writing is VSTO2005SE and this allows users to write add-ins for the Microsoft

Office suite that can use either VB.NET or C# as the programming language. The documentation that comes with WaterObjects includes more description on VSTO and how to use it. Note that this is a step up in complexity from regular WaterObjects.NET development.

Q. How do I design a WaterObjects Program?

Whether or not you are doing the programming yourself you'll need to base your design on what you are trying to achieve with the program. First it will be necessary to document the goals of the application. In the software development industry this is typically done from the user's point of view and is called creating "user stories". To that end, put yourself into the shoes of the end-users for your program and document the workflows that the user would expect to encounter. This can be as detailed as it needs to be including how the user would start the program, and what they do when the program is running.

Options for starting a WaterObjects program will depend on the nature of the program developed, but may include:

1. Starting from the External Tools Menu from within WaterCAD/GEMS/ HAMMER,
2. Starting from a desktop shortcut to a stand alone executable,
3. Starting some 3rd party application (such as Excel) and accessing add-in menus.

In addition to starting the program you'll need to define the inputs and the expected outputs. Inputs may include human entered input or file based input (such as a Water model, or tabular data) and output may include things like raw data, reports, graphs and tables in desired formats (e.g. an Excel spreadsheet, Oracle database or a Notepad file). In arriving at the outputs the details of any specific calculations will need to be documented. Finally, you'll need to determine where you want to store the output from your calculations. Choices for storing output may include:

1. Custom results file (binary, XML, text or other format),
2. Within a 3rd party application (such as MS Access or Excel),
3. Within WaterCAD/GEMS/HAMMER using User Data Extensions.

The above process sounds like it may be tricky, particularly when some of the answers potentially require some advance knowledge of how things are going to turn out. This is precisely why in software development an iterative development approach is commonly adopted. In an iterative approach a the overall program requirements are kept initially to a minimum and then staged in bite sized pieces with the progress of the development being demonstrated by the programmer to the stakeholders at regular intervals. This process is sometime called "Agile" software development. More can be found out about Agile development by searching on-line.

Chapter

Quick Start Lessons 2

Building a Network and Performing a Steady-State Analysis

Extended Period Simulation

Scenario Management

Reporting Results

Automated Fire Flow Analysis

Water Quality Analysis

Darwin Designer to Optimize the Setup of a Pipe Network

Darwin Designer to Optimize a Pipe Network

Scenario Energy Costs

Pressure Dependent Demands

Criticality and Segmentation

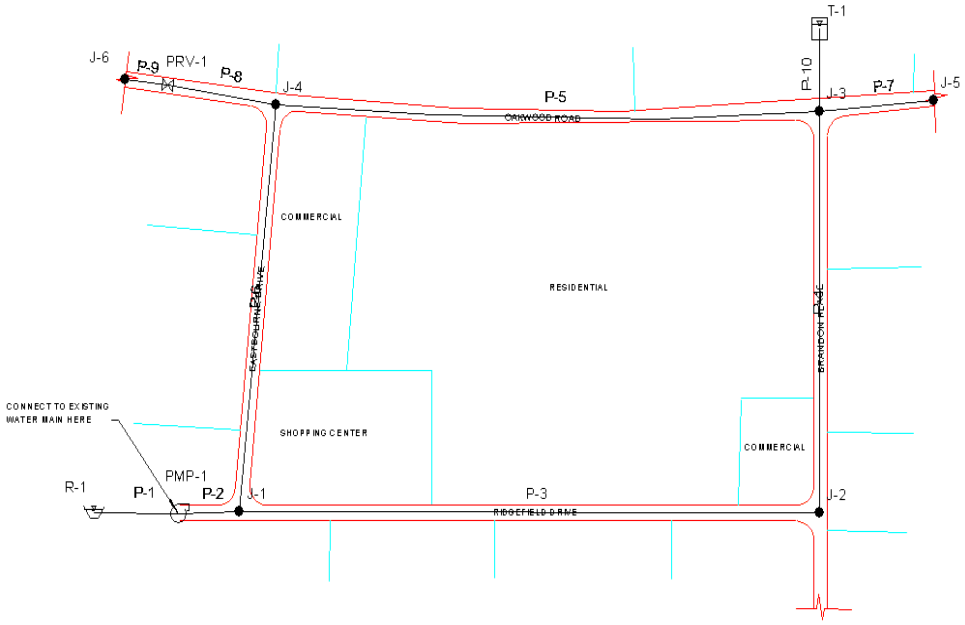
Flushing

Building a Network and Performing a Steady-State Analysis

In constructing a distribution network for this lesson, you do not need to be concerned with assigning labels to pipes and nodes, because Bentley WaterCAD V8*i* will assign labels automatically. When creating a schematic drawing, pipe lengths are entered manually. In a scaled drawing, pipe lengths are automatically calculated from the position of the pipes' bends and start and stop nodes on the drawing pane.

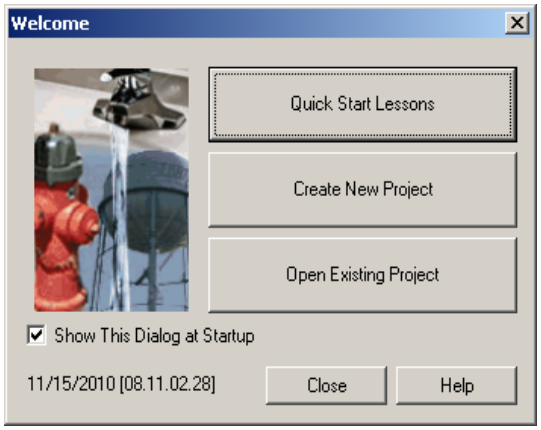
In this network, the modeling of a reservoir connected to a pump simulates a connection to the main water distribution system. Simplifying the network in this way can approximate the pressures supplied to the system at the connection under a range of demands. This type of approximation is not always applicable, and care should be taken when modeling a network in this way. It is more accurate to trace the network back to the source.

In this lesson, you will create and analyze the network shown below. You will use a scaled background drawing for most of the network; however, four of the pipes are not to scale and will have user-defined lengths.

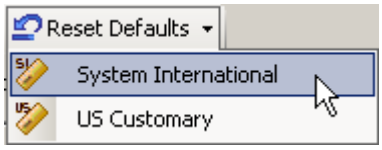


Step 1: Create a New Project File

1. From the welcome dialog, click Create New Project and an untitled project opens. Or click File > New to create a new project.

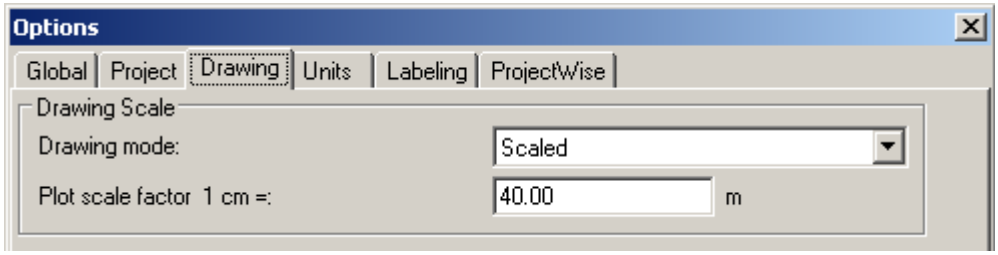


2. Click the Tools menu and select the Options command. Click the Units tab. Since you will be working in System International units, click the Reset Defaults button and select System International.



3. Verify that the Default Unit System for New Project is set to System International. If not, select from the menu.

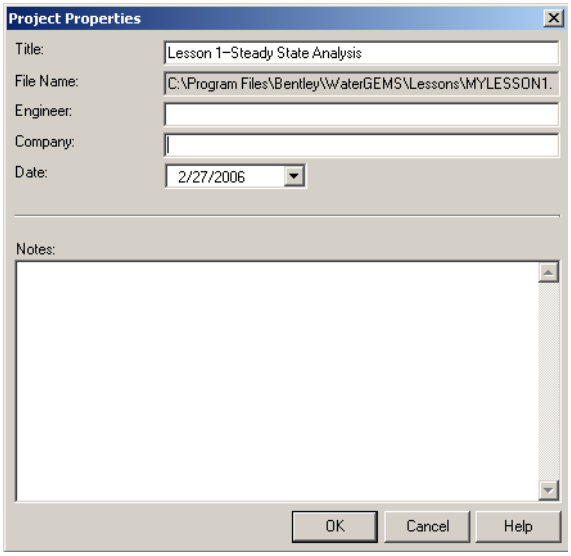
4. Click the Drawing tab to make sure Drawing Mode is set to Scaled.



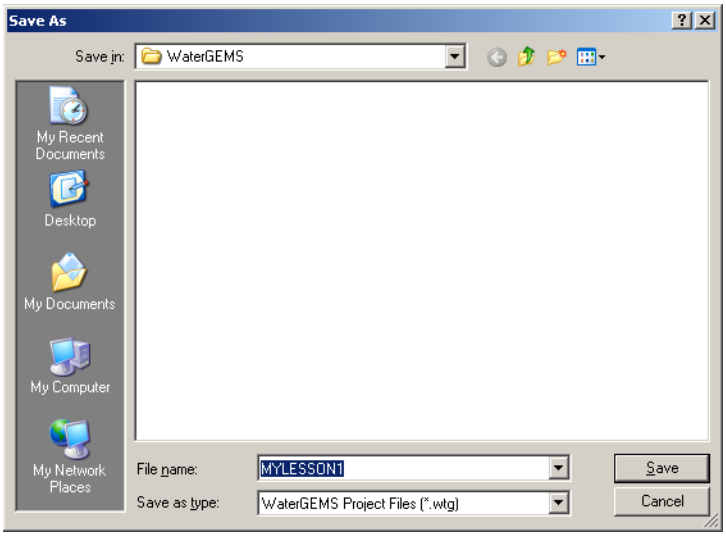
5. Set the Plot Scale Factor 1 cm = **40** m.

6. Click OK.

7. Set up the project. Choose File > Project Properties and name the project **Lesson 1—Steady State Analysis** and click OK.



8. Choose **File > Save as**. In the Save File As dialog box, browse to the **My Documents/Bentley/WaterGEMS** folder.

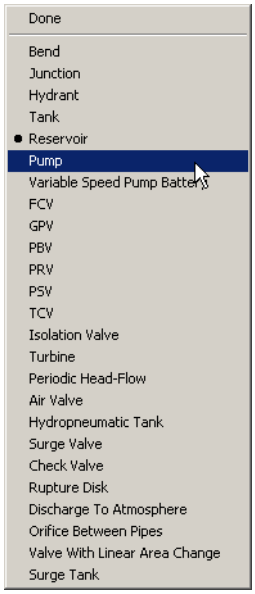


9. Enter the file name **MYLESSON1.WTG** for your project, and click **Save**.

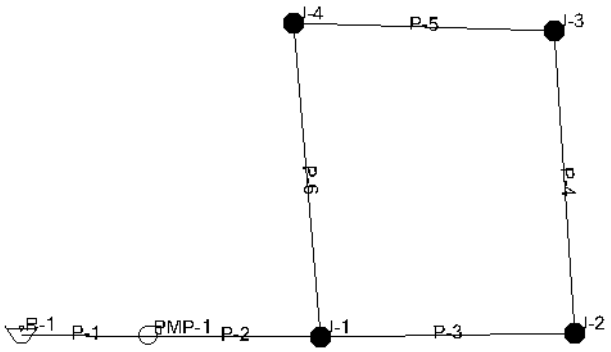
Step 2: Lay out the Network

1. Select Pipe  from the layout toolbar.
2. Move the cursor on the drawing pane and right click to select Reservoir from the menu or click  from the toolbar.

3. Click to place R-1.
4. Move the cursor to the location of pump P-1. Right-click and select Pump from the shortcut menu.



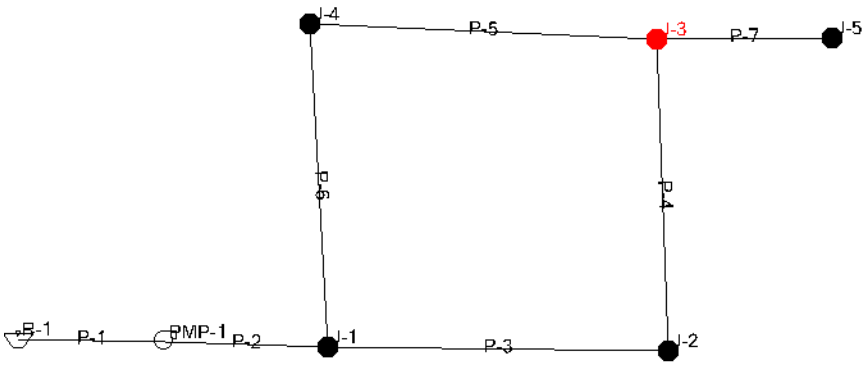
5. Click to place it.
6. Right click to select Junction from the menu and click to place J-1.
7. Click to place junctions J-2, J-3, and J-4.
8. Click on J-1 to finish.
9. Right-click and choose Done from the menu.



10. Create J-5.
    a. Select the **Pipe** layout tool again.
    b. Click junction J-3.

c. Move the cursor to the location of J-5, and click to insert the element.

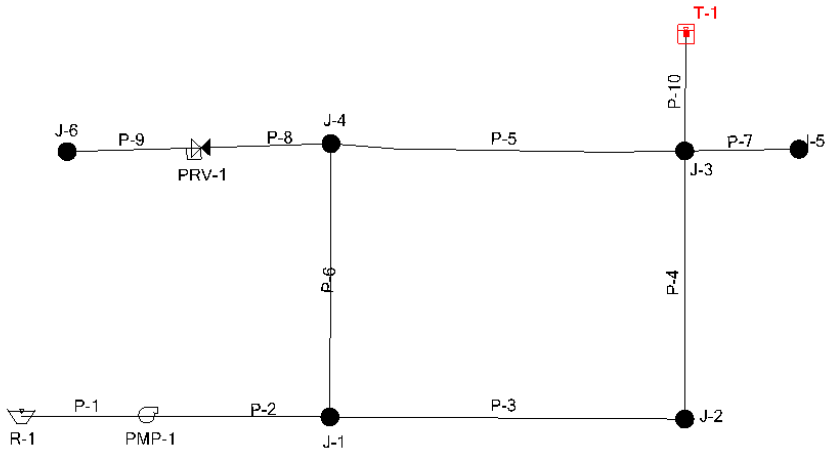
d. Right-click and select **Done**.



11. Lay out junction **J-6** and the **PRV** by selecting the **Pipe** layout tool and placing the elements in their appropriate locations.

    Be sure to lay out the pipes in numerical order (P-7 through P-9), so that their labels correspond to the labels in the diagram. Right-click and select **Done** from the menu to terminate the Pipe Layout command.

12. Insert the tank, T-1, using the **Pipe** layout tool. Pipe **P-10** should connect the tank to the network if you laid out the elements in the correct order.



13. Save the network by clicking **Save**  or choose File > Save.

Step 3: Enter and modify data

- **Dialog Boxes**—You can use the Select tool  and double-click an element to bring up its Properties editor.

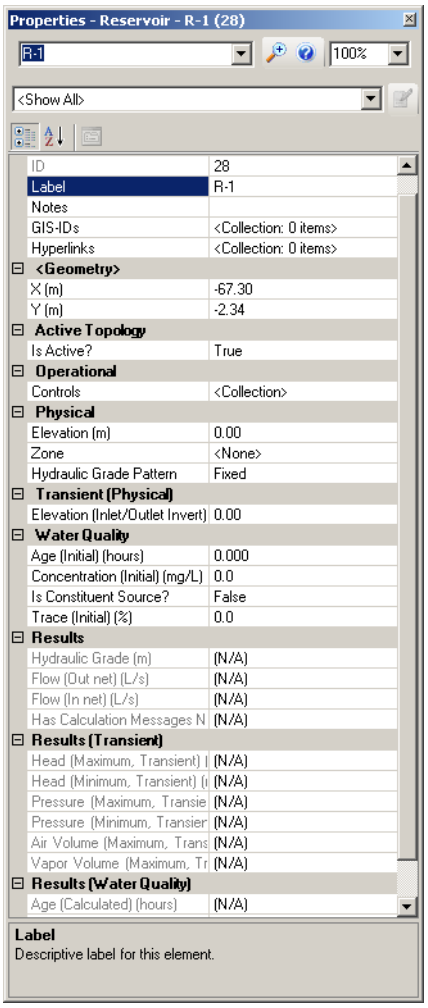
- **FlexTables**—You can click FlexTables  to bring up dynamic tables that allow you to edit and display the model data in a tabular format. You can edit the data as you would in a spreadsheet.

- **User Data Extensions**—The User Data Extensions  feature (Tools menu > User Data Extensions) allows you to import and export element data directly from XML files.
- **Alternative Editors**—Alternatives are used to enter data for different “What If?” situations used in Scenario Management.

Entering Data through the Properties Editor

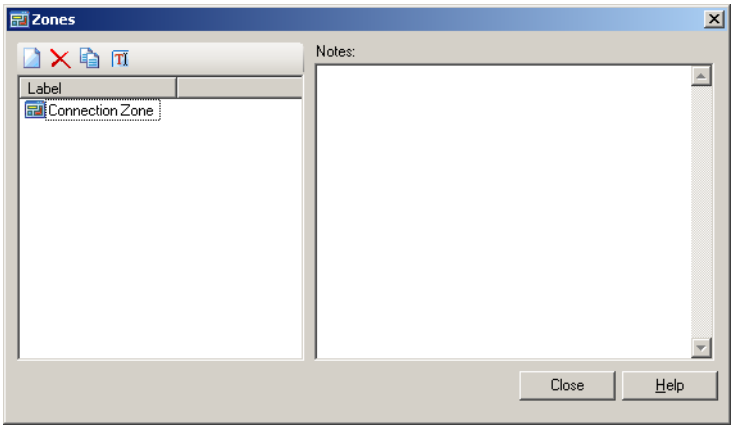
To access an element's property editor, double-click the element.

1. Open the **Reservoir Editor** for reservoir R-1.



2. Enter the Elevation as 198 (m).
3. Set Zone to **Connection Zone**.
   a. Click the **Zone** menu and select the **Edit Zones** command, which will open the **Zone Manager**.
   b. Click **New**  .

c. Enter a label for the new pressure zone called Connection Zone.

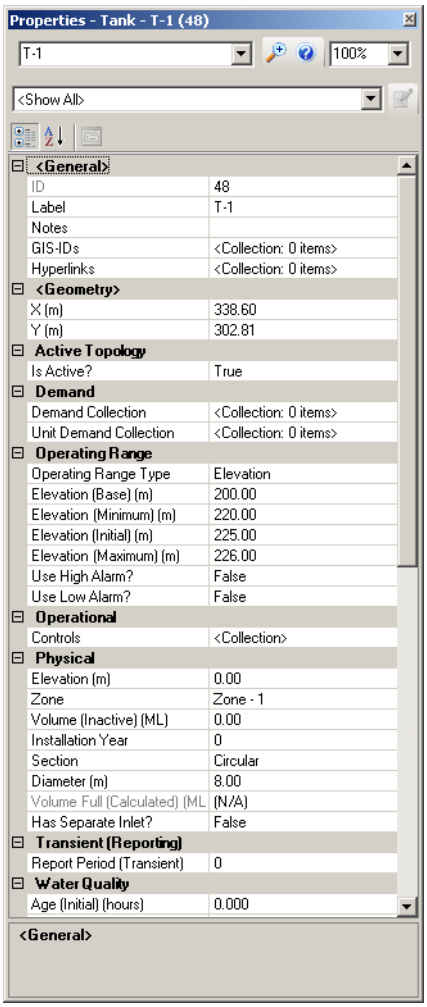


d. Click Close.

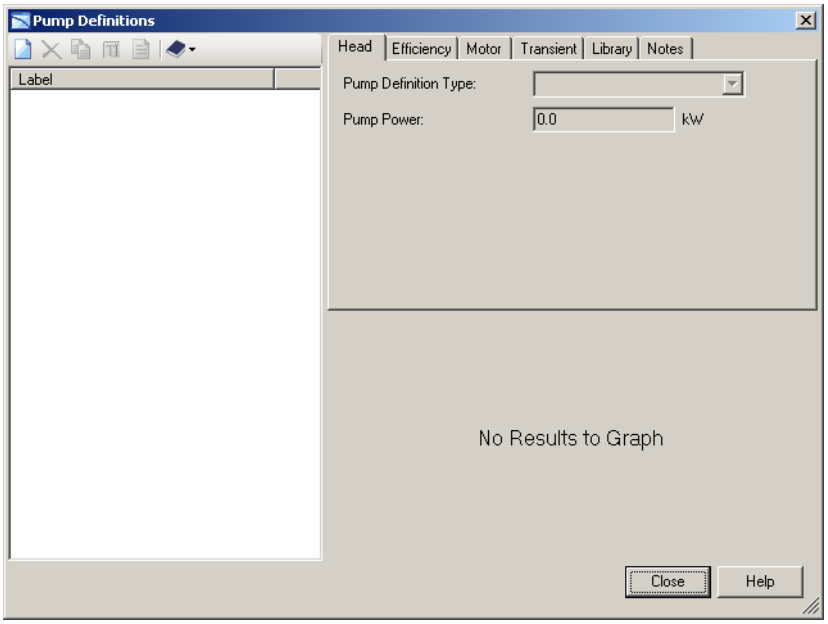
e. Select the zone you just created from the Zone menu.

4. Click tank **T-1** in the drawing to highlight it and enter the following:
   Elevation (Base) = 200 m
   Elevation (Minimum) = 220 m
   Elevation (Initial) = 225 m
   Elevation (Maximum) = 226 m
   Diameter = 8 m

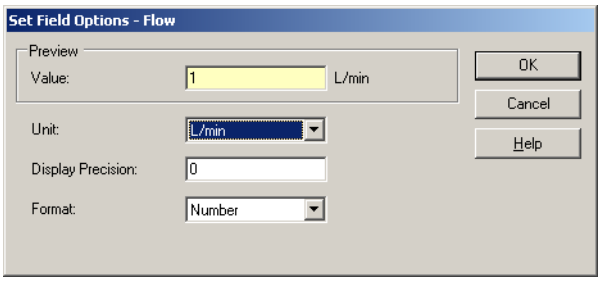
Section = Circular
Set the Zone to Zone 1 (You will need to create Zone-1 in the Zone Manager as described above.)



5. Click pump PMP-1 in the drawing to highlight it.
   a. Enter 193 (m) for the Elevation.
   b. Click in the Pump Definition field and click on Edit Pump Definitions from the drop-down list to open the Pump Definitions manager.



   c. Click New  to create a new pump definition.
   d. Leave the default setting of Standard (3 Point) in the Pump Definition Type menu.
   e. Right click on the Flow column and select the Units and Formatting command.
   f. In the Set Field Options box set the Units to L/min.

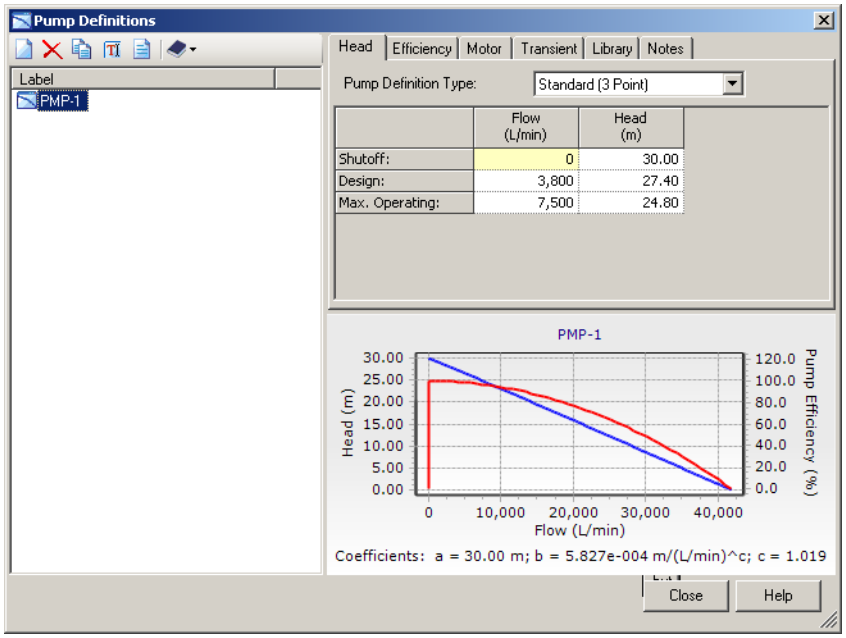


   g. Click OK.

h. Enter the following information:

	Flow (L/min)	Head (m)
Shutoff:	0.00	30.00
Design:	3,800.00	27.40
Max. Operating:	7,500.00	24.80

i. Highlight **Pump Definition - 1** and click the **Rename** button. Change the name to **PMP-1**.

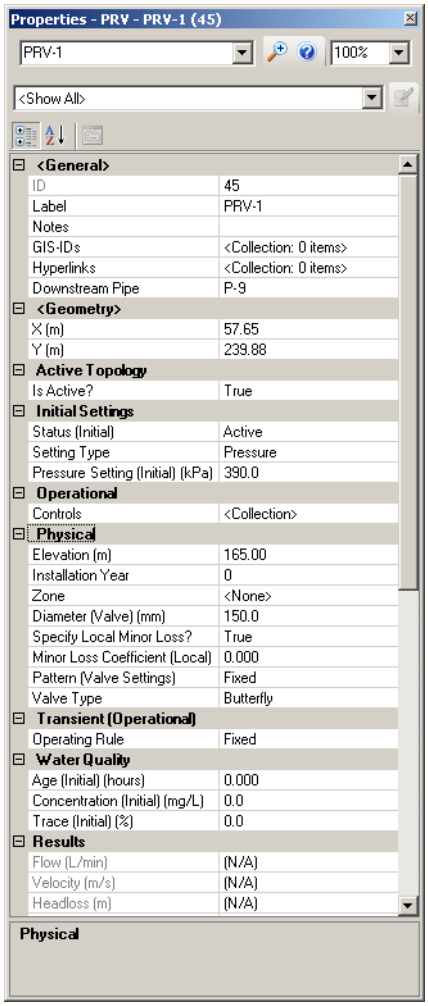


j. Click **Close**.

k. In the Properties editor, select **PMP-1** from the **Pump Definition** menu.

6. Highlight valve **PRV-1** in the drawing. Enter in the following data:
Status (Initial) = Active
Setting Type= Pressure
Pressure Setting (Initial)= 390 kPa
Elevation =165 m

Diameter (Valve) = 150 mm
Create Zone-2 and set the valve's Zone field to Zone-2.

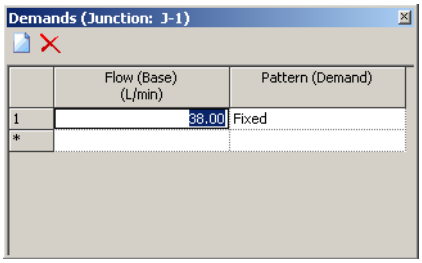


7. Enter the following data for each of the junctions. Leave all other fields set to their default values.

Junction	Elevation (m)	Zone	Demand (l/min)
J-1	184	Zone-1	38
J-2	185	Zone-1	31
J-3	184	Zone-1	34
J-4	183	Zone-1	38
J-5	185.5	Zone-1	350
J-6	165	Zone-2	356

In order to add the demand, click the ellipsis  in the Demand Collection

field to open the Demand box, click New, and type in the value for Flow (L/min).



Specify user-defined lengths for pipes P-1, P-7, P-8, P-9 and P-10.

a. Click pipe **P-1** to open the Pipe Editor.

b. Set **Has User Defined Length?** to **True**. Then, enter a value of **0.01 m** in the **Length (User Defined)** field.

   Note that the default display precision will cause only "0" to be displayed. To change display precision, right click the column heading and select **Units and Formatting** to open the **Set Field Options** dialog; from here you can change the **Display Precision** to the desired value and click **OK**.

   Since you are using the reservoir and pump to simulate the connection to the main distribution system, you want headloss through this pipe to be negligible. Therefore, the length is very small and the diameter will be large.

c. Enter **1000 mm** as the diameter of P-1.

Physical	
Zone	<None>
Diameter (mm)	1,000.0
Material	Ductile Iron
Hazen-Williams C	130.0
Has User Defined Length?	True
Length (User Defined) (m)	0.01
Has Check Valve?	False
Specify Local Minor Loss?	True
Minor Loss Coefficient (Local)	0.000
Installation Year	0

d. Change the lengths (but not the diameters) of pipes P-7 through P-10 using the following user-defined lengths:
   P7 = Length (User Defined): 400 m
   P8 = Length (User Defined): 500 m
   P9 = Length (User Defined): 31 m
   P-10 = Length (User Defined): 100 m

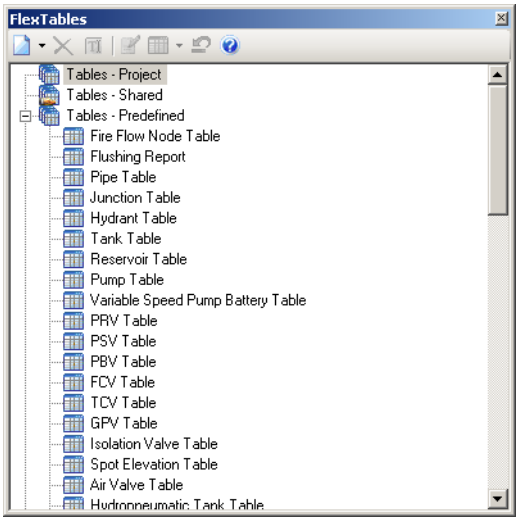
e. Close the Properties editor.

Step 4: Entering Data through FlexTables

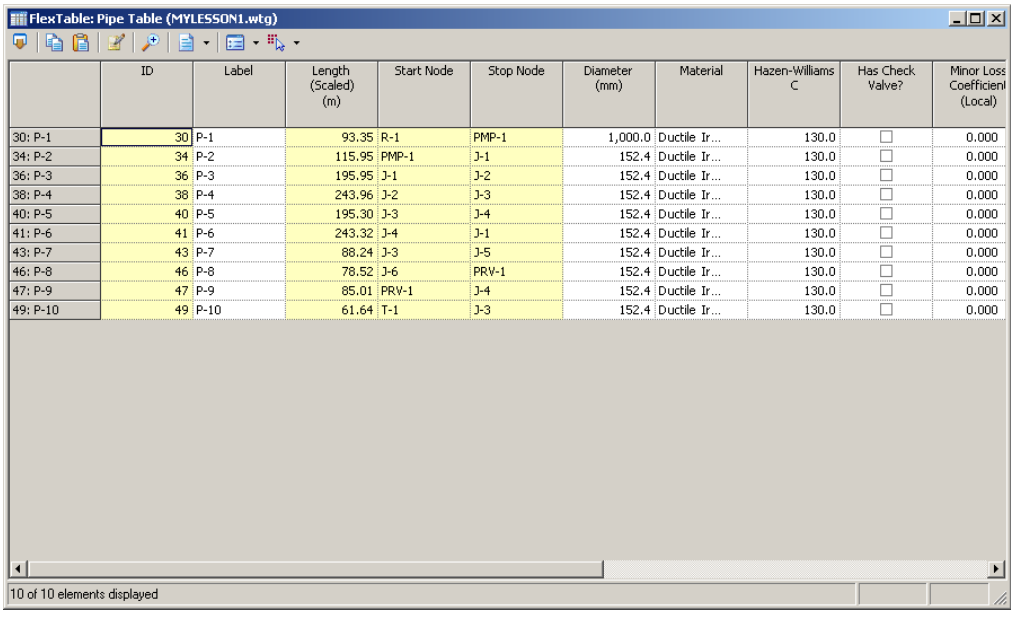
It is often more convenient to enter data for similar elements in tabular form, rather than to individually open the properties editor for an element, enter the data, and then select the next element. Using FlexTables, you can enter the data as you would enter data into a spreadsheet.

To use FlexTables

1. Click FlexTables  or choose View > FlexTables.



2. Double-click **Pipe Table**. Fields that are white can be edited, yellow fields can not.

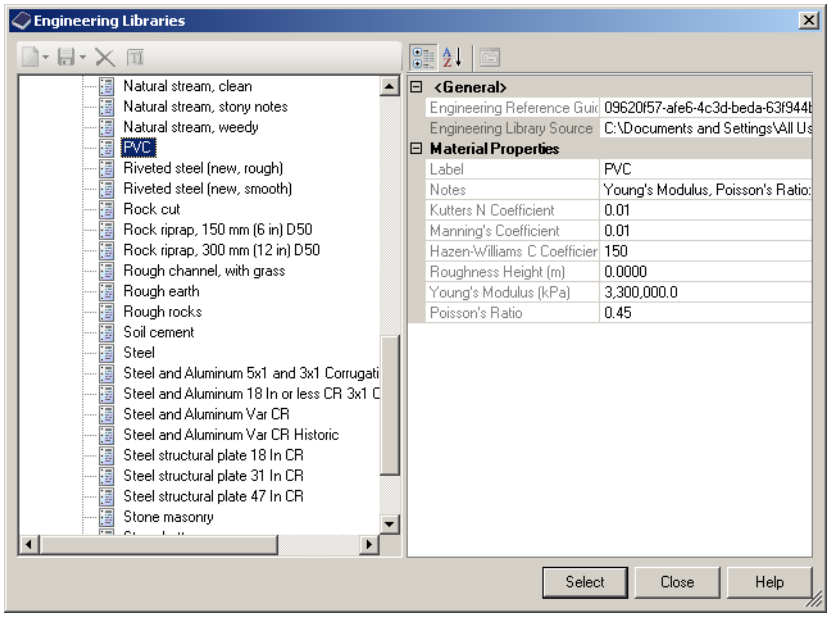


	ID	Label	Length (Scaled) (m)	Start Node	Stop Node	Diameter (mm)	Material	Hazen-Williams C	Has Check Valve?	Minor Loss Coefficient (Local)
30: P-1	30	P-1	93.35	R-1	PMP-1	1,000.0	Ductile Ir...	130.0	☐	0.000
34: P-2	34	P-2	115.95	PMP-1	J-1	152.4	Ductile Ir...	130.0	☐	0.000
36: P-3	36	P-3	195.95	J-1	J-2	152.4	Ductile Ir...	130.0	☐	0.000
38: P-4	38	P-4	243.96	J-2	J-3	152.4	Ductile Ir...	130.0	☐	0.000
40: P-5	40	P-5	195.30	J-3	J-4	152.4	Ductile Ir...	130.0	☐	0.000
41: P-6	41	P-6	243.32	J-4	J-1	152.4	Ductile Ir...	130.0	☐	0.000
43: P-7	43	P-7	88.24	J-3	J-5	152.4	Ductile Ir...	130.0	☐	0.000
46: P-8	46	P-8	78.52	J-6	PRV-1	152.4	Ductile Ir...	130.0	☐	0.000
47: P-9	47	P-9	85.01	PRV-1	J-4	152.4	Ductile Ir...	130.0	☐	0.000
49: P-10	49	P-10	61.64	T-1	J-3	152.4	Ductile Ir...	130.0	☐	0.000

3. For each of the pipes, enter the diameter and the pipe material as follows:

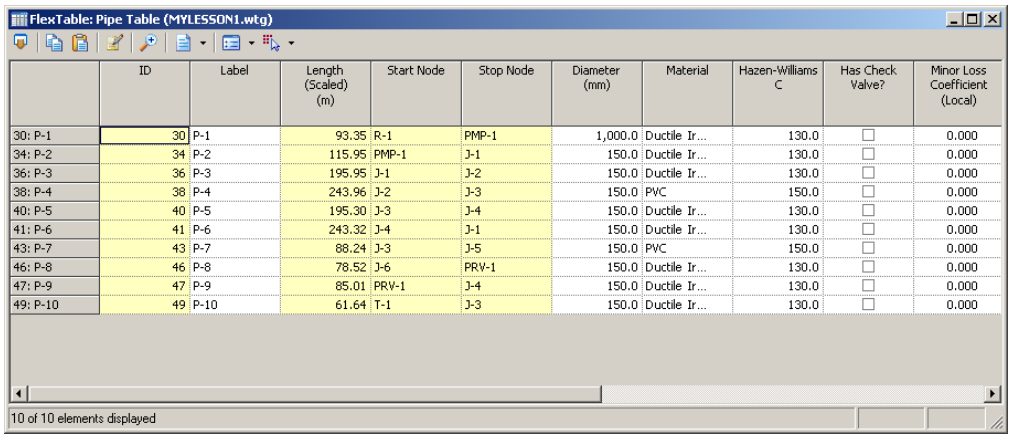
Pipe	Material	Diameter (mm)
P-1	Ductile Iron	1000
P-2	Ductile Iron	150
P-3	Ductile Iron	150
P-4	PVC	150
P-5	Ductile Iron	150
P-6	Ductile Iron	150
P-7	PVC	150
P-8	Ductile Iron	150
P-9	Ductile Iron	150
P-10	Ductile Iron	150

4. In order to enter the material type, click the ellipsis  to open the Engineering Libraries box. Click on Material Libraries > Material Libraries.xml and then click the appropriate material type and then click Select.



5. Notice that the C values for the pipes will be automatically assigned to preset values based on the material; however, these values could be modified if a different coefficient were required.

6. Leave the other data set to their default values. Click to exit the table when you are finished.



	ID	Label	Length (Scaled) (m)	Start Node	Stop Node	Diameter (mm)	Material	Hazen-Williams C	Has Check Valve?	Minor Loss Coefficient (Local)
30: P-1	30	P-1	93.35	R-1	PMP-1	1,000.0	Ductile Ir...	130.0		0.000
34: P-2	34	P-2	115.95	PMP-1	J-1	150.0	Ductile Ir...	130.0		0.000
36: P-3	36	P-3	195.95	J-1	J-2	150.0	Ductile Ir...	130.0		0.000
38: P-4	38	P-4	243.96	J-2	J-3	150.0	PVC	150.0		0.000
40: P-5	40	P-5	195.30	J-3	J-4	150.0	Ductile Ir...	130.0		0.000
41: P-6	41	P-6	243.32	J-4	J-1	150.0	Ductile Ir...	130.0		0.000
43: P-7	43	P-7	88.24	J-3	J-5	150.0	PVC	150.0		0.000
46: P-8	46	P-8	78.52	J-6	PRV-1	150.0	Ductile Ir...	130.0		0.000
47: P-9	47	P-9	85.01	PRV-1	J-4	150.0	Ductile Ir...	130.0		0.000
49: P-10	49	P-10	61.64	T-1	J-3	150.0	Ductile Ir...	130.0		0.000

10 of 10 elements displayed

Step 5: Run a Steady-State Analysis

1. Click  to open the **Calculation Options** manager.

2. Double-click **Base Calculation Options** under the **Steady-State/EPS Solver** heading to open the **Properties** editor. Make sure that the **Time Analysis Type** is set to **Steady State**.

Calculation Times	
Simulation Start Date	1/1/2000
Time Analysis Type	Steady State
Use simple controls during ste	True
Is EPS Snapshot?	False
Start Time	12:00:00 AM

   Close the **Properties** editor and the **Calculation Options** manager.

3. Click Compute  to analyze the model.

4. When calculations are completed, the Calculation Summary and User Notifications open.

5. A blue light is an informational message, a green light indicates no warnings or issues, a yellow light indicates warnings, and a red light indicates issues.

6. Click to close the Calculation Summary and User Notifications dialogs.

7. Click to Save  project.

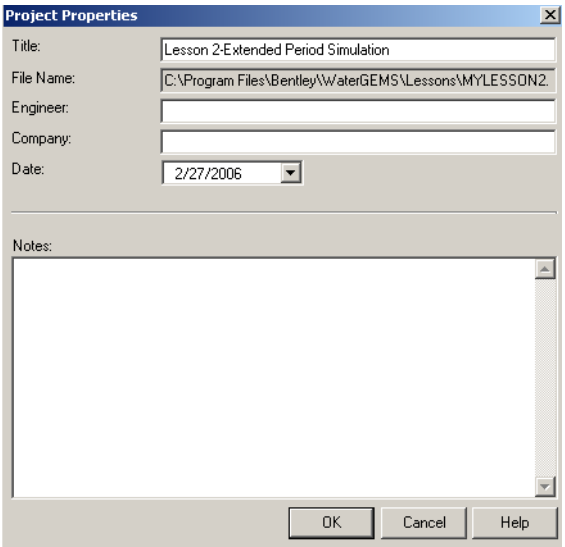
Extended Period Simulation

This lesson will illustrate how Bentley WaterCAD V8*i* can model the behavior of a water distribution system through time using an extended period simulation (EPS). An EPS can be conducted for any duration you specify. System conditions are computed over the given duration at a specified time increment. Some of the types of system behaviors that can be analyzed using an EPS include how tank levels fluctuate, when pumps are running, whether valves are open or closed, and how demands change throughout the day.

This lesson is based on the project created in Building a Network and Performing a Steady-State Analysis. If you have not completed it, then open the project **LESSON2.WTG** from the Bentley\WaterGEMS\Lesson directory. If you completed Lesson 1, then you can use the **MYLESSON1** file you created.

To open the existing project

1. Open **MYLESSON1.WTG.**
2. After you have opened the file, choose **File > Save As**.
3. Enter the filename **MYLESSON2** and click **Save**.
4. Choose **File > Project Properties**, and change the Project Title to **Lesson 2—Extended Period Simulation**.



5. Click OK.

Step 1: To Create Demand Patterns

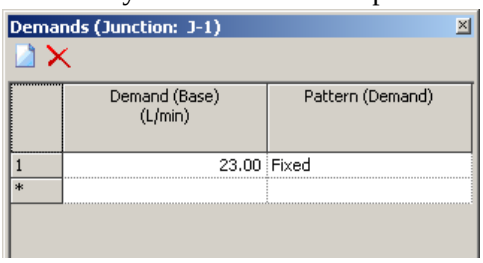
Water demand in a distribution system fluctuates over time. For example, residential water use on a typical weekday is higher than average in the morning before people choose work, and is usually highest in the evening when residents are preparing dinner, washing clothes, etc. This variation in demand over time can be modeled using demand patterns. Demand patterns are multipliers that vary with time and are applied to a given base demand, most typically the average daily demand.

In this lesson, you will be dividing the single fixed demands for each junction node in Lesson 1 into two individual demands with different demand patterns. One demand pattern will be created for residential use, and another for commercial use. You will enter demand patterns at the junction nodes through the junction editors.

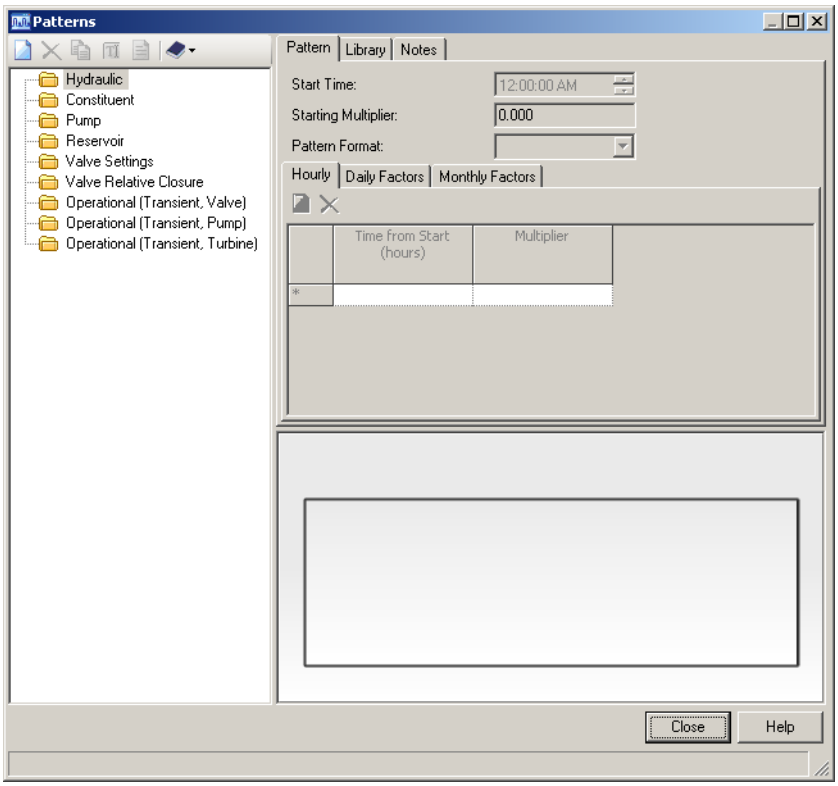
1. Open the Properties editor for Junction J-1 (double-click junction J-1) and click the ellipsis  in the Demand Collection field to open the Demands box.



2. By default, the demand pattern is set to Fixed. Enter **23** l/min for Flow. (If field already has a number from previous lesson, type over it.



3. Click in the Pattern (Demand) field and click the ellipsis  to open the Patterns manager.



4. Highlight the Hydraulic folder and click New  to create a hydraulic pattern.

   a. Rename the new pattern **Residential**.

   b. Leave the Start Time **12:00:00 AM**.

   c. Enter 0.5 as the Starting Multiplier.

   d. In the Pattern Format menu select **Stepwise**.

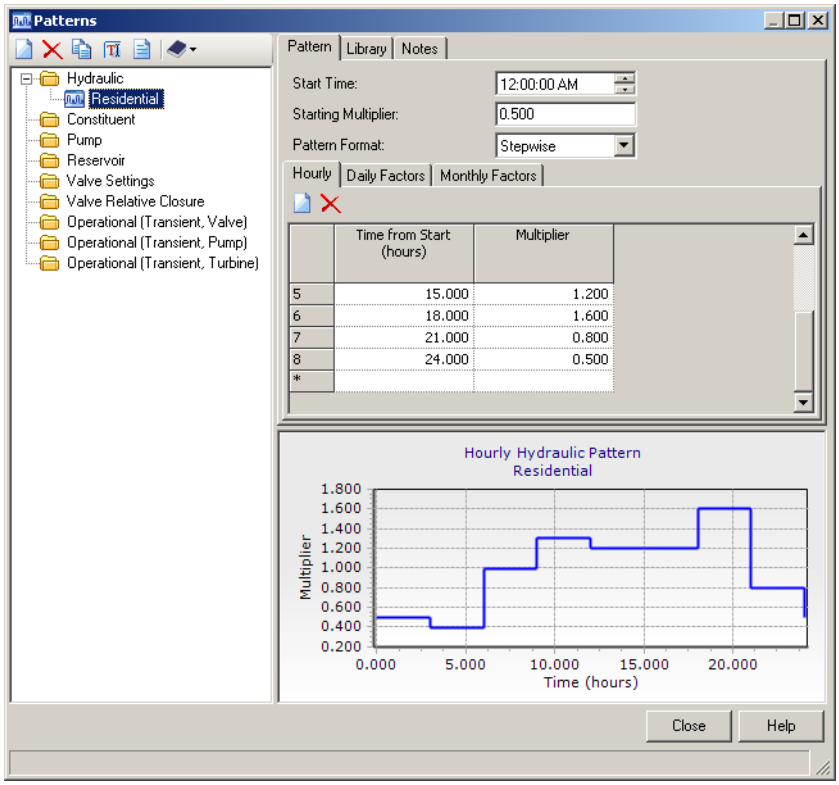
      The resulting demand pattern will have multipliers that remain constant until the next pattern time increment is reached.

      Note that the multiplier for the last time given (24 hrs.) must be the same as the Starting Multiplier (0.5). These values are equal because the demand curve represents a complete cycle, with the last point the same as the first.

e. Under the Hourly tab, enter the following times and multipliers:

Time from Start	Multiplier
3	.4
6	1
9	1.3
12	1.2
15	1.2
18	1.6
21	.8
24	.5

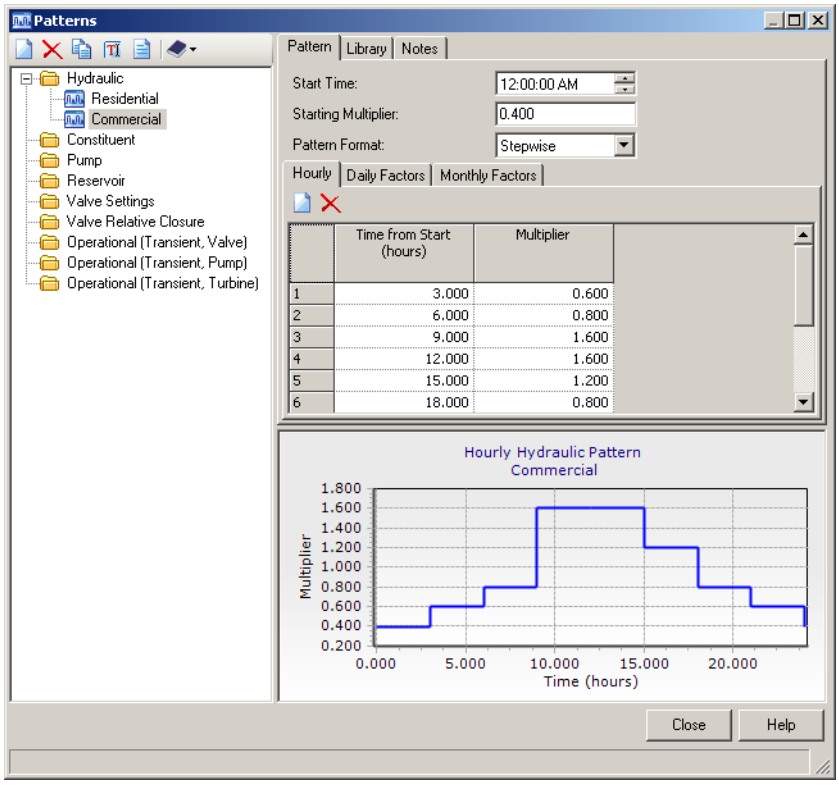
f. The Residential Patterns dialog box should look like the following:



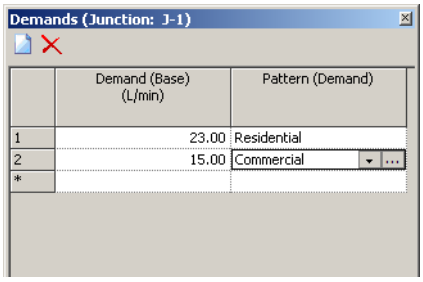
5. Click New  to create a new hydraulic pattern for commercial demands.
   a. Rename the new pattern **Commercial**.
   b. Leave the Start Time **12:00:00 AM**.
   c. Enter 0.4 as the Starting Multiplier.
   d. In the Pattern Format menu select **Stepwise**.
   e. Under the Hourly tab, enter the following times and multipliers:

Time from Start	Multiplier
3	.6
6	.8
9	1.6
12	1.6
15	1.2
18	.8
21	.6
24	.4

f. The Commercial Patterns dialog box should look like the following:



6. Click Close.
7. In the Demands dialog box, in the Pattern field, select Residential from the menu.
8. In the second row, enter a flow of 15 l/min and select Commercial as the pattern for this row.



9. Close the Demands dialog box.

10. Choose Demand Collection in the properties for junctions J-2, J-3, J-4, J-5 and J-6 and enter the following demand data using the **Residential** and **Commercial** demand patterns already created.

Junction	Residential Demand (l/min)	Commercial Demand (l/min)
J-2	23	8
J-3	23	11
J-4	23	15
J-5	350	N/A
J-6	280	76

11. Now, you will set up an additional demand pattern to simulate a three-hour fire at node J-6.

    a. In the Demand Collection field for J-6, click the ellipsis  to insert an additional Flow of 2000 l/min in row three of the Demands table.

    b. Click the Pattern column for row three and select the ellipsis  to open the Pattern Manager.

    c. Click New  to create a new Hydraulic pattern.

    d. Rename the new pattern **3-Hour Fire**

    e. Leave the Start Time **12:00:00 AM**

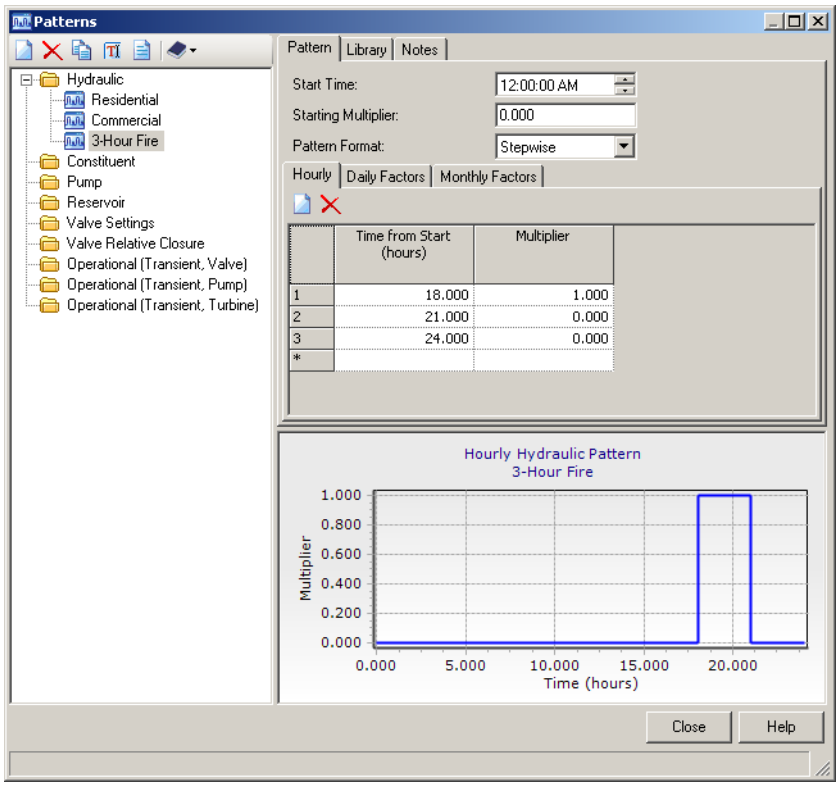
    f. Enter 0.00 as the Starting Multiplier.

    g. Select the Stepwise format.

    h. Under the Hourly tab, enter the following times and multipliers:

Time from Start	Multiplier
18	1
21	0
24	0

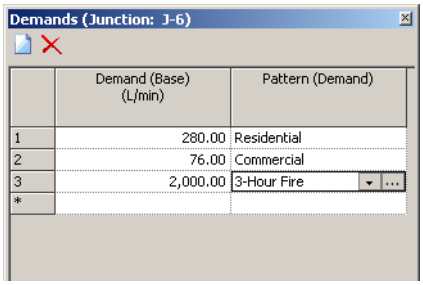
i. After you have filled in the table, look at the Graph in the lower section of the Patterns box.



j. The value of the multiplier is zero, except for the period between 18 and 21 hours, when it is 1.0. Since the input the demand as 2000 l/min., the result will be a 2000 l/min. fire flow at junction J-6 between hours 18 and 21.

k. Click Close.

12. Select the new pattern, **3-Hour Fire**, from the Pattern selection box in row three of the demands table.



13. Close the Demands dialog box.

14. Close the Properties editor.

Step 2: To run an Extended Period Simulation (EPS)

1. Click Calculation Options  to open the Calculation Options dialog.
2. Double-click Base Calculation Options under Steady State/EPS Solver to open the properties manager and select EPS from the Time Analysis Type menu.

Calculation Times	
Simulation Start Date	1/1/2000
Time Analysis Type	EPS
Start Time	12:00:00 AM
Duration (hours)	24.000
Hydraulic Time Step (hours)	1.000
Reporting Time Step	<All>

3. Click Compute  to analyze the model.
4. When there are errors or warnings then the User Notifications dialog box opens. A blue light is an informational message, a yellow light indicates warnings, and a red light indicates issues.
5. Close the User Notifications dialog box and other open dialogs..
6. Click Save  or choose File > Save to save the project.

Scenario Management

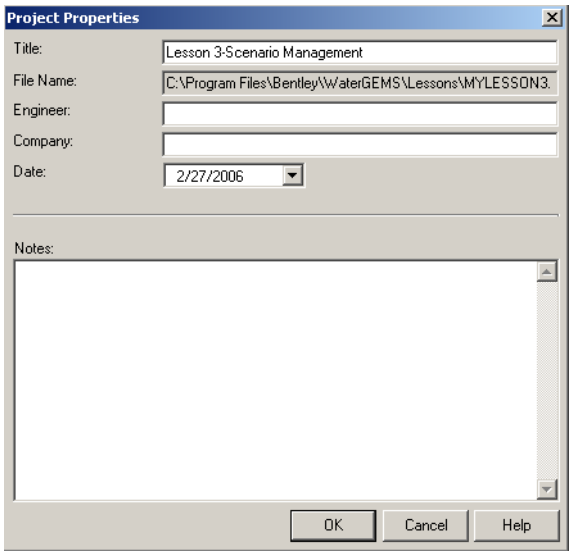
One of the many project tools in Bentley WaterCAD V8*i* is Scenarios Management. Scenarios allow you to calculate multiple “What If?” situations in a single project file. You may wish to try several designs and compare the results, or analyze an existing system using several different demand alternatives and compare the resulting system pressures.

A scenario is a set of Alternatives, while alternatives are groups of actual model data. Scenarios and alternatives are based on a parent/child relationship where a child scenario or alternative inherits data from the parent scenario or alternative.

In Lessons 1 and 2, you constructed the water distribution network, defined the characteristics of the various elements, entered demands and demand patterns, and performed steady-state and extended period simulations. In this lesson, you will set up the scenarios needed to test four “What If?” situations for our water distribution system. These “What If?” situations will involve changing demands and pipe sizes. At the end of the lesson, you will compare all of the results using the Scenario Comparison tool.

To open the existing project

1. This lesson is based on the project created in the Extended Period Simulation lesson. If you have not completed it, then open the project LESSON3.WTG from the Bentley\WaterGEMS\Lesson directory. If you completed Lesson 2, then you can use the MYLESSON2 file you created.
2. After you have opened the file, choose **File > Save As**.
3. Enter the filename **MYLESSON3** and click **Save**.
4. Choose **File > Project Properties**, and change the Project Title to **Lesson 3—Scenario Management**.

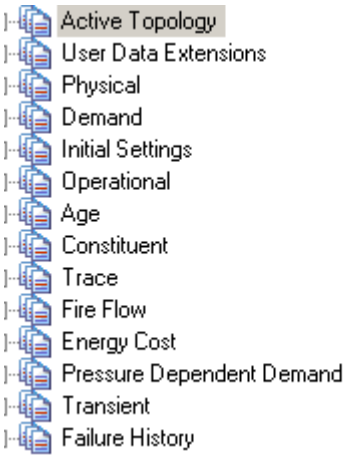


5. Click OK.

Step 1: Create a New Alternative

First, you need to set up the required data sets, or alternatives. An alternative is a group of data that describes a specific part of the model.

There are 14 alternative types:



In this example, you need to set up a different physical or demand alternative for each design trial you want to evaluate. Each alternative will contain different pipe size or demand data.

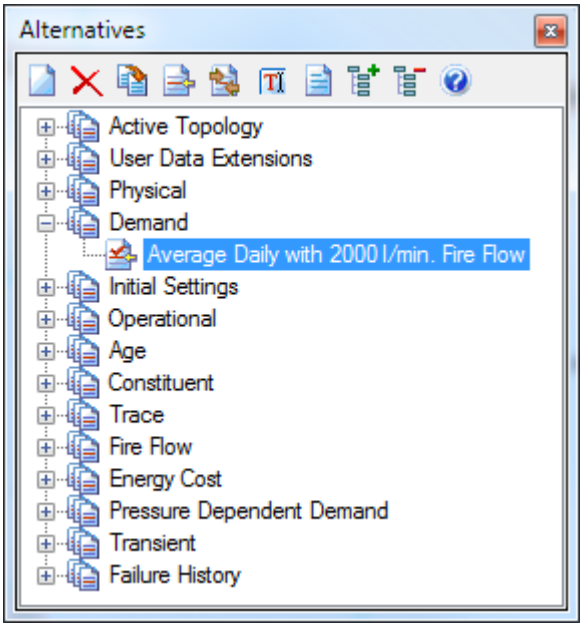
In Bentley WaterCAD V8*i* , you create families of alternatives from base alternatives. Base alternatives are alternatives that do not inherit data from any other alternative. Child alternatives can be created from the base alternative. A Child alternative inherits the characteristics of its parent, but specific data can be overridden to be local to the child. A child alternative can, in turn, be the parent of another alternative.

1. Click **Analysis** > **Alternatives** or click  .
2. Click to open the Demand alternative. The Base Demand alternative contains the demands for the current distribution system.

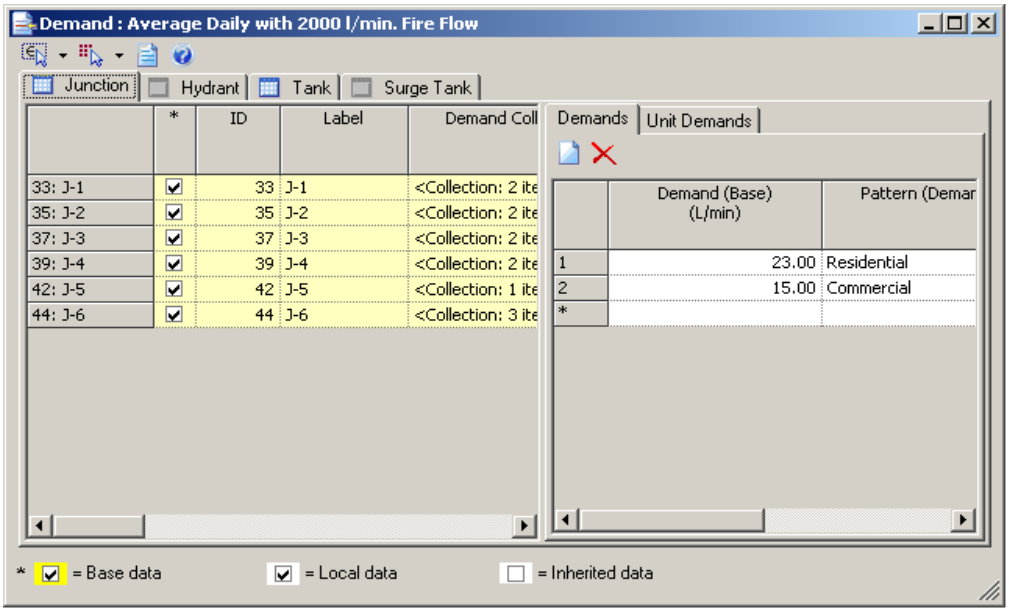
3. Change the default demand name.

   a. Click Rename  or right click to Rename.

   b. Enter the new name, **Average Daily with 2000 l/min. Fire Flow**.



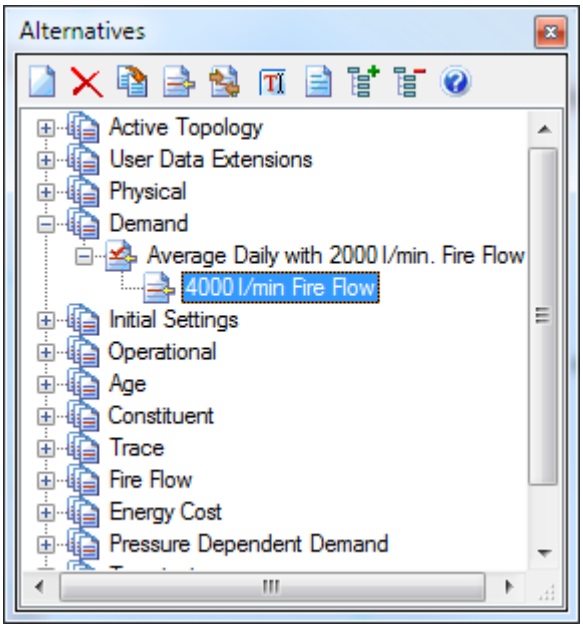
   c. Double-click on the alternative to open the Demand alternative manager.



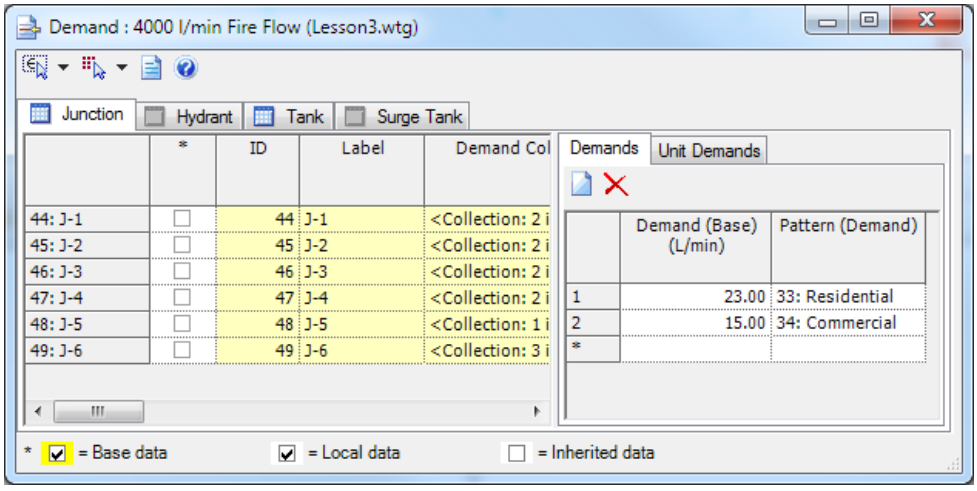
4. Now you should add a child of the base-demands alternative, because the new alternative will inherit most data. Then, you can locally change the data that you want to modify. You will modify the existing demand data by increasing the fire flow component at node J-6 from 2000 l/min. to **4000 l/min**.

   a. In the Alternatives manager, right-click the **Average Daily with 2000 l/min. Fire Flow alternative**, then select **New** > **Child Alternative**.

   b. Highlight the new alternative and click **Rename**. Enter a label of **4000 l/min Fire Flow** for the new Alternative.

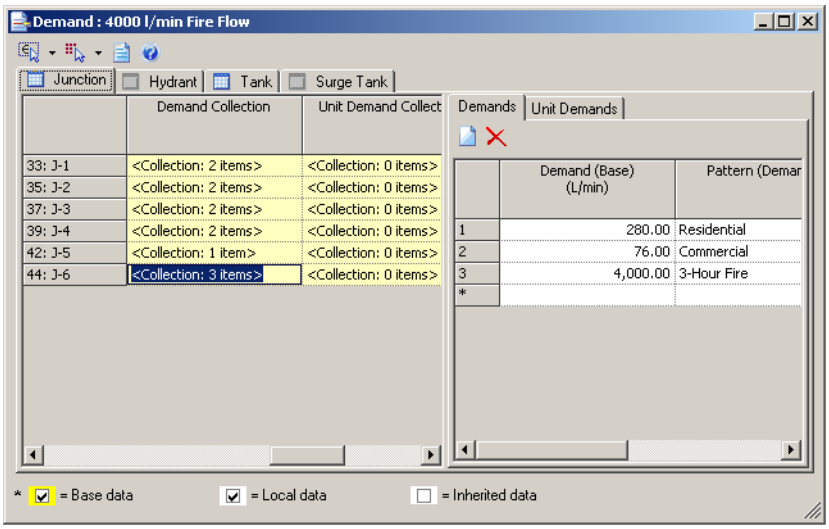


   c. Double-click to open the Demand Alternatives editor for the new alternative which shows the data that was inherited from the parent alternative.



If you change any piece of data, the check box will become selected because that record is now local to this alternative and not inherited from the parent.

5. Click in the Demand Collection column for node **J-6**. Change the 2000 l/min. fire demand to **4000 l/min**.



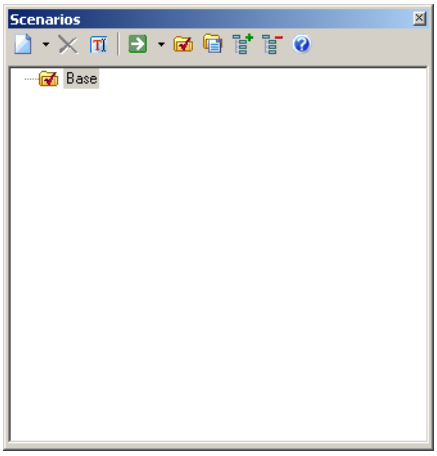
6. Close the Demand Alternative Editor.
7. Close the Alternatives Manager

Step 2: To create and edit Scenarios

Alternatives are the building blocks of a scenario. A scenario is a set of one of each of the types of alternatives, plus all of the calculation information needed to solve a model.

Just as there are base, parent, and child alternatives, there are also base, parent, and child scenarios. The difference is that instead of inheriting model data, scenarios inherit sets of alternatives. To change the new scenario, change one or more of the new scenario's alternatives. For this lesson, you will create a new scenario for each different set of conditions you need to evaluate.

1. Choose **Analysis > Scenarios** or click  to open Scenarios.



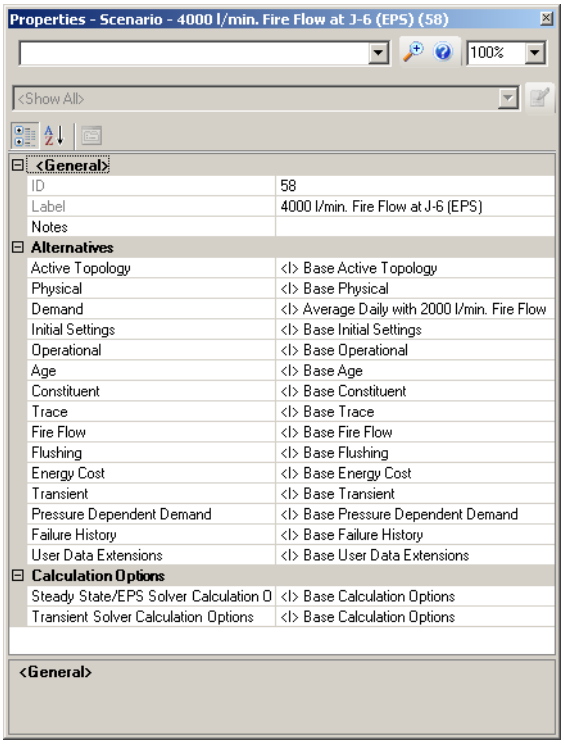
   There is always a default Base Scenario that is composed of the base alternatives. Initially, only the Base is available, because you have not created any new scenarios.

2. Click Rename  to rename the Base Scenario to **2000 l/min., 3-hour Fire Flow at J-6 (EPS)**.

3. Create a child scenario from the existing base scenario to incorporate the new demand alternative.

   a. Right-click on the **2000 l/min., 3-hour Fire Flow at J-6 (EPS)** scenario and select **New** > **Child Scenario**.

   b. Highlight the new scenario and click **Rename**. Enter a scenario name of **4000 l/min. Fire Flow at J-6 (EPS)**. Double-click the scenario to open the Properties editor for the scenario.

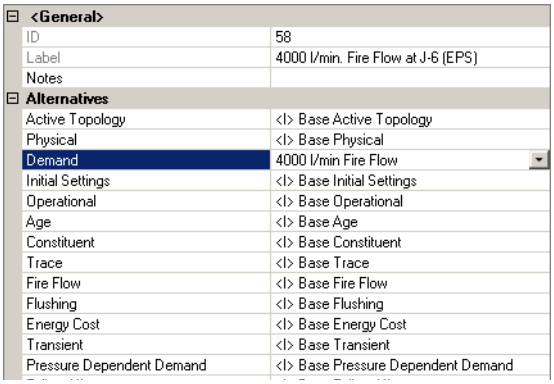


The new scenario lists the alternatives as inherited from the base scenario.

4. Your new Child Scenario initially consists of the same alternatives as its parent scenario. Set the Demand Alternative to the new alternative you created, **4000 l/min. Fire Flow**.

   a. Click in the **Demand** field

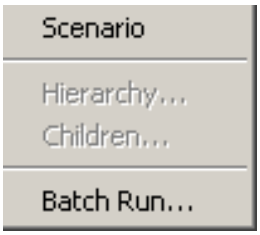
   b. From the menu, select the **4000 l/min. Fire Flow** alternative.



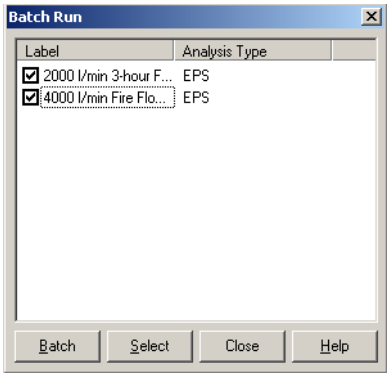
   The new alternative is no longer inherited from the parent, but is local to this scenario.

Step 3: To calculate both of the scenarios using the Batch Run tool

1. In the Scenarios manager, click **Compute Scenario**  and then **Batch Run**



2. Select both check boxes next to the scenario names in the Batch Run dialog box.

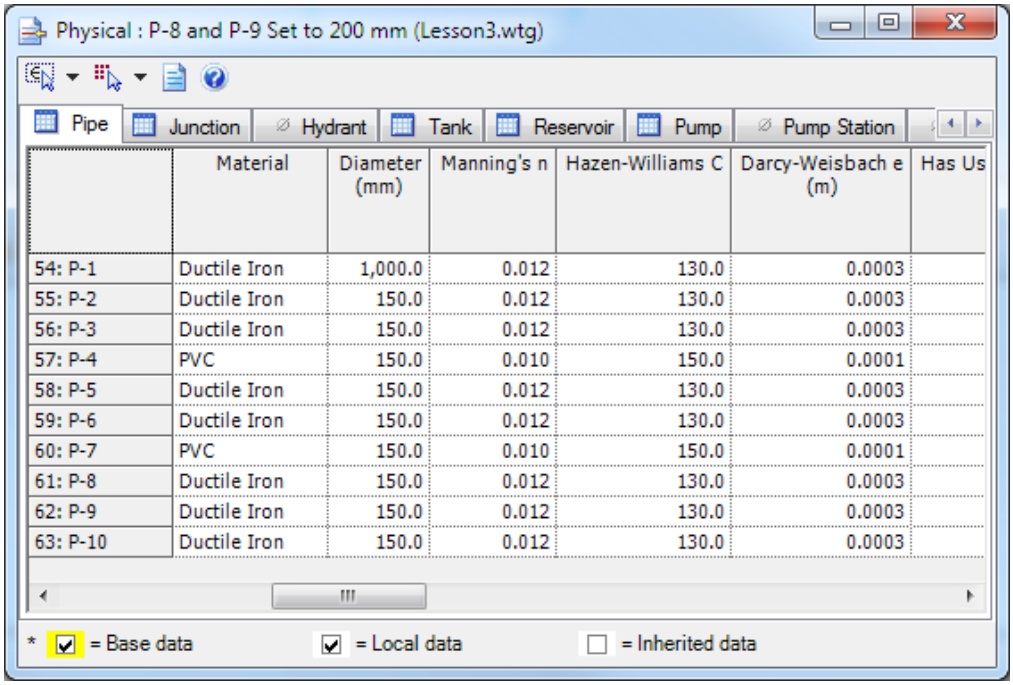


3. Click **Batch**.
4. Click **Yes** at the prompt to run the batch for two scenarios.
5. After computing finishes, click **OK**.

Step 4: To create a Physical Alternative

You need to further examine what is going on in the system as a result of the fire flow, and find solutions to any problems that might have arisen in the network as a result. You can review output tables to quickly see what the pressures and velocities are within the system, and create new alternatives and scenarios to capture your modifications.

1. Click **Analysis** > **Alternatives**. Under **Physical**, highlight **Base Physical**. Right-click and select **New** > **Child Alternative**.
2. Rename the new Child Alternative **P-8 and P-9 Set to 200 mm**.
3. Double-click the newly created physical alternative to open the Physical alternative editor. In the Pipe tab for this Alternative, change the diameter for pipes **P-8** and **P-9** to **200** mm.



4. **Close** the alternative editor dialog.

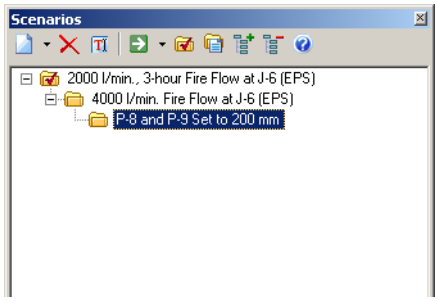
5. Create a new scenario having a new physical alternative with the pipe sizes for P-8 and P-9 increased to 200 mm.

   a. Click  or choose **Analysis > Scenarios**.

   b. Select **4000 l/min. Fire Flow at J-6 (EPS)** in the list of Scenarios.

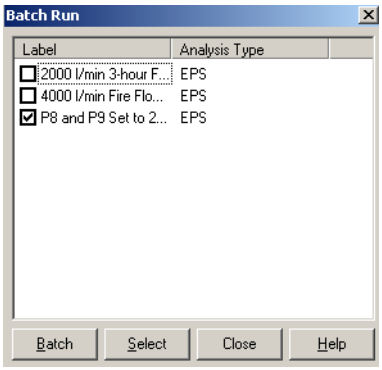
   c. Click New, and select Child Scenario.

   d. Name the new Scenario **P-8 and P-9 Set to 200 mm**.



6. Double click scenario **P-8 and P-9 Set to 200 mm** to open the Properties editor for the scenario. Click **Physical** and select the **P-8 and P-9 Set to 200 mm** alternative.

7. In the Scenarios manager, click **Compute** > **Batch Run** and select the check box for **Pipes P-8 and P-9 Set to 200 mm**.



8. Click **Batch** and then **Yes** to confirm and run the Scenario.

9. Click **OK** after the run is complete.

10. Close the open boxes and save the project.

Reporting Results

An important feature in all water distribution modeling software is the ability to present results clearly. This lesson outlines several of Bentley WaterCAD V8*i* reporting features, including:

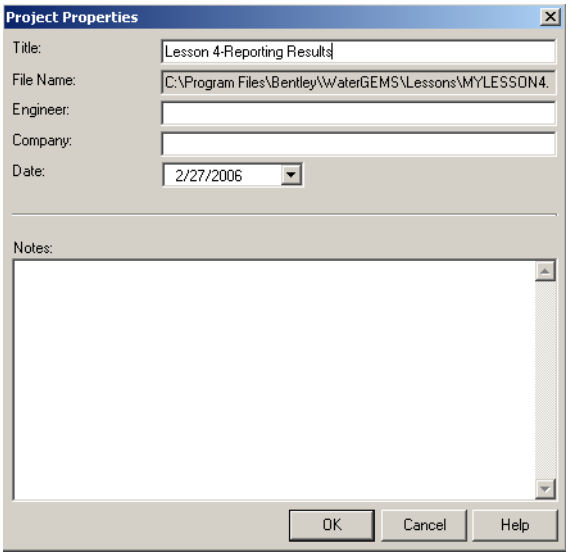
- **Reports**, which display and print information on any or all elements in the system.
- **Element Tables (FlexTables)**, for viewing, editing, and presentation of selected data and elements in a tabular format.
- **Profiles**, to graphically show, in a profile view, how a selected attribute, such as hydraulic grade, varies along an interconnected series of pipes.
- **Contouring**, to show how a selected attribute, such as pressure, varies throughout the distribution system.
- **Element Annotation**, for dynamic presentation of the values of user-selected variables in the plan view.
- **Color Coding**, which assigns colors based on ranges of values to elements in the plan view. Color coding is useful in performing quick diagnostics on the network.

For this lesson, you will use the system from the Scenario Management lesson, saved as **MYLESSON3** in the Bentley\WaterGEMS\Lesson directory. If you did not complete this lesson, you may use the file **LESSON4.WTG** (**LESSON4.DWG** in AutoCAD).

To open the existing project

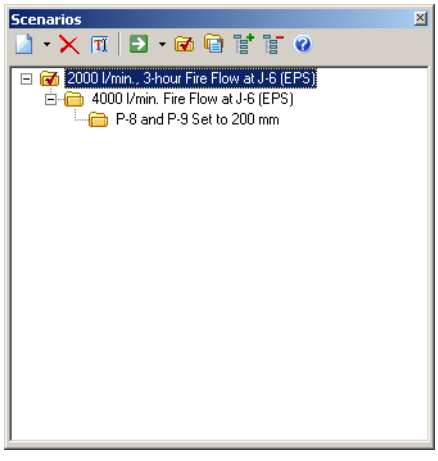
1. Open **MYLESSON3.WTG**.
2. Select **File > Save As**.

3. Enter the filename **MYLESSON4**, and click **Save**.
4. Select **File > Project Properties**, and change the Project Title to **Lesson 4 - Reporting Results**.



Reports

1. Choose Analysis > Scenarios or click  to open Scenarios.
2. Select the **2000 l/min., 3 hour fire flow at J-6 (EPS)** scenario.

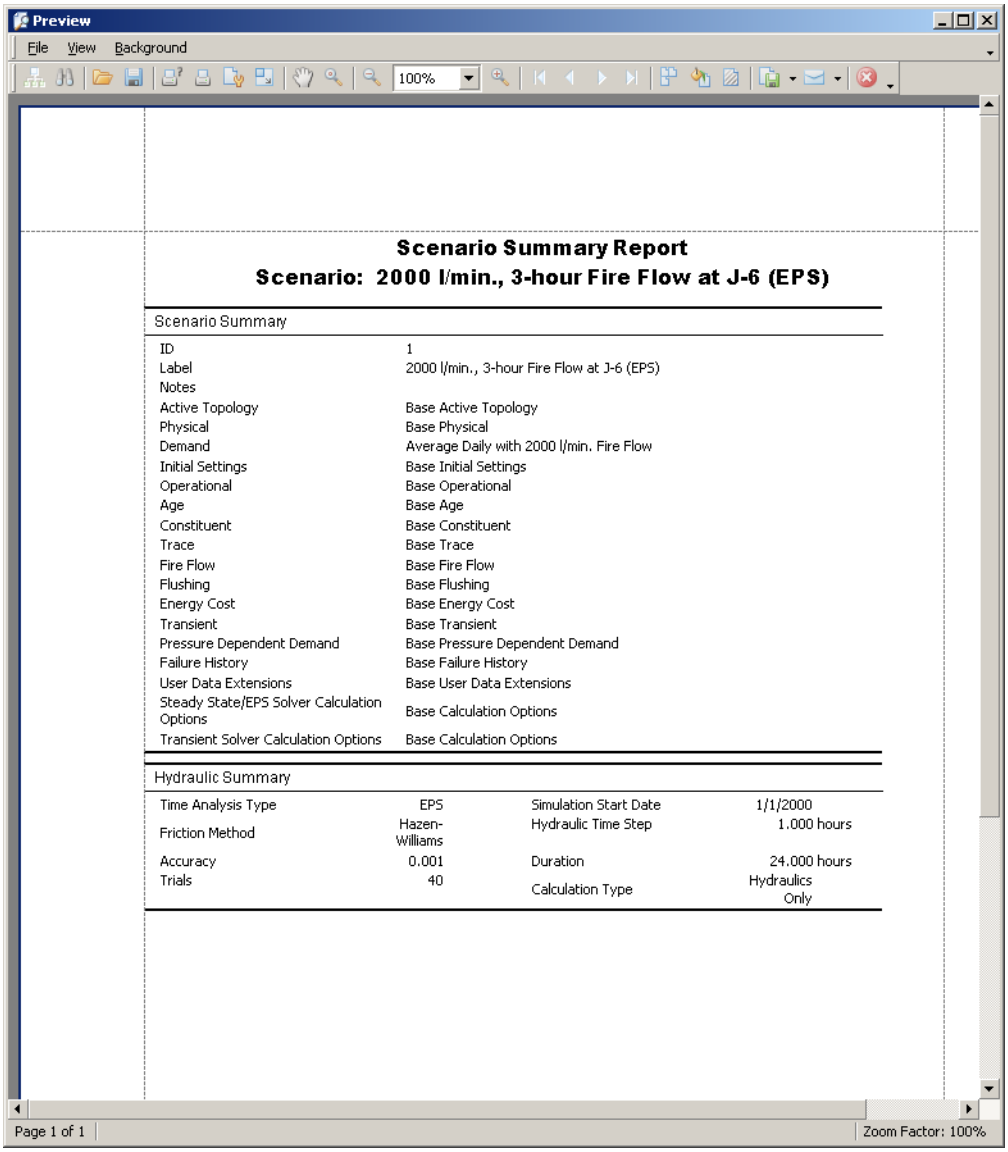


3. Click  to compute the Scenario.

4. Click Report > Scenario Summary

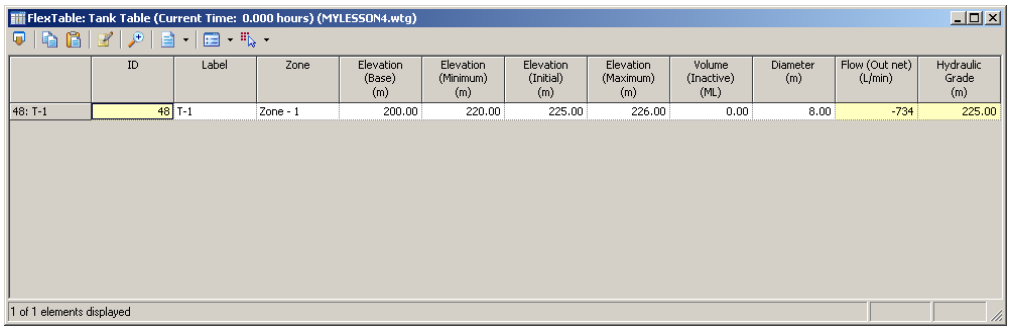


5. The report opens.

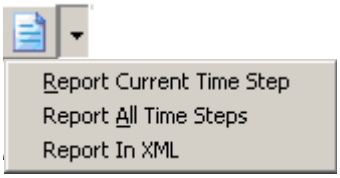


6. You can use the toolbar to save, print or copy the results to another program.
7. Close the Scenario Summary.

8. Choose Report > Element Tables > Tank.

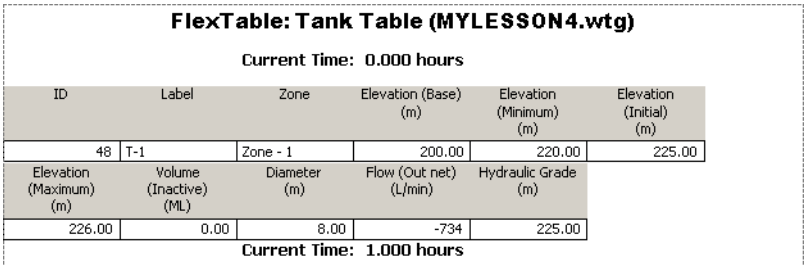


9. Click **Report** and select for either the Current Time Step or All Time Steps.



10. Use the **Page** icons  to navigate through the report.

    Every element can generate a report in the same general format, which includes the name of the calculated scenario and information describing the element's properties and results in detail.



FlexTable: Tank Table (MYLESSON4.wtg)

Current Time: 0.000 hours

ID	Label	Zone	Elevation (Base) (m)	Elevation (Minimum) (m)	Elevation (Initial) (m)
48	T-1	Zone - 1	200.00	220.00	225.00

Elevation (Maximum) (m)	Volume (Inactive) (ML)	Diameter (m)	Flow (Out net) (L/min)	Hydraulic Grade (m)
226.00	0.00	8.00	-734	225.00

Current Time: 1.000 hours

You can print this report using these icons. 

The report will print in the exact format seen on the screen.

11. Close the report, and then exit the Tank FlexTable.

FlexTable

When data must be entered for a large number of elements, clicking each element and entering the data can be time consuming. FlexTable elements can be changed using the global edit tool, or filtered to display only the desired elements. Values that are entered into the table will be automatically updated in the model. The tables can also be customized to contain only the desired data. Columns can be added or removed, or you can display duplicates of the same column with different units.

FlexTables are dynamic tables of input values and calculated results. White columns are editable input values, and yellow columns are non-editable calculated values. When data is entered into a table directly, the values in the model will be automatically updated. These tables can be printed or copied into a spreadsheet program.

Global Edit and Filtering are very useful tools. For example, if you decide to evaluate how the network might operate in five years. Assume that the C factor for 5-year old ductile iron pipe reduces from 130 to 120. It would be repetitive to go through and edit the pipe roughness through the individual pipe dialog boxes, particularly when dealing with a large system. Instead, you will use the filter tool in this example to filter out the PVC pipes, and then use global edit tool to change the pipe roughness on the ductile iron pipes only.

To use Global Edit and Filtering

1. Set up a new Alternative and Scenario to capture the changes to the C values.

    a. Click **Analysis > Alternatives**. Highlight the **P-8 and P-9 Set to 200 mm** Physical Alternative and click **New Child Alternative**.

    b. Rename the new Alternative **5-yr.-old D.I.P.**

    c. Close the Alternatives manager.

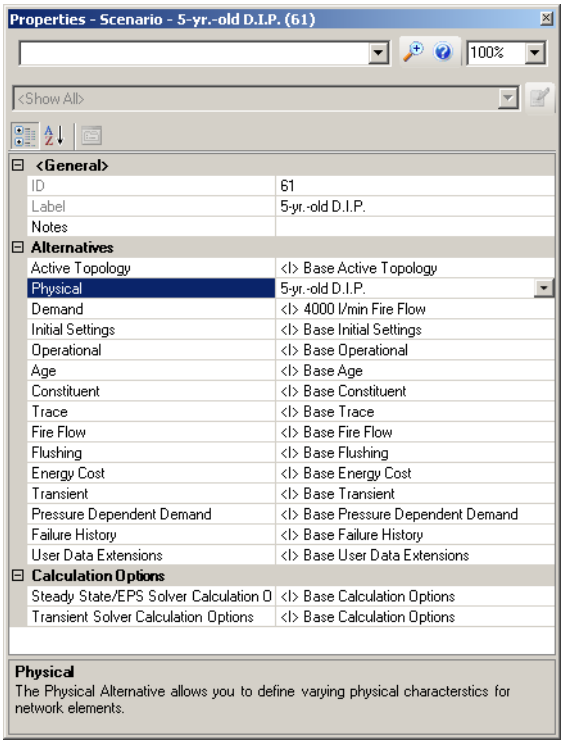
    d. Click **Analysis > Scenarios**.

    e. Select the **P-8 and P-9 Set to 200 mm** scenario.

    f. Right click and select New > Child Scenario.

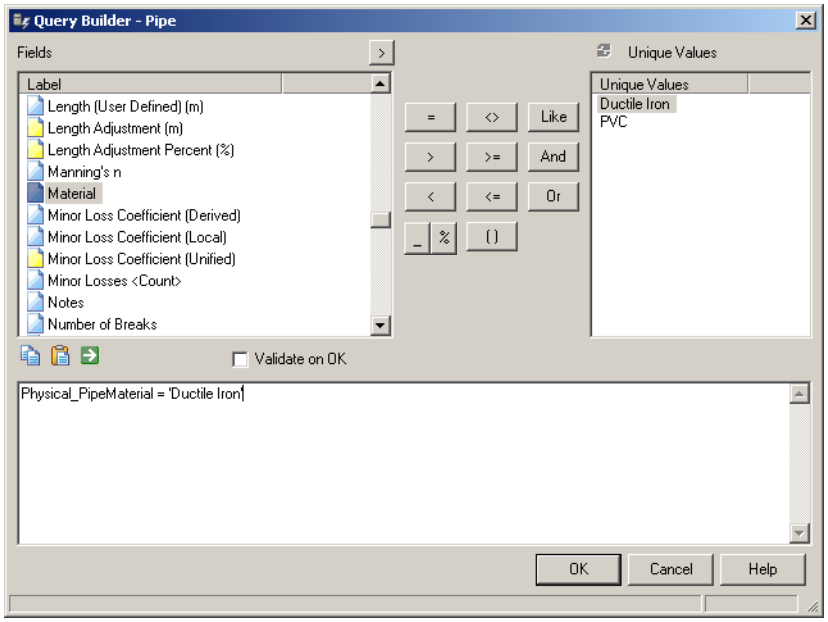
    g. Rename the new scenario **5-yr.-old D.I.P.**

h. Double-click the new scenario to open the Properties editor. Change the Physical alternative to **5-yr.-old D.I.P.**



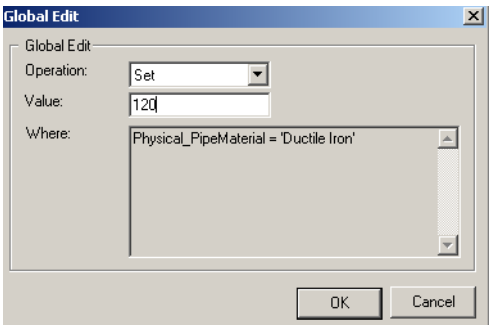
2. Click Report > Element Tables > Pipe.
3. Right-click the **Material** column and choose **Filter > Custom** from the menu.
4. The query builder opens.
   a. Double-click on **Material** in the **Fields** list.
   b. Click the = equal sign.
   c. Click  to select the Unique Values for Material

d. Double-click **Ductile Iron**.



e. Click **Apply**  , then click **OK**.

5. Use the Global Edit tool to modify all of the roughness values in the table.
   a. Right-click the Hazen-Williams C column and select **Global Edit**.
   b. Select **Set** from the Operation list.
   c. Enter **120** into the Value field.

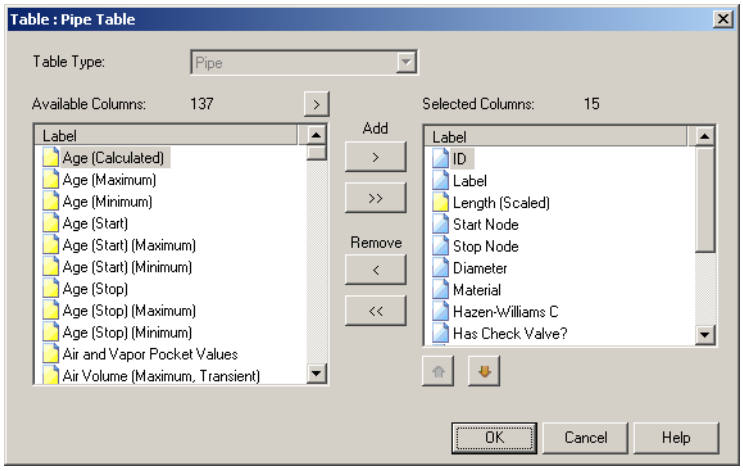


   d. Click **OK**. All of the values are now set to 120.

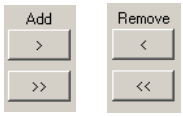
6. To deactivate the filter, right-click anywhere in the dialog box and click **Filter** > **Reset** from the menu. Click **Yes** to reset the filter.

7. You may also wish to edit a table by adding or removing columns using the Table Manager.

   a. Click Edit  to open the table editor.



   b. Scroll through the list on the left to view the types of data available for placement in the table. You can select an item to add or remove from the table.



   c. You can adjust the order which the columns will be displayed by using the arrows below Selected Columns  .

   d. Click Ok to save your changes or Cancel to exit the table without making change.

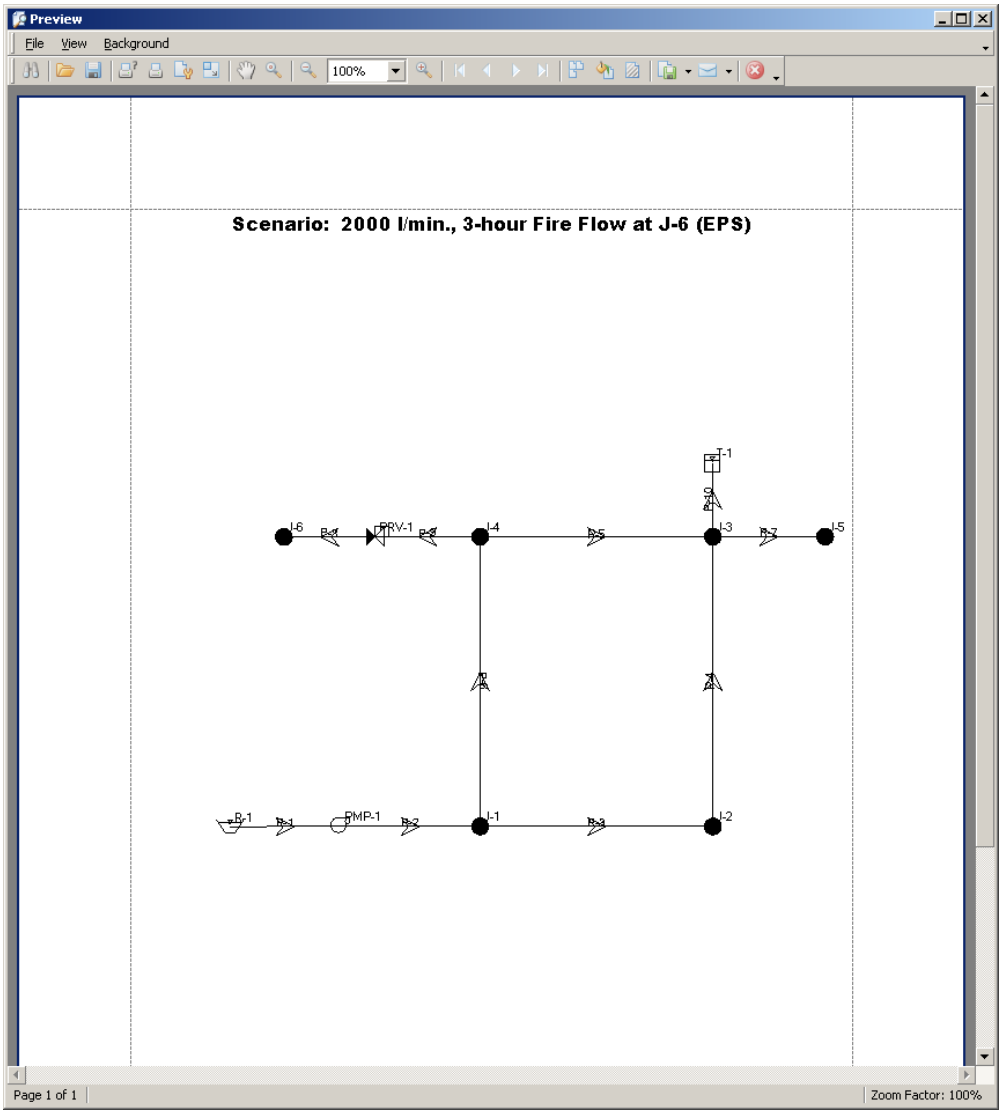
8. Click to exit the table.
9. Choose Analysis > Scenarios > Compute Scenario > Batch Run.
10. Check 5-yr.-old D.I.P., and then click Batch.
11. Click to exit the table when you are finished.

Create a Print Preview and Profile

1. To create a print preview of the distribution system, click File > Print Preview > Fit to Page.

   This option will create a preview of the entire system regardless of what the screen shows.

The print preview opens in a separate window, which can then be printed or copied to the clipboard.



2. Close the print preview window.

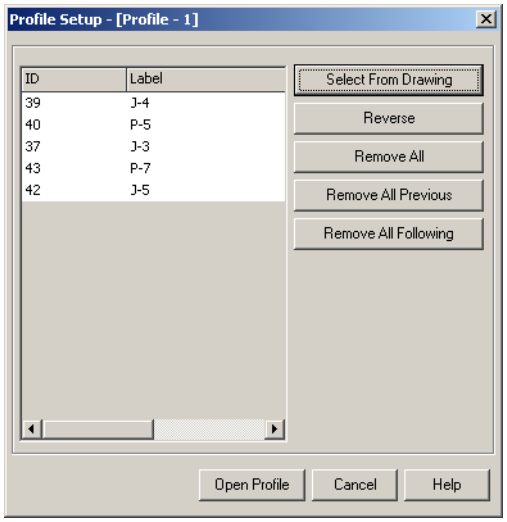
3. To create a profile view, choose **View > Profiles**, or click **Profile**  in the toolbar. This activates the Profiles manager.

4. Click New to open the Profile Setup dialog box, and then click **Select From Drawing** to choose the element to profile.

5. The dialog box closes and select opens. Choose a few elements to include in the profile and click Done .

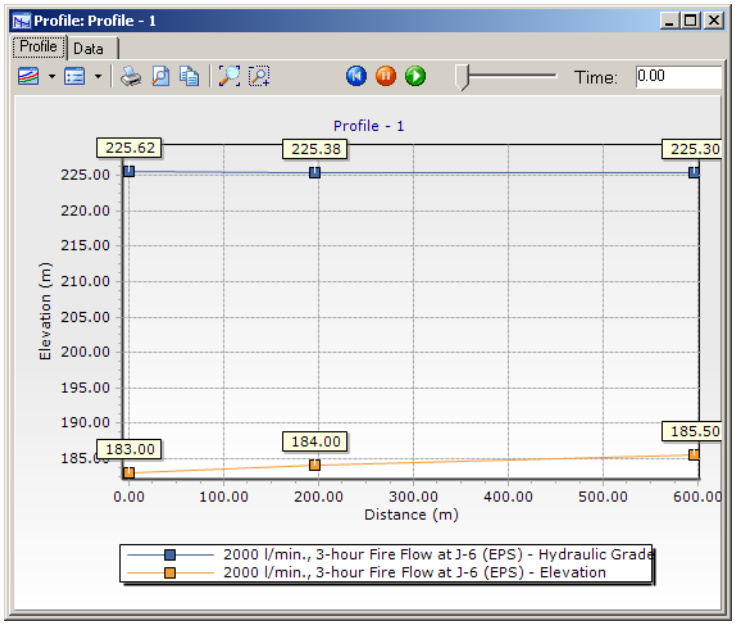


6. The Profile Setup dialog box opens with the selected elements appearing, in order, in the list.



Click Open Profile to view the profile.

7. After you create the profile, you can make adjustments to its appearance by clicking **Profile Series Options** or **Chart Options** .

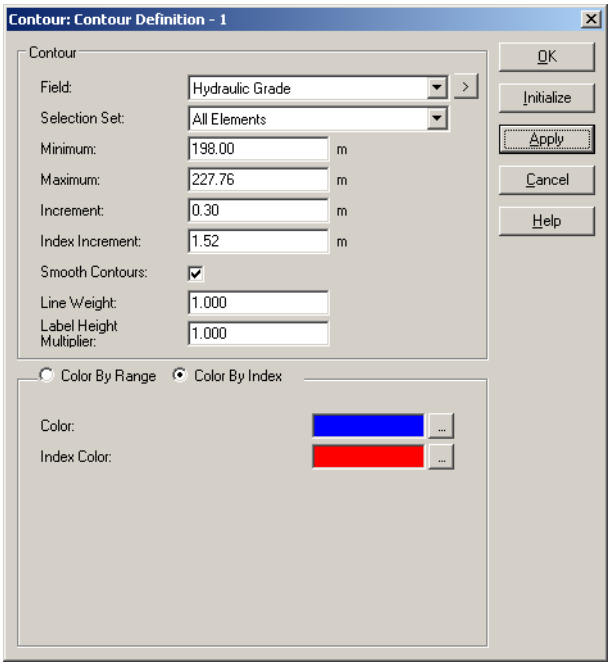


8. The graph can be printed or copied to the clipboard.
9. Close the Profile window.
10. Close the Profile manager.

To Create a Contour

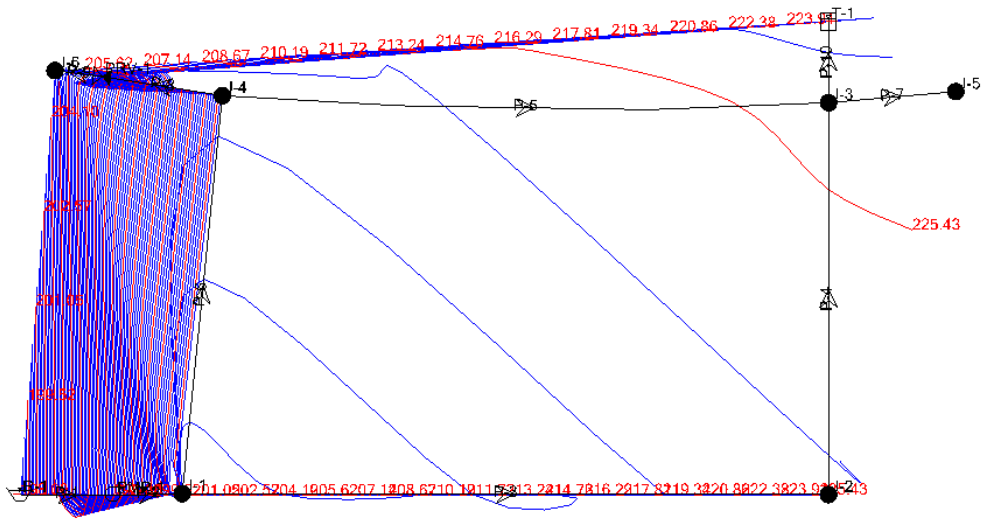
The contouring feature in Bentley WaterCAD V8*i* enables you to generate contours for reporting attributes such as elevation, pressure, and hydraulic grade. You can specify the contour interval, as well as color code the contours by index values or ranges of values. In this lesson, you will contour based on hydraulic grade elevations.

1. Choose View > Contours or click Contours  .
2. Click New in the Contour Manager.
3. Choose Hydraulic Grade from the Field menu.
4. Choose All Elements in the Selection Set menu.
5. Click **Initialize** and select **Full Range** to update the Minimum and Maximum HGL elevations.
6. Make sure **Color by Index** is selected
7. Select Smooth Contours to improve the overall appearance of the drawing.



8. Click OK.

9. View result in the drawing pane.

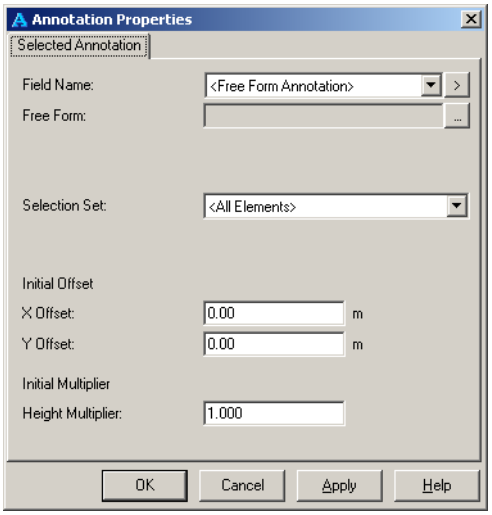


10. Close the Contour Manager.

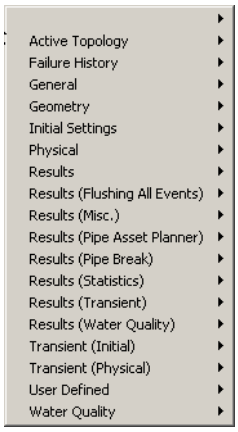
Element Symbology

When you want to label network attributes use the Annotation feature. With it, you can control which values are displayed, how they are labeled, and how units are expressed.

1. Choose View > Element Symbology > New > New Annotation.



2. Select the Field Name to annotate.



3. Enter additional information into the other fields as needed.
4. Click Apply.
5. The drawing will now display all of the annotations. You can try changing the properties of an element and recalculating. The annotations will update automatically to reflect any changes in the system.
6. If the annotation is crowded, you can click and drag the annotation to move it.
7. Click OK.

Color Coding

1. Choose View > Element Symbology and click an element to create a new color coding for that element type.
2. Right-click the element and choose New > Color Coding or click New > New Color Coding from the toolbar.
3. The Color Coding dialog box allows you to set the color coding for links, nodes, or both.
   a. Select Diameter from the Field Name menu.
   b. In the table, enter values of **150**, **200**, and **1000 mm** with colors of **red**, **blue**, and **green**, respectively.

	Value (mm)	Color
0	150.0	255, 0, 0
1	200.0	0, 0, 192
2	1,000.0	0, 128, 0
*		

   c. Click **Calculate Range > Full Range** to get the minimum and maximum values for the variable displayed at the top of the dialog box. The maximum must be higher than the minimum.
   d. Then, click **Initialize**  and the model will select the color coding ranges in the table automatically.

	Value (mm)	Color
0	156.4	0, 255, 0
1	160.4	0, 255, 255
2	164.4	0, 0, 255
3	168.4	255, 0, 255
4	172.4	255, 0, 0

   e. Click **OK** to generate the Color Coding.
4. You can add a legend to the drawing. Right-click on the color coding and select Insert Legend from the menu. You can move the legend in the drawing by clicking the mouse and dragging the legend.

5. Close any open dialog boxes.

6. Save  the project.

Automated Fire Flow Analysis

One of the primary goals of a water distribution system is to provide adequate capacity to fight fires. Bentley WaterCAD V8*i* automated fire flow analysis can be used to determine if the system can meet the fire flow demands while maintaining minimum pressure constraints. Fire flows can be computed for all nodes in the system, or you can create a selection set consisting of specific nodes where you wish to test available flow.

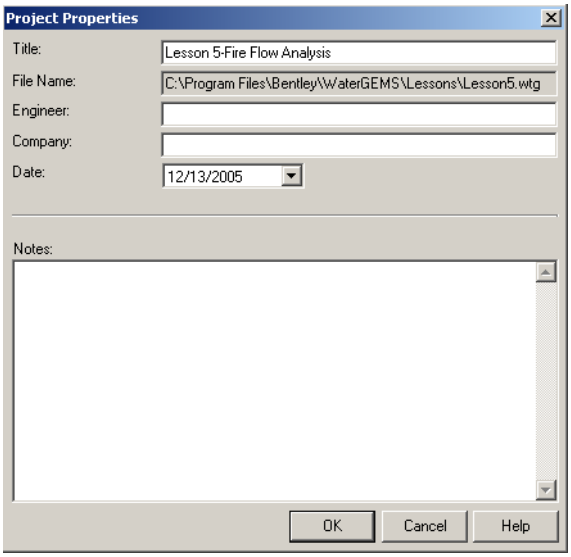
Fire flows are computed at each node by iteratively assigning demands and computing system pressures. The model assigns the fire flow demand to a node and checks the model, checking to see if all pressure and velocity constraints are met at that demand. If a constraint is not met, the flow is reduced until the constraint is just met; if all constraints are exceeded, the fire flow is increased until the constraint is barely met within a tolerance. The analysis automatically rechecks the system pressures if a constraint is violated. Iterations continue until the constraints are met, or until the maximum number of iterations is reached.

The purpose of this example is to walk you through the steps to create, calculate, and analyze a fire-flow scenario. This lesson again uses the distribution system from the previous lessons.

Step 1: Inputting Fire Flow Data

1. Start Bentley WaterCAD V8*i* and open the **LESSON5.wtg** file, found in the Bentley\WaterGEMS\Lesson folder.
   Or
   if you have previously completed the Reporting Results lesson, you can use your **MYLESSON4** file.
2. Choose File > Save As and save as MYLESSON5.

3. Choose File > Project Properties and name the title of the project **Lesson 5—Fire Flow Analysis**.



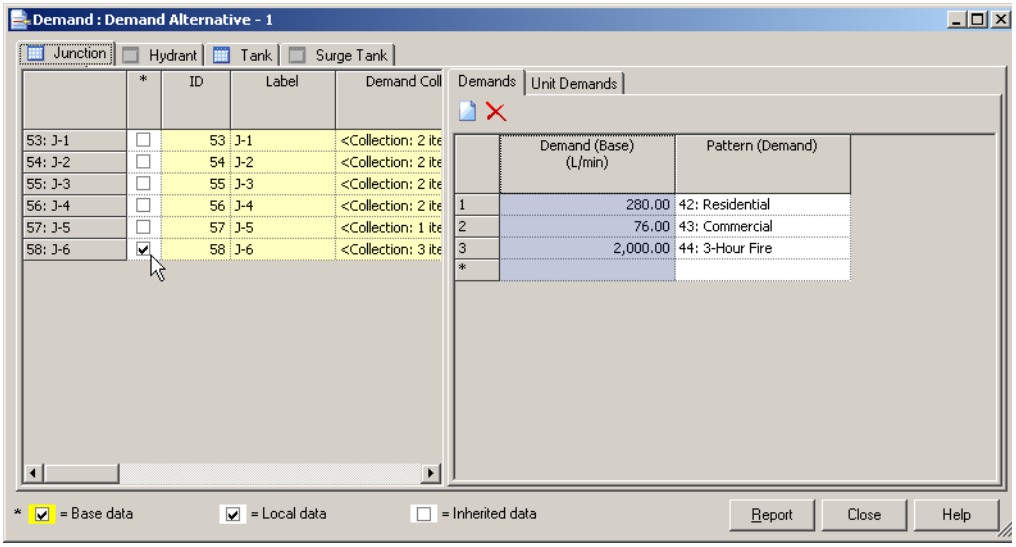
4. Click OK.

5. Previously, you ran an analysis with a fire flow at node J-6 by manually adding a large demand to the individual node. Before running the automated fire flow analysis, you will create a new Demand Alternative, removing that demand. In the U.S., fire flows are generally added to max day demands.

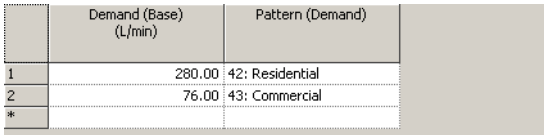
   a. Click Analysis > Alternatives.

   b. Expand the Demand Alternative and select Average Daily with 2000 l/min Fire Flow. Right-click and select New > Child Alternative.

   c. Double-click to open the new alternative and put a check in the box for J-6.



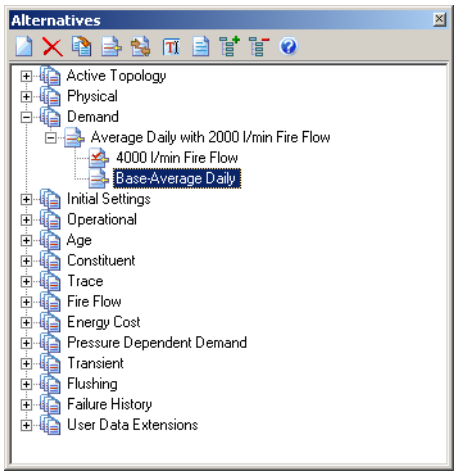
d. In the Demands tab, select the row with 2,000 Flow and 3-Hour Fire and click  to delete it.



	Demand (Base) (L/min)	Pattern (Demand)
1	280.00	42: Residential
2	76.00	43: Commercial
*		

e. Click Close to exit the Demand Alternative.

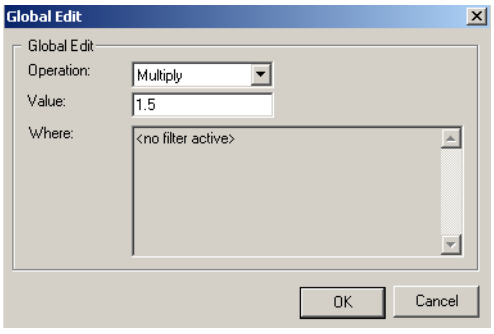
6. Click to Rename  this Alternative **Base-Average Daily**.



7. You are going to analyze the fire flows by adding to the Maximum Day Demands, which are 1.5 times the Average Day Demands.

   a. Right-click on Base-Average Daily then select New > Child Alternative.

   b. Double click to open the Alternative and highlight the J-1 row. Right-click the Demands column and select **Global Edit**. Set the Operation to multiply, and enter a value of **1.5**.



   c. Click **OK**.

d. Repeat step b and c for J-2 through J-6. Click **Close** to exit the Demand Alternative.

e. Click to Rename  this Alternative **Max. Day**.

8. Select the **Fire Flow** alternative and expand to select the Base-Fire Flow Alternative.

9. Click Open  to set up the Base-Fire Flow Alternative.

   a. In the **Fire Flow (Needed)** field, enter **3000 l/min.**

   b. In the **Fire Flow (Upper Limit)** field enter **6000 l/min.**

   c. **Apply Fire Flows By** should be set to **Adding to Baseline Demand**.

      This selection means that when Bentley WaterCAD V8*i* performs the analysis, the fire flow will be added to any demands already assigned to the junction. Alternatively, you could have selected to replace these demands, so that the fire flow would represent the total demand at the node.

   d. Pressure Constraints Pressure (Residual Lower Limit) and Pressure (Zone Lower Limit) should be set to **150 kPa**.

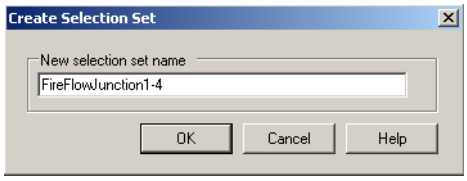
   e. Leave the check box for **Use Minimum System Pressure Constraint** cleared, so that the minimum pressure will only be checked for the zone a particular node is in.

      If you had multiple zones within your project and wanted to insure that a minimum system-wide pressure constraint was met, you could check the **Use Minimum System Pressure Constraint** box and enter it in the box provided. This box is grayed out until the check box is activated.

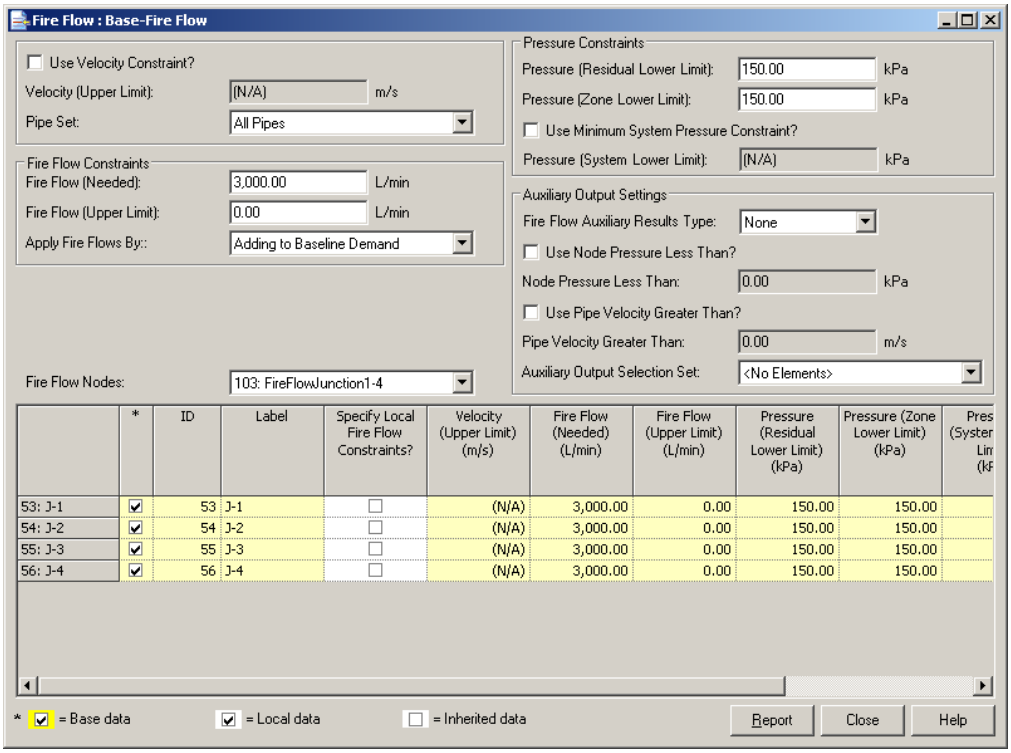
   f. Create a selection set to choose from the Fire Flow Nodes drop-down menu. For this example, a fire flow analysis is only needed for the junctions at the four street corners in our drawing.

   g. The Fire Flow Alternative manager can remain open. In the drawing and while pressing the <Shift> key, click nodes **J-1**, **J-2**, **J-3**, and **J-4.**

   h. Right-click and select **Create Selection Set**, then name the set FireFlowJunction1-4 and click OK.



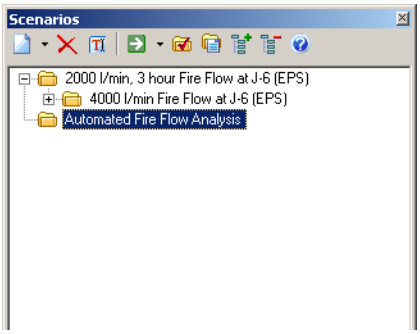
i. In the Fire Flow Alternative manager, select FireFlowJunction1-4 from the Fire Flow Nodes drop-down menu.



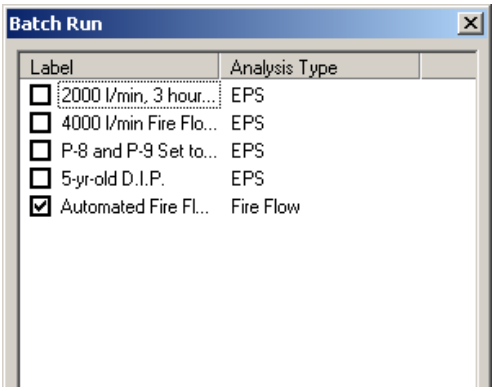
10. Close the Fire Flow Alternative manager.

Step 2: Calculating a Fire Flow Analysis

1. Click **Analysis > Scenarios** or click  .
2. In the Scenarios dialog, click New > Base Scenario.
3. Name the new Scenario **Automated Fire Flow Analysis**.



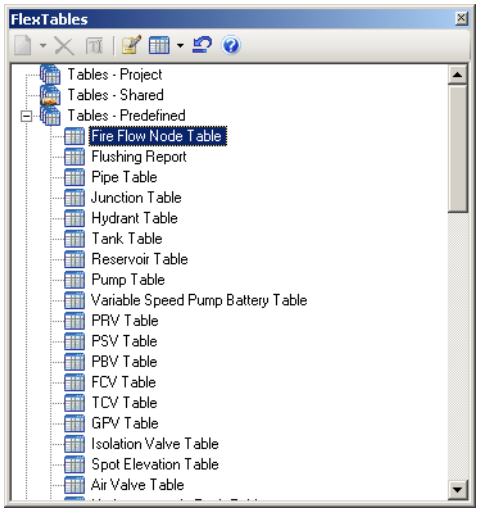
4. Double-click the **Automated Fire Flow Analysis** scenario to open the properties editor.
   a. Change the Physical Alternative to **P-8 and P-9 Set to 200 mm**.
   b. Change the Demand to **Max. Day** and leave all other Alternatives set to their defaults.
   c. Close the properties dialog.
5. In the Scenarios manager, make Automated Fire Flow Analysis the current scenario by highlighting it and clicking the Make Current button  .
6. Click the **Analysis > Calculation Options**, double-click on **Base under Steady State/EPS** and set the **Calculation Type** to **Fire Flow**.
7. Close the Properties editor.
8. Run the Scenario.
   a. From the Scenarios Manager click Batch Run.
   b. Check **Automated Fire Flow Analysis**, and clear the other Scenarios, if necessary.



   c. Click **Batch** to run the analysis, and **Yes** at the confirmation prompt.
   d. When the calculation is complete, click **OK** and close the Scenarios Manager.

Step 3: Viewing Fire Flow Results

1. Make sure that **Automated Fire Flow Analysis** is selected in the Scenario list box.
2. Click **View** > **FlexTables**. Under **Tables - Predefined**, double-click the **Fire Flow Node Table**.



In the Satisfies Fire Flow Constraints column, all of the boxes are checked except for the nodes that you did not analyze, because the specified needed flow of 3000 l/min. was available and minimum pressures were exceeded.

For nodes J-1 and J-3, pressures were computed for the Fire Flow Upper Limit of 6000 l/min. because none of the node pressures ever dropped below specified minimum pressures and no velocity constraint was specified.

Nodes J-2 and J-4 reached their minimum residual pressures at flows slightly below the maximum of 6000 l/min.

The report contains the Minimum System Pressure (excluding the current node being flowed) and its location.

3. When you are finished reviewing the report, click **Close** in the Bentley WaterCAD V8*i* Fire Flow Report dialog box and save your file as **MYLESSON5**.

Note: **Another good way to review an automated fire flow analysis is to use color coding. If you have a situation where no nodes meet the pressure constraints for the needed fire flow, you can color code these nodes in the plan view for easy identification.**

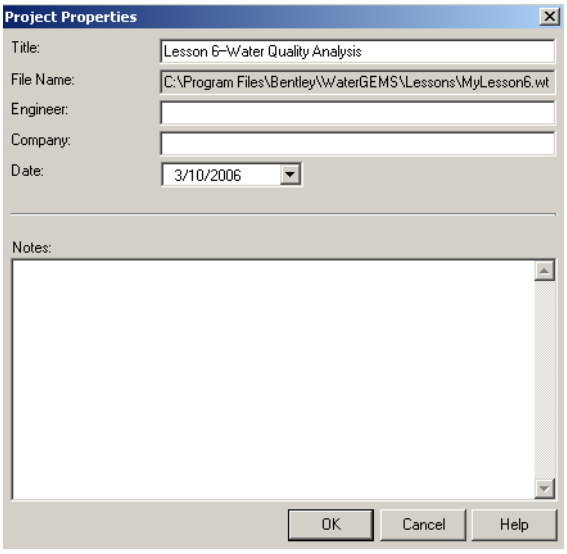
Water Quality Analysis

In conjunction with Extended Period simulations, Bentley WaterCAD V8*i* is capable of performing a water quality analysis to compute water age, constituent concentration, or percentage of water from a given node (trace analysis). Using these features, you can look at factors such as residence time in tanks, chlorine residuals throughout the system, and which tank or reservoir is the primary water source for different areas in your system.

This lesson uses the file called LESSON6.wtg (LESSON6.DWG in the AutoCAD version), located in the Bentley\WaterGEMS\Lesson directory.

To open the existing lesson

1. Open **Lesson6.wtg**.
2. After you have opened the file, choose **File > Save As**.
3. Enter the filename **MYLESSON6** and click **Save**.
4. Choose **File > Project Properties**, and change the Project Title to **Lesson 6—Water Quality Analysis**.



5. Click OK.

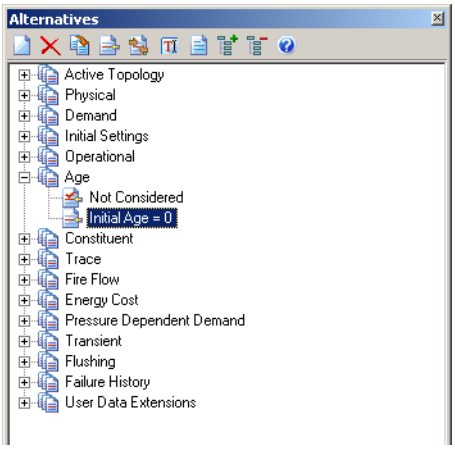
The water distribution system has already been set up for you. It has one reservoir and one tank. The system serves primarily residential areas, with some commercial water use as well. There are two pumps connected to the reservoir. However, under normal conditions, only one pump will be in use. A background drawing has been included for reference.

If you would like to turn off the .DXF background in the WaterCAD V8*i* version, clear the background check box in the Background Layers pane (View > Background Layers).

Step 1: Computing Water Age

You will begin by running an age analysis for water in the system, assuming an initial age of 0 for all nodes. The water from the reservoir will be an infinite supply of new water, so the age of water elsewhere in the system will be a reflection of time from the start of the run and how long ago the water left the reservoir. The analysis will be run for a 2-week period (336 hours), in order to determine the equilibrium point of the system.

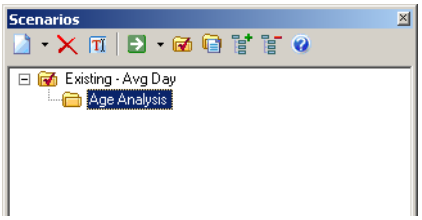
1. Choose **Analysis** > **Alternatives** or click  .
2. Select Age Alternative and click New  to create a new age alternative.
3. Name the new alternative **Initial Age = 0**. Since you are assuming an initial age of 0 everywhere in the system, you do not need to enter any initial ages.



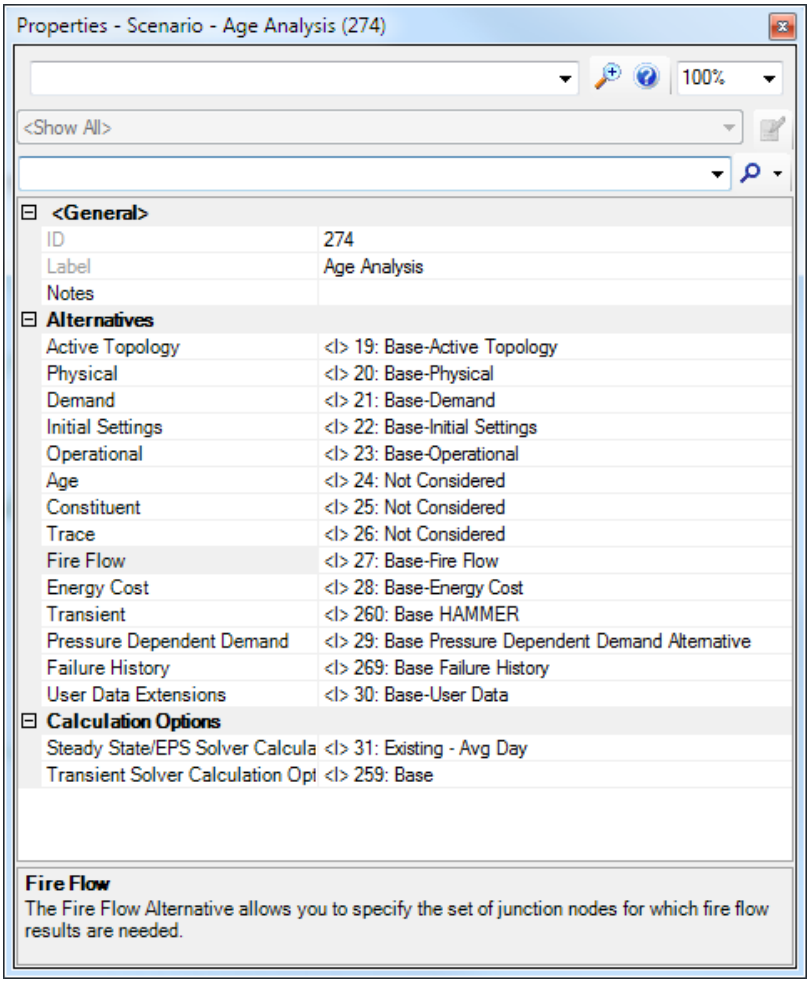
4. Next, set up a new Scenario to run an Extended Period Simulation incorporating the new Alternative.

   a. Click **Analysis** > **Scenarios**; note that the **Existing - Avg Day** scenario already exists.

   b. Highlight the **Existing - Avg Day** scenario and click **New** > **Child Scenario** and enter **Age Analysis** as the new scenario name.



   c. Double-click on the new scenario to open the properties editor. In the Age Alternative field select **Initial Age = 0**, from the drop-down menu.



   d. Close the properties box.

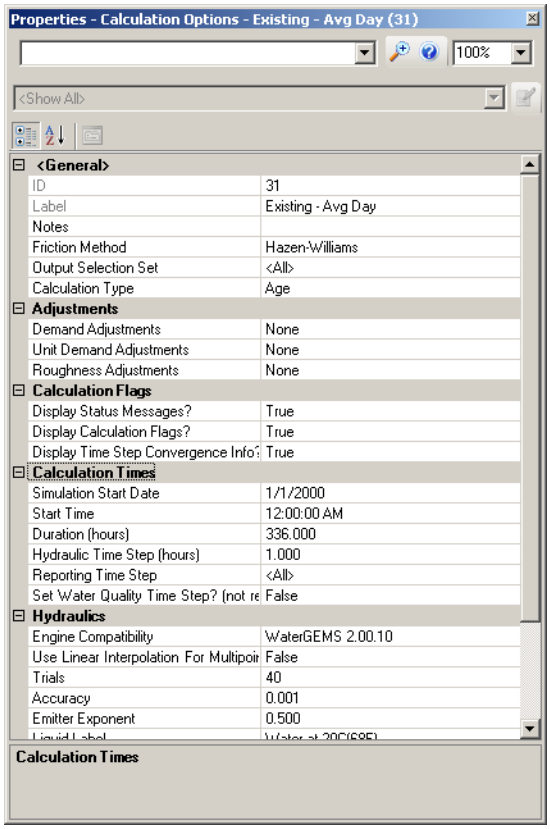
e. Click **Analysis** > **Calculation Options** tab and double click **Existing - Avg Day** to view the settings for this Scenario. **Extended Period Analysis (EPS)** should already be selected as the **Time Analysis Type**.

f. Set the **Calculation Type** to **Age**

g. Leave the default **Start Time** of **12:00:00 AM**.

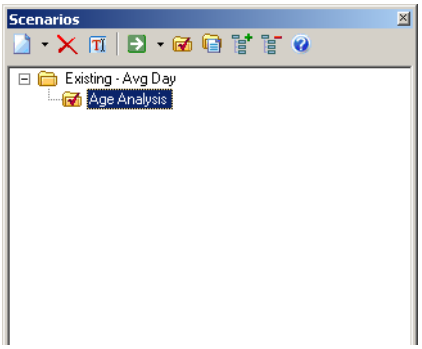
h. Set a **Duration** of **336** hours.

i. Leave the default **Hydraulic Time Step** of **1** hour.



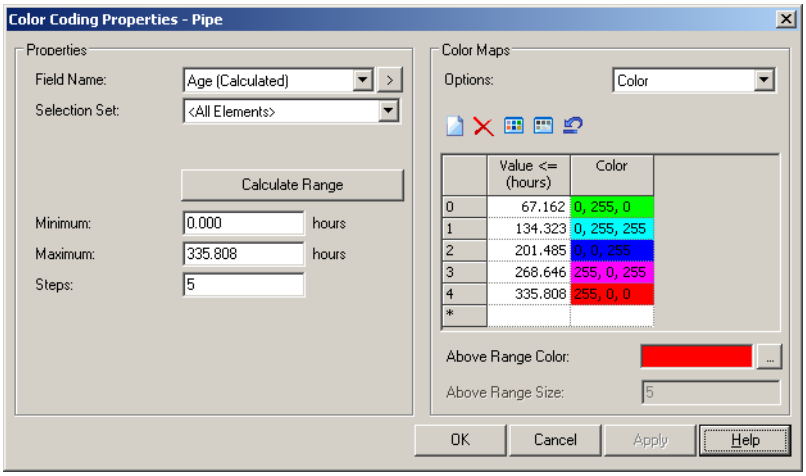
j. Close the properties editor.

5. Click **Analysis** > **Scenarios** tab and make **Age Analysis** current by highlighting it and clicking the **Make Current** button  .



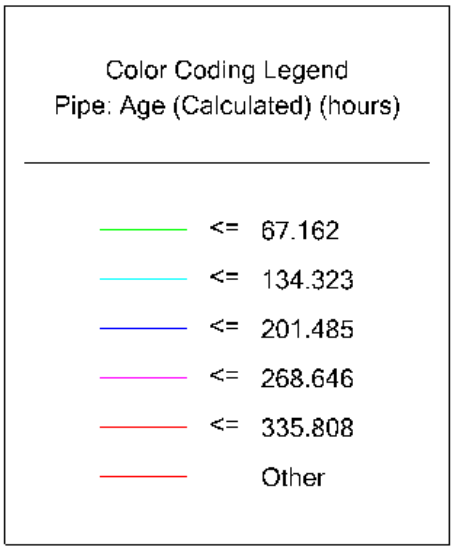
6. Click **Compute**  and then close the **Calculation Summary**.
7. Click **View** > **Element Symbology** manager.
8. Right-click on **Pipe** and select **New** > **Color Coding**.
9. Select **Age (Calculated)** as the **Field Name**.
10. Click Calculate Range  and select **Full Range**.
11. Click **Initialize**  to set up a default color scheme. Accept this default scheme.

    If you get a message about Bentley WaterCAD V8*i* being unable to determine the limits for mapping, make sure that Age Analysis is selected in the Scenario drop-down list, in the toolbar.
12. Click **Apply**.



13. Click **OK**.

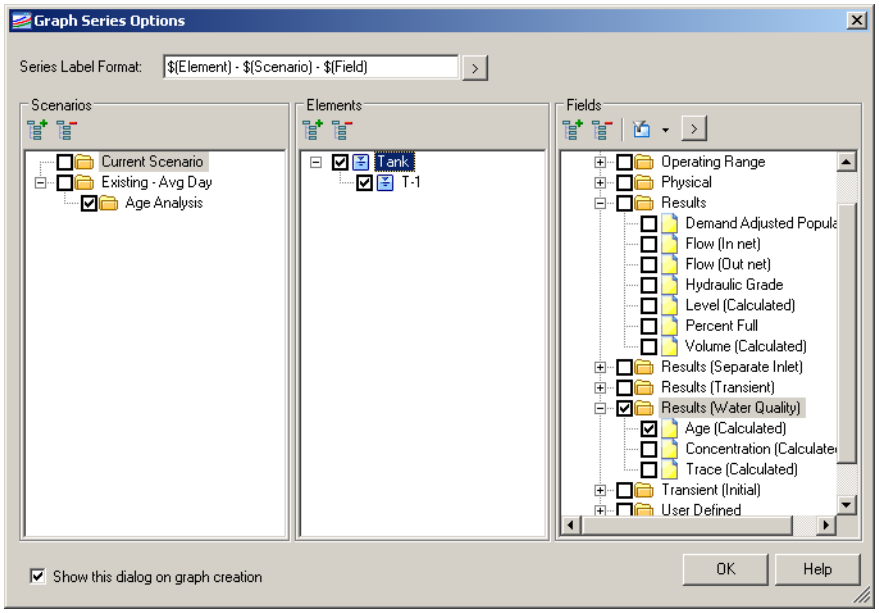
14. In the Element Symbology manager, right-click on **Age (Calculated)** and click **Insert Legend**. Click in the drawing to place the legend.



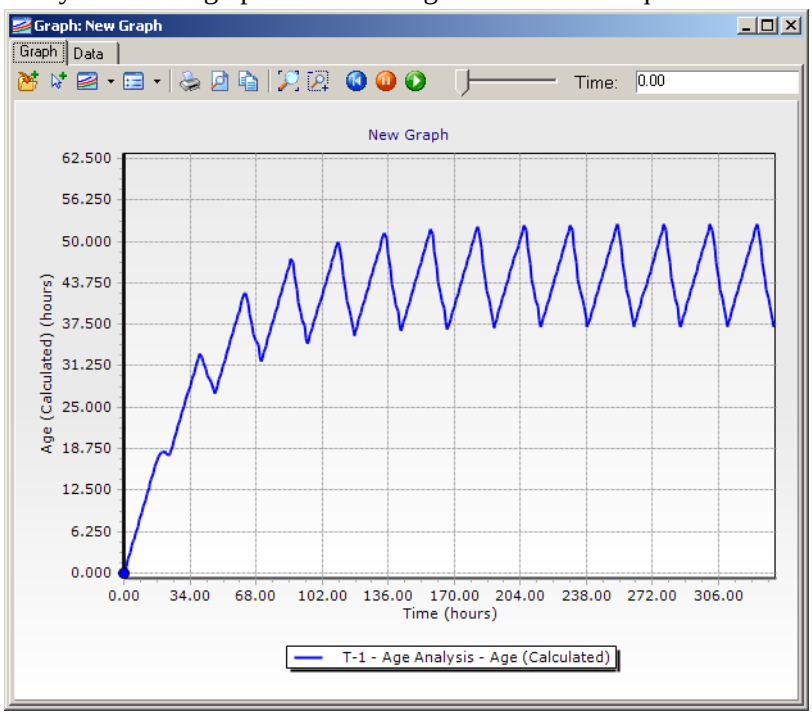
15. A good way to check if your network has had sufficient time to reach an equilibrium point is to look at **Age vs. Time graphs** for your elements.

    a. Right-click on Tank **T-1** and select **Graph**

    b. In the Graph Series Option dialog make sure that **Age Analysis** is checked in the **Scenarios** column, **Tank** and **T-1** are checked in the **Elements** column, and **Results (Water Quality)** and **Age (Calculated)** are checked in the **Fields** column. Uncheck all other boxes.



c. Click **OK**.
From the graph, you can see that once a repeating pattern is reached, the age of the water fluctuates between approximately 38 and 52 hours in 24-hour periods. Looking at these equilibrium ranges for various nodes can help guide you in setting up initial water age values in subsequent runs.



d. Close all open dialogs.

Step 2: Analyzing Constituent Concentrations

In this portion of the lesson, you will look at chlorine residuals in the system over time. Bentley WaterCAD V8*i* stores information on constituent characteristics in a file called a constituent library. You will add information for chlorine to this library, set up initial concentrations in the system, and run the simulation.

1. Choose **Analysis > Alternatives**.
2. Click the Constituent Alternative and click New.
3. Name the new alternative **Chlorine Injection** and double-click to open.
4. Click the **Ellipsis (…)** next to the Constituent drop-down menu to open the Constituents manager.

5. Click the already created Chlorine Label and enter the data below into the dialog box.

Label	Chlorine
Bulk Reaction Rate	-0.10 (mg/L)^(1-n)/day
First Order Wall Reaction Rate	-0.08 m/day
Diffusivity	1.2e-9m^2/s

6. Check the **Unlimited Concentration** box, and close the dialog.
7. Click **Close** to exit the Constituents window. You should now be back in the Constituent Alternative Editor.
8. Select **Chlorine** from the Constituent list box.
9. On the same **Constituent Alternative-1** editor window, go to each tab for each of the different valves as well as the pump tab and set the **Concentration (Initial)** for each to **1 mg/l**.
10. Click the **Junction** tab, and initialize the chlorine concentrations by entering a value of **1 mg/l** at each junction node. (Right-click the column heading and use Global Edit to set the **Concentration (Initial)** fields.)
11. In the Reservoir tab, enter a **Concentration (Initial)** value of **2.0 mg/l** for the reservoir.
12. Set the tank's **Concentration (Initial)** to **0.5 mg/l**.
13. Close the **Editor** and the **Alternatives Manager**.
14. Now, open the **Scenario Manager** (Analysis > Scenarios) and set up a new Scenario in order to run the Constituent Analysis.
    a. Create a new Child off of the Age Analysis Scenario by highlighting it and clicking **New > Child Scenario**.
    b. Enter **Chlorine Analysis** as the new scenario name.
    c. Double click the newly created **Chlorine Analysis** scenario to open its property grid. In the **Constituent Alternative** dropdown, select **Chlorine Injection**.
15. Click the **Analysis** menu and select **Calculation Options**. Double-click **Existing - Avg Day** to open the Properties grid and set the **Calculation Type** field to **Constituent**.
16. Click **Close** to exit the dialog box.

17. In the **Scenarios Manager**, click **Compute Batch Run**.
18. Deselect **Age Analysis**.
19. Select **Chlorine Analysis**, then click **Batch** to run the model.
20. Click **Yes** and **OK** to accept the message boxes.
21. Select **Chlorine Analysis** as the current Scenario.
22. Close the **Scenario Manager** dialog box.
23. Set up color coding. This time, color code by Calculated Concentration instead of Calculated Age. Scroll through the time steps to view how the concentrations change throughout the network. When you look at your results using color coding, tables, and graphs, try to discover what better initial values for chlorine concentration might be.

Step 3: Performing a Trace Analysis

A trace analysis determines the percentage of water at all nodes and links in the system from a specific source node (the trace node). In systems with more than one source, it is common to perform multiple trace analyses using the various source nodes as the trace nodes in successive analyses. For this run, you will perform a trace analysis to determine the percentages of water coming from the tank.

1. Click **Analysis > Alternatives**.
2. Click the Trace alternative to highlight it.
3. Click **New**.
4. Name the new alternative **Trace Analysis for Tank**, and double-click it to open the alternative editor.
5. In the Trace Element list box, select the tank, **T-1** (click the ellipsis button to select it from the drawing).
6. Close the editor.
7. Close the Alternatives Manager.

8. Next, set up a new scenario to run an Extended Period Simulation incorporating the new alternative.
    a. Select **Analysis > Scenarios**.
    b. Create a new child for the Age Analysis scenario by highlighting it and clicking **Add > Child Scenario**.
    c. Enter **Trace Analysis** as the new scenario name.
    d. Double-click the new scenario to open the Properties editor. Change the **Age** Alternative to **Not Considered**. Change the **Trace** Alternative to **Trace Analysis for Tank**.

e. Close the Properties editor.

f. Click **Analysis** > **Calculation Options**. Double-click **Existing - Avg Day** and change the **Calculation Type** to **Trace**.

g. Click **Close** to exit the dialog box.

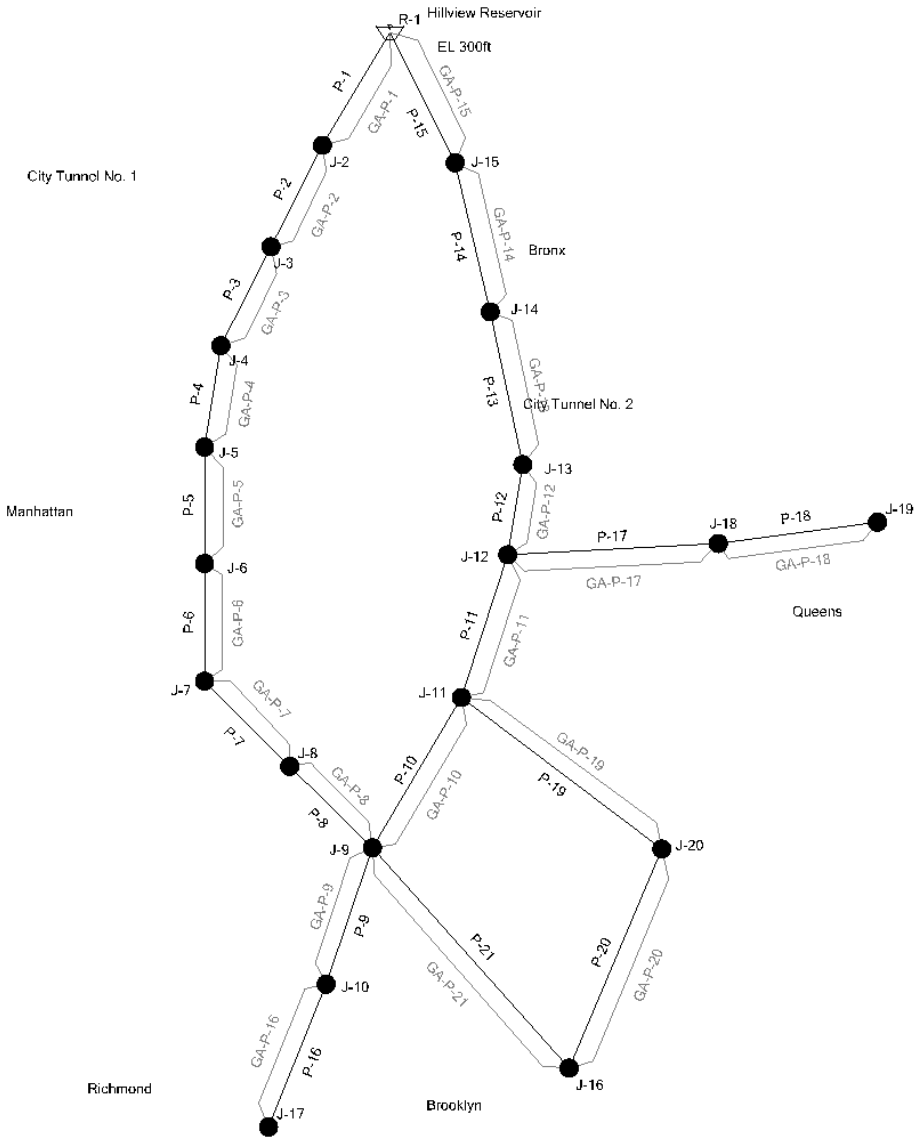
9. Click **Analysis** > **Scenarios**, then **Compute** > **Batch Run**.
10. Select the new **Trace Analysis** scenario, make it the current scenario, and click **Batch**.
11. Use color coding (by Calculated Trace), tables, and graphs to view the results of this run. As you scroll through the time periods, notice how the colors spread outward from the tank during periods when the tank is draining, and recede when the tank begins to fill. For more information on reporting features, Reporting Results.
12. Close the open dialog boxes and save this project.

Darwin Designer to Optimize the Setup of a Pipe Network

In this lesson, you use Darwin Designer to optimize the setup of a pipe network.

Step 1: Creating the Darwin Designer Optimization

1. In Bentley WaterCAD V8*i* choose File > Open.
2. Browse to the WaterGEMS\Samples\Designer directory and open **DesignerSample1.wtg**.

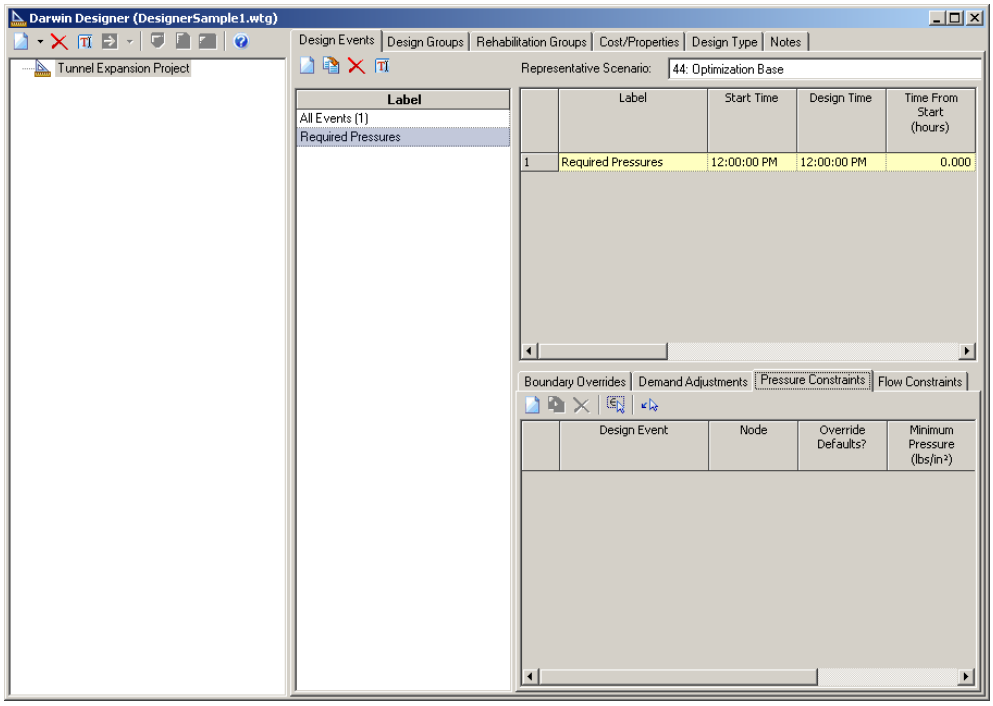


3. Click **Analysis** > **Darwin Designer**.
4. In the Darwin Designer window, click **New** > New Designer Study.
5. Highlight the new design and click the Rename button. Enter **Tunnel Expansion Project**.
6. If needed, click the **Design Events** tab.

7. In the Representative Scenario field, select **Optimization Base** from the drop-down list.

8. Click **New**.

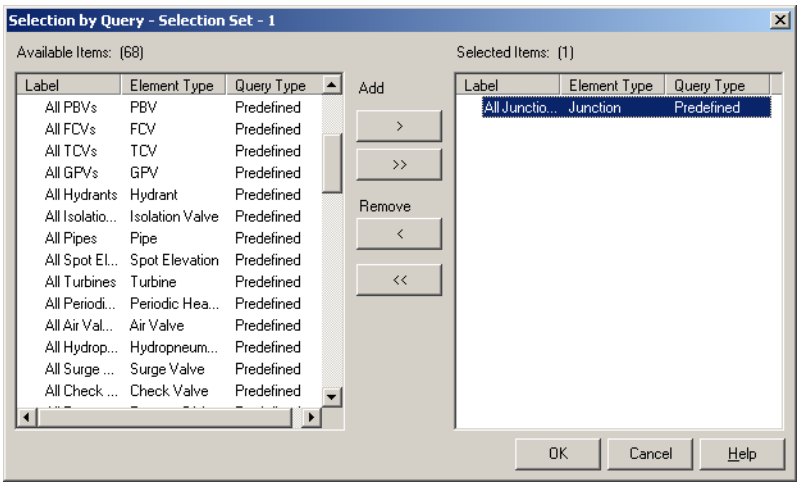
9. Highlight the new design event and click the Rename button. Enter **Required Pressures**, and click **OK**.



10. Set pressure constraints for all junctions.

    a. First, create a new selection set containing all of the junctions in the model. Click **View** > **Selection Sets**.

    b. Click **New** > **Create From Query**. Double-click the **All Junctions** query, then click **OK.**

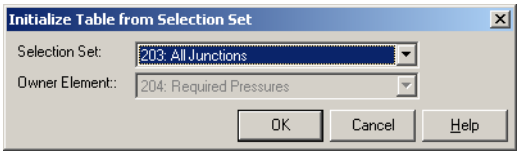


    c. Rename the new selection set **All Junctions**.

    d. Back in Darwin Designer, click the **Pressure Constraints** tab.

    e. Click the **Initialize Table from Selection Set** button.

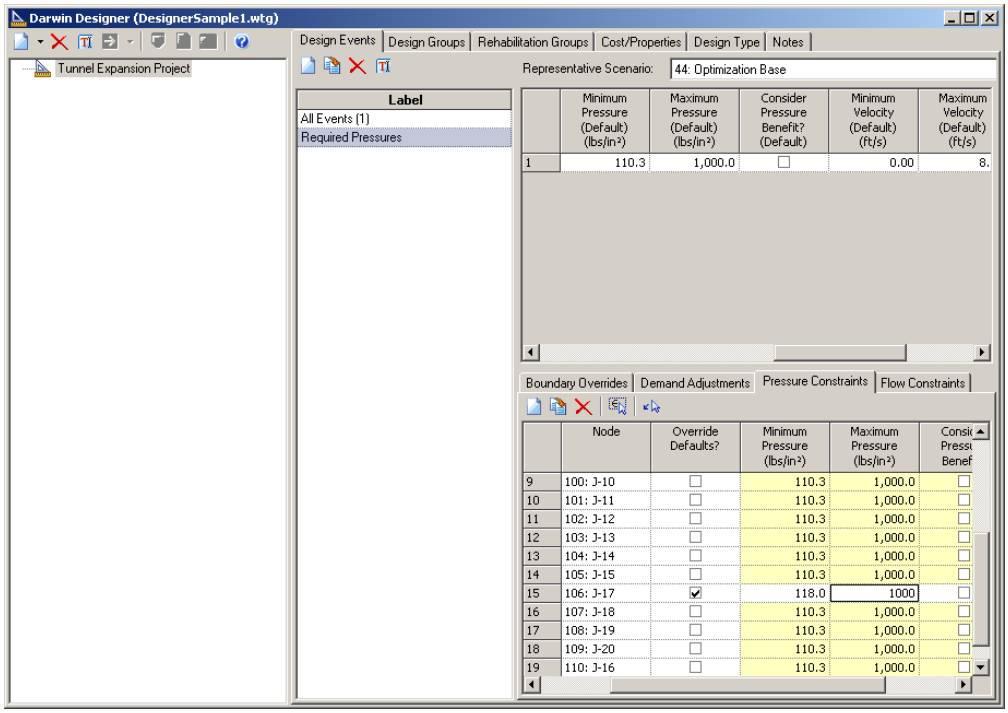
    f. Select **All Junctions** from the **Selection Set** drop-down list, then click **OK**.



    g. In the table in the upper right of the Designer dialog, set the **Minimum Pressure (Default)** value to **110.33 psi** (HGL = 255 ft.).

    h. In the table in the upper right of the Designer dialog, set the **Maximum Pressure (Default)** value to **1000 psi**. For this example, maximum pressure is not a consideration, so if you set it to a high value it won't affect the calculations.

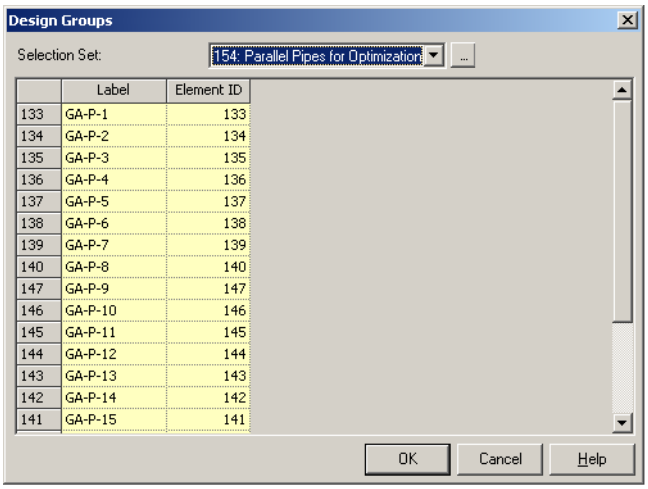
11. Customize junction J-17 to require a minimum pressure of **118.03 psi**.
    a. In the Pressure Constraints area, scroll so you can see junction **J-17**.
    b. Select the **Override Defaults?** check box.
    c. Type a minimum pressure of **118.03 psi**. and a **Maximum Pressure** of **1000 psi**.



12. Click the **Design Groups** tab.

13. Click **Select Elements for Design Group** button  . This button lets you automatically create one design group for each pipe in the network or for a particular set of pipes.

    a. In the Selection Sets drop-down list, select **Parallel Pipes for Optimization**. This highlights a selection set containing a specific subset of the pipes in your network.



    b. Click **OK**.

14. Add a option group for your optimization.

    a. Click the **Cost/Properties** tab.

    b. Highlight **New Pipe** in the tree-view.

    c. Click **New > Design Option Groups**.

    d. Name the new table **New Pipe Sizes**.

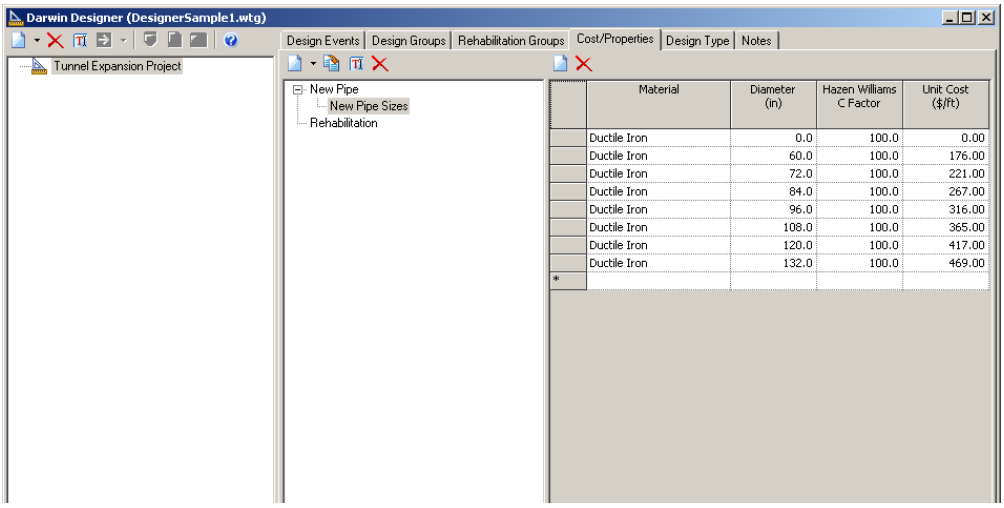
    e. Type the following pipe material, size, roughness coefficient, and cost:

New Pipe Parameters

Material	Diameter (in.)	Hazen Williams C Factor	Unit Cost ($/ft)
Ductile Iron	0	100	0.00
Ductile Iron	60	100	176.00
Ductile Iron	72	100	221.00
Ductile Iron	84	100	267.00

New Pipe Parameters

Material	Diameter (in.)	Hazen Williams C Factor	Unit Cost ($/ft)
Ductile Iron	96	100	316.00
Ductile Iron	108	100	365.00
Ductile Iron	120	100	417.00
Ductile Iron	132	100	469.00



15. Create a new optimized design run.

    a. In the **Designs** tree-view, right-click **Tunnel Expansion Project** and select **New > New Optimized Design Run**.

    Or, click the **New** button and select **New Optimized Design Run**.

    b. Name the design run **Optimized Design**.

16. Select the design event you want to use, **Required Pressures**, by making sure the **Is Active?** check box is checked.

17. Click the **Design Groups** tab.

    a. Make sure the **Is Active?** check boxes for all of the design groups are checked.

    b. Right-click the **Cost/Properties** column heading.

    c. Select **Global Edit**.

d. Choose **New Pipe Sizes** as the option group you want to use and click **OK**.

18. Click the **Options** tab.

a. Set the GA Parameters as follows (most of these are the default settings, with the exception of Random Seed and Penalty Factor):

GA Parameters

GA Parameter	Value
Maximum Era Number	6
Era Generation Number	150
Population Size	50
Cut Probability	1.7
Splice Probability	60.0
Mutation Probability	1.5
Random Seed	0.4
Penalty Factor	25000000

b. Set the Stopping Criteria as follows:

Stopping Criteria

Stopping Criteria	Value
Maximum Trials	50000
Non Improvement Generations	200

c. Set the Top Solutions, Solutions to Keep to **3**. This sets how many results will be available as results (see Step 2: Viewing Results later in the lesson).

19. Click **Compute** to calculate the optimized design.

While the calculation proceeds, Bentley WaterCAD V8*i* displays the Darwin Designer Run Progress dialog box, which displays the following information:

– Fitness—In this case, you were calculating based on cost. So, the best fitness is the least costly solution that the GA (Genetic Algorithm) found.

– Cost ($)—The lowest cost found by the calculation displays here.

- Benefit—Measured pressure improvement in the network. This is 0 because the lesson only considers cost and not pressure benefit.
- Violation—The largest violation of established pressure and flow boundaries, such as maximum or minimum pressures, displays here. If there were a violation, you would use the results area Pressure and/or Flow tabs (in the results pane of the main Darwin Designer window) to look for the actual violations.
- Generations—The maximum value for generations is determined by the Maximum Era Number and Era Generation Number you set in the Options > GA Parameters. The actual number of generations that get calculated depend on the Options > Stopping Criteria you set.
- Trials—The maximum value for trials is determined by what you set in Options > Stopping Criteria. Note that you can set a number larger than (Maximum Era Number)*(Era Generation Number)*(Population Size), but calculations beyond that number (for this example, the value is 45,000) are less likely to produce significant improvements.

  Also, note that the Messages tab might report you exceeded the maximum number of trials. This is usually because Darwin Designer must complete all of the generations before ending a trial, so it is possible that completing generations will cause a few excess trials to be calculated.

20. After the calculation is finished, click **Close** to close the Darwin Designer Run Progress dialog box.

Step 2: Viewing Results

After you calculate the optimized design results display. You can review results and look for violations of parameters.

1. From the hierarchy pane, you can click on the Solutions folder or any of the individual solutions for more detail. Select the solution you want to see: **Solution 1**.

   You can click the **Graph** button  to view the solutions plotted; each solution is color coded; use the color code as a key when viewing graphs.

   Solutions are ranked by fitness, with Solution 1 being the best.

2. In the Solutions tab, if you scroll down, you can see there are seven pipes that changed from the default. These are the pipes that Darwin added to the scenario to provide the optimal solution:

New Pipes

Pipe	Diameter (in.)	Cost
GA-P-3	72	1613300.00
GA-P-7	120	4003200.00
GA-P-16	96	8342400.00
GA-P-17	96	9859200.00
GA-P-18	84	6408000.00
GA-P-19	72	3182400.00
GA-P-21	60	4646400.00

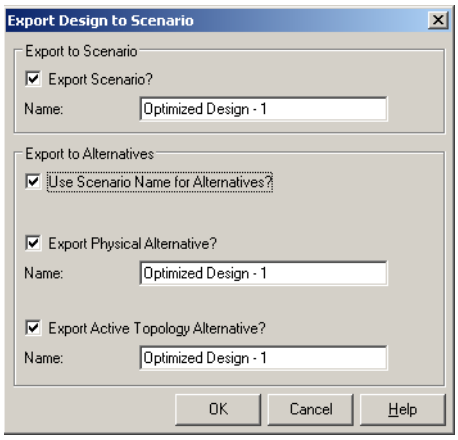
3. The Rehabilitation Groups and Flow results under the Simulated Results tab are empty because this lesson does not use those.
4. Click the **Pressure** results under the Simulated Results tab. This displays the maximum and minimum pressure constraints you set on the junctions and the actual pressures calculated by Darwin Designer.

Step 3: Using Results

After you calculate the optimized design results display. You can use the results to create graphs and reports.

1. Solution 1 clearly provides the least expensive solution. Export the solution to Bentley WaterCAD V8*i* so you can use it.
   a. Select **Solution 1** in hierarchy under the Solutions folder.
   b. Click the **Export to Scenario** button  . The Export Design to Scenario dialog box opens.
   c. Select all check boxes to export to the various alternatives.

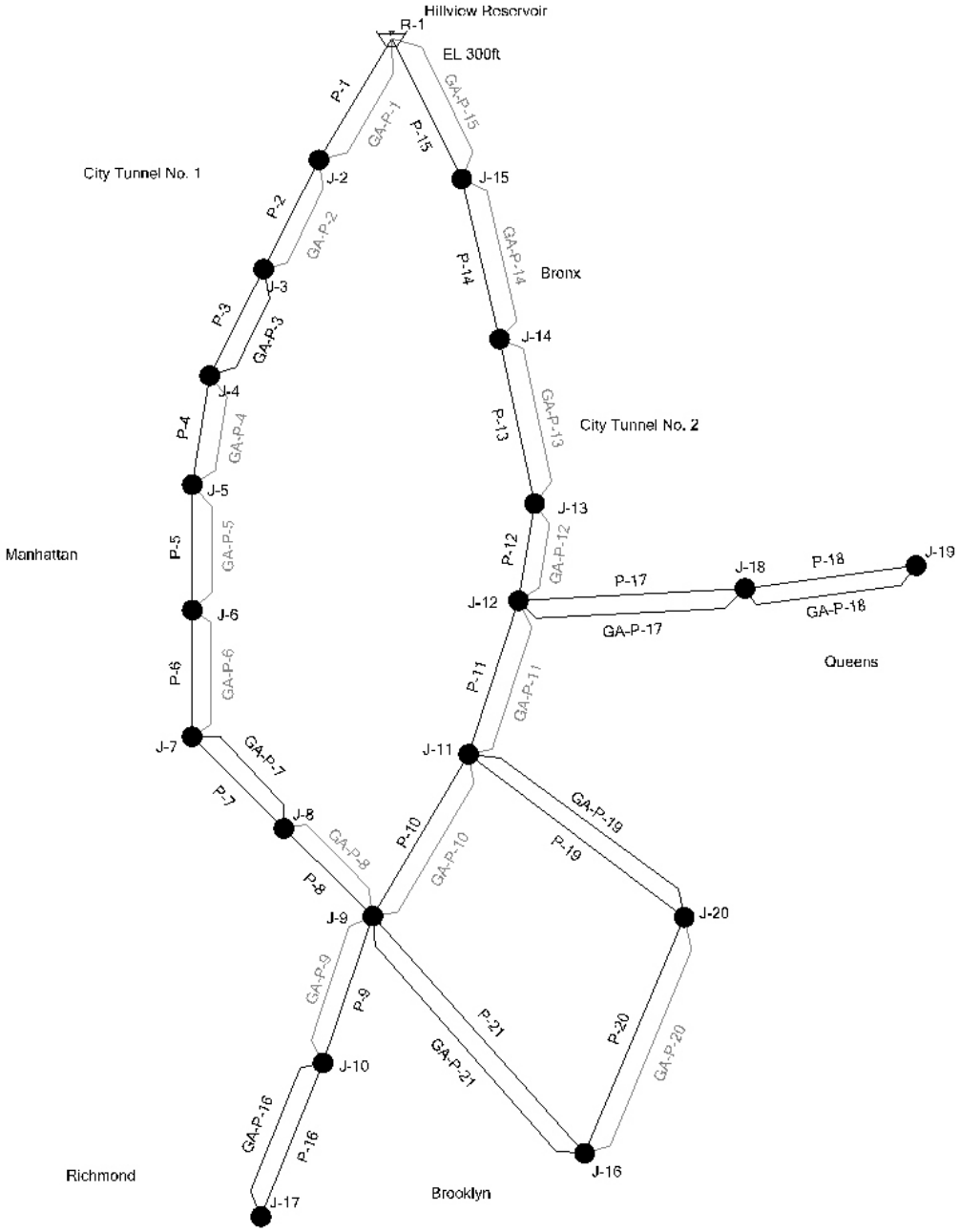
d. Name the scenarios you want to export, such as **Optimized Design - 1**. The name you choose must be unique; there cannot already exist a scenario with the same name.



e. Click **OK**.

2. Click **Close** to close Darwin Designer.

3. In Bentley WaterCAD V8*i* , select the scenario you exported from the **Scenario** drop-down list. Notice the parallel pipes that have been added to the base network. These are the pipes that meet the optimized design calculated by Darwin Designer.



Darwin Designer to Optimize a Pipe Network

In this lesson, you use Darwin Designer to optimize the setup of a pipe network.

There are three scenarios:

- **Existing System** representing current system conditions
- **Future Condition** representing the system expansion layout
- **Optimization Base** representing the base scenario that Designer will optimize.

There are two design goals:

- New pipes to be sized are pipes 54, 68, 70, 72, 74, 76.
- Old pipes need to be rehabilitated by applying possible actions including cleaning pipe, relining pipe, and leaving the pipe as it is (no action or do nothing to a pipe).

The design criteria are:

- Minimum pressure of 45 psi at all demand junctions
- Maximum pressure of 100 psi at all demand junctions
- Filling each tank to or above the initial tank level

Getting Started

1. Browse to your \Bentley\WaterGEMS\Samples\Designer directory, and open **DesignerSample2.wtg**.
2. If needed, select **Existing System** from the Scenario drop-down list and click the Make Current button. This displays the current network.

   Notice that the Existing scenario comprises two types of pipe:

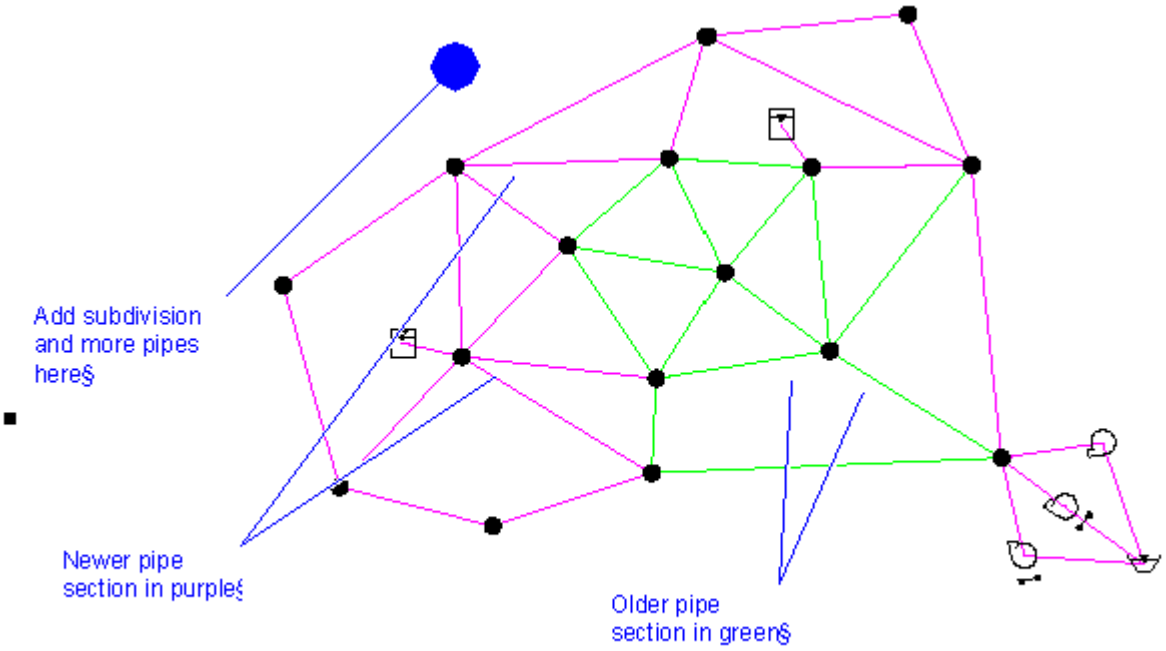
   – In green, there are older pipes, perhaps representing an old downtown section

   – In purple, there are newer pipes, perhaps representing newer additions to the water supply network

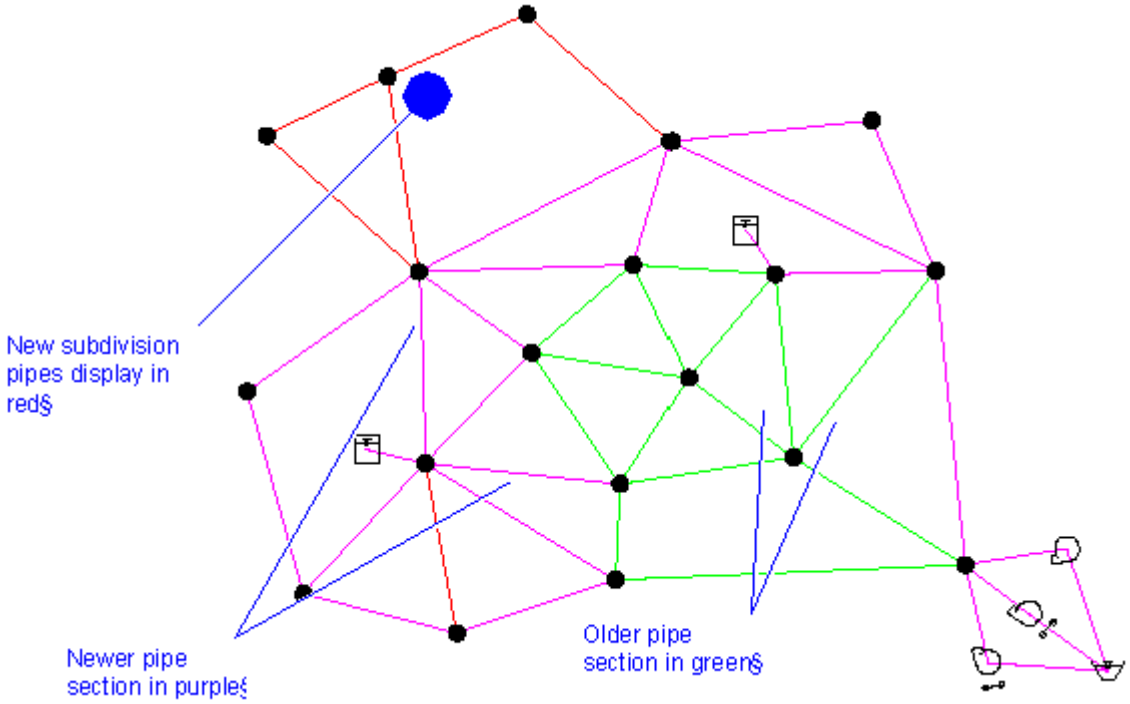
   Note: The Future conditions portion of the model may display in gray for you. You can control visibility of inactive elements by toggling the "Display inactive topology" setting under Tools > Options > Global.

3. Click **Compute** to calculate the system pressures and tank levels for the Existing Condition.

If you want, you can inspect the pressures and tank volumes, but the purpose for calculating this condition was for a tank level comparison between the Existing and Future Condition scenarios in a later step.



4. Close the Calculation Summary and User Notifications windows.
5. Select **Future Condition** from the Scenario drop-down list. If needed, click **Zoom Extents**  to view the entire network in the window.



6. Click **Compute** to calculate the system pressures and tank levels for the Future Condition.

7. Close the Calculation Summary and User Notifications windows.

8. Review the pressure at junctions using color coding.

   a. Click **View > Element Symbology**. Right-click on Junction in the list and select **New** > **Color Coding**. The Color Coding dialog box opens.

   b. Set the **Field Name** to **Pressure**.

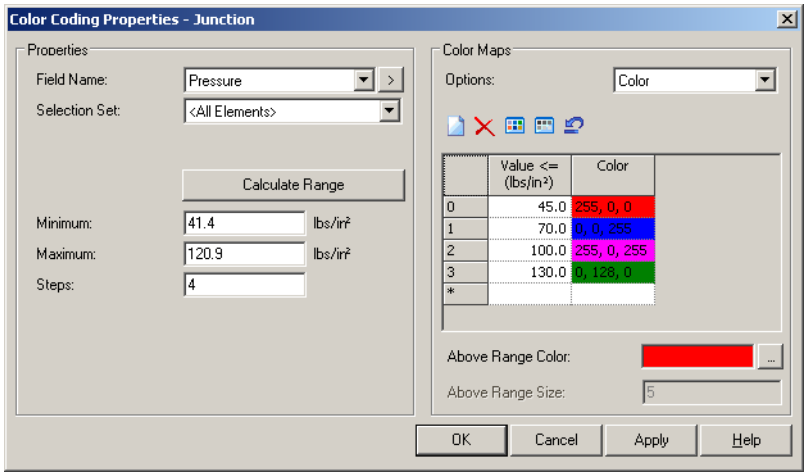
   c. Click the **Calculate Range** button and select **Quick Range**. Change the number of **Steps** to **4**.

d. In the Color Maps section, click the **New** button. Set the **Value** <= for this first row to **45** psi and the **Color** to **Red**. Create 3 more rows using the following data:

Junction Color Coding Settings

Value <=	Color
70	Blue
100	Magenta
130	Green

e. The Color Coding dialog should now look like this:



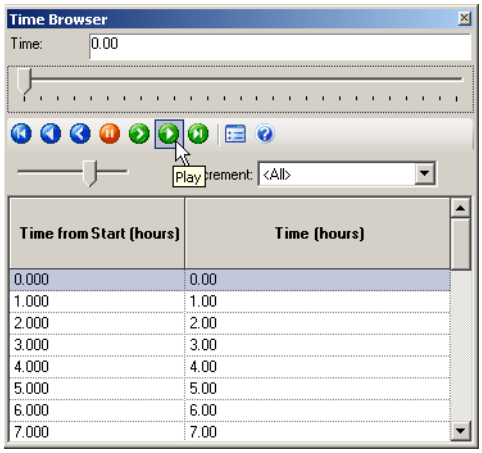
f. Click **OK** to apply the color coding and close the dialog.

For this lesson, one objective is to keep the junction pressures above 45psi. So, when you run the simulation, watch for red junctions which indicate unacceptably low pressure.

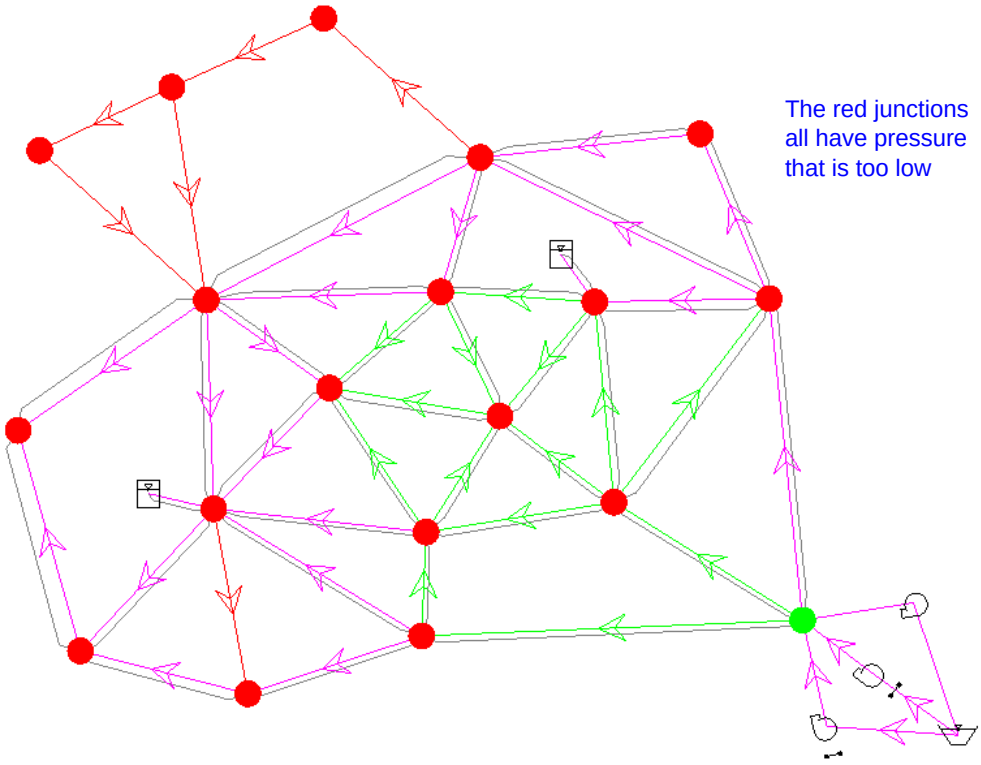
9. Run an animation to see what happens in the network over the course of 24 hours.

a. Click **Analysis** > **Time Browser**.

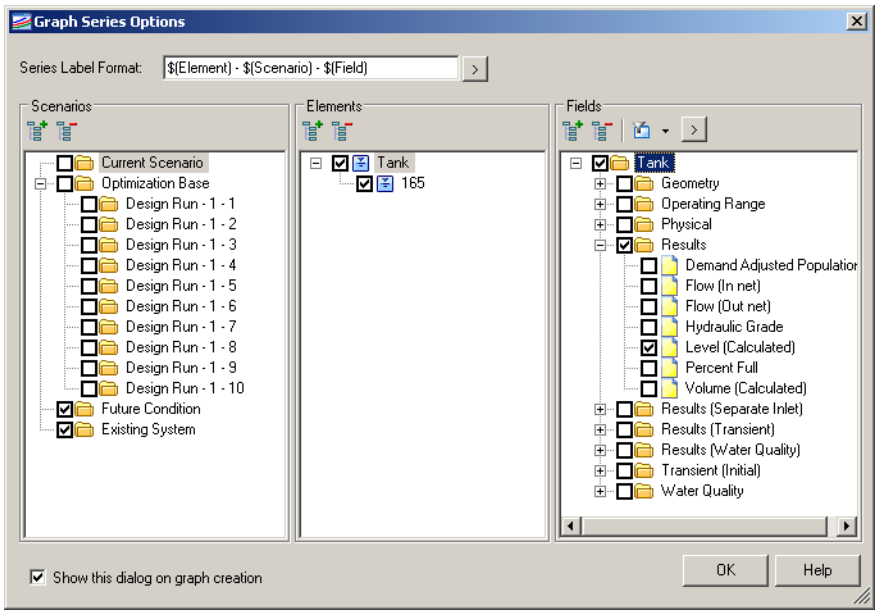
b. Click **Play** to run the animation.



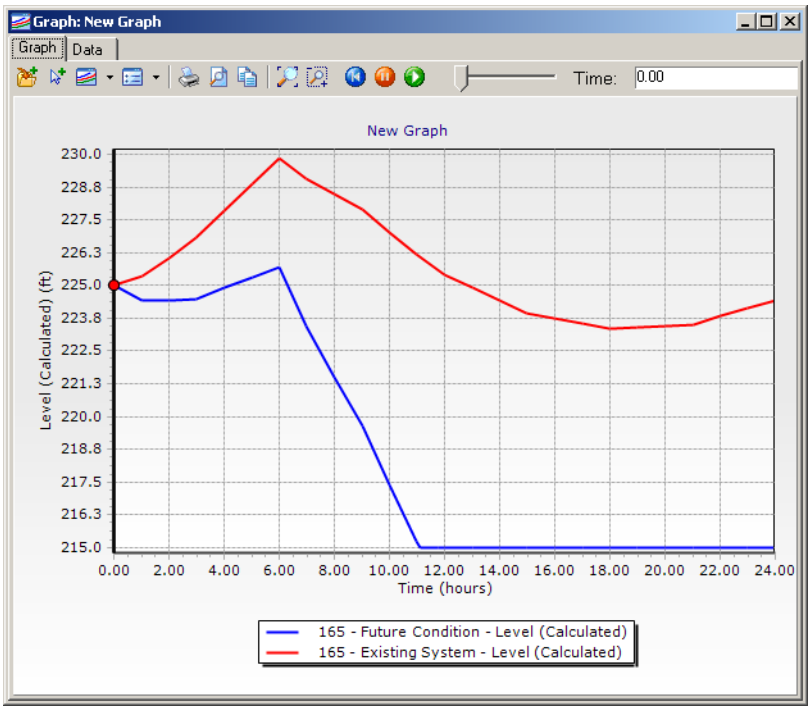
c. Notice, at hour 6 there is a low pressure junction and by hour 12, most of the junctions are showing a low pressure.



10. Use Graphs to check the levels of the tanks.
    a. Right-click the tank labeled **165** and select **Graph**.
    b. We want the graph to show the water levels for tank 165 in the Existing scenario and also the Future Condition scenario. In the Graph Series Options dialog, check the box for **Existing System** in the **Scenarios** list pane.
    c. In the **Fields** list pane, uncheck **Flow (Out Net)** and check **Level (Calculated)**.



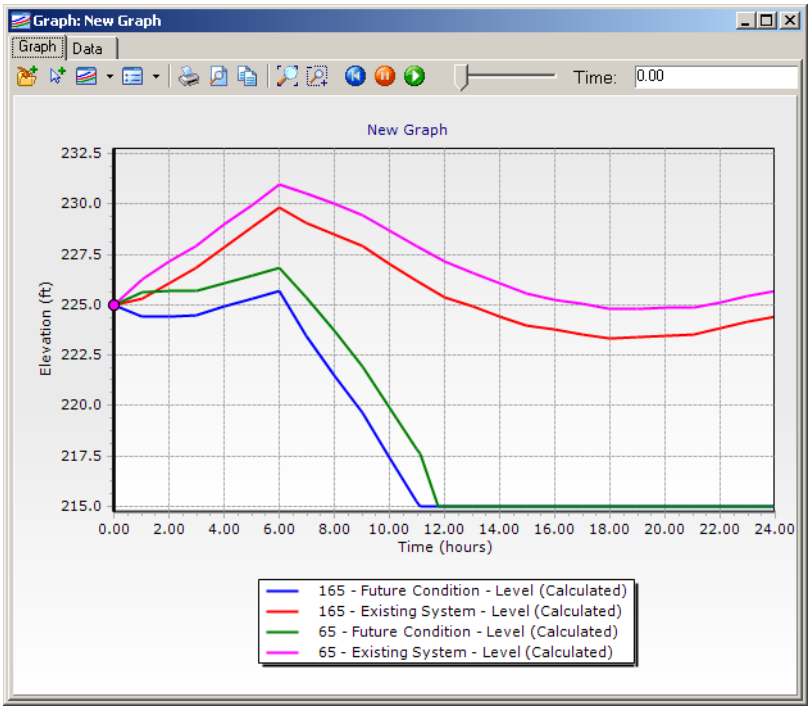
d. Click **OK**.



e. Notice that by hour 11, Tank 165 is empty and does not refill.

f. Click the **Add to Graph** button. 

g. In the drawing pane, click tank **65** then right-click and select **Done**.



h. Notice that by hour 12, Tank 65 is also empty.

i. Close the graph window.

11. You need to use Darwin Designer and some analysis in Bentley WaterCAD V8*i* to change the existing pipe network to:

– Keep junction pressures above 45psi

– Keep the two water tanks filled

Set Up for Darwin Designer

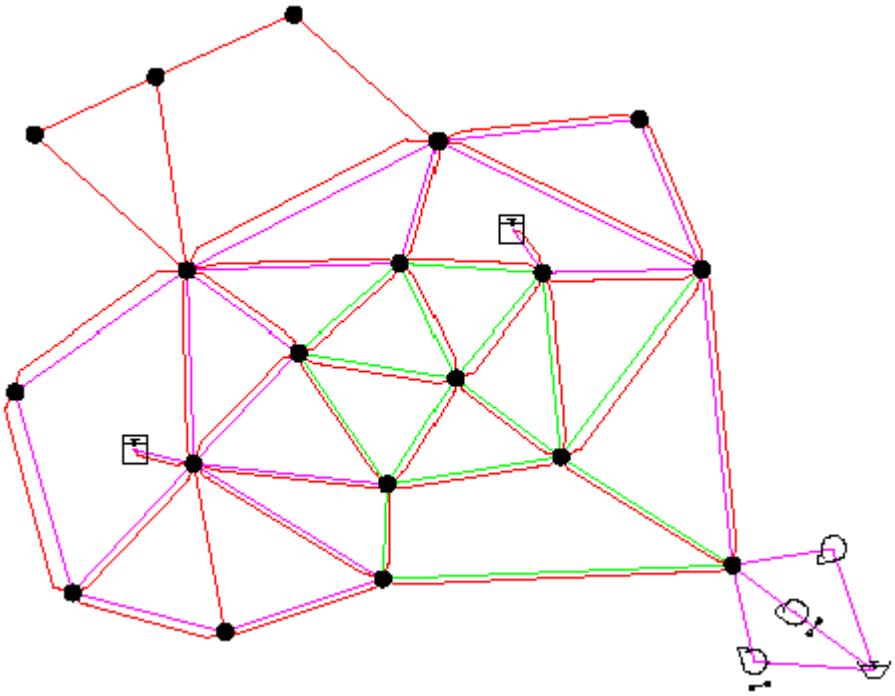
With Darwin Designer, you need to consider two ways of accomplishing a cost-effective design: create new or parallel pipes and rehabilitate existing pipes. Clearly, the new subdivision will get new pipes. And, as you can design an appropriate size for these new pipes, there is no need for parallel pipes and there are no existing pipes on which to perform rehabilitation.

With that in mind, you would create a parallel pipe option for all existing pipes. This parallel pipe option should include a variety of sizes so Darwin Designer has flexibility to choose the most efficient size. Additionally, the pipe sizes must include a 0 diameter, which lets Darwin Designer calculate the efficiency of the system with the pipe absent (without installing the parallel pipe). There are four options in this tutorial for existing pipe:

- Install parallel pipe
- Clean existing pipe
- Reline existing pipe
- Take no action

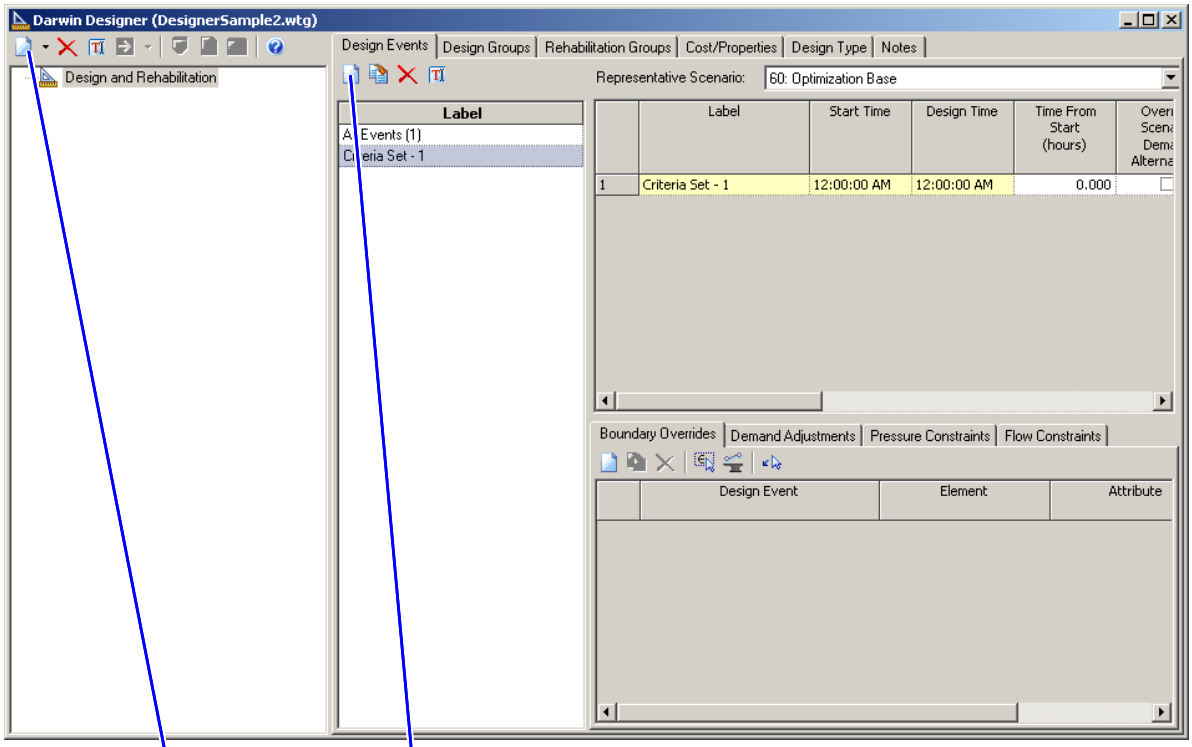
1. Select **Optimization Base** from the Scenario drop-down list.

   This is the future network set up for Darwin Designer optimization. Notice that parallel pipes have been added next to all the existing pipes. All new pipes—parallel and new ones for the subdivision—are colored red.



2. Click **Analysis** > **Darwin Designer**.
3. Create a new designer study, called **Design and Rehabilitation**.
   a. Click the **New** button and select **New Designer Study**.
   b. Rename the study **Design and Rehabilitation**.
4. If needed, select **Optimization Base** from the Representative Scenario drop-down list.

5. Create a new design event, called **Criteria Set - 1**.
   a. In the **Design Events** tab click **New**.
   b. Highlight the new design event and click **Rename**.
   c. Enter the name **Criteria Set - 1 and click OK.**

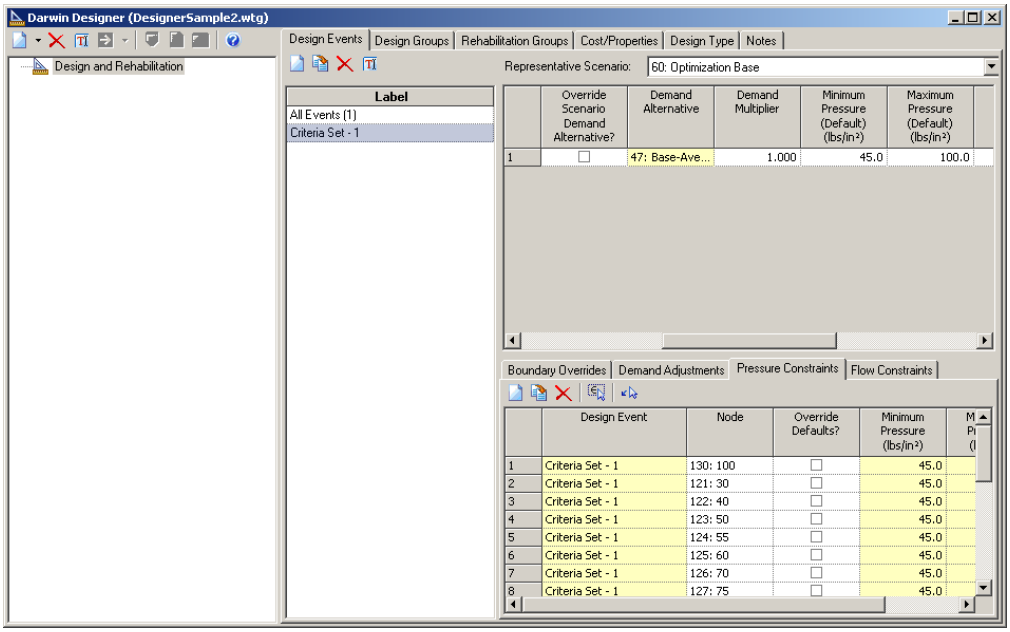


6. Set up the Design Event.
   a. Scroll to the right and set the default minimum and maximum pressure constraints:
      - **Minimum Pressure (Default)** to **45** psi
      - **Maximum Pressure (Default)** to **100** psi.
   b. Click the **Pressure Constraints** tab at the bottom.
   c. Click the **Select From Drawing** button. 

d. In the **Select** toolbar, click the **Query** button and select **Network** > **All Junctions**. Then right-click and select **Done**.



e. Note that the Pressure Constraints table now contains entries for each junction in the model.



7. Click the **Design Groups** tab.
8. Click **New** to create design groups. You need to create design groups for all new or potentially new pipes, which include:
   - All pipes labeled in the model with a **P** (these are parallel pipes)
   - All new pipes: 54, 68, 70, 72, 74, 76

   Do not include existing pipes in any of these groups, because these need to be in a rehabilitation group.
9. Click the **Rehabilitation Groups** tab. Create rehabilitation groups containing pipes grouped as follows:
   - 4, 8, 30, 32, 34 36
   - 2, 10, 12, 14, 16, 18, 20, 22, 24, 26, 28, 48
   - 6, 78
   - 38, 40, 42, 66

– 44, 46, 50, 58, 62, 80

– 52, 56, 60, 64

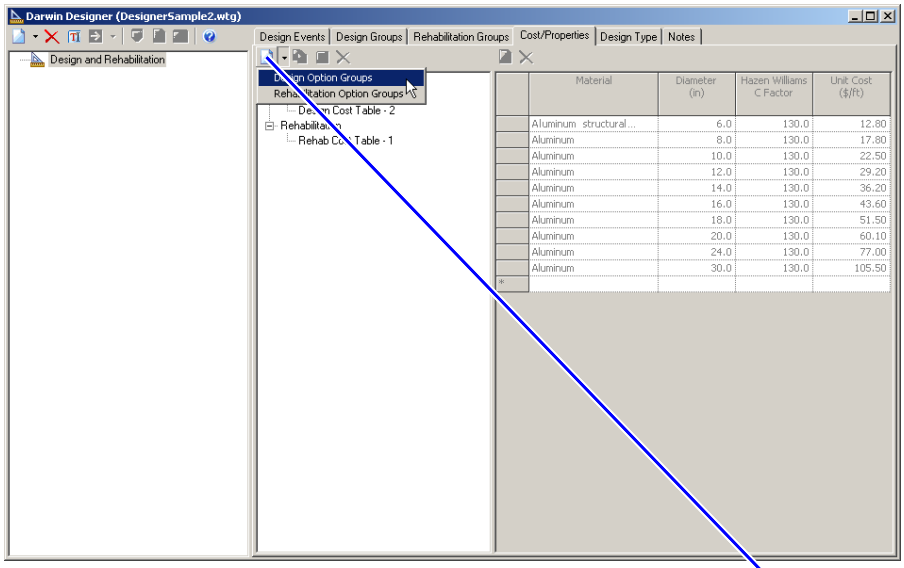
You might consider grouping pipes based on size or age. To create a Rehab group:

a. Click **New**.

b. If desired, rename the Rehab group and click **OK**.

c. Click the **Select Elements for Demand Group** button to choose the pipes you want to include in the group.

10. Click the **Cost/Properties** tab. Create two design option groups and one rehabilitation option group.



d. Click **New** > **Design Option Groups** to create a new table.

e. Rename the table **Design Cost Table - 1**.

f. Enter the data below into the table. The first table contains a pipe diameter of 0. All parallel pipes will use this option group. Including a diameter of 0 lets Darwin Designer consider not adding a parallel pipe if that pipe is not needed for the optimal solution.

Design Cost Table - 1

Material	Diameter (in.)	Hazen Williams Roughness	Unit Cost ($/ft.)
Aluminum structural plate in 32 CR	6	130	12.80
Aluminum	8	130	17.80
Aluminum	10	130	22.50
Aluminum	12	130	29.20
Aluminum	14	130	36.20
Aluminum	16	130	43.60
Aluminum	18	130	51.50
Aluminum	20	130	60.10
Aluminum	24	130	77.00
Aluminum	30	130	105.50
Aluminum	0	130	0.00

g. Create a second design costs table named **Design Cost Table - 2**. (You can duplicate the table you just created and delete the row for 0 diameter.) This table is the same as the first one except it does not have a pipe diameter of 0 and is used for new pipes. New pipes must have a minimum diameter because their existence is a requirement, unlike the parallel pipes.

Design Cost Table - 2

Material	Diameter (in.)	Hazen Williams Roughness	Unit Cost ($/ft.)
Aluminum structural	6	130	12.80
Aluminum	8	130	17.80
Aluminum	10	130	22.50
Aluminum	12	130	29.20
Aluminum	14	130	36.20
Aluminum	16	130	43.60
Aluminum	18	130	51.50
Aluminum	20	130	60.10
Aluminum	24	130	77.00
Aluminum	30	130	105.50

11. Create a single rehabilitation option groups table containing three actions: Clean, Relining, and Do Nothing. A do-nothing action is necessary so Darwin Designer can consider not rehabilitating some pipes. Each of these actions must reference three functions, one for each column in the table.
12. Click **New** > Rehabilitation Option Groups to create a new rehabilitation option table.
    a. Rename the table **Rehab Cost Table - 1**.
    b. Type the name of an action you want to create, such as **Clean**.
    c. Click the cell under Pre-Rehabilitation Diameter Vs. Post-Rehabilitation Diameter and click the **Ellipsis (…)** button to create a new function. The Rehabilitation Functions manager opens.
    d. Click **New > New Pre-Rehabilitation Diameter Vs. Post-Rehabilitation Diameter Function**.
    e. Name the function, **Function - 0**.

f. Enter your diameter data (inside pipe diameter) into the table on the right side of the dialog. We recommend you included all of the diameters of pipe in the table. (If you do not, Darwin Designer will use interpolation to calculate the diameters you do not include.) In this case, the function does not change the diameter of any pipes.

Function - 0 Diameter Data

Pre-Rehab Diameter (in.)	Post-Rehab Diameter (in.)
6	6
8	8
10	10
12	12
14	14
16	16
18	18
20	20

13. In the Rehabilitation Functions manager, click **New > Pre-Rehabilitation Vs. Post-Rehabilitation Unit Cost**.

    a. Rename it **Function - 1**.

    b. Enter the data for pipe diameter and unit cost as follows:

Function -1 Diameter vs. Unit Cost

Diameter (in.)	Unit Cost($/ft.)
6	17.00
8	17.00
10	17.00
12	17.00
14	18.20

Function -1 Diameter vs. Unit Cost (Cont'd)

Diameter (in.)	Unit Cost($/ft.)
16	19.80
18	21.60
20	23.50
30	25.50

14. In the Rehabilitation Functions manager, click **New > Pre-Rehab Diameter Vs. Post-Rehab Roughness Function**.
    a. Rename it **Function - 2**.
    b. Enter the data for pipe diameter and roughness as follows:

Function -2 Pre-Rehab Diameter vs. Post-Rehab Roughness

Diameter (in.)	Roughness
6	130
8	130
10	130
12	130
14	130
16	130
18	130
20	130

15. Create another Function called **Cost Function - Reline 1**. This is the cost for relining pipes. Use these values:

Relining Diameter vs. Cost

Diameter (in.)	Unit Cost ($/ft.)
6	26.20
8	27.80
10	34.10
12	41.40
14	50.20
16	58.50
18	66.20
20	76.80
24	109.20
30	142.50

16. Create a final function called **Cost Function - Do Nothing**. This function is required if you need Darwin Designer to consider not rehabilitating an existing pipe as an option.

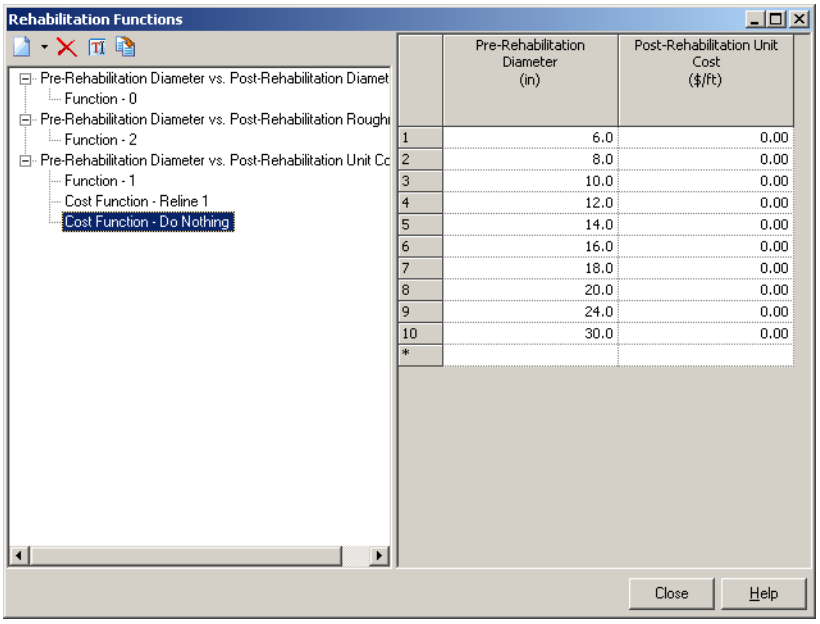
Do Nothing Cost

Diameter (in.)	Unit Cost ($/ft.)
6	0.00
8	0.00
10	0.00
12	0.00
14	0.00
16	0.00

Do Nothing Cost

Diameter (in.)	Unit Cost ($/ft.)
18	0.00
20	0.00
24	0.00
30	0.00

17. The Rehabilitation Functions manager should now look like this:



18. Click **Close** to close the Rehabilitation Functions manager.
19. For the **Action**: **Clean**:
    a. In the **Pre-Rehabilitation Diameter vs. Post-Rehabilitation Diameter Function** cell, select **Function - 0** from the list.
    b. In the **Pre-Rehabilitation Diameter vs. Post-Rehabilitation Unit Cost Function** cell, select **Function 1** from the drop-down list.
    c. In the **Pre-Rehabilitation Diameter vs. Post-Rehabilitation Roughness Function** cell, select **Function 2** from the drop-down list.

20. Type a new Action, called **Relining 1**.

    a. Set the **Pre-Rehabilitation Diameter vs. Post-Rehabilitation Diameter Function** cell to **Function - 0**.

    b. Set the **Pre-Rehabilitation Diameter vs. Post-Rehabilitation Unit Cost Function** cell to **Cost Function - Reline 1**.

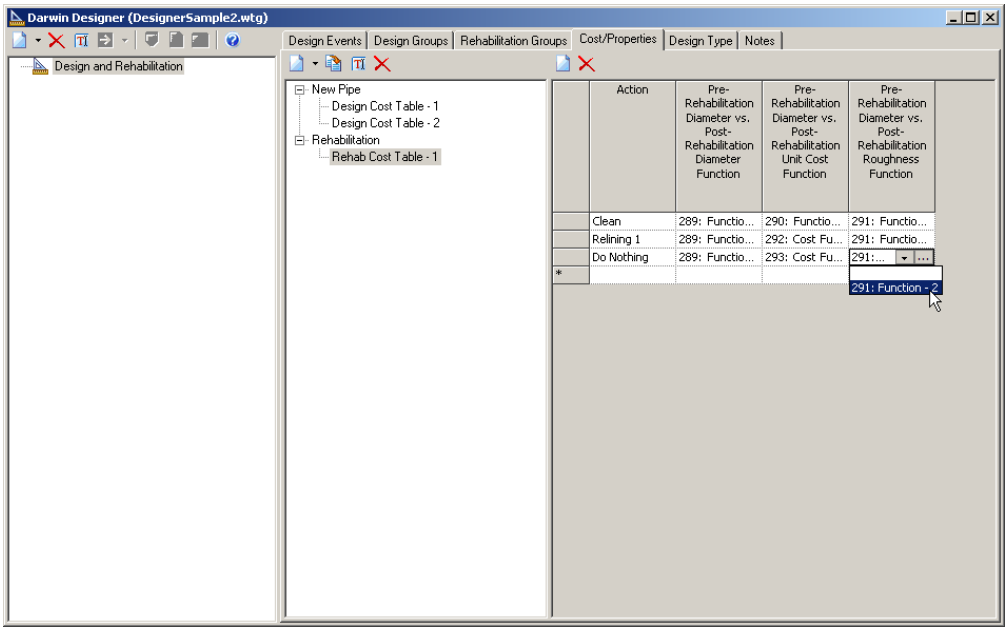
    c. Set the **Pre-Rehabilitation Diameter vs. Post-Rehabilitation Roughness Function** cell to **Function - 2**.

21. Type a new Action called **Do Nothing**.

    a. Set the **Pre-Rehabilitation Diameter vs. Post-Rehabilitation Diameter Function** cell to **Function - 0**.

    b. Set the **Pre-Rehabilitation Diameter vs. Post-Rehabilitation Unit Cost Function** cell to **Cost Function - Do Nothing**.

    c. Set the **Pre-Rehabilitation Diameter vs. Post-Rehabilitation Roughness Function** cell to **Function - 2**.

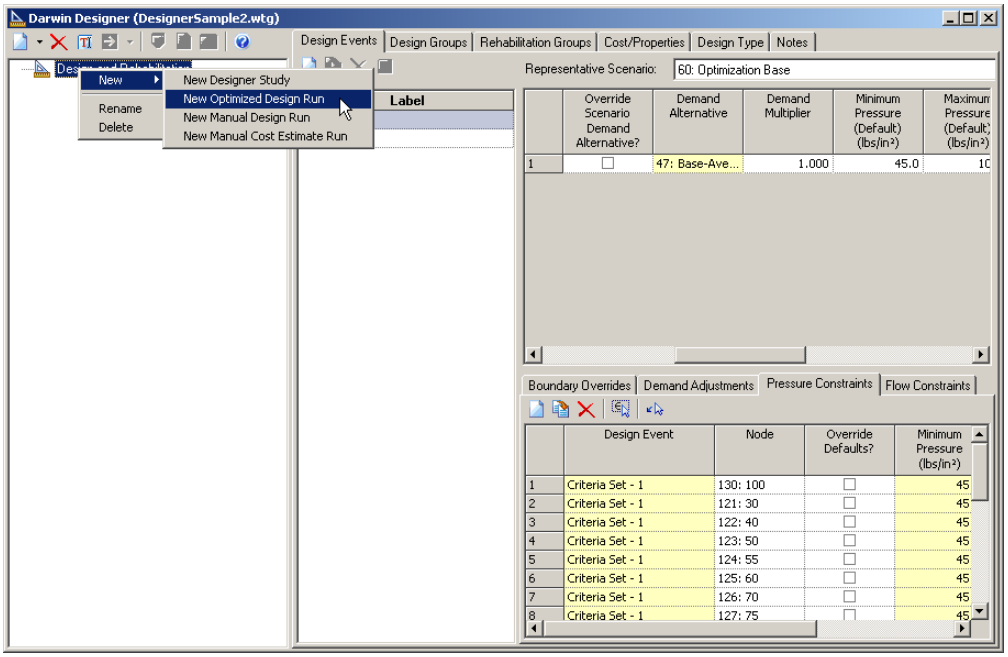


22. Click the **Design Type** tab to set the genetic algorithm parameters. Set the **Objective Type** to **Minimize Cost**. You are not considering any benefits to increasing system flow or pressure.

Create the Optimized Design Run

The design run uses your setup and applies it to the network.

1. Right-click the **Design and Rehabilitation** design run in the left pane, and select **New Optimized Design Run**.

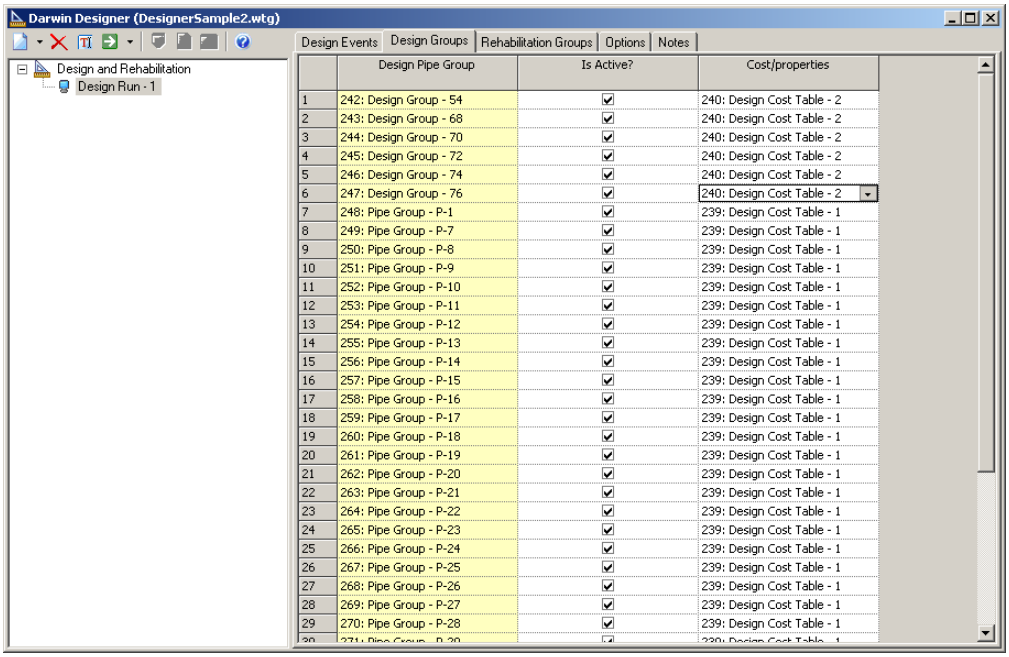


2. Rename the optimized design run as **Design Run -1**.
3. In the **Design Events** tab, make sure the **Is Active?** check box is checked for the Design Event **Criteria Set -1**. This enables the selected design event for the current run.
4. Click the **Design Groups** tab.
5. Make sure the **Is Active?** check box is checked for all of the design groups.

6. Select the design option group used by your design groups.

   a. All groups containing parallel pipes need to use **Design Cost Table 1**, for that option group contains data for a pipe size of 0. Parallel pipes have the prefix **P**.

   b. All groups containing new, single pipes need to use **Design Cost Table 2**, for that option group does not use a 0 pipe size.



7. Click the **Rehab Groups** tab.

   a. Make sure all the groups are set as **Active**.

   b. Set all the groups to use your rehab option group. (Right-click the heading of the check box column and globally edit them.)

8. Click the **Options** tab to set the GA parameters for the optimization.

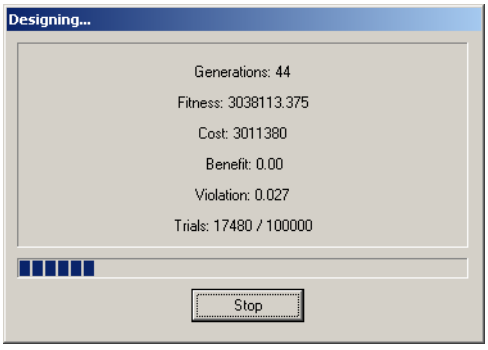
   – Under Stopping Criteria, set Maximum Trials to **100000**.

   – Under Top Solutions, set Solutions to Keep to **5**.

Calculate and Verify the Optimal Solution

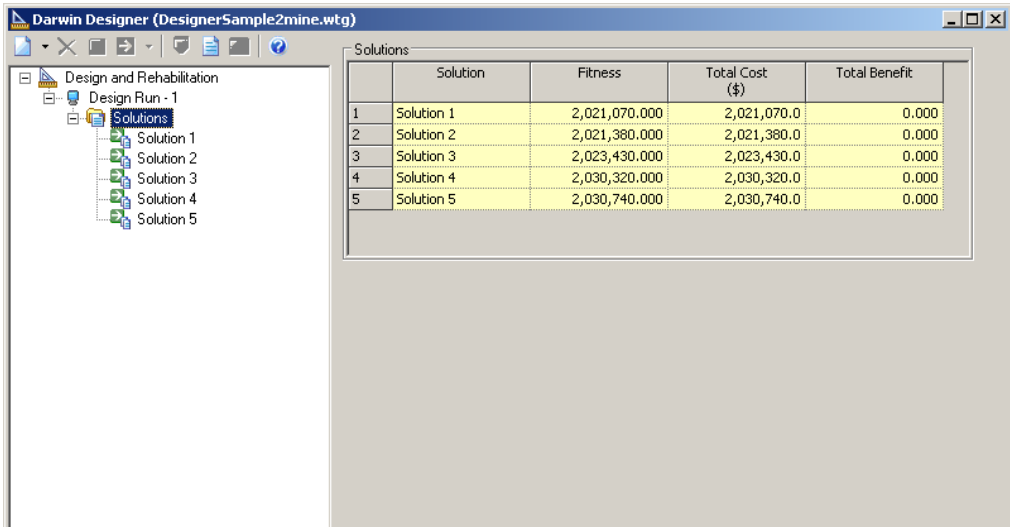
After you calculate your solutions, it is important that you look at them and verify they do what you need.

1. Click **Compute**. A dialog box opens that displays the progress and certain statistics of the calculation.



2. After the calculation is complete, click **Close**. (If the calculation did not complete successfully, you would check the Messages window.)

   Under the Solutions folder you see five solutions numbered 1 through 5 These are the five top solutions Darwin Designer has calculated. Highlight the Solutions folder to display a summary of each of the top solutions.



   Solutions are stored in order of optimization fitness, with Solution 1 providing a better calculated solution than Solution 2, which has a better calculated solution that Solution 3, etc.

3. Export the solutions to your model, so you can review tank levels.

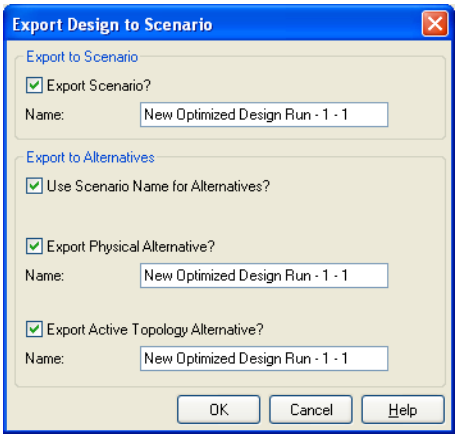
Note that the optimization calculations consider your pressure requirements (that pressure be greater than 45 psi) but not your tank levels.

a. Highlight Solution 1.

b. Click **Export to Scenario**.  The Export Design to Scenario dialog box opens.

c. Click the **Use Scenario Name for Alternatives** check box. The default name is the design run name plus an incremental number starting at 1.

d. Check the **Export Physical Alternative?** and **Export Active Topology Alternative?** checkboxes.



e. Click **OK**. This exports Solution 1.

f. Select **Solution 2** from the solutions drop-down list.

g. Export Solution 2.

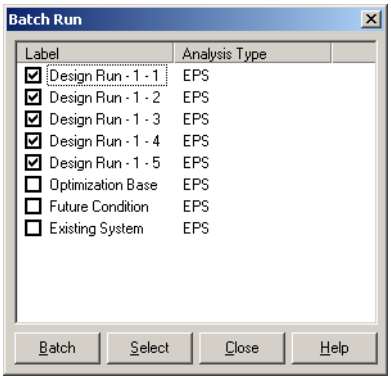
h. Export the remaining solutions in turn.

4. **Close** the Darwin Designer window so you can review the solutions you exported.
5. Click **Analysis** > **Scenarios** to open the **Scenarios** manager.

6. Compute the scenarios you exported in a batch run. This lets you graph those results and look at what is happening with your tank levels.

   a. Click the black down arrow next to the compute button at the top of the scenario manager and choose **Batch Run**.

   b. Select the Scenarios you want to run (Design Run - 1 - 1, Design Run - 1 - 2, Design Run - 1 - 3, Design Run - 1 - 4, and Design Run - 1 - 5).



   c. Click **Batch**, click Yes in the prompt, and close the message boxes that appear before and after the calculations.

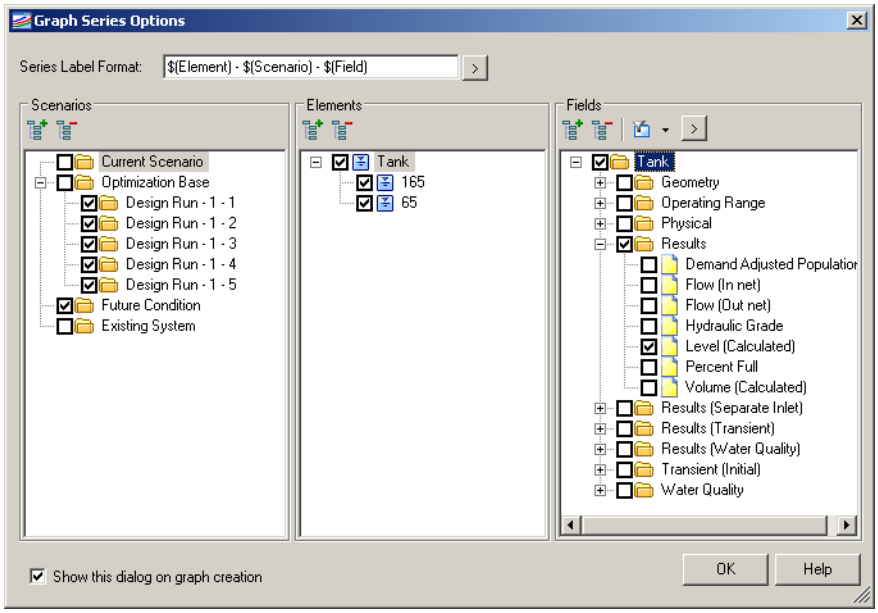
   d. After the batch run finishes, close the **Scenarios** manager and the **User Notifications** dialogs.

7. You will use graphing to inspect your tank levels. Click **View** > **Graphs**.

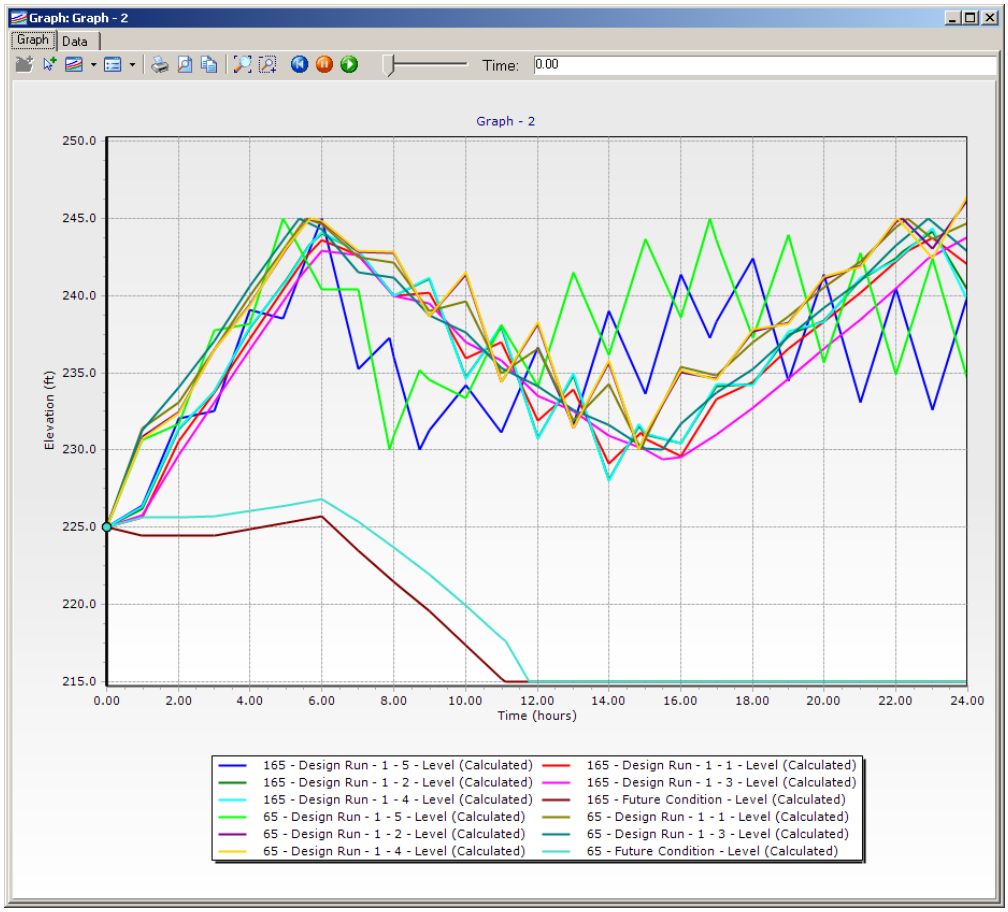
   a. Click the **New** button and select **Line Series Graph**. A Select toolbar appears to allow you to select the elements you want to graph from the drawing view. Click on both tanks, then right-click and select **Done**.

   b. In the Scenarios list of the the Graph Series Options dialog, check the boxes next to the **Design Run - 1 - 1**, **Design Run - 1 - 2**, **Design Run - 1 - 3**, **Design Run - 1 - 4**, **Design Run - 1 - 5**, and **Future Condition** scenarios (uncheck Optimized Base if it is checked).

c. In the Fields list uncheck the **Flow (Out Net)** box and check the **Level (Calculated)** box.



d. Click **OK**.



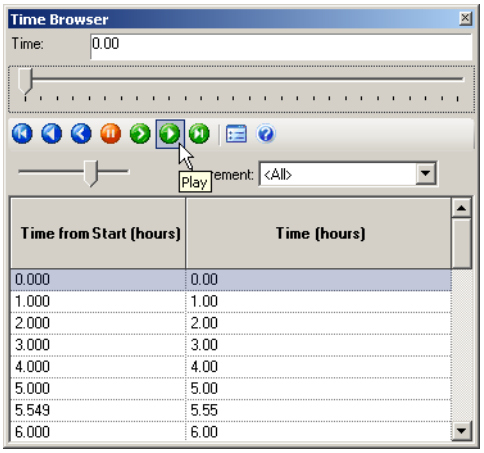
e. Review the graph. Notice that each of the design runs are able to keep the tanks full.

f. While all of the design runs do keep the tanks full, Solution 1 is the best optimal solution that meets your pressure and tank fill requirements while minimizing costs.

8. Close the Graph window.

9. In the Scenario drop-down list, choose **Design Run - 1 - 1**, which represents Solution 1 that Darwin Designer calculated. From looking at the results in the graph, you know this solution keeps your tanks full.

10. Inspect your tank pressure by animating the scenario over 24 hours. Click **Analysis** > **Time Browser**. Click **Play**.



Note the color coding for pressure:

– <= 45 psi is red

– <= 70 psi is blue

– <= 100 psi is magenta

– <= 130 psi is green

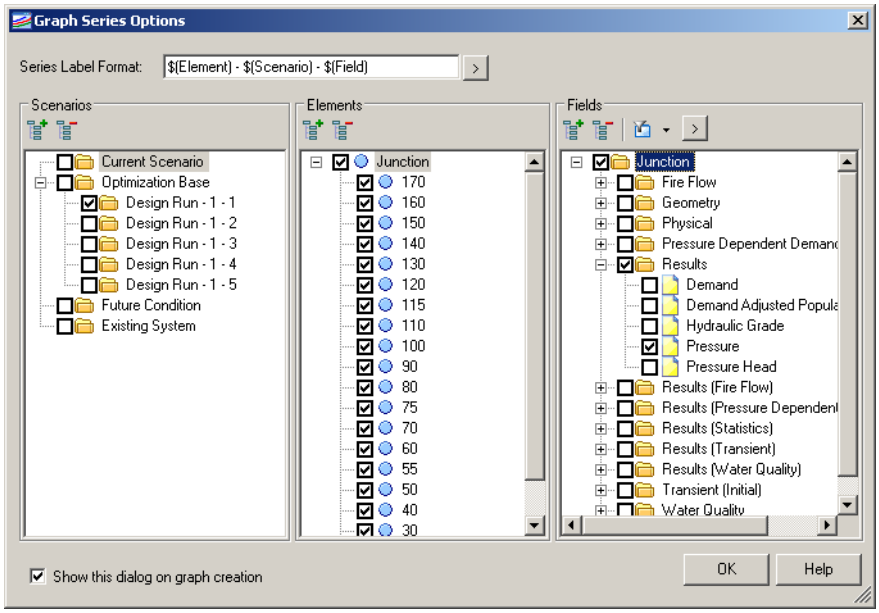
11. Make sure none of the junctions are red during the animation.

12. Inspect a graph of junction pressures.

a. Click **Edit** > **Select by Element** > **Junction** to select all of the junctions.

b. Right-click one of the junctions and select **Graph**. Click **Yes** to the prompt asking if you want to graph all of the selected elements.

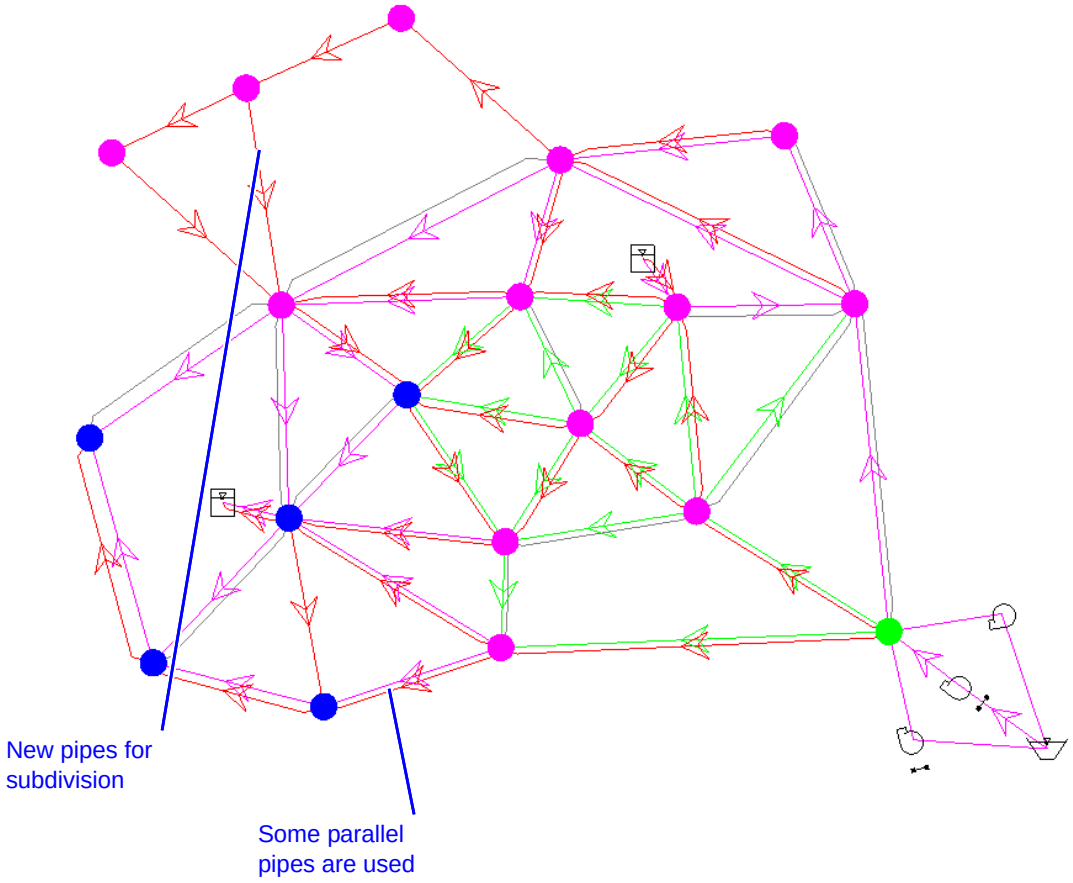
c. In the Graph Series Options dialog, uncheck the **Hydraulic Grade** Field and check the **Pressure** box.



d. Click **OK**.

e. The Graph dialog box opens and displays pressures for the junctions you selected. Note that none of the junctions fall below 45 psi.





Scenario Energy Costs

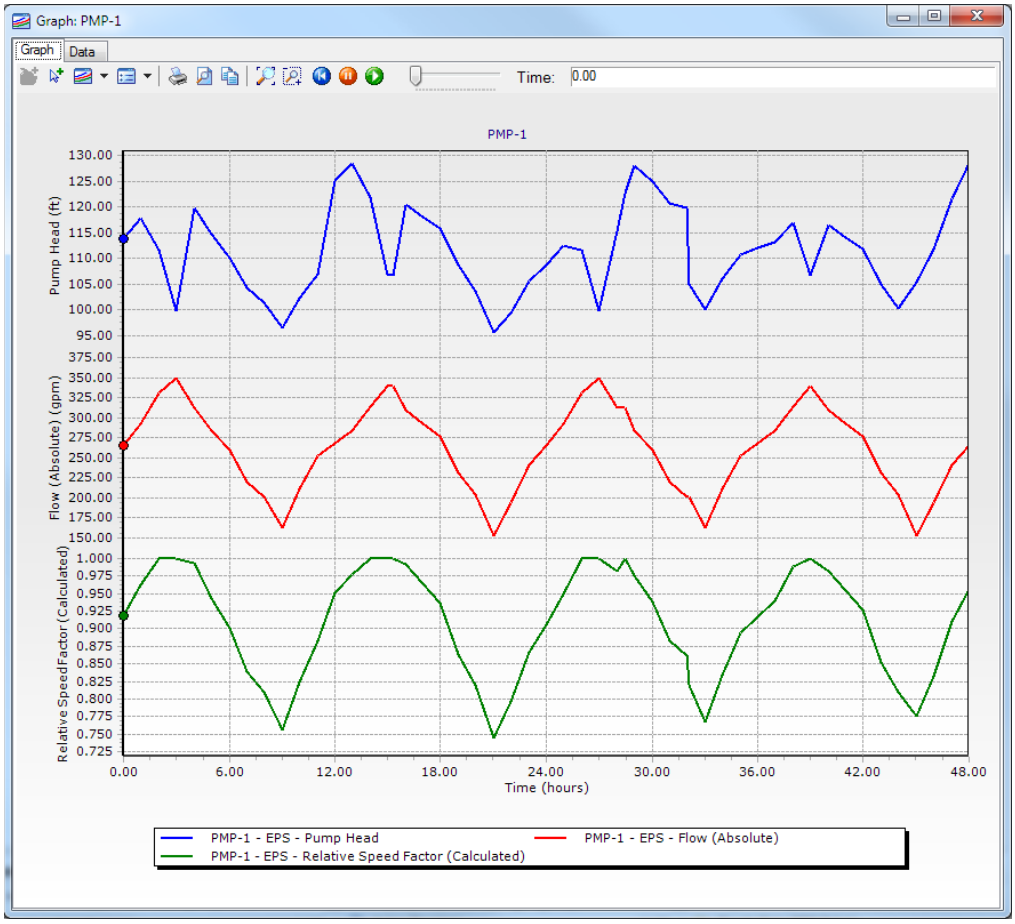
Scenario energy costs calculates energy usage and cost based on an extended period simulation (EPS). It also determines a number of intermediated values such as efficiency, power, and peak energy use.

The steps in running an energy cost calculation

1. Run EPS simulation.
2. Open energy cost manager and set up energy pricing.
3. Select scenario and run energy cost calculation.
4. Review Results.

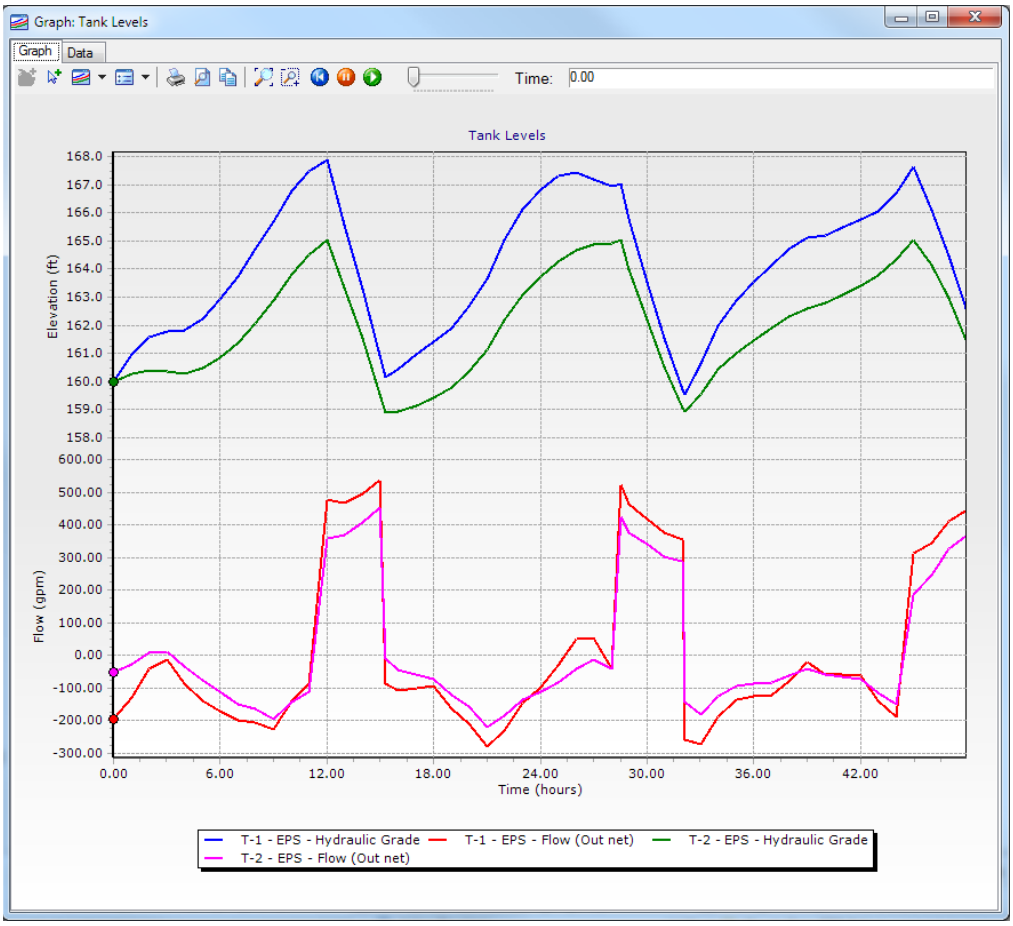
Step 1: Run EPS Model

1. Open the **EngCostLessonStart.wtg** file in the Lessons directory.
2. **Compute** the model  .
3. Choose **View** > **Graphs** and double-click on the **PMP-1** graph.



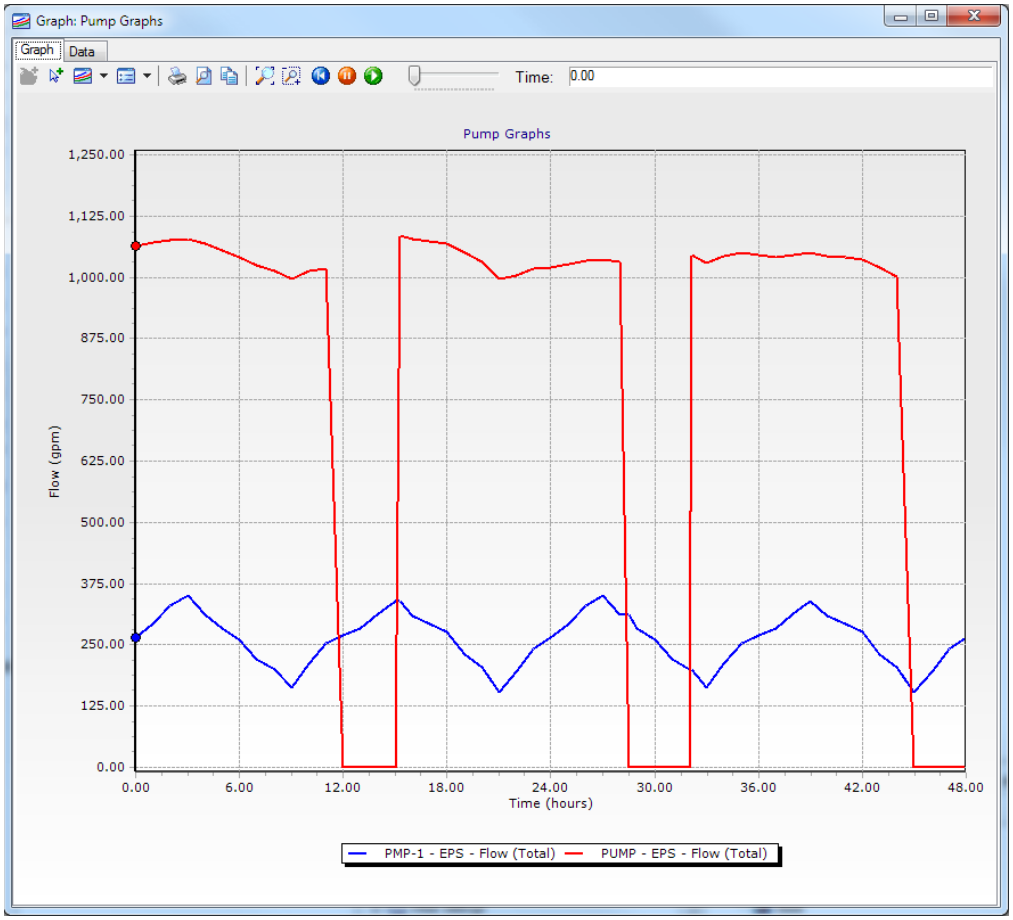
Notice that the pump reaches 100% full speed several times.

4. Close the graph. In the Graphs manager, double-click the **Tank Levels** graph.



The tanks fill gradually during this run and empty slightly quicker when the main PUMP cycles off.

5. Close the graph. In the Graphs manager, double-click the **Pump Graphs** graph.



You can see the relative flow of the main pump and the booster bump.

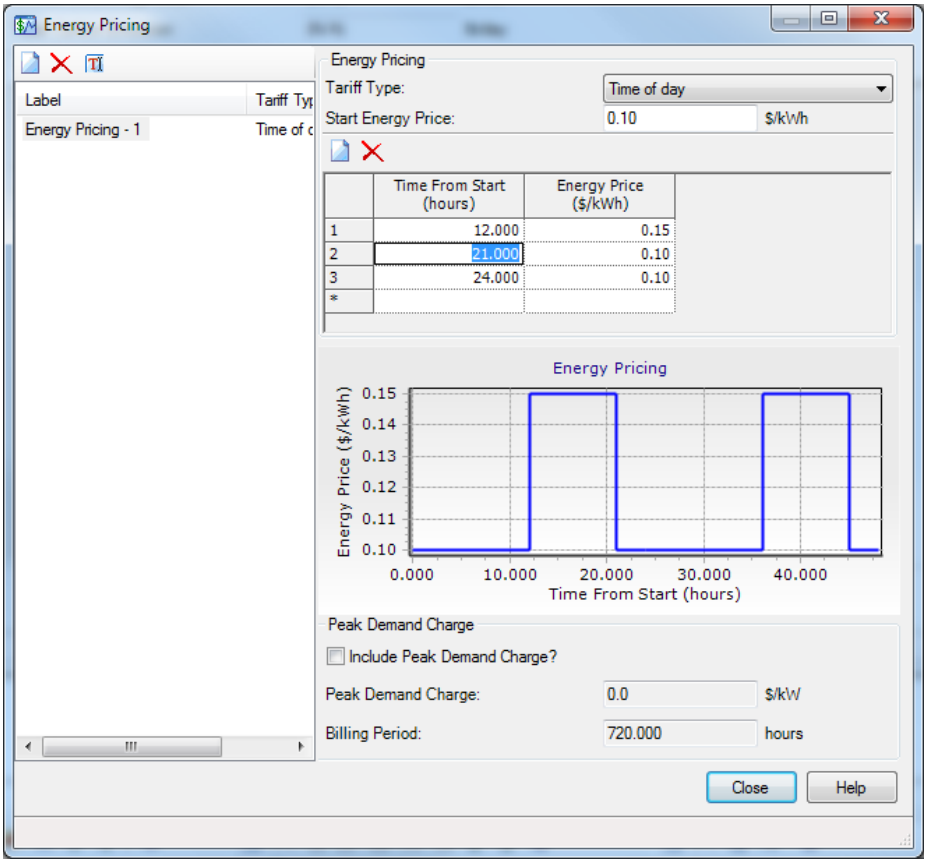
6. Close the graph and the Graphs manager.
7. Save the file as MYLESSON11.

Step 2: Setting up energy pricing

1. Click **Analysis > Scenario Energy Cost** or click  from the toolbar.
2. Click the **Energy Pricing** button  .

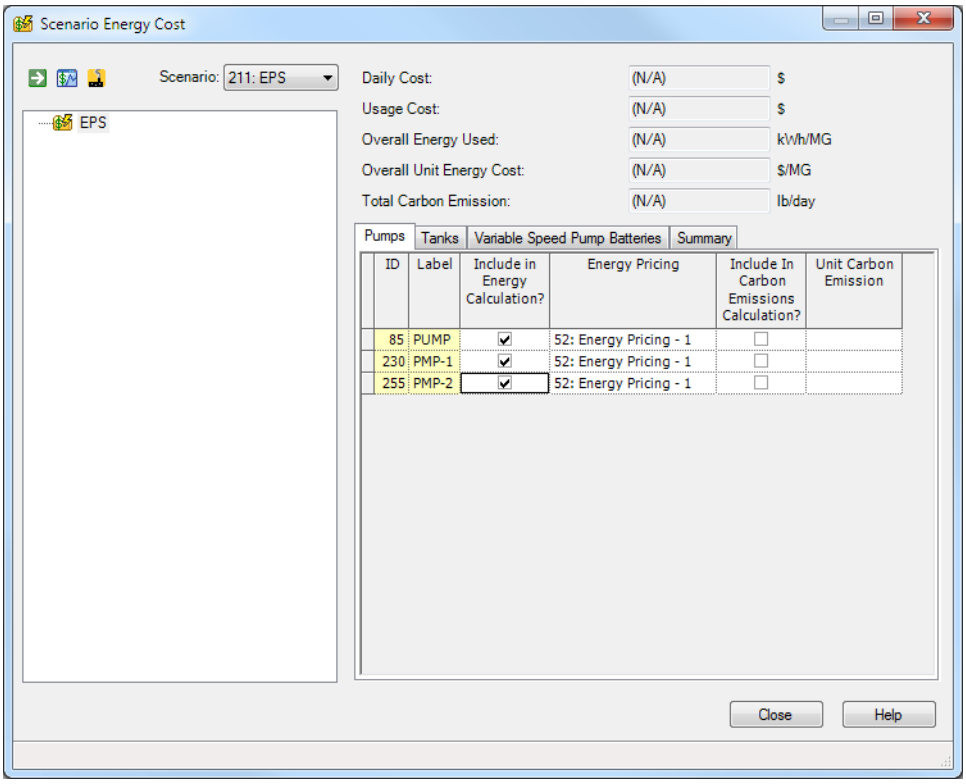
3. Type the following information into the corresponding fields:
   Start Energy Price = .10

Time From Start	Energy Price
12	.15
21	.10
24	.10



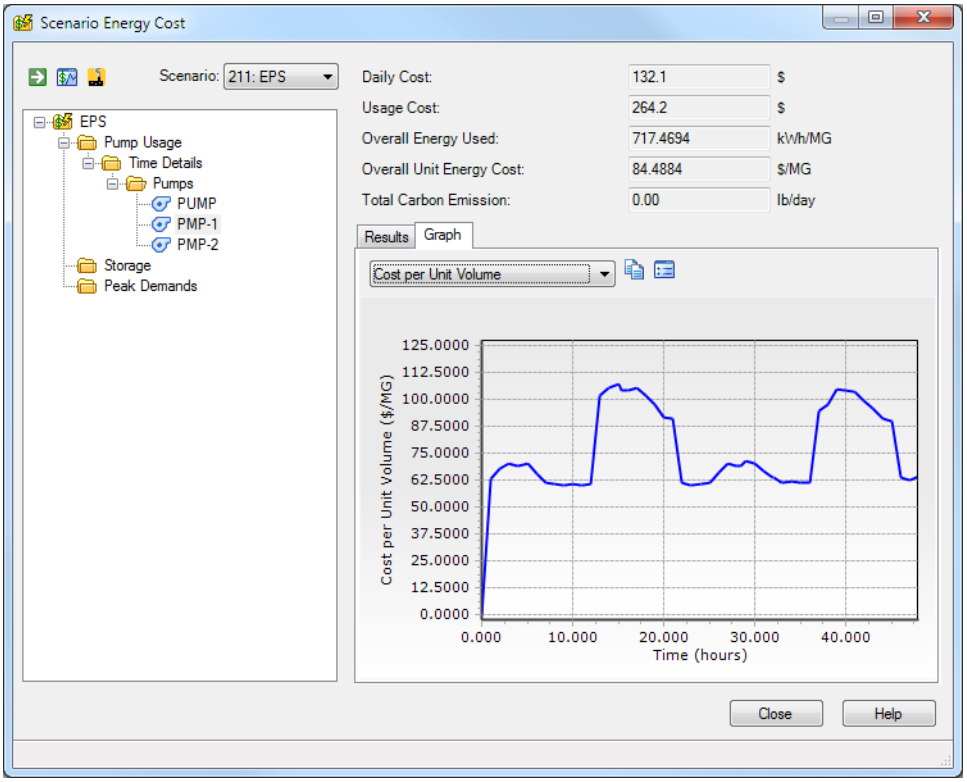
4. Close the **Energy Pricing** dialog.
5. In the Energy Cost Manager, select **EPS** from the **Scenario** menu.
6. In the Pumps tab, check the **Include in Energy Calculation?** boxes for each of the pumps.

7. Click the **Tanks** tab. Make sure the **Include in Energy Calculation?** boxes are checked for both tanks.



Step 3: Run the energy cost analysis

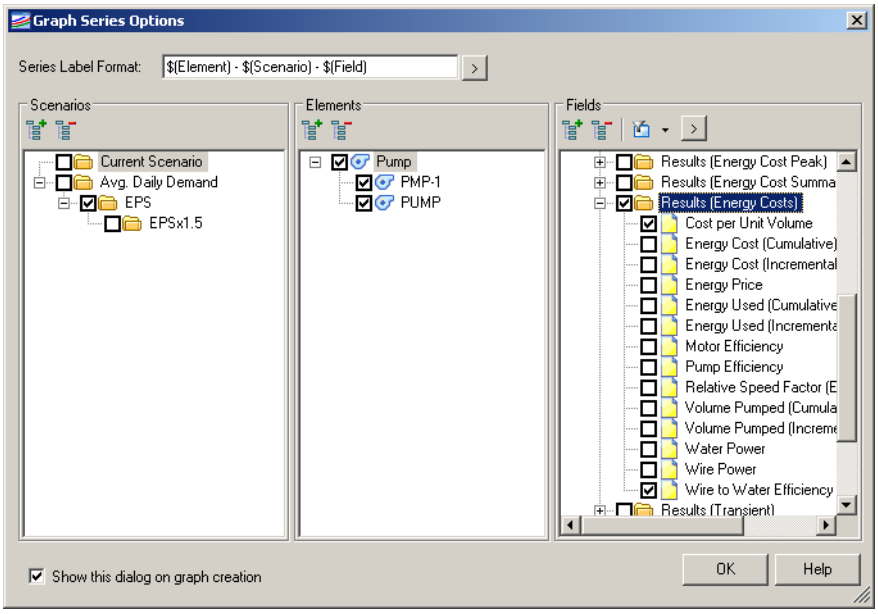
1. In the Energy Costs manager, click **Compute**  .
2. Review the overall summary. Highlight the **Pump Usage** entry in the list. You can see that the efficiency of the constant speed PUMP is higher than that of the variable speed PMP-1 and PMP-2 was not called during this run.
3. Highlight **PMP-1** and click the **Graph** tab. Change the attribute being graphed to **Cost per Unit Volume** and see how the cost changes as a result of pump status and time of day energy charges.



Step 4: Making graphical comparisons between pumps

1. Close the Energy Cost manager.
2. In the drawing, click **PMP-1** and then, while holding down the **<Ctrl>** key, click on the **PUMP** element. Right-click and select **Graph** to open the **Graph Series Option** manager.
3. Uncheck the **Flow (Total)** checkbox and expand the **Results (Energy Costs)** category (click the + button)

4. Check the **Wire to Water Efficiency** and **Cost per Unit Volume** boxes.



5. Click **OK** to open the graph.



The efficiency of the constant speed pump is higher than the variable speed pump whenever it is on. The cost per volume pumped is comparable since the PUMP usually pumps against a higher head. In order to view the head attribute, click the Graph Series Options button and check the **Pump Head** box under the **Results** folder.

6. Click **OK**.
7. PUMP pumped into a pressure zone that required a higher pump head.
8. Click the **Add to Graph Manager** button to save the graph, enter a name and click **OK**, and then close the graph window.

Pressure Dependent Demands

Pressure dependent demands (PDD) are used to simulate situations where a change in pressure affects the quantity of water used.

To use PDD

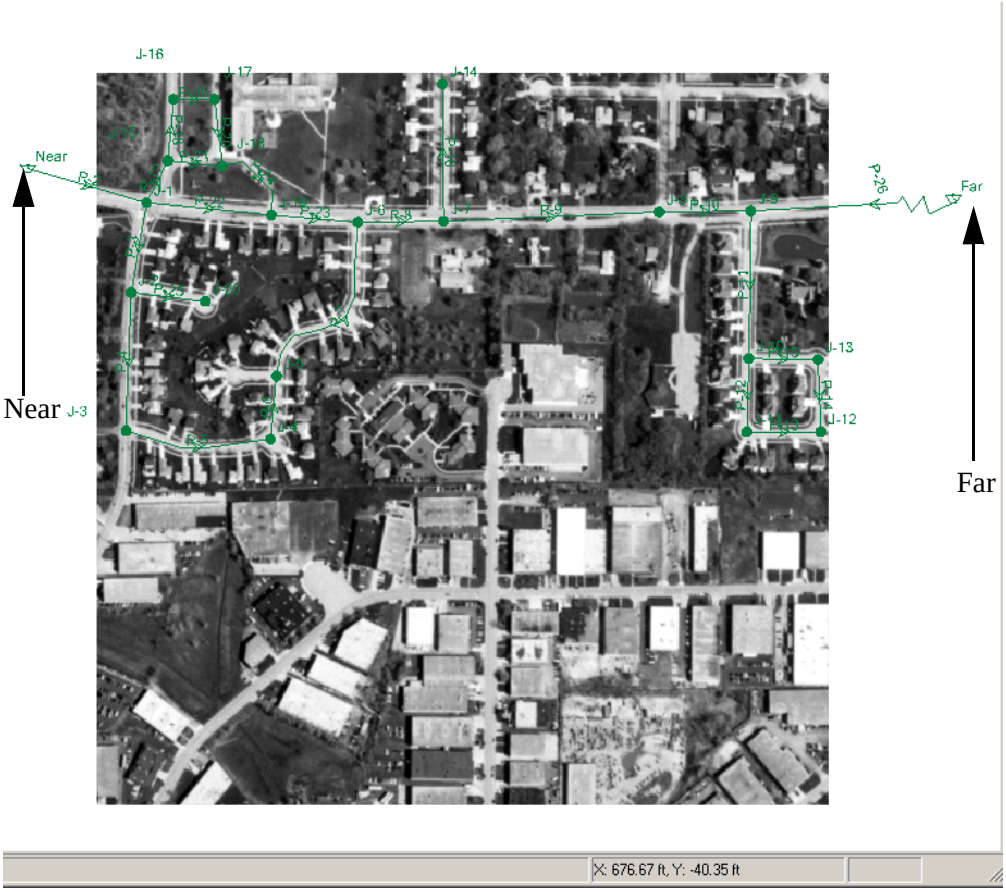
1. Set up a model.
2. Create a PDD function.
3. Create a scenario that assigns a PDD function to an alternative.
4. Run the scenario.

This lesson uses the example of a neighborhood that receives water from two sources, reservoirs that are near and far and both have a hydraulic grade of 150 ft. In this lesson, you will simulate the system without considering PDD and all elements operating. Then the analysis will be run with PDD. In order to simulate a situation where pressure significantly drops, the Near source is taken out of service and the behavior with and without consideration of PDD is made.

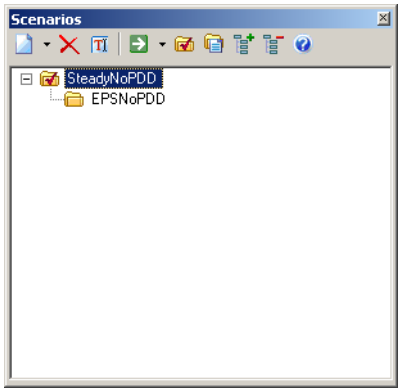
The starter file consists of a model with two non-PDD scenarios, SteadyNoPD and EPSNoPDD. The demands have been loaded and the diurnal demand function has been created.

Step 1: Run the initial NoPDD Model

1. Open the **PDDLessonStart.wtg** file in the Lessons directory.
2. The Near source is on the left and the Far source is on the right.

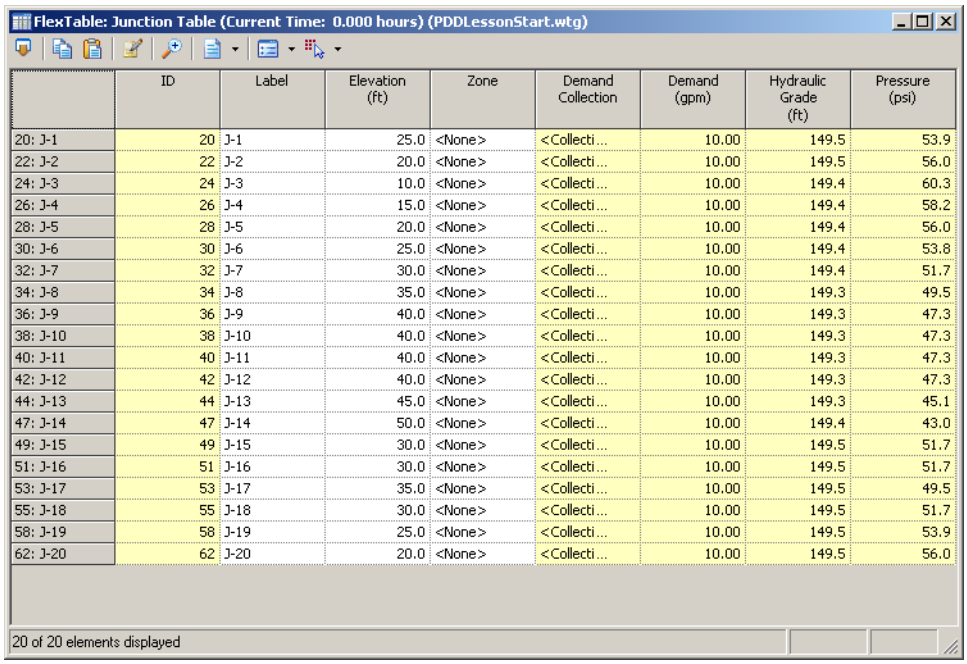


3. Click Scenarios  or **Analysis** > **Scenarios** to verify the current scenario is **SteadyNoPDD**.



4. Compute the model  and make sure results are green, then close the Calculation Summary.

5. Click **Report** > **Element Tables** > **Junction**.



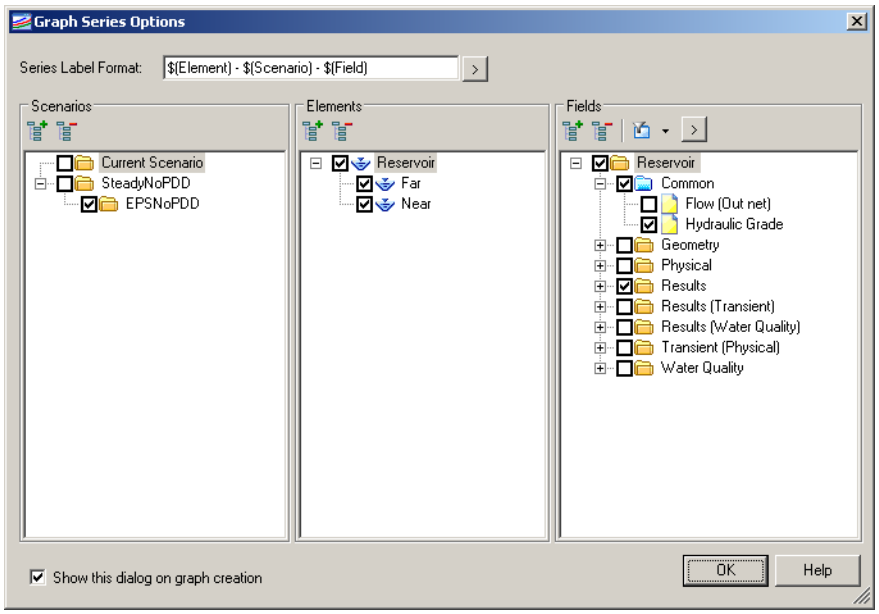
	ID	Label	Elevation (ft)	Zone	Demand Collection	Demand (gpm)	Hydraulic Grade (ft)	Pressure (psi)
20: J-1	20	J-1	25.0	<None>	<Collecti...	10.00	149.5	53.9
22: J-2	22	J-2	20.0	<None>	<Collecti...	10.00	149.5	56.0
24: J-3	24	J-3	10.0	<None>	<Collecti...	10.00	149.4	60.3
26: J-4	26	J-4	15.0	<None>	<Collecti...	10.00	149.4	58.2
28: J-5	28	J-5	20.0	<None>	<Collecti...	10.00	149.4	56.0
30: J-6	30	J-6	25.0	<None>	<Collecti...	10.00	149.4	53.8
32: J-7	32	J-7	30.0	<None>	<Collecti...	10.00	149.4	51.7
34: J-8	34	J-8	35.0	<None>	<Collecti...	10.00	149.3	49.5
36: J-9	36	J-9	40.0	<None>	<Collecti...	10.00	149.3	47.3
38: J-10	38	J-10	40.0	<None>	<Collecti...	10.00	149.3	47.3
40: J-11	40	J-11	40.0	<None>	<Collecti...	10.00	149.3	47.3
42: J-12	42	J-12	40.0	<None>	<Collecti...	10.00	149.3	47.3
44: J-13	44	J-13	45.0	<None>	<Collecti...	10.00	149.3	45.1
47: J-14	47	J-14	50.0	<None>	<Collecti...	10.00	149.4	43.0
49: J-15	49	J-15	30.0	<None>	<Collecti...	10.00	149.5	51.7
51: J-16	51	J-16	30.0	<None>	<Collecti...	10.00	149.5	51.7
53: J-17	53	J-17	35.0	<None>	<Collecti...	10.00	149.5	49.5
55: J-18	55	J-18	30.0	<None>	<Collecti...	10.00	149.5	51.7
58: J-19	58	J-19	25.0	<None>	<Collecti...	10.00	149.5	53.9
62: J-20	62	J-20	20.0	<None>	<Collecti...	10.00	149.5	56.0

Note that the pressures range from 43 to 60 psi.

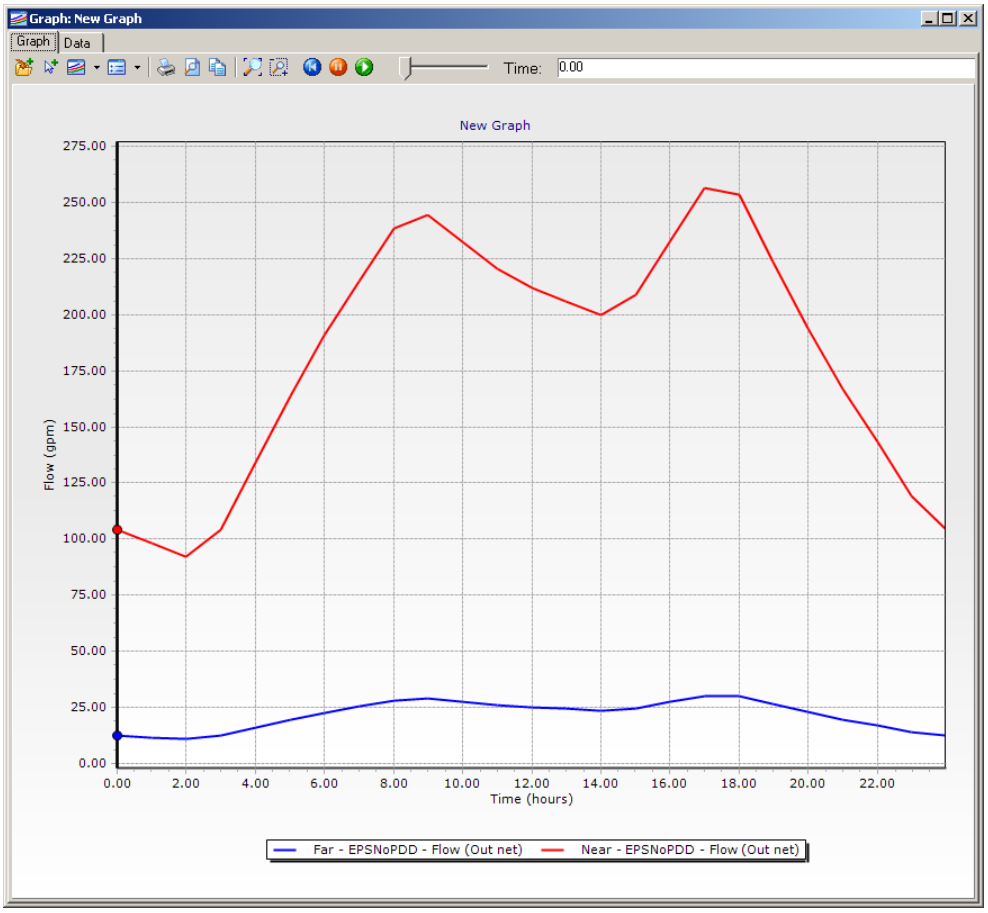
6. Close the FlexTable.

7. Click **Analysis** > **Scenarios** and select **EPSNoPDD** and make it the current scenario  .

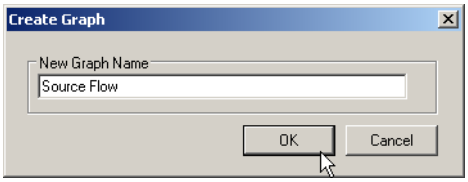
8. Compute the scenario  , make sure user notifications are green, then close the Calculation Summary.
9. In the drawing, hold the **<Ctrl>** key and click the **Near** reservoir, then the **Far** reservoir, and then right-click and select **Graph**.
10. Make sure the **Flow (Out net)** box is checked.



11. Click **OK** to view the graph.

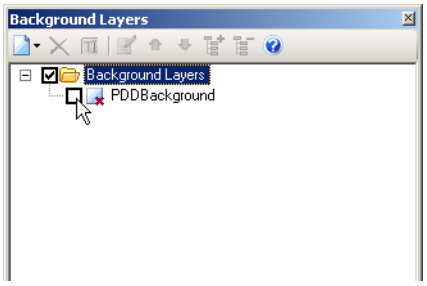


12. Click **Add to Graph Manager**  to save the graph and name it **Source Flow**.

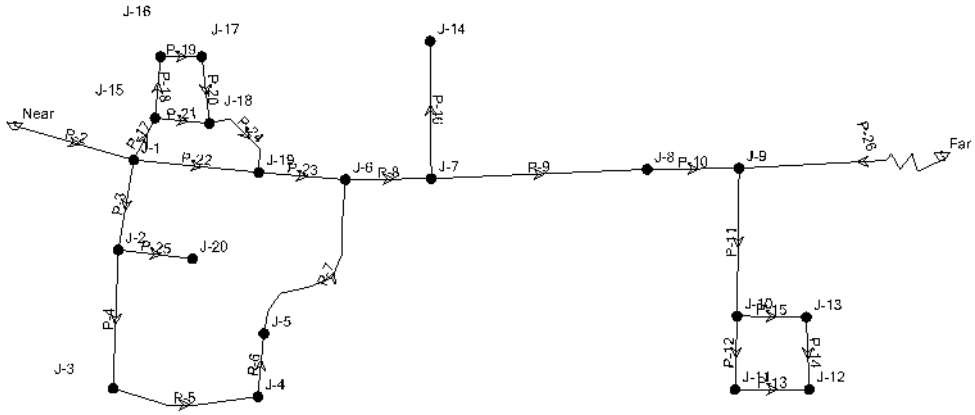


13. Click **OK** and then close the graph.

14. If you want to turn off the background layers of the drawing choose View > Background Layers and uncheck the box next to **PDD Background**.

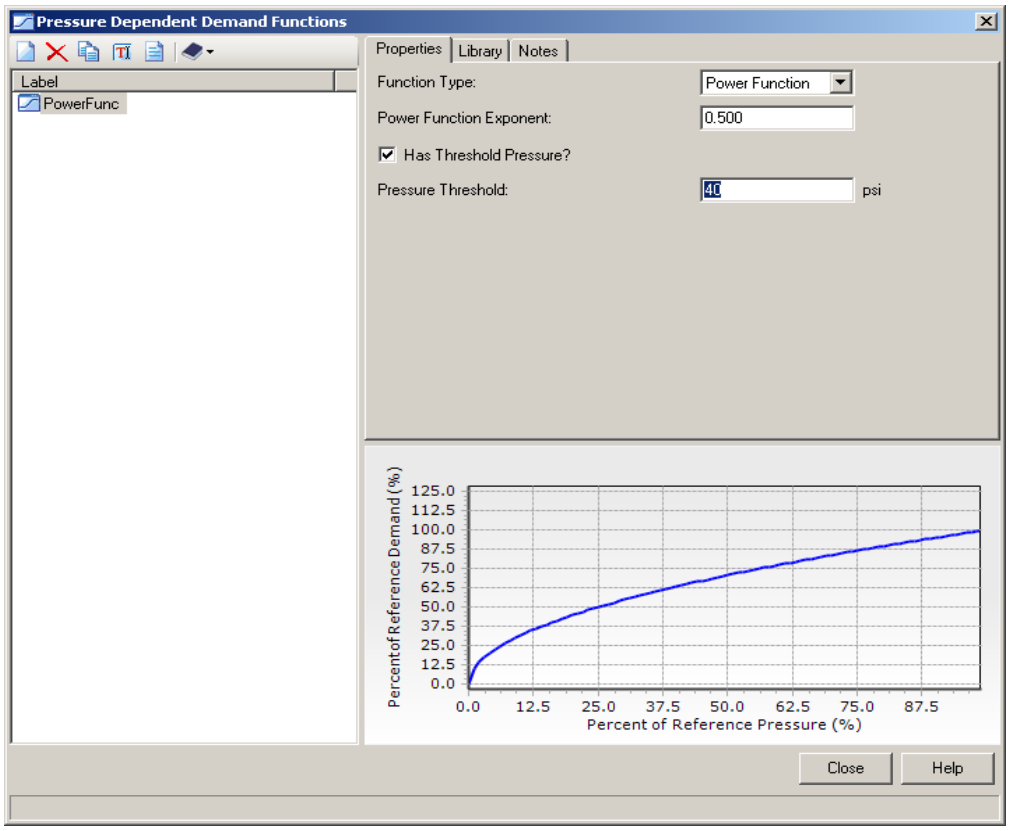


15. Without the background image the drawing will look like the following:



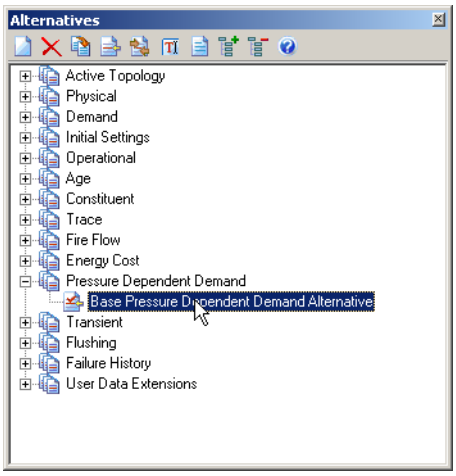
Step 2: Setting up PDD function

1. Click **Components** > **Pressure Dependent Demand Functions**. Click New and then rename the function to **PowerFunc**.
2. Has Threshold Pressure? should be checked and type in 40 for the pressure threshold.

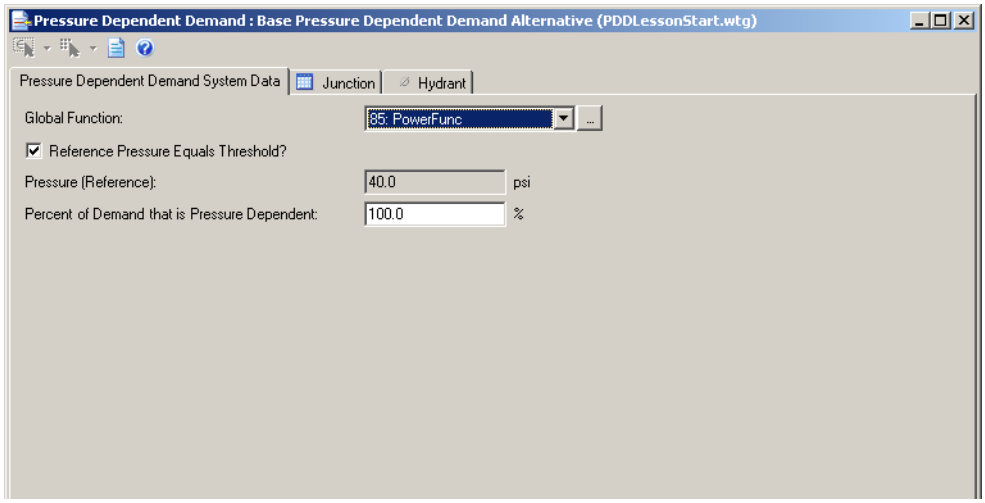


3. Close the PDD Functions manager.

4. Click **Analysis** > **Alternatives**. Expand the **Pressure Dependent Demand** alternative and double-click the **Base Pressure Dependent Demand** alternative to edit it.



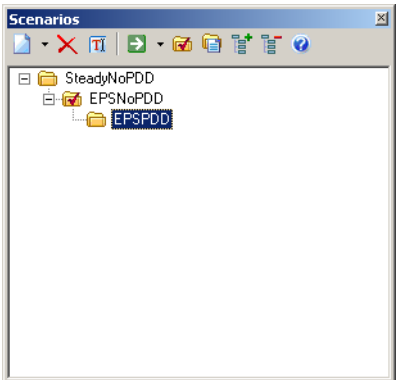
5. Select **PowerFunc** from the **Global Function** menu.



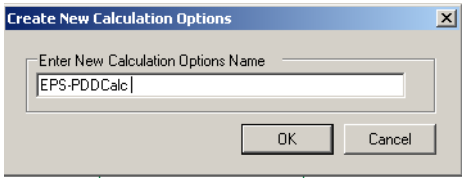
6. Close the alternative editor.

Step 3: Run the model with PDD

1. Click **Analysis** > **Scenarios** and create a child scenario of **EPSNoPDD**. Right-click on **EPSNoPDD** and select **New** > **Child Scenario** and rename the new scenario **EPSPDD.**

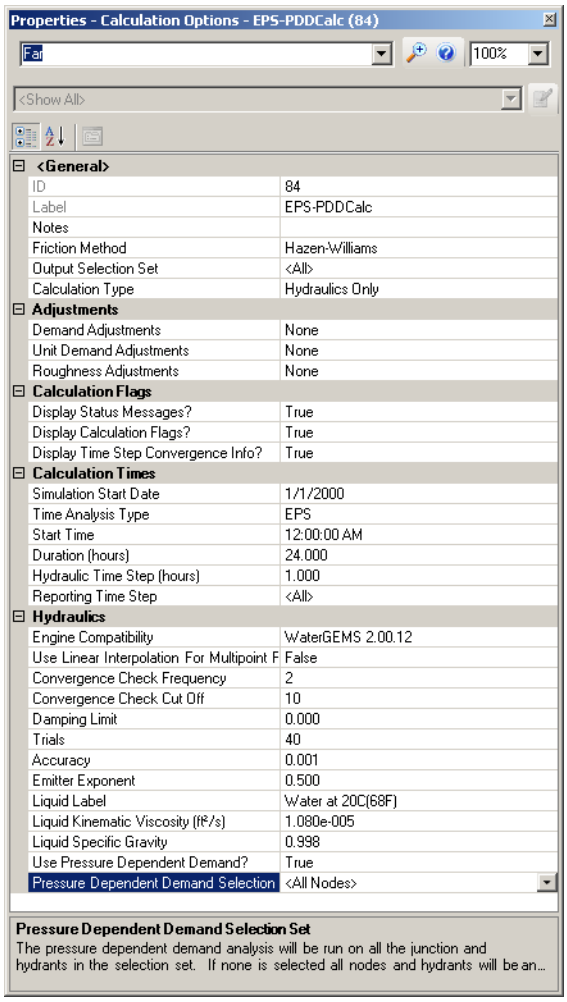


2. Double-click on the **EPSPDD** scenario to open the scenario Properties editor. Under Calculations Options, click the **Steady State/EPS Solver Calculation Options** menu and select **New**. Rename the new option EPS-PDDCalc and then click OK.

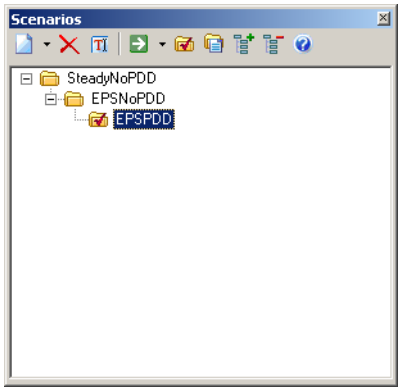


3. Make **EPSPDD** the current scenario.
4. Click **Analysis** > **Calculation Options** and double-click on **EPS-PDDCalc** to open the Properties editor.

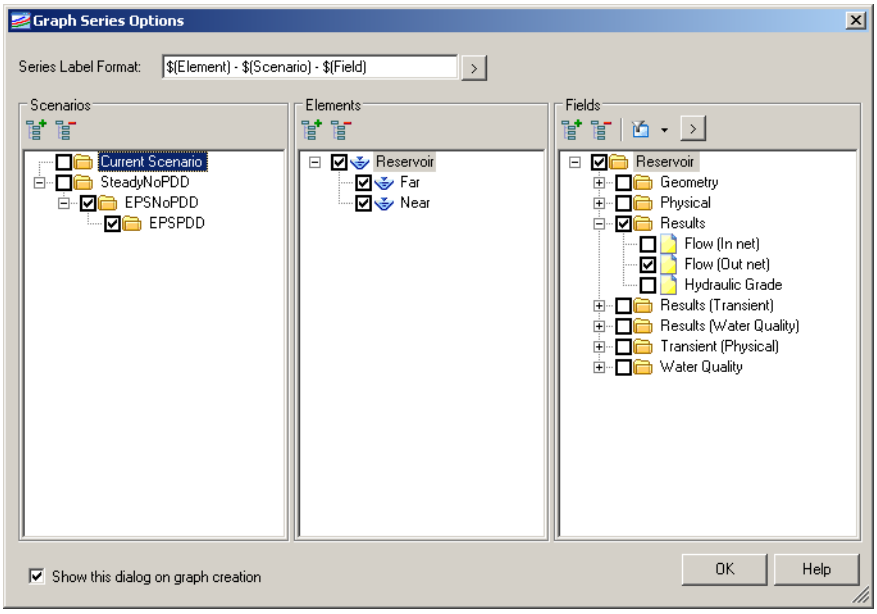
5. Set **Time Analysis Type** to **EPS**.
   Set **Use Pressure Dependent Demand?** to **True**.
   Set **Pressure Dependent Demand Selection** to **<All Nodes>**.



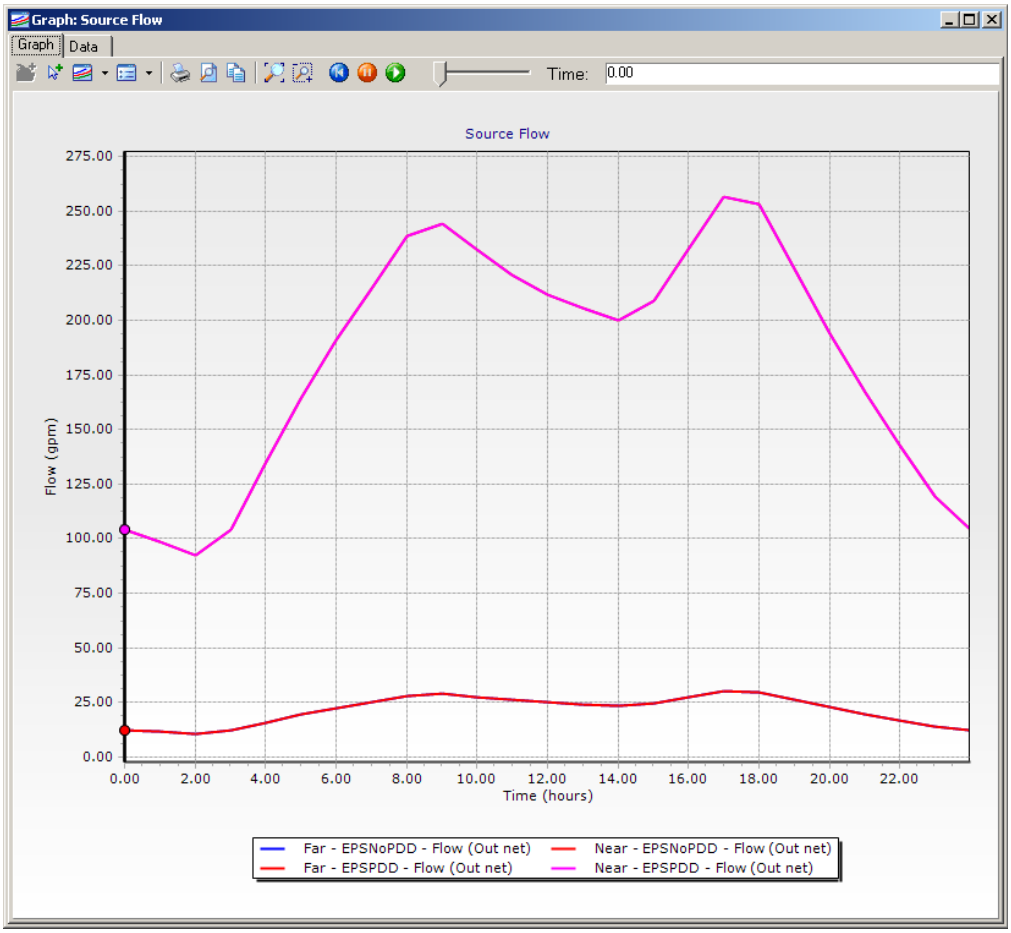
6. In the Scenarios manager, make the **EPSPDD** scenario current, then click **Compute**.



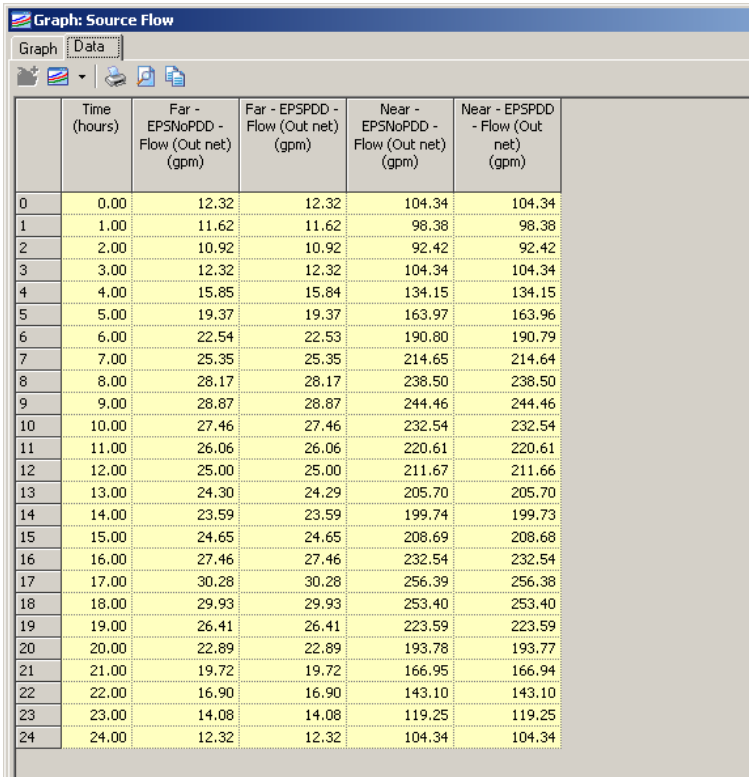
7. Review the calculation summary and then close it.
8. Review the results by plotting a graph of flow vs. time. Click **View** > **Graphs** and double-click on the **SourceFlow** graph.
9. Click **Graph Series Options**  and check both **EPSNoPDD** and **EPSPDD** and then **OK**.



10. There are four lines on the graph but only two are visible. This is because the lines for both scenarios are identical.



11. Click the Data tab to see that the pressure did not drop below the reference pressure during the run.



Graph: Source Flow

Graph | Data

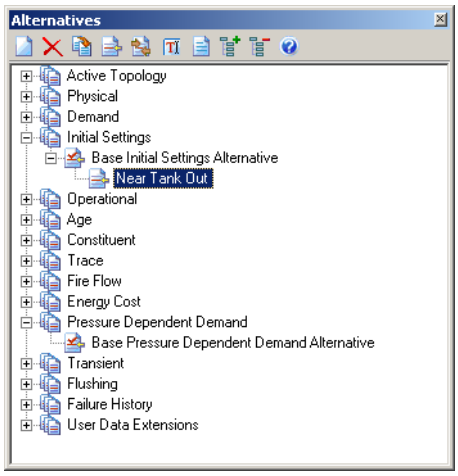
	Time (hours)	Far - EPSNoPDD - Flow (Out net) (gpm)	Far - EPSPDD - Flow (Out net) (gpm)	Near - EPSNoPDD - Flow (Out net) (gpm)	Near - EPSPDD - Flow (Out net) (gpm)
0	0.00	12.32	12.32	104.34	104.34
1	1.00	11.62	11.62	98.38	98.38
2	2.00	10.92	10.92	92.42	92.42
3	3.00	12.32	12.32	104.34	104.34
4	4.00	15.85	15.84	134.15	134.15
5	5.00	19.37	19.37	163.97	163.96
6	6.00	22.54	22.53	190.80	190.79
7	7.00	25.35	25.35	214.65	214.64
8	8.00	28.17	28.17	238.50	238.50
9	9.00	28.87	28.87	244.46	244.46
10	10.00	27.46	27.46	232.54	232.54
11	11.00	26.06	26.06	220.61	220.61
12	12.00	25.00	25.00	211.67	211.66
13	13.00	24.30	24.29	205.70	205.70
14	14.00	23.59	23.59	199.74	199.73
15	15.00	24.65	24.65	208.69	208.68
16	16.00	27.46	27.46	232.54	232.54
17	17.00	30.28	30.28	256.39	256.38
18	18.00	29.93	29.93	253.40	253.40
19	19.00	26.41	26.41	223.59	223.59
20	20.00	22.89	22.89	193.78	193.77
21	21.00	19.72	19.72	166.95	166.94
22	22.00	16.90	16.90	143.10	143.10
23	23.00	14.08	14.08	119.25	119.25
24	24.00	12.32	12.32	104.34	104.34

Step 4: Running non-PDD models with outage

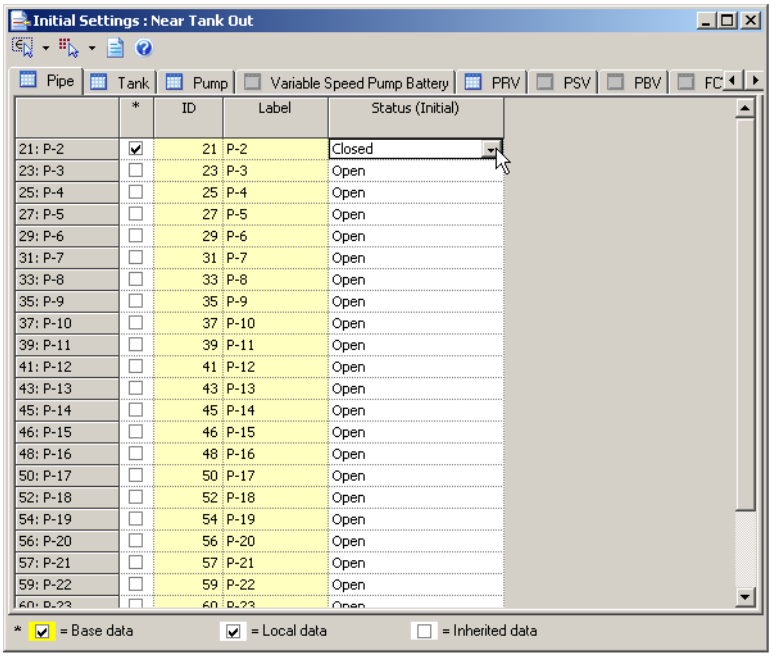
In order to examine the effect of a drop in pressure, create a scenario where the pressures will drop. In this example, Near tank will be taken out of service. Create a new scenario where pipe P-2 is closed.

1. Click **Analysis** > **Alternatives**. Expand the Initial Settings alternative node and right-click the **Base Initial Settings Alternative**. Select **New** > **Child Alternative**.

2. Rename the new alternative to **Near Tank Out**.



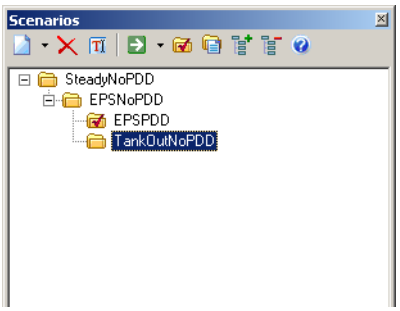
3. Double-click on **Near Tank Out** and change the initial status of **P-2** to **Closed**. When the status has been changed to Closed a check shows in the first column to show that it is different from its parent.



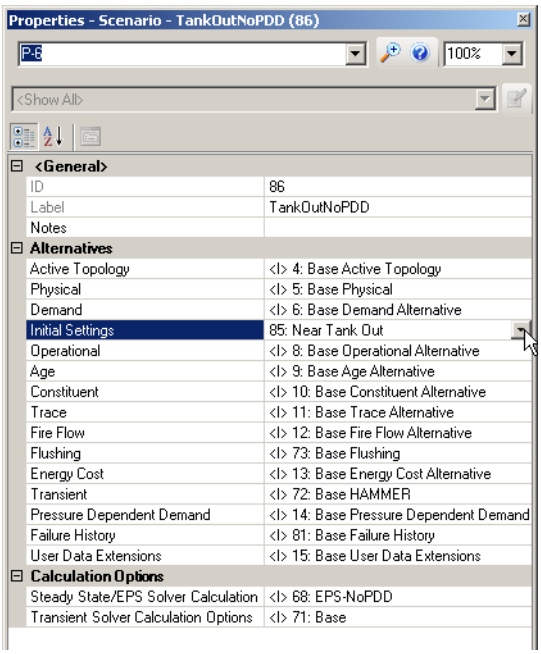
	*	ID	Label	Status (Initial)
21: P-2	☑	21	P-2	Closed
23: P-3	☐	23	P-3	Open
25: P-4	☐	25	P-4	Open
27: P-5	☐	27	P-5	Open
29: P-6	☐	29	P-6	Open
31: P-7	☐	31	P-7	Open
33: P-8	☐	33	P-8	Open
35: P-9	☐	35	P-9	Open
37: P-10	☐	37	P-10	Open
39: P-11	☐	39	P-11	Open
41: P-12	☐	41	P-12	Open
43: P-13	☐	43	P-13	Open
45: P-14	☐	45	P-14	Open
46: P-15	☐	46	P-15	Open
48: P-16	☐	48	P-16	Open
50: P-17	☐	50	P-17	Open
52: P-18	☐	52	P-18	Open
54: P-19	☐	54	P-19	Open
56: P-20	☐	56	P-20	Open
57: P-21	☐	57	P-21	Open
59: P-22	☐	59	P-22	Open

4. Close the alternative editor.

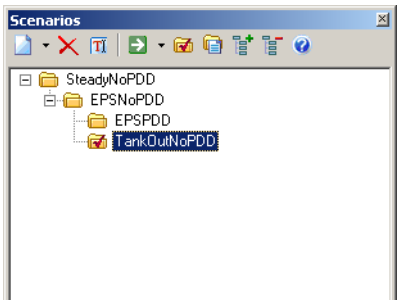
5. In the Scenarios manager create a new child scenario from **EPSNoPDD** called **TankOutNoPDD**.



6. Double-click the new scenario to open the scenario Properties editor. Change the **Initial Alternative** to **Near Tank Out** and then close the Properties editor.

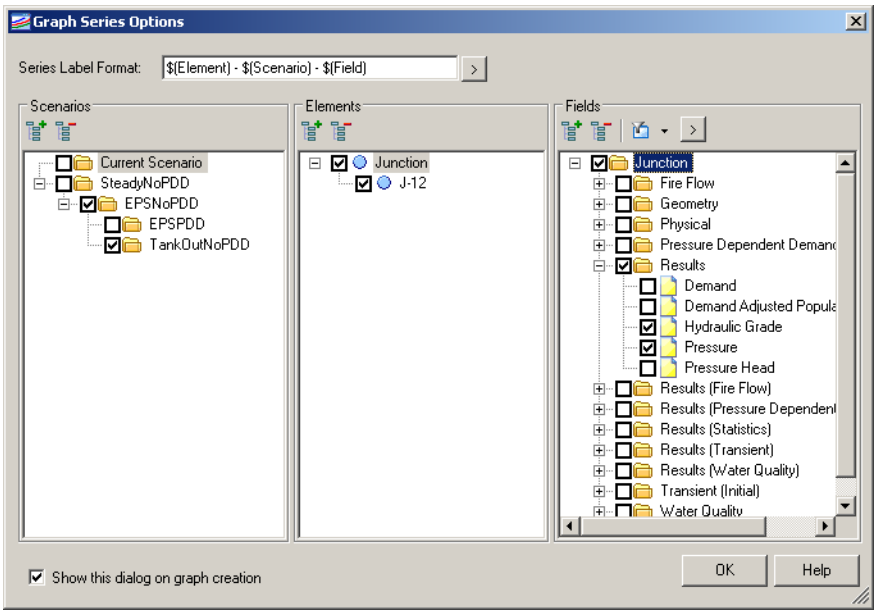


7. Make the **TankOutNoPDD** the current scenario and then click **Compute**.

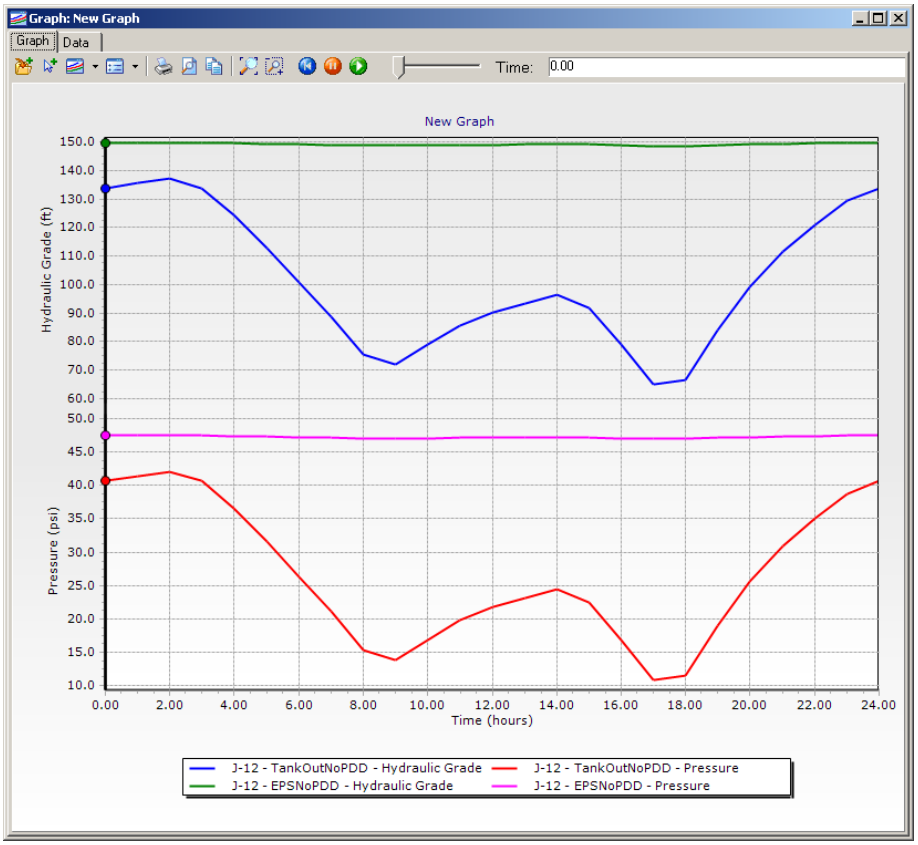


8. Review the calculation summary and then close it.
9. Right-click on **J-12** and select **Graph**.

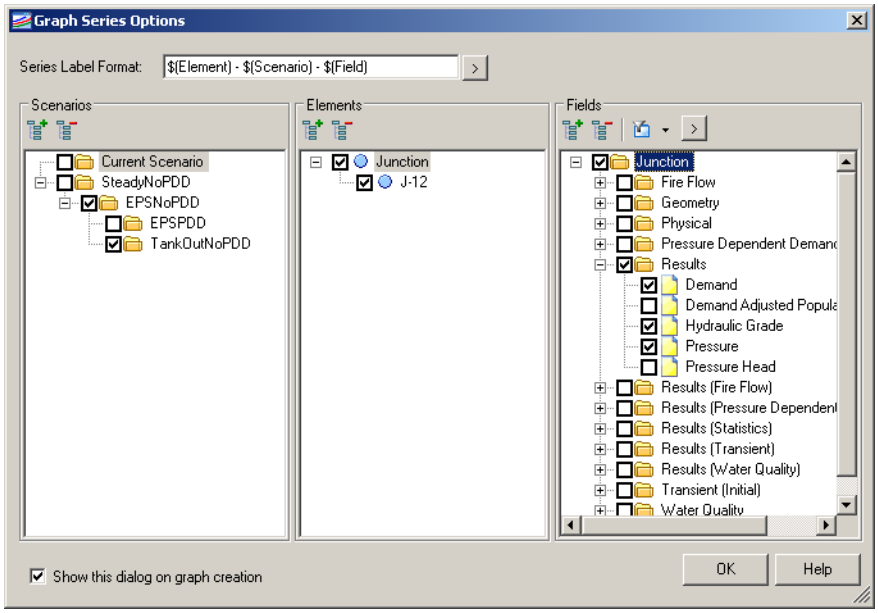
10. In Graph Series Options check the boxes for the **EPSNoPDD** and **TankOut-NoPDD** scenarios. Check the box next to the **Pressure** field (Hydraulic Grade is checked by default; leave it checked) and click **OK**.



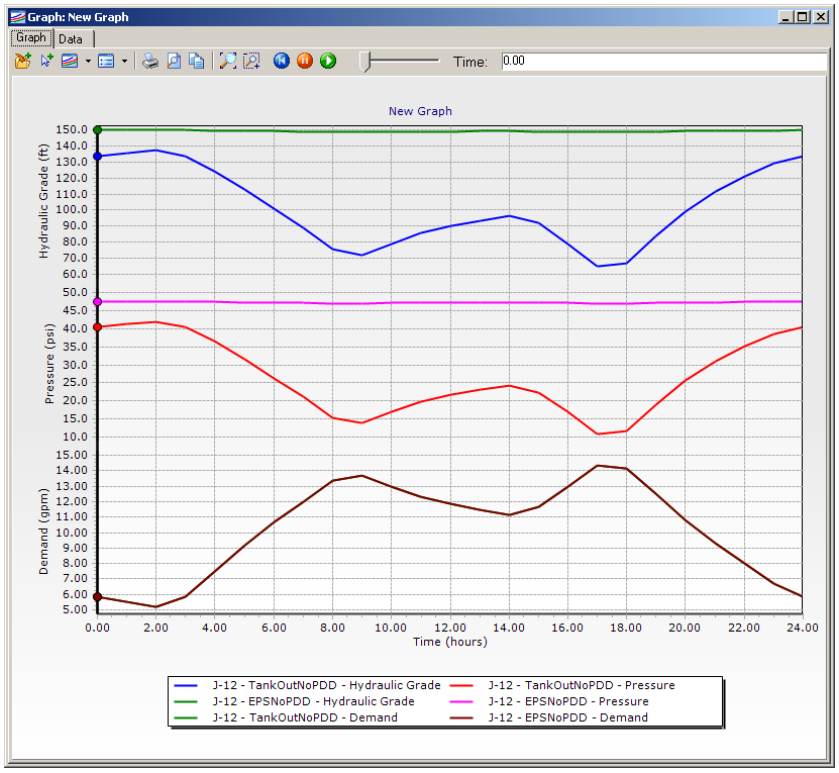
11. When the Near Tank is out of service there is a significant drop in pressure.



12. Examine the effect of the drop in pressure on Demand. Click the **Graph Series Options** button. In the Graph Series Options manager check **Demand** and then **OK**.



13. The demand did not change with pressure because it is not a PDD run; demand is independent of pressure, so there is a single line for Demand. Notice that when flow increases due to the time of day, there is not a corresponding drop in flow because of pressure drop.



14. Click the **Add to Graph Manager** button, rename the graph as **Pressure Demand J-12** and click **OK**.

15. Close the graph.

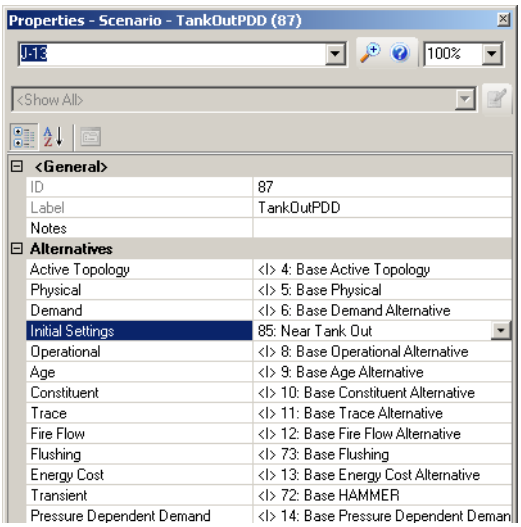
Step 5: Run PDD model with outage

1. Click **Analysis** > **Scenarios**.

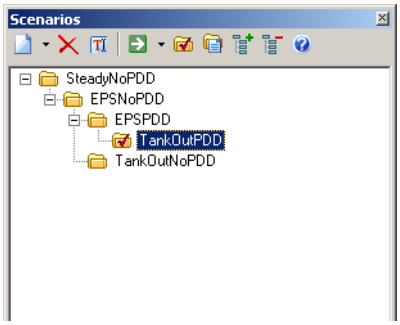
2. Right-click **EPSPDD** and select **New** > **Child Scenario**. Rename the new scenario **TankOutPDD**.

3. Double-click on **TankOutPDD** to open the scenario Properties editor.

4. Set the **Initial Settings** alternative to **Near Tank Out**.

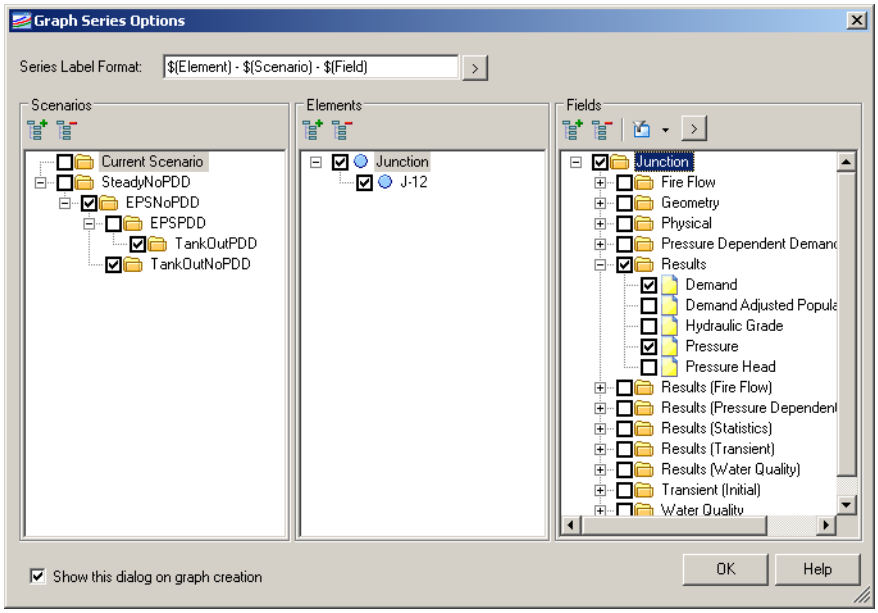


5. Close the Properties editor and make the **TankOutPDD** scenario current.

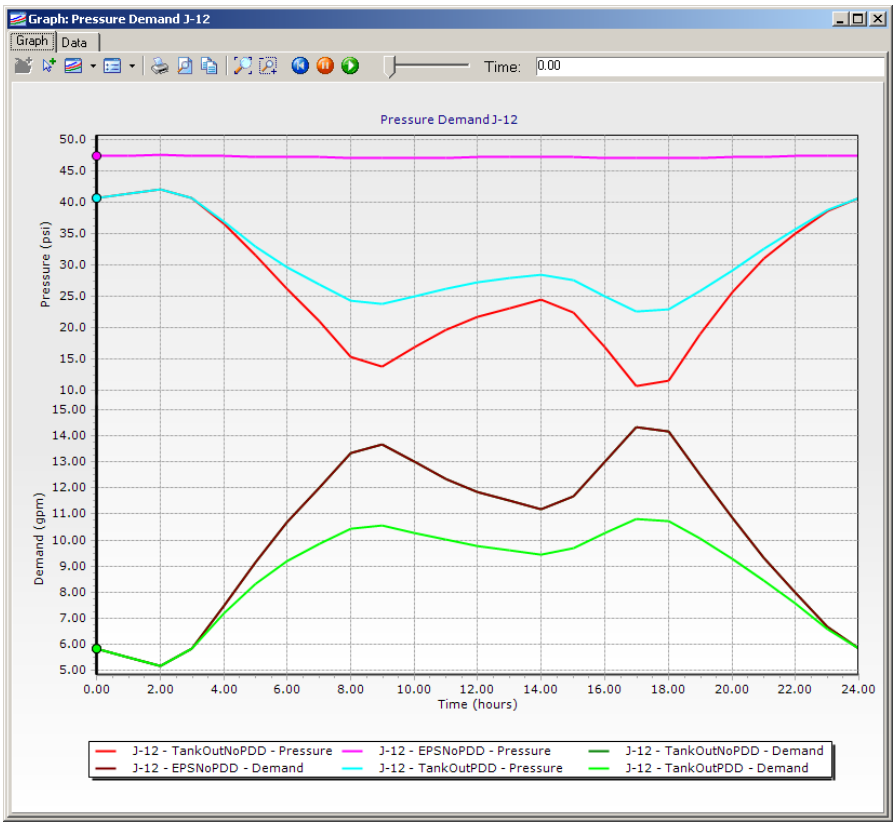


6. **Compute** the scenario, review the calculation summary, and close it.
7. Click **View** > **Graphs** and open the **Pressure Demand J-12** graph.

8. Click the **Graph Series Options** button and check **TankOutPDD** in the list of Scenarios, uncheck **Hydraulic Grade** in the list of Fields, and then click **OK**.



9. When PDD is used, the demand decreases when the pressure drops, so the overall pressure drop is not as great as when the pressure dependency of demands is ignored.

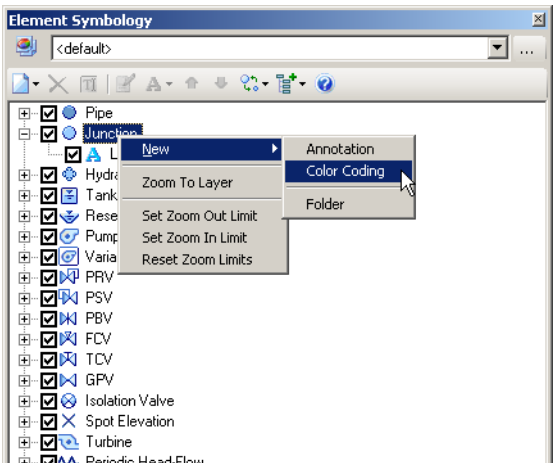


10. Close the graph.

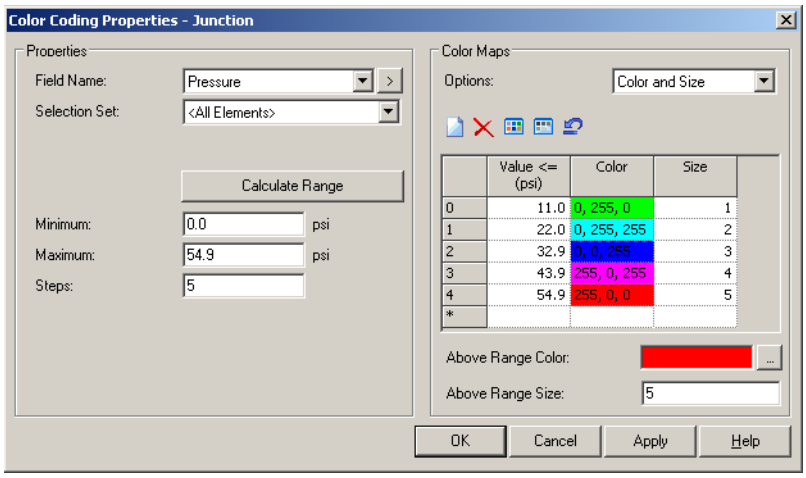
Step 6: Animating Results

1. Click **Analysis** > **Scenarios** and make the **TankOutNoPDD** scenario current.
2. Click **View** > **Element Symbology** and expand the Junction entry.

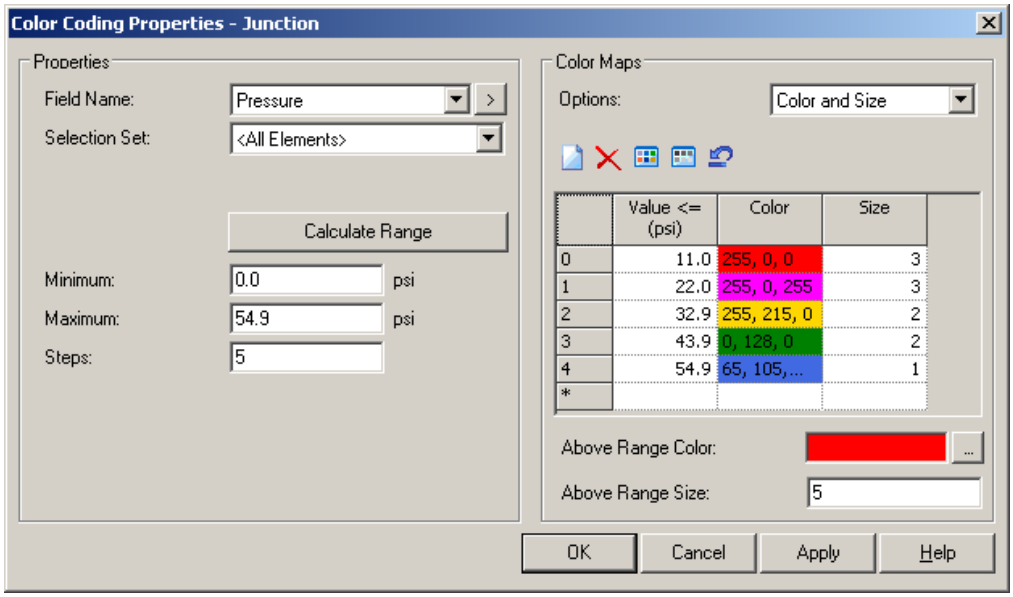
3. Right-click on **Junction** and then select **New** > **Color Coding**.



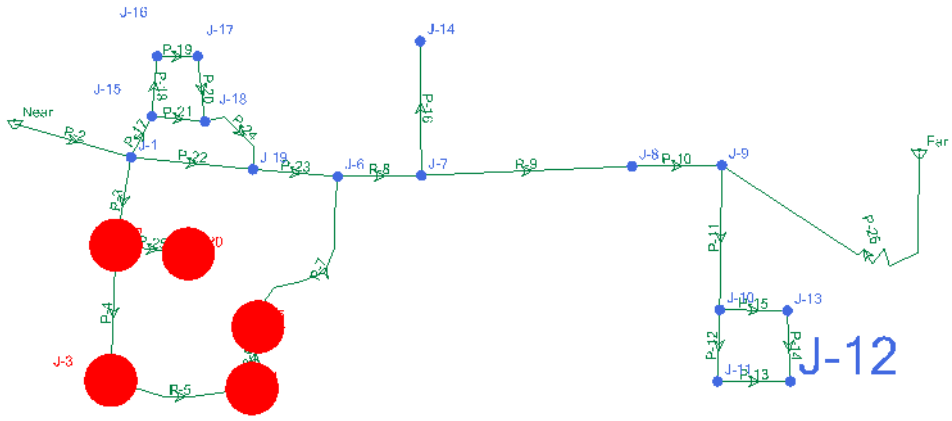
4. Select **Pressure** from the Field Name menu. Click the **Calculate Range** button and select **Full Range**. Select **Color and Size** from the Options menu. Click the **Initialize** button.



5. Manually edit the range and the color and size fields to look like the following example. The colors, in order of appearance are: Red, Magenta, Gold, Green, and Royal Blue. The sizes are 3, 3, 2, 2, and 1.



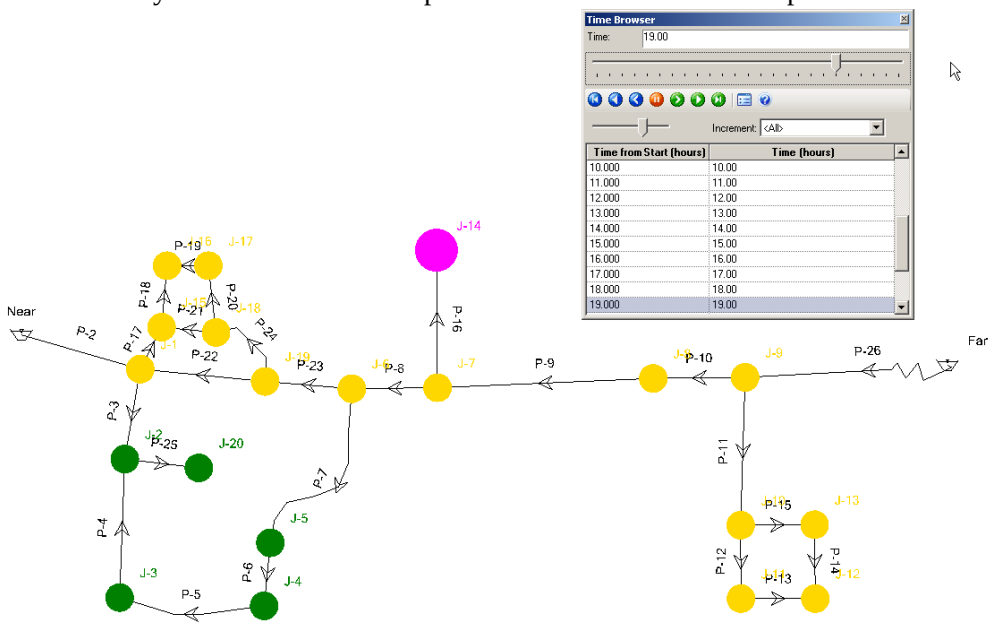
6. Click **OK**.



7. Click **Analysis** > **Time Browser** and click **Play**  . Observe how the colors and pressures change over the course of a day. Then click **Pause**  .

8. Click **Analysis** > **Scenarios** and select the **TankOutPDD** scenario. Make it current, compute, and then close the calculation summary.

9. Click Play and observe how the pressures in this run do not drop as low.



10. Pause the animation. Click **View** > Background Layers and check the PDDBackground box.



11. Close the open dialogs.

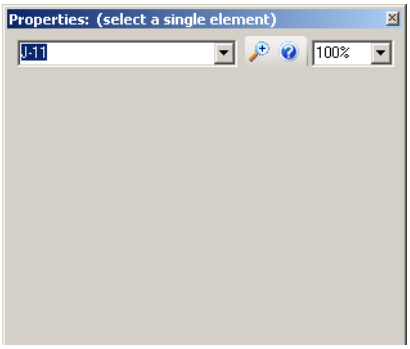
Criticality and Segmentation

In order to conduct a criticality analysis, WaterGEMS must identify the segments to be removed from service. Once the options have been set in the Criticality Studies level of the Segmentation and Criticality manager, you must decide which scenario is to be used for the analysis and set the rules for use of valving in the options tab.

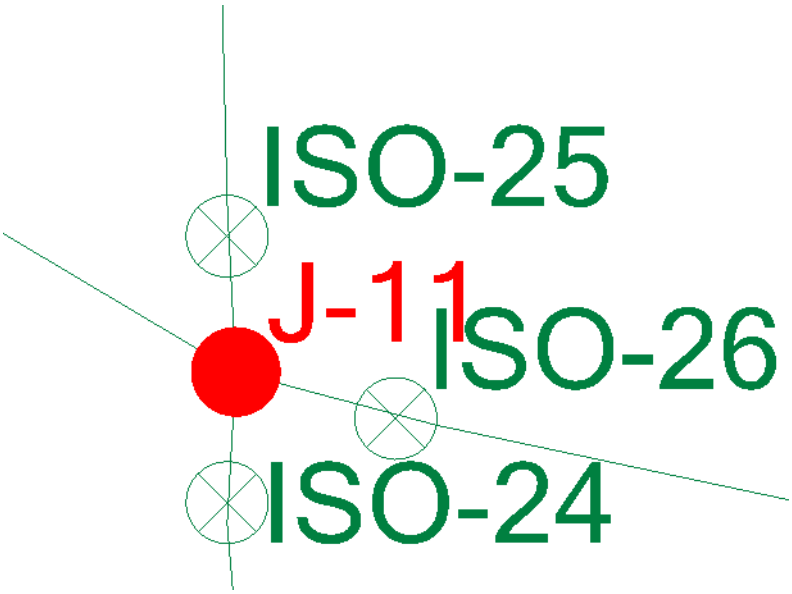
This lesson assumes that you have already constructed a model that has isolating valves and that these valves reference pipes and pressure dependent demand functions that have been set up.

Step 1: Check the Isolation Valves

1. Open **CritStart.wtg** from the Lessons folder.
2. Use **Pan**  to look at the placement of isolation valves (or hold the middle mouse button to pan).
3. Click **Edit** > **Find Element** and type **J-11** in the field, then click **Find**.

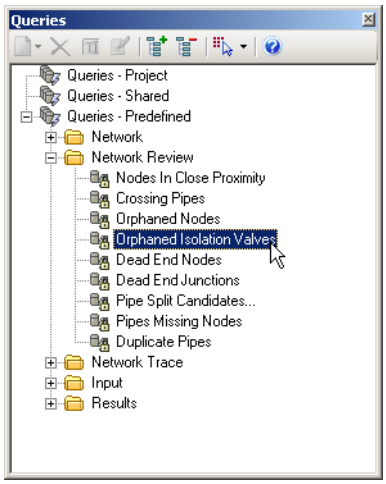


4. Click Zoom Window  to draw a box around J-11.



5. Check for valves not assigned to pipes.

   a. Click **View** > **Queries**. Under **Queries - Predefined**, expand the **Network Review** folder and double-click **Orphaned Isolation Valves**.

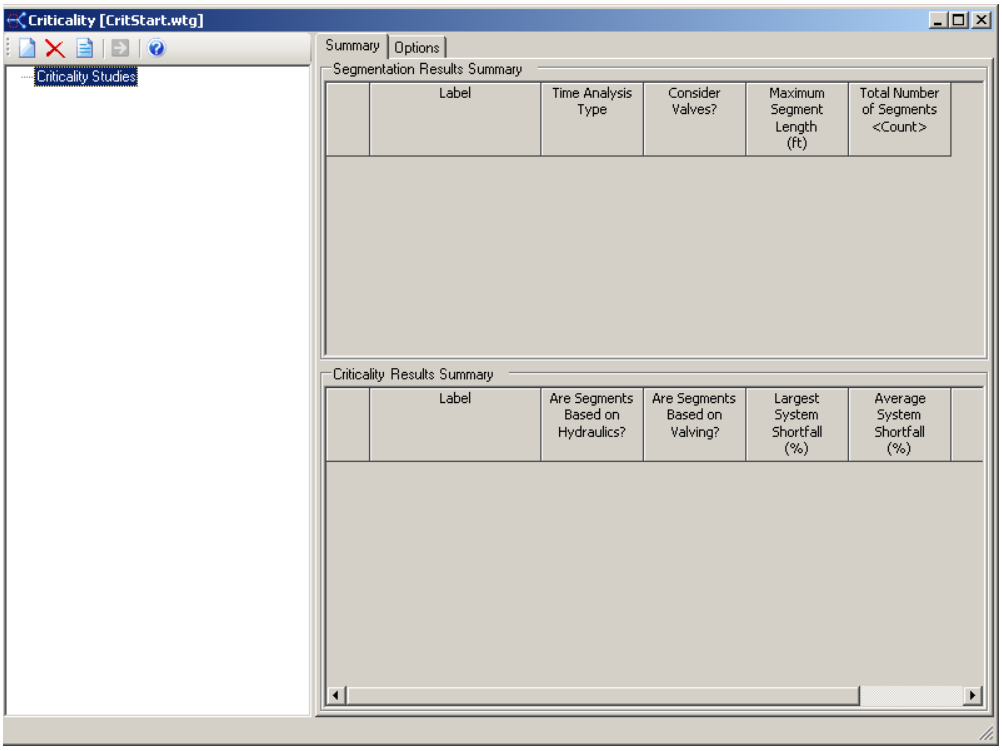


   b. All valves are assigned, however if the query turned up orphaned valves then you could delete the isolation valve, leave it orphaned, or select the valve and choose the menu from Referenced Pipe and select the pipe where the valve is located.

6. Close the Queries manager.

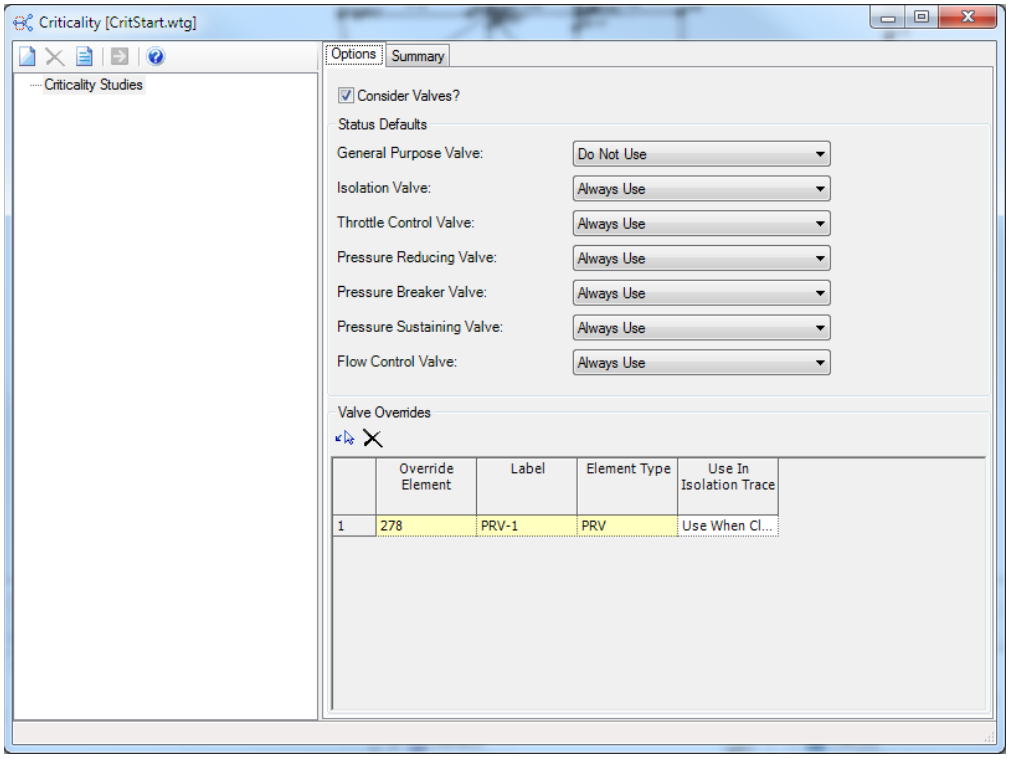
Step 2: Start the Criticality Manager and set up segmentation

1. Click **Analysis** > **Criticality** or click the **Criticality** button  .

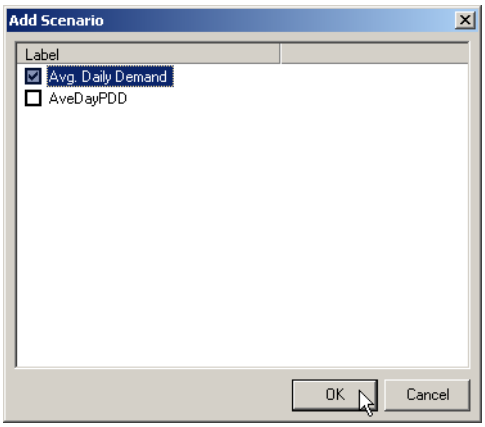


2. Click the **Options** tab and verify that **Consider Valves** is checked and that

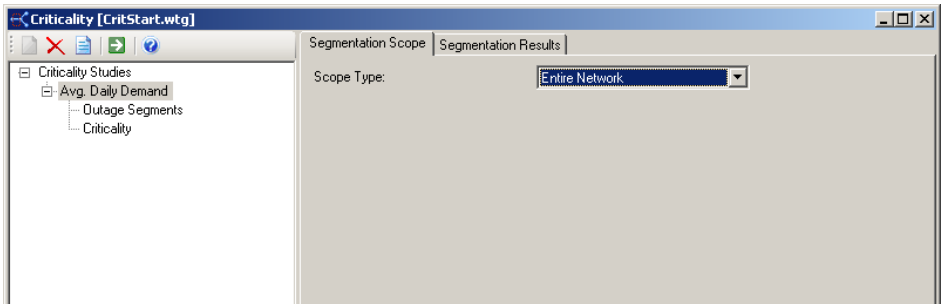
Always Use is selected in the **Isolation Valve** field.



3. Click **New**  . In the Add Scenario dialog, check **Avg. Daily Demand** and click **OK**.



4. Select **Entire Network** from the **Scope Type** menu.



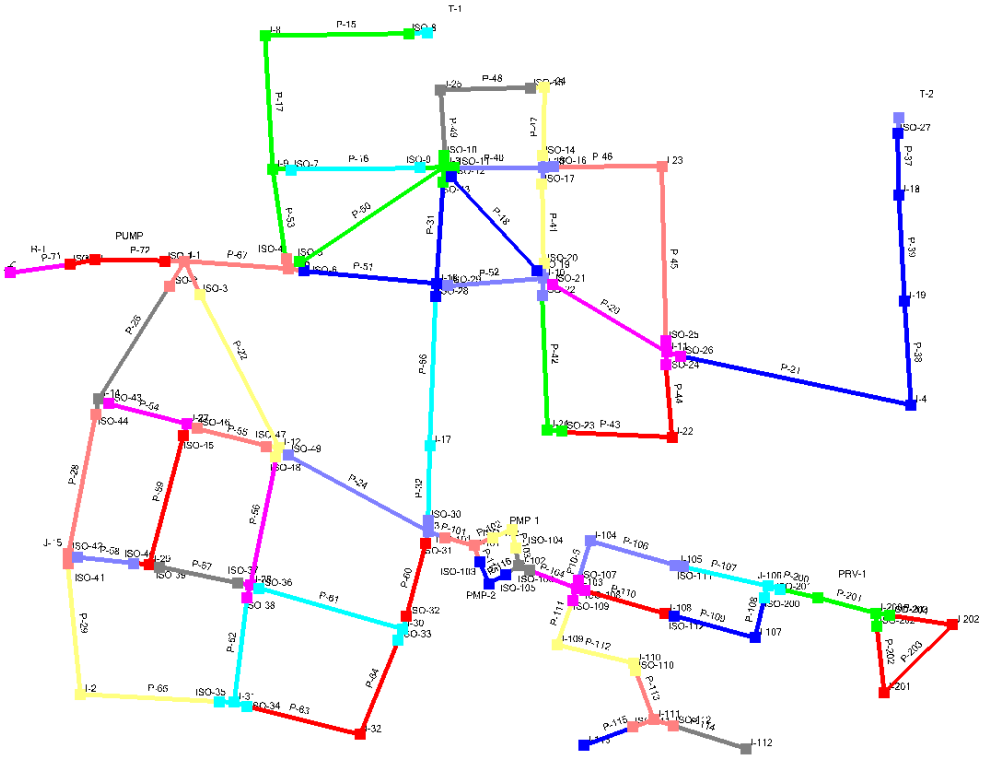
5. Click **Compute**  to perform the segmentation analysis, and click Yes at the prompt.

   Label - List of segments that were identified in the analysis. If Use Valves was not checked, there is one pipe per segment and the label of the pipe is listed next to the segment name. In this case, Use Valves was checked so the segments consist of a variety of pipes and nodes.

   General statistics are given for each segment.

   Affected Elements - The elements that make up or bound the segment.

6. Click Highlight Segments  to view the color coded segments in the drawing.



The results of segmentation can be advantageous. You can identify which segments require successfully operating a large number of valves in order to achieve a shutdown.

7. Right-click on the **Isolation Nodes <Count>** column and select **Sort** > **Sort Descending**.

15	Segment - 16	6	7	2	1,625.97	324.74	255, 255, 12...
27	Segment - 28	6	6	1	1,493.51	444.92	255, 0, 255, 0
21	Segment - 22	5	5	1	2,095.09	507.51	255, 128, 12...
13	Segment - 14	4	4	1	1,225.20	240.57	255, 128, 12...
18	Segment - 19	4	4	1	913.06	186.78	255, 0, 0, 255
22	Segment - 23	4	4	1	1,098.58	215.70	255, 255, 0,...
25	Segment - 26	4	4	1	1,476.87	463.36	255, 255, 25...
29	Segment - 30	4	4	1	1,206.44	420.09	255, 128, 12...
30	Segment - 31	4	4	1	1,677.73	513.67	255, 255, 0,...
38	Segment - 39	4	4	1	1,971.05	387.01	255, 255, 0,...
2	Segment - 3	3	4	2	983.26	193.06	255, 0, 255, 0
6	Segment - 7	3	3	1	1,184.10	103.80	255, 128, 12...
11	Segment - 12	3	4	2	2,382.40	742.87	255, 0, 255, 0
14	Segment - 15	3	3	1	364.53	71.58	255, 255, 0,...
20	Segment - 21	3	3	1	1,086.15	213.27	255, 0, 255,...
24	Segment - 25	3	3	1	1,504.80	163.17	255, 255, 12...
26	Segment - 27	3	3	1	925.16	181.65	255, 255, 0, 0
28	Segment - 29	3	3	1	814.27	275.41	255, 0, 255,...
32	Segment - 33	3	3	1	1,013.44	198.99	255, 255, 12...
36	Segment - 37	3	3	1	1,006.64	197.65	255, 0, 255,...
39	Segment - 40	3	3	1	1,990.13	390.76	255, 128, 12...
40	Segment - 41	3	3	1	1,007.70	197.86	255, 255, 12...

The segments at the top of the list usually prove to be the most difficult to isolate and may require investigation to make them less susceptible to issues that arise due to an inoperative valve.

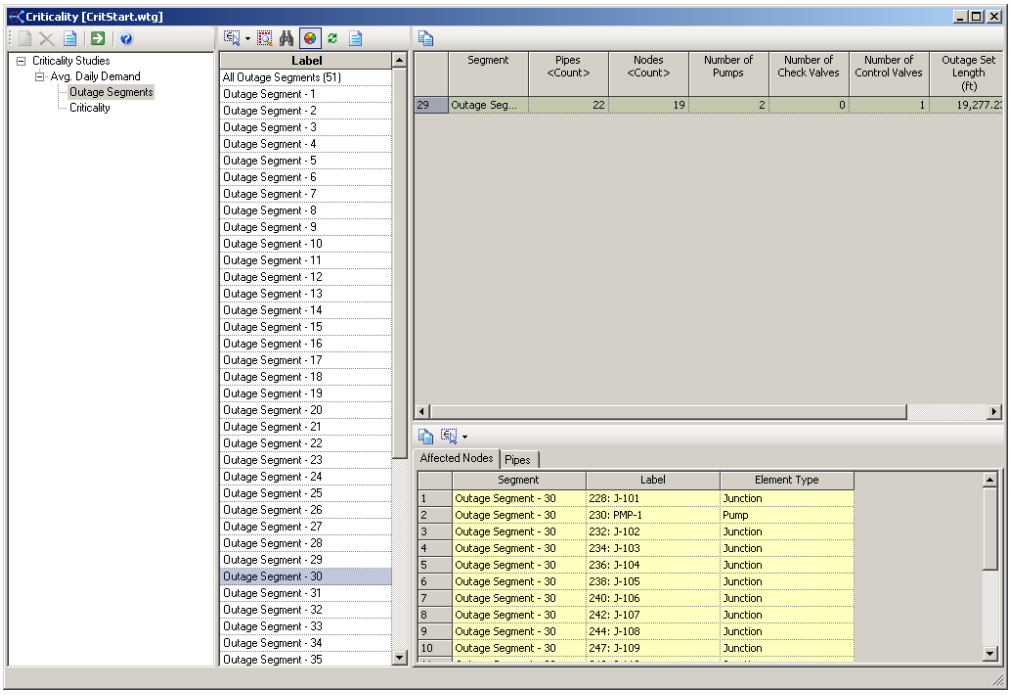
Step 3: Perform outage analysis to identify if isolating a segment causes other segments to be isolated

1. Click on **Outage Segments** and then **Compute**  **,** and click **Yes** at the prompt**.**

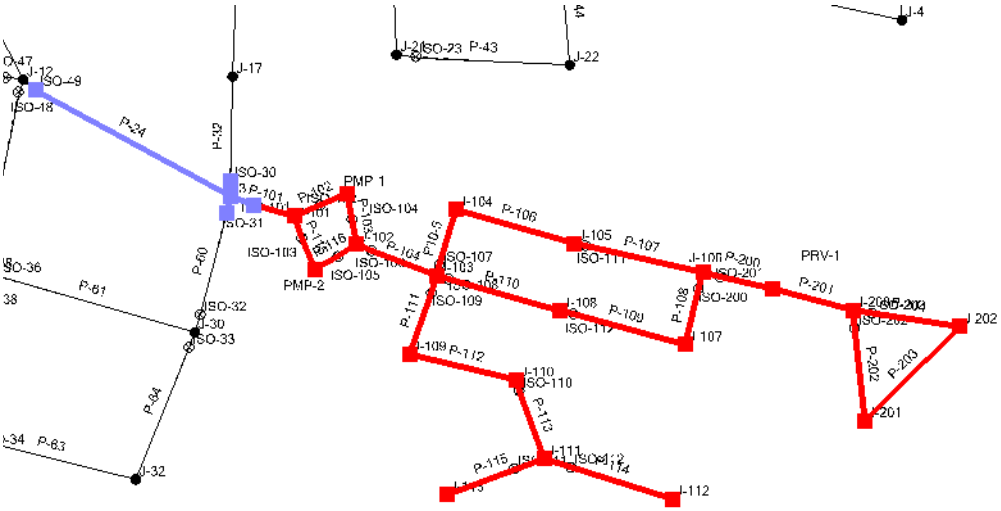
2. Right-click on **Outage Set Length** and select **Sort** > **Sort Descending** to find out which segments have outages that will cause significant downstream outages.

	Segment	Affected Elements <Count>	Pipes <Count>	Number of Pumps	Number of Check Valves	Number of Control Valves	Outage Set Length (ft)
	Outage Seg...	19	22	2	0	1	18,915.43
	Outage Seg...	18	21	2	0	1	17,376.80
	Outage Seg...	15	17	0	0	1	14,090.48
	Outage Seg...	14	16	0	0	1	12,810.29
	Outage Seg...	4	5	0	0	1	2,779.08
	Outage Seg...	3	3	0	0	0	2,367.70
	Outage Seg...	2	3	0	0	0	2,362.72
	Outage Seg...	2	2	0	0	0	1,141.13
	Outage Seg...	0	0	0	0	0	0.00
	Outage Seg...	0	0	0	0	0	0.00
	Outage Seg...	0	0	0	0	0	0.00
	Outage Seg...	0	0	0	0	0	0.00
	Outage Seg...	0	0	0	0	0	0.00
	Outage Seg...	0	0	0	0	0	0.00
	Outage Seg...	0	0	0	0	0	0.00
	Outage Seg...	0	0	0	0	0	0.00
	Outage Seg...	0	0	0	0	0	0.00

3. Select the segement with the highest **Outage Set Length** from the **Label** column, click **Highlight Segments**  to view the color coded segments in the drawing.



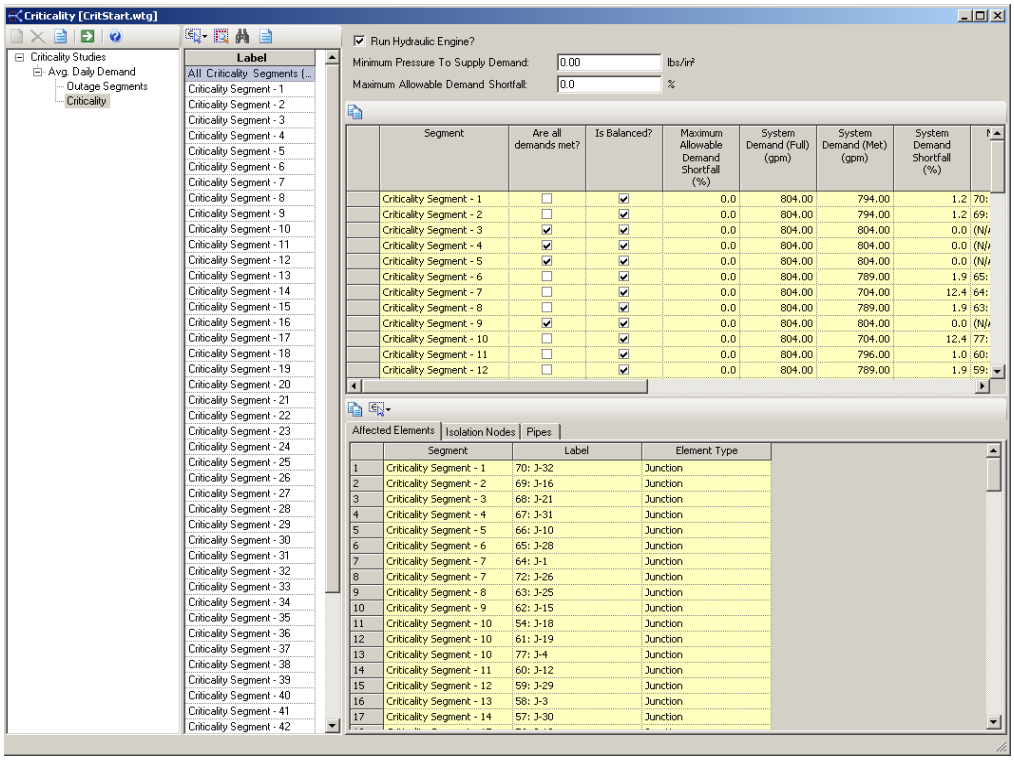
4. View the drawing to see that the pipe with the highest Outage Set Length is in blue and the downstream outage segments that will be out of service are in red.



Step 4: Run criticality analysis

The most important function of criticality analysis is the ability of the system to meet demands given a segment outage. A form of this analysis is the case where the shortfalls are determined solely based on connectivity. If the node is connected back to the source, it is assumed the demands are met. This type of run does not involve the hydraulic engine and runs very fast.

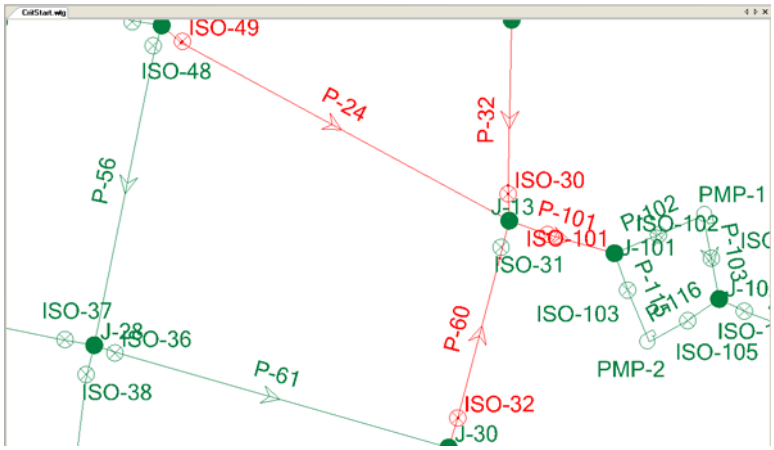
1. Select **Criticality** and make sure **Run Hydraulic Engine?** is unchecked. Then click **Compute** .



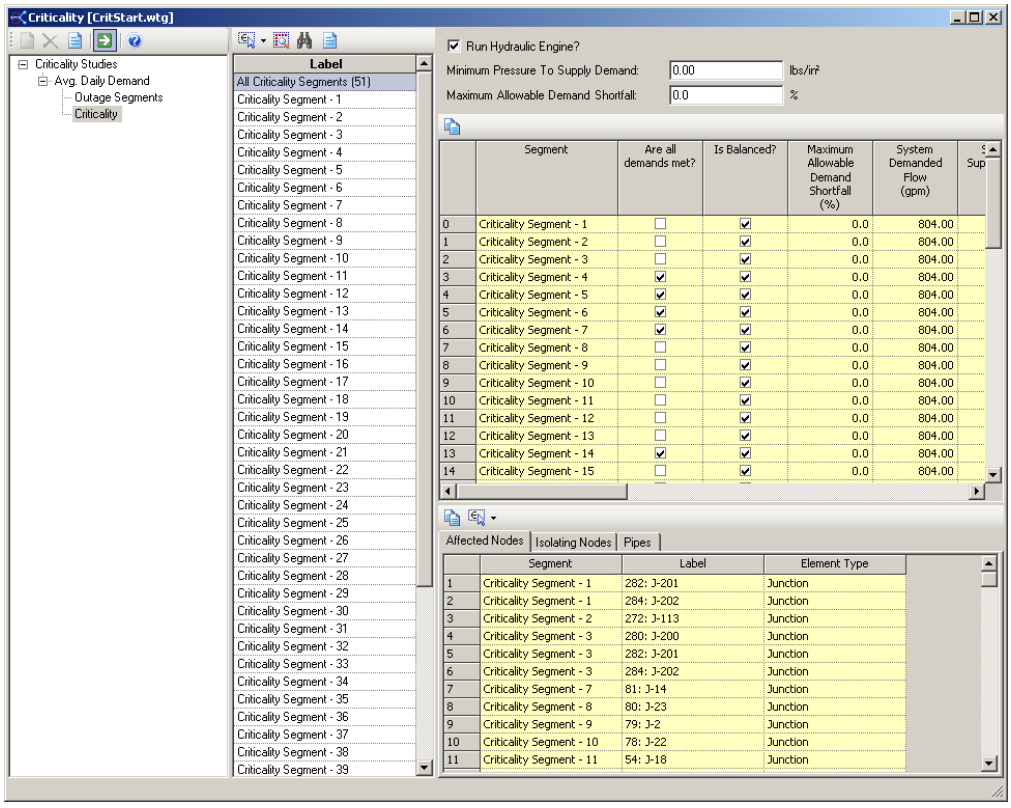
2. Right-click on the **System Demand Shortfall** column and select **Sort** > **Sort Descending**.

	Segment	Are all demands met?	Is Balanced?	Maximum Allowable Demand Shortfall (%)	System Demand (Full) (gpm)	System Demand (Met) (gpm)	System Demand Shortfall (%)	
	Criticality Segment - 15	☐	☑	0.0	804.00	528.00	34.3	56:
	Criticality Segment - 27	☐	☑	0.0	804.00	543.00	32.5	232
	Criticality Segment - 30	☐	☑	0.0	804.00	543.00	32.5	232
	Criticality Segment - 42	☐	☑	0.0	804.00	543.00	32.5	232
	Criticality Segment - 26	☐	☑	0.0	804.00	547.00	32.0	234
	Criticality Segment - 7	☐	☑	0.0	804.00	704.00	12.4	64:
	Criticality Segment - 10	☐	☑	0.0	804.00	704.00	12.4	77:
	Criticality Segment - 20	☐	☑	0.0	804.00	712.00	11.4	247
	Criticality Segment - 36	☐	☑	0.0	804.00	724.00	10.0	79:
	Criticality Segment - 13	☐	☑	0.0	804.00	749.00	6.8	58:
	Criticality Segment - 25	☐	☑	0.0	804.00	752.00	6.5	236
	Criticality Segment - 19	☐	☑	0.0	804.00	764.00	5.0	251

3. Select the segment with the highest Outage Set Length (as determined in Step 3) from the Label list and then click **Zoom**  .



4. Now run a criticality analysis that uses the hydraulic network engine to determine the impact of segment outages. Check the **Run Hydraulic Engine?** box and click **Compute**  .



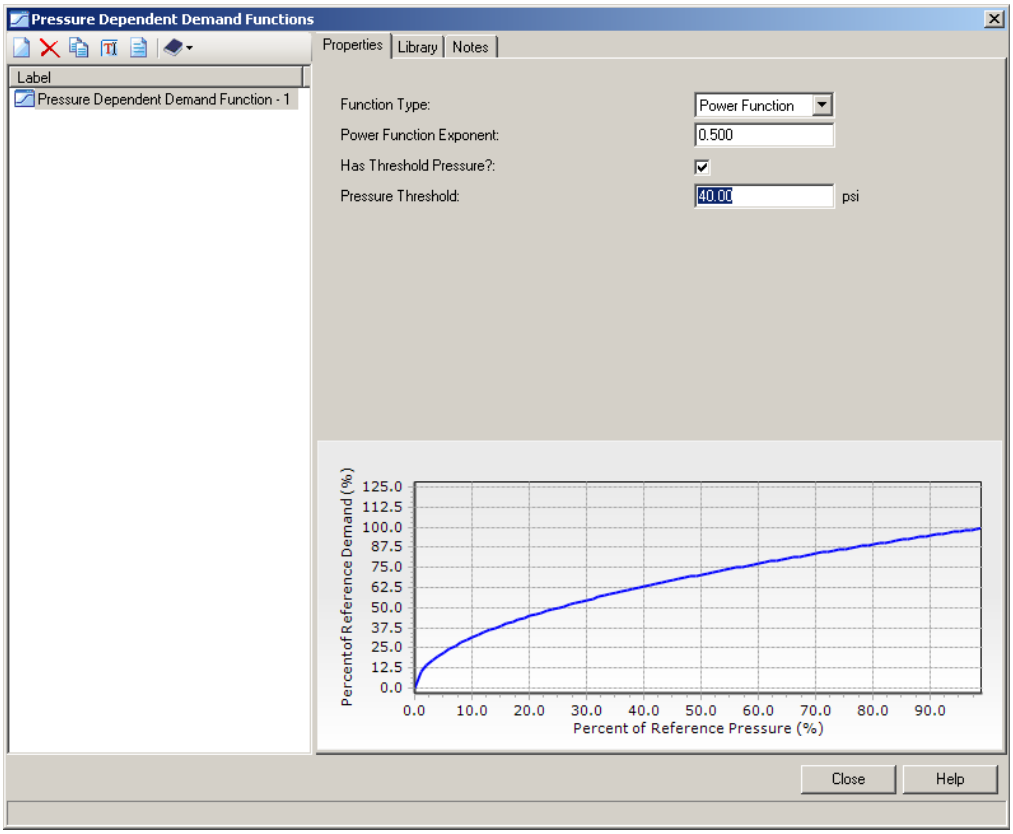
The System Demand Shortfall percentages are the same as the run without hydraulic calculations. This is because the flows are delivered to all nodes that are connected regardless of the pressure.

Step 5: Run criticality analysis hydraulic with PDD

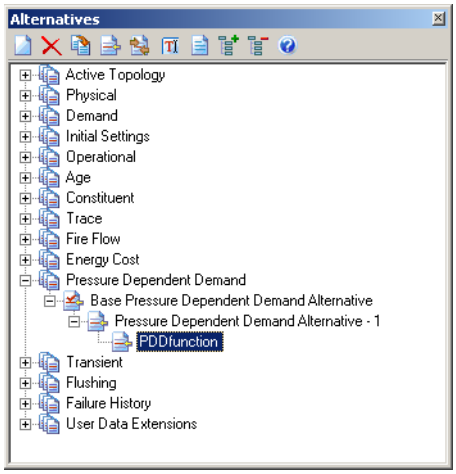
While other types of runs can indicate which segment outages cause the most demand to be isolated from the system, they are not the way to determine the impact on nodes that remain connected to the source but receive much less flow due to the outage.

In order to make these calculations, the demand in the system must be modeled using pressure dependent demands (PDD).

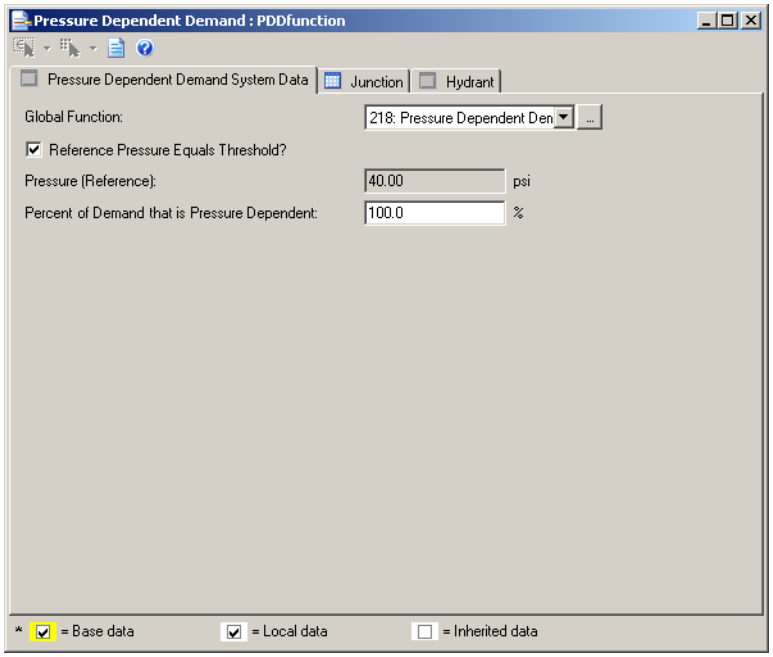
1. Close the Criticality manager and click **Components** > **Pressure Dependent Demand Functions**.
2. Set the **Pressure Threshold** to **40** psi and then close the PDD Function manager.



3. Click **Analysis** > **Alternatives**, expand the **Pressure Dependent Demand** node and select **PDDfunction**.

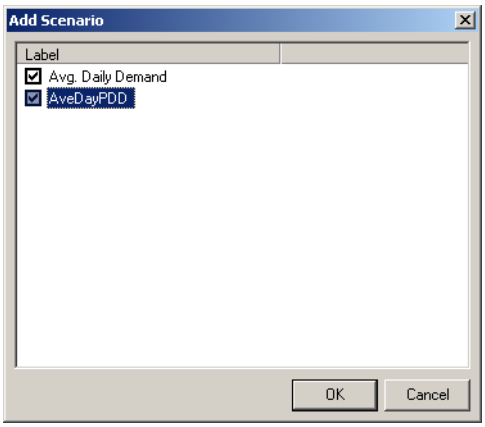


4. Double-click the **PDDfunction** alternative to verify which PDD function is being used, that the reference pressure (the pressure at which all demand is met) is equal to the threshold pressure, and that 100% of the demand is pressure dependent.



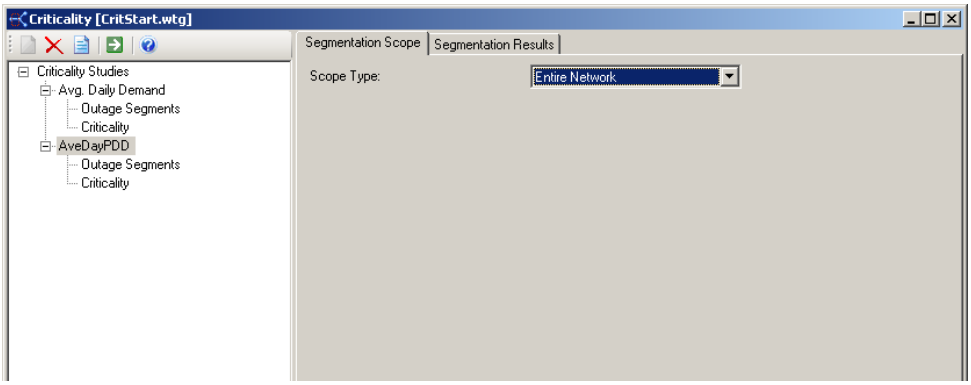
5. Close the alternative editor and the Alternatives manager.

6. Click **Analysis** > **Criticality**. Highlight Criticality Studies and click the New button. Check the box for **AveDayPDD**.

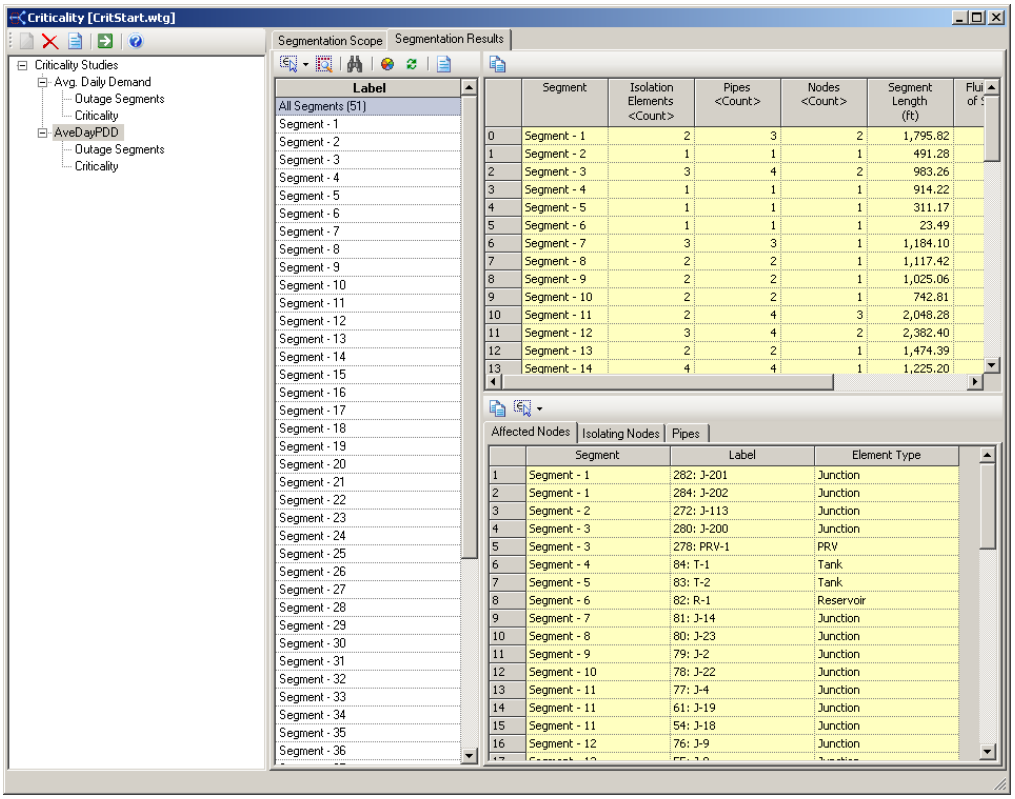


7. Click **OK**.

8. On the **Segmentation Scope** tab, select **Entire Network** from the **Scope Type** menu.



9. Select **AveDayPDD** and click **Compute**  . Click **Yes** in the prompt that appears.



The segmentation results are the same as the first scenario because the same valving is used.

10. Select **Criticality** below **AveDayPDD.** Check the **Run Hydraulic Engine?** box and click **Compute**  .

11. Right click on the **System Demand Shortfall** column and select **Sort** > **Sort Descending**.

	Segment	Are all demands met?	Is Balanced?	Maximum Allowable Demand Shortfall (%)	System Demanded Flow (gpm)	System Supplied Flow (gpm)	System Demand Shortfall (%)	Node with Largest Percent Demand Shortfall	De Shou Wors
29	Criticality Segment - 30	☐	☑	0.0	804.00	524.74	34.7	56: J-13	
39	Criticality Segment - 40	☐	☑	0.0	804.00	539.83	32.9	232: J-102	
40	Criticality Segment - 41	☐	☑	0.0	804.00	539.83	32.9	232: J-102	
41	Criticality Segment - 42	☐	☑	0.0	804.00	539.83	32.9	232: J-102	
38	Criticality Segment - 39	☐	☑	0.0	804.00	543.81	32.4	234: J-103	
15	Criticality Segment - 16	☐	☑	0.0	804.00	674.29	16.1	64: J-1	
10	Criticality Segment - 11	☐	☑	0.0	804.00	684.71	14.8	77: J-4	
8	Criticality Segment - 9	☐	☑	0.0	804.00	704.95	12.3	79: J-2	
33	Criticality Segment - 34	☐	☑	0.0	804.00	708.46	11.9	247: J-109	
27	Criticality Segment - 28	☐	☑	0.0	804.00	731.78	9.0	58: J-3	
37	Criticality Segment - 38	☐	☑	0.0	804.00	748.40	6.9	236: J-104	
28	Criticality Segment - 29	☐	☑	0.0	804.00	749.76	6.7	57: J-30	
32	Criticality Segment - 33	☐	☑	0.0	804.00	757.88	5.7	251: J-111	
12	Criticality Segment - 13	☐	☑	0.0	804.00	758.59	5.6	75: J-17	
36	Criticality Segment - 37	☐	☑	0.0	804.00	762.00	5.2	240: J-106	
14	Criticality Segment - 15	☐	☑	0.0	804.00	764.70	4.9	73: J-27	
34	Criticality Segment - 35	☐	☑	0.0	804.00	768.22	4.5	244: J-108	
22	Criticality Segment - 23	☐	☑	0.0	804.00	769.63	4.3	65: J-28	

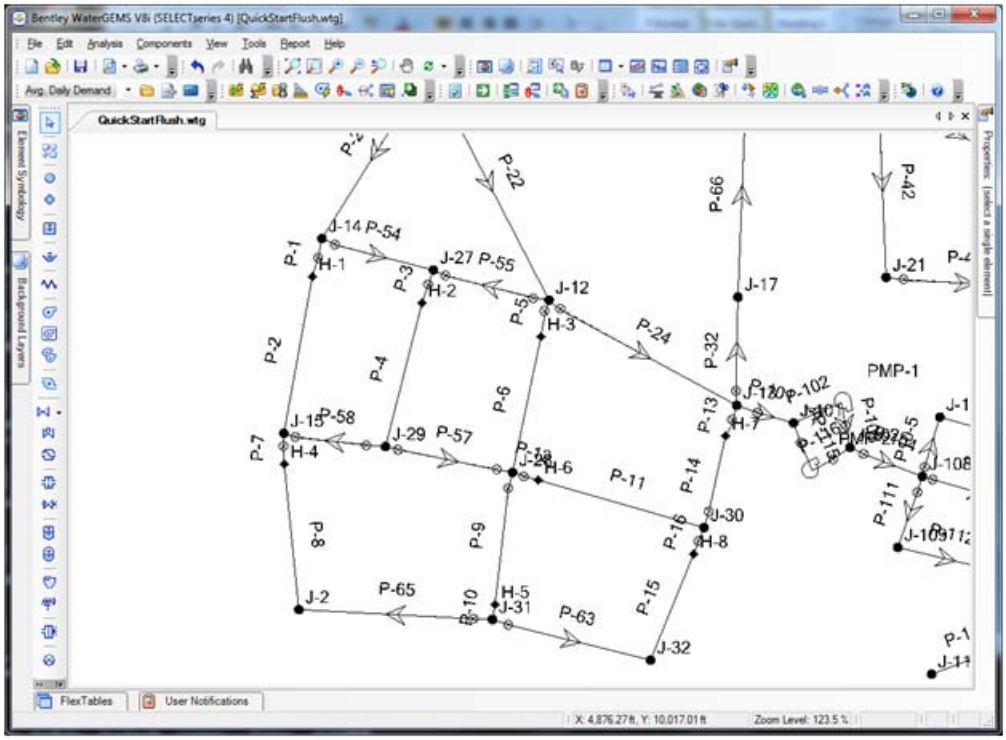
Notice that the shortfalls have increased over the previous runs because the runs that incorporate PDD account for the impact on nodes that receive water but at a lower pressure than under normal circumstances.

12. Close the Criticality manager.

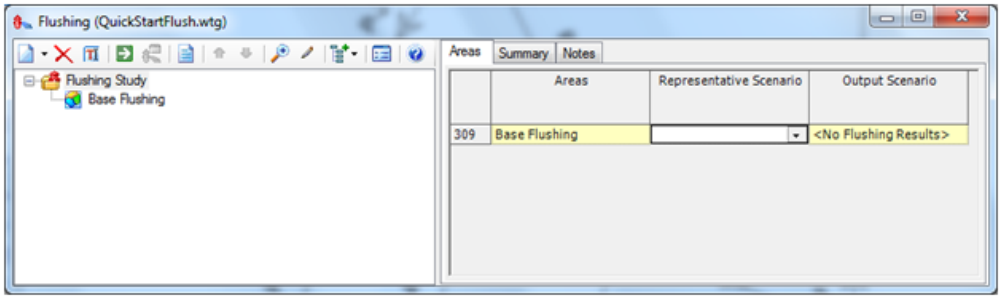
Flushing

In this Quick Start lesson, you will set up and run a series of conventional flushes (no valve operation) and one unidirectional flush.

1. Open the model called QuickStartFlush. Then zoom to the south west portion of the model View > Zoom > Window so that it looks like below:

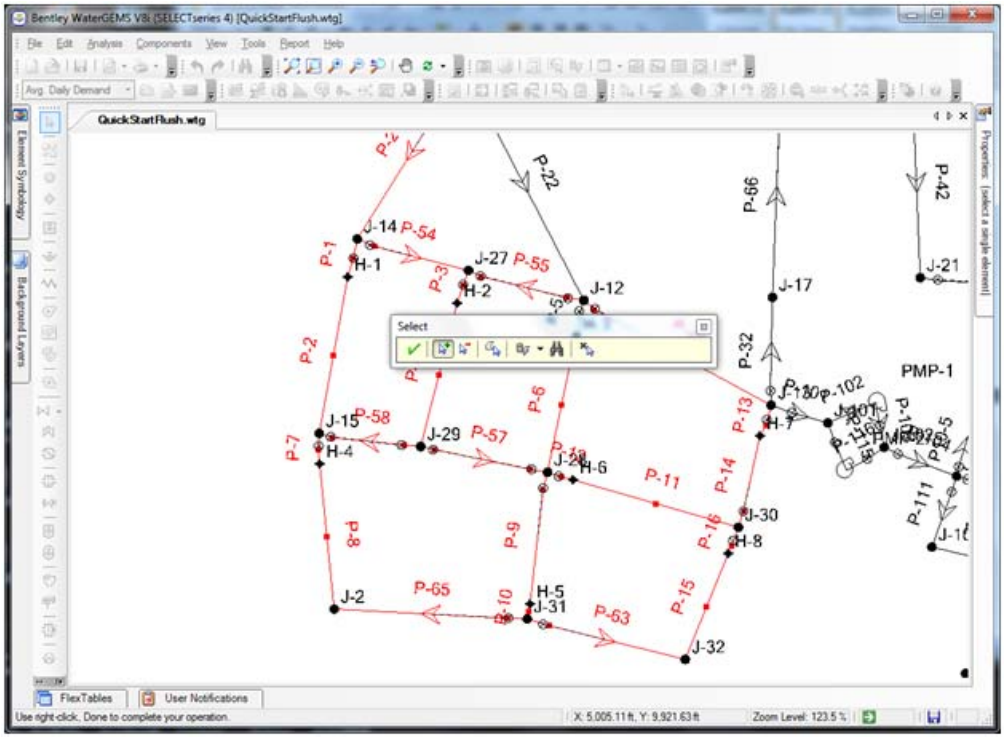


2. Pick Analysis > Flushing or click on the Flushing button on the toolbar [show button]. This opens the flushing manager dialog.

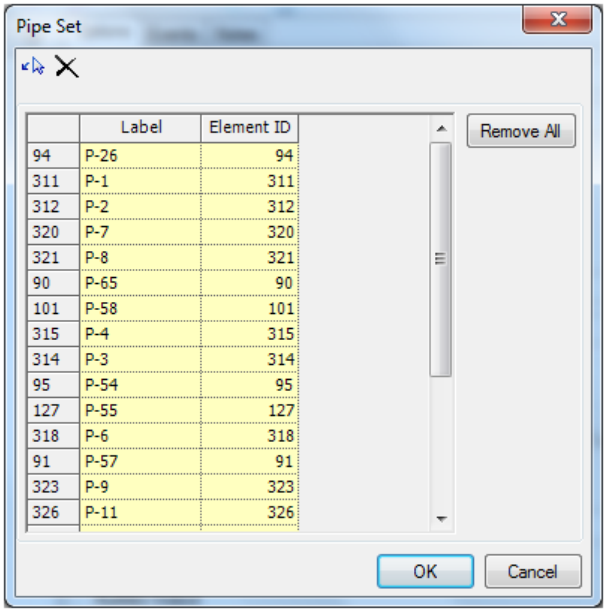


3. Pick Avg. Daily Demand as the Representative Scenario in the right pane.
4. In the left pane, highlight "Base Flushing", pick the Rename button (third from left on top) and change the name to "Conventional".

5. In the right pane, create a pipe set for which you will calculate flushing properties by picking the ellipse next to Pipe Set, and Select in Drawing. Select the pipes shown below. (It may be necessary to zoom in to some of the shorter pipes to select them.)

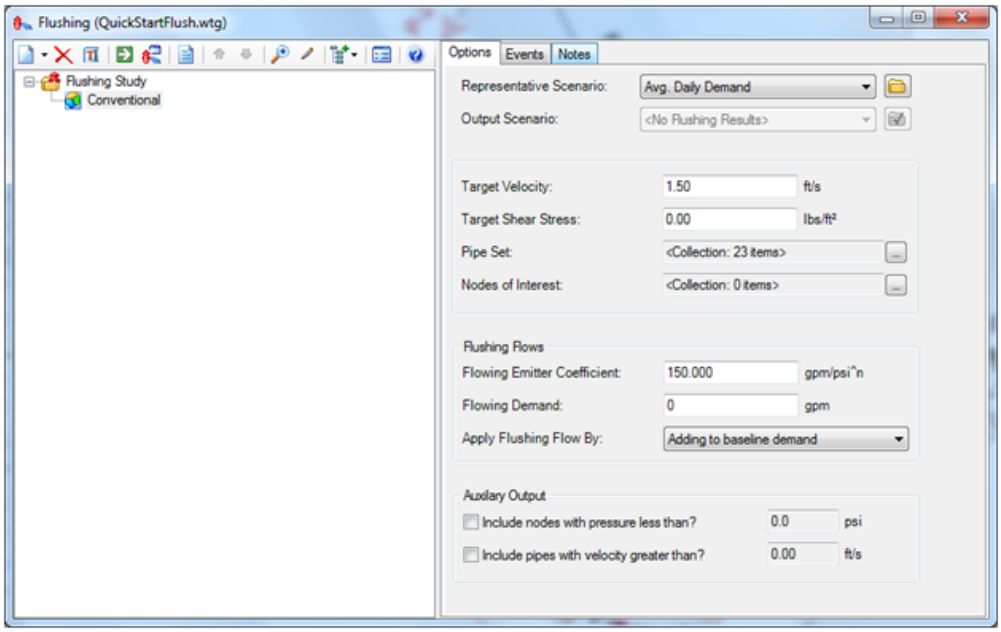


6. Pick the check box to complete the selection.

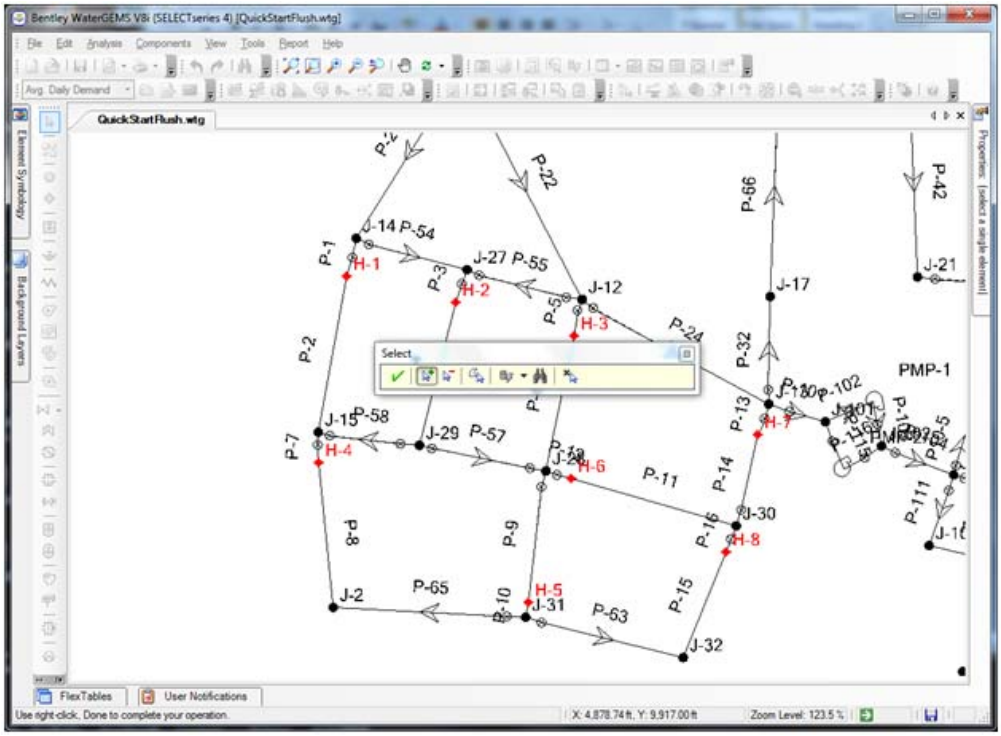


7. Pick OK..

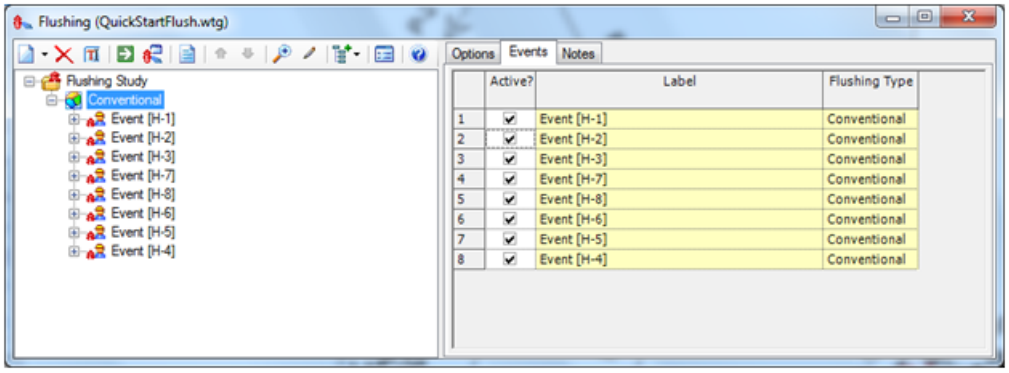
8. Set the Flow Emitter Coefficient to 150. The dialog should look like this:



9. Pick the Events tab over the right pane and then while highlighting "Conventional" in the left pane, pick New (first button on left top) > New Conventional Events (Batch) and select hydrants H-1 through H-8 and click the check mark.

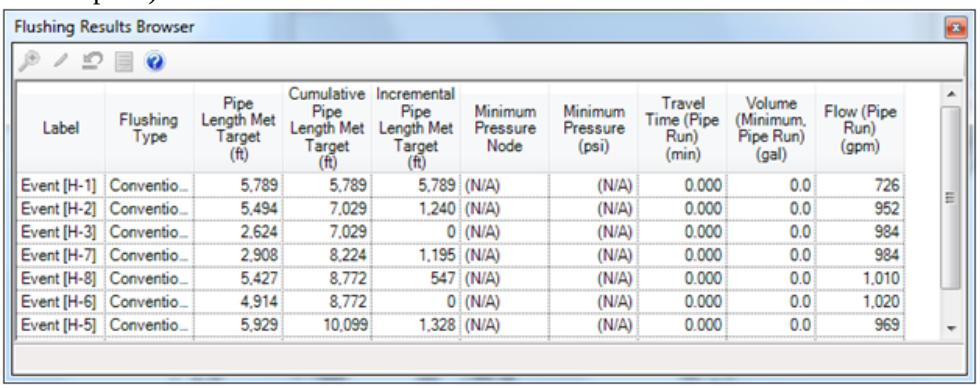


The Flushing Manager will look like this:



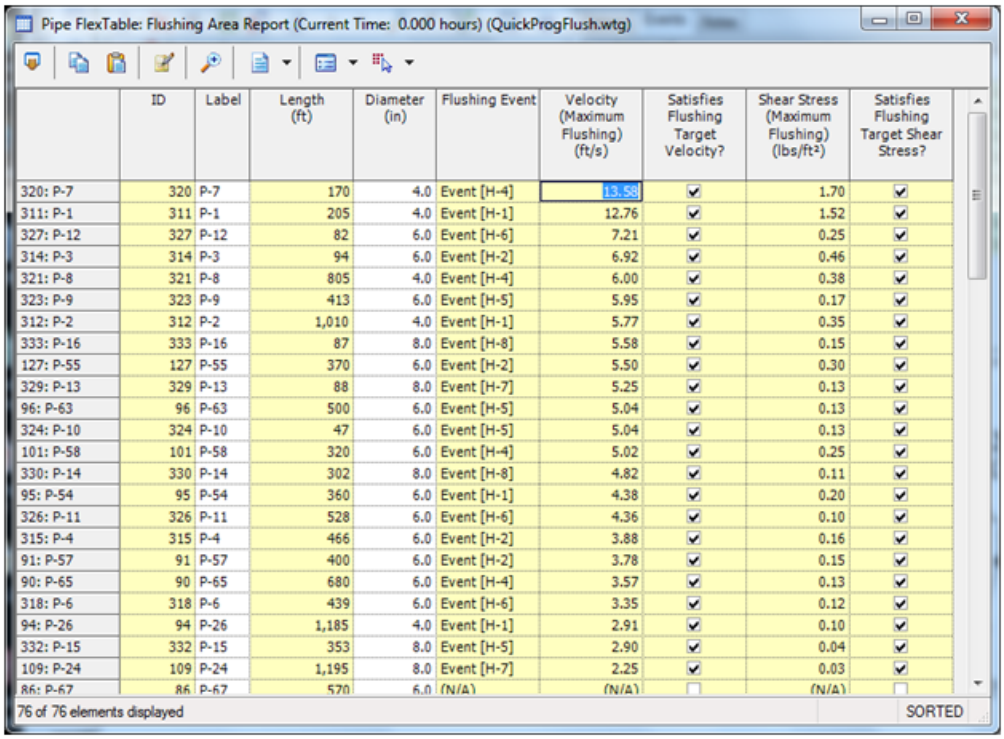
You can run the 8 events in sequence by clicking the Compute button (fourth from left in left pane).

10. To check the results, open the Flushing Results Browser (fifth button from left in left pane). It shows the effect of each event.



Label	Flushing Type	Pipe Length Met Target (ft)	Cumulative Pipe Length Met Target (ft)	Incremental Pipe Length Met Target (ft)	Minimum Pressure Node	Minimum Pressure (psi)	Travel Time (Pipe Run) (min)	Volume (Minimum, Pipe Run) (gal)	Flow (Pipe Run) (gpm)
Event [H-1]	Conventio...	5,789	5,789	5,789	(N/A)	(N/A)	0.000	0.0	726
Event [H-2]	Conventio...	5,494	7,029	1,240	(N/A)	(N/A)	0.000	0.0	952
Event [H-3]	Conventio...	2,624	7,029	0	(N/A)	(N/A)	0.000	0.0	984
Event [H-7]	Conventio...	2,908	8,224	1,195	(N/A)	(N/A)	0.000	0.0	984
Event [H-8]	Conventio...	5,427	8,772	547	(N/A)	(N/A)	0.000	0.0	1,010
Event [H-6]	Conventio...	4,914	8,772	0	(N/A)	(N/A)	0.000	0.0	1,020
Event [H-5]	Conventio...	5,929	10,099	1,328	(N/A)	(N/A)	0.000	0.0	969

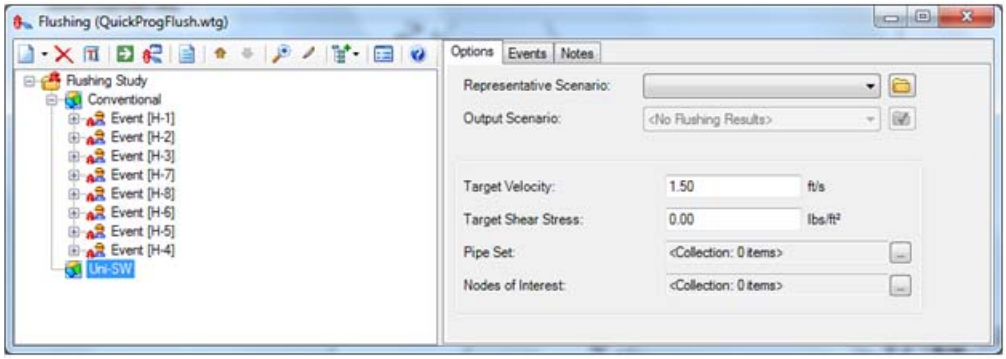
11. Close this Browser and open the Flushing Area flex table. View > Flex Tables > Flushing Area Report. Right click on the Velocity Maximum Flushing column and Sort > Descending (or Filter on Velocity > 0). This table shows the Velocity and Shear Stress for the pipes in the Pipe Set. All of these exceeded the Target.



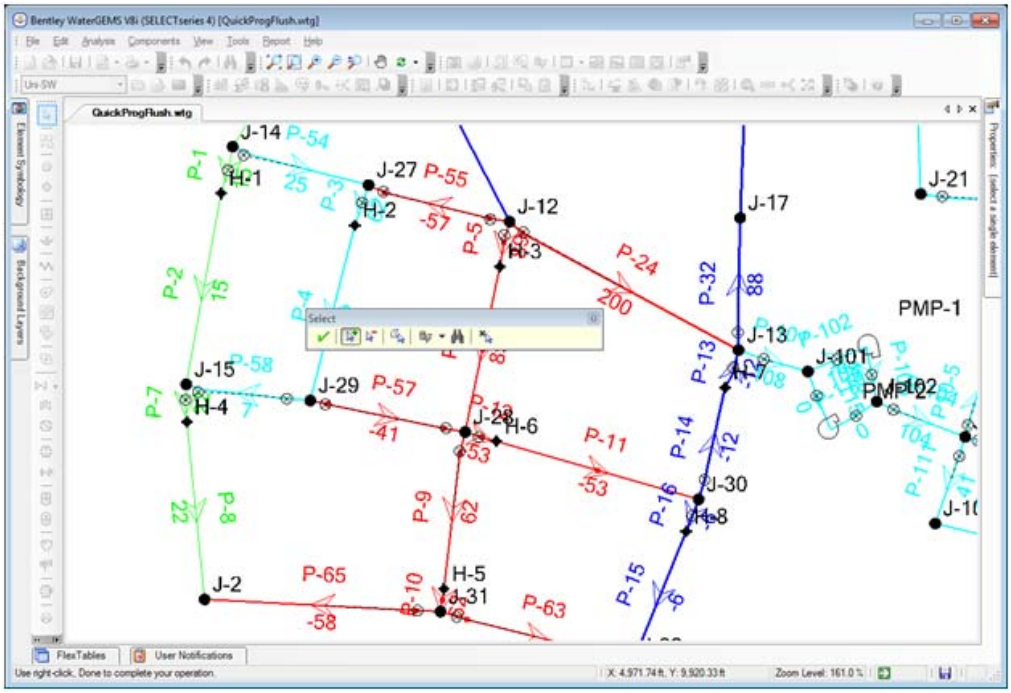
	ID	Label	Length (ft)	Diameter (in)	Flushing Event	Velocity (Maximum Flushing) (ft/s)	Satisfies Flushing Target Velocity?	Shear Stress (Maximum Flushing) (lbs/ft²)	Satisfies Flushing Target Shear Stress?
320: P-7	320	P-7	170	4.0	Event [H-4]	13.58	☑	1.70	☑
311: P-1	311	P-1	205	4.0	Event [H-1]	12.76	☑	1.52	☑
327: P-12	327	P-12	82	6.0	Event [H-6]	7.21	☑	0.25	☑
314: P-3	314	P-3	94	6.0	Event [H-2]	6.92	☑	0.46	☑
321: P-8	321	P-8	805	4.0	Event [H-4]	6.00	☑	0.38	☑
323: P-9	323	P-9	413	6.0	Event [H-5]	5.95	☑	0.17	☑
312: P-2	312	P-2	1,010	4.0	Event [H-1]	5.77	☑	0.35	☑
333: P-16	333	P-16	87	8.0	Event [H-8]	5.58	☑	0.15	☑
127: P-55	127	P-55	370	6.0	Event [H-2]	5.50	☑	0.30	☑
329: P-13	329	P-13	88	8.0	Event [H-7]	5.25	☑	0.13	☑
96: P-63	96	P-63	500	6.0	Event [H-5]	5.04	☑	0.13	☑
324: P-10	324	P-10	47	6.0	Event [H-5]	5.04	☑	0.13	☑
101: P-58	101	P-58	320	6.0	Event [H-4]	5.02	☑	0.25	☑
330: P-14	330	P-14	302	8.0	Event [H-8]	4.82	☑	0.11	☑
95: P-54	95	P-54	360	6.0	Event [H-1]	4.38	☑	0.20	☑
326: P-11	326	P-11	528	6.0	Event [H-6]	4.36	☑	0.10	☑
315: P-4	315	P-4	466	6.0	Event [H-2]	3.88	☑	0.16	☑
91: P-57	91	P-57	400	6.0	Event [H-2]	3.78	☑	0.15	☑
90: P-65	90	P-65	680	6.0	Event [H-4]	3.57	☑	0.13	☑
318: P-6	318	P-6	439	6.0	Event [H-6]	3.35	☑	0.12	☑
94: P-26	94	P-26	1,185	4.0	Event [H-1]	2.91	☑	0.10	☑
332: P-15	332	P-15	353	8.0	Event [H-5]	2.90	☑	0.04	☑
109: P-24	109	P-24	1,195	8.0	Event [H-7]	2.25	☑	0.03	☑
86: P-67	86	P-67	570	6.0	(N/A)	(N/A)	☐	(N/A)	☐

12. Close the Flex Table.

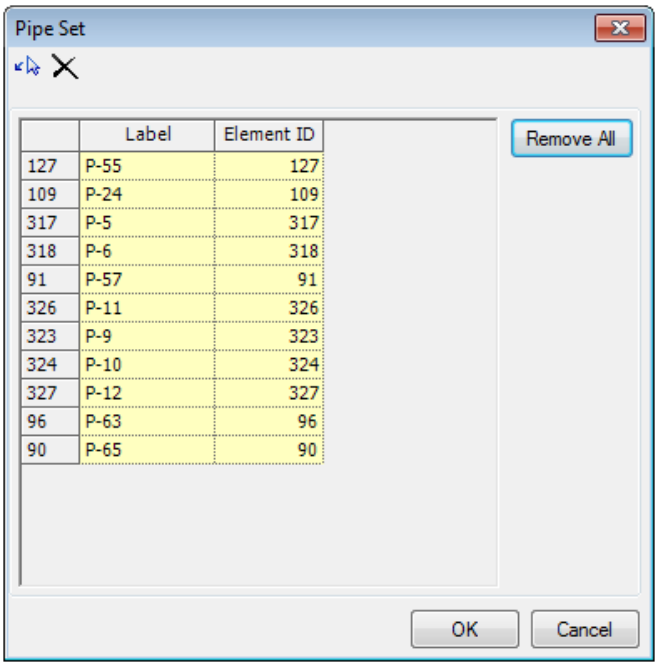
13. Next you will set up a unidirectional flushing event to increase the velocity in a run of pipes along the southwest edge of the system. Highlight Flushing study in the left pane, pick new (first button on top), pick New Unidirectional Event. Highlight "Area1", pick Rename and call it "Uni-SW".



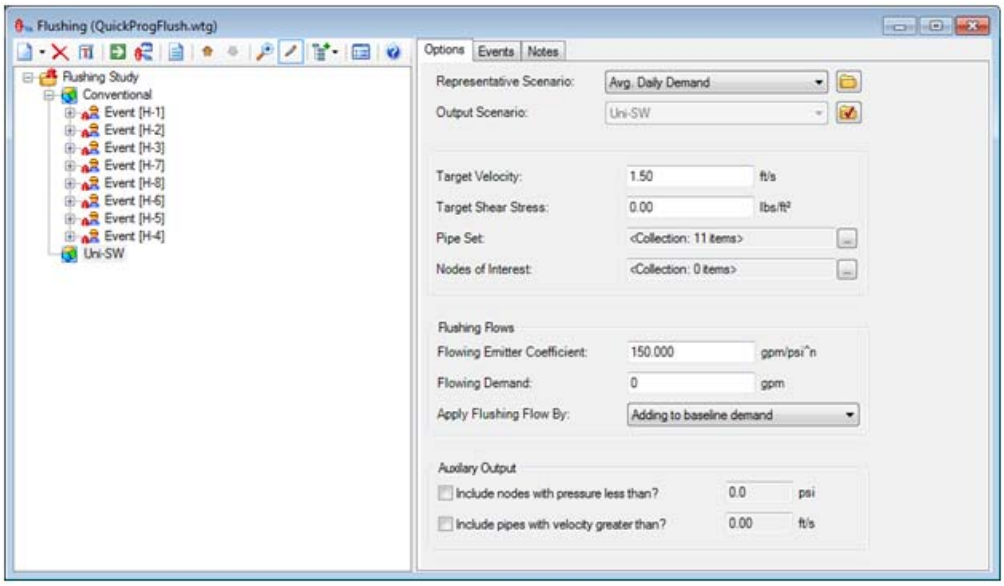
14. Pick Avg. Day Demand as the Representative Scenario, set the Emitter coefficient to 150 and create the Pipe Set as shown.



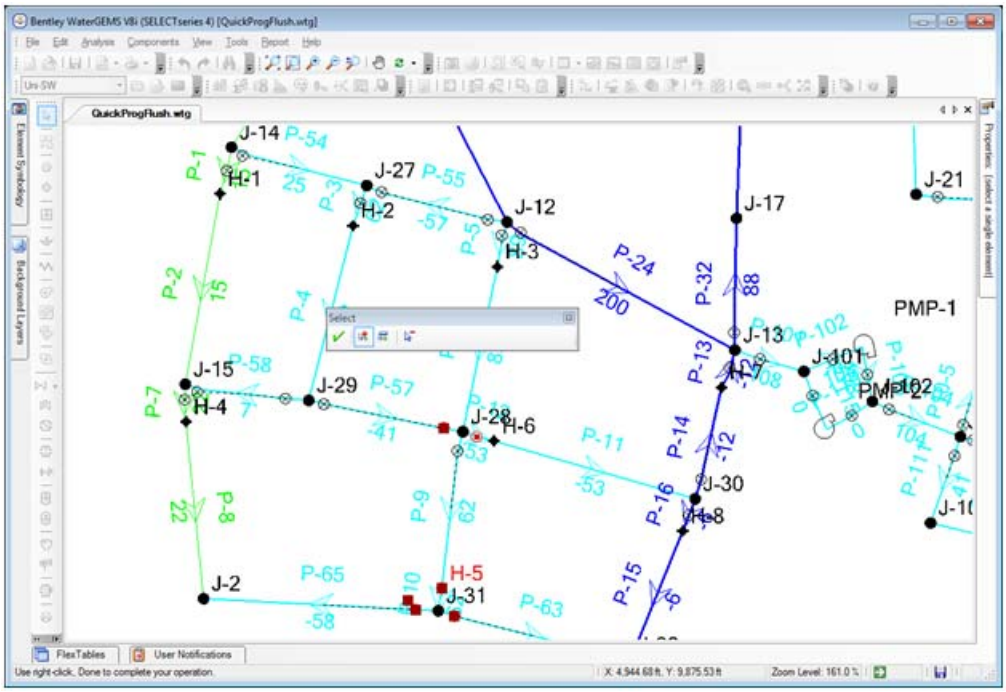
15. Pick the check and view the Pipe set.



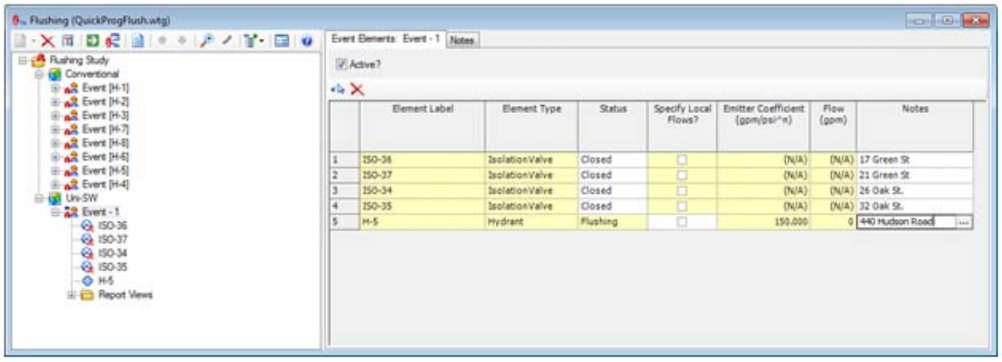
16. Click OK.



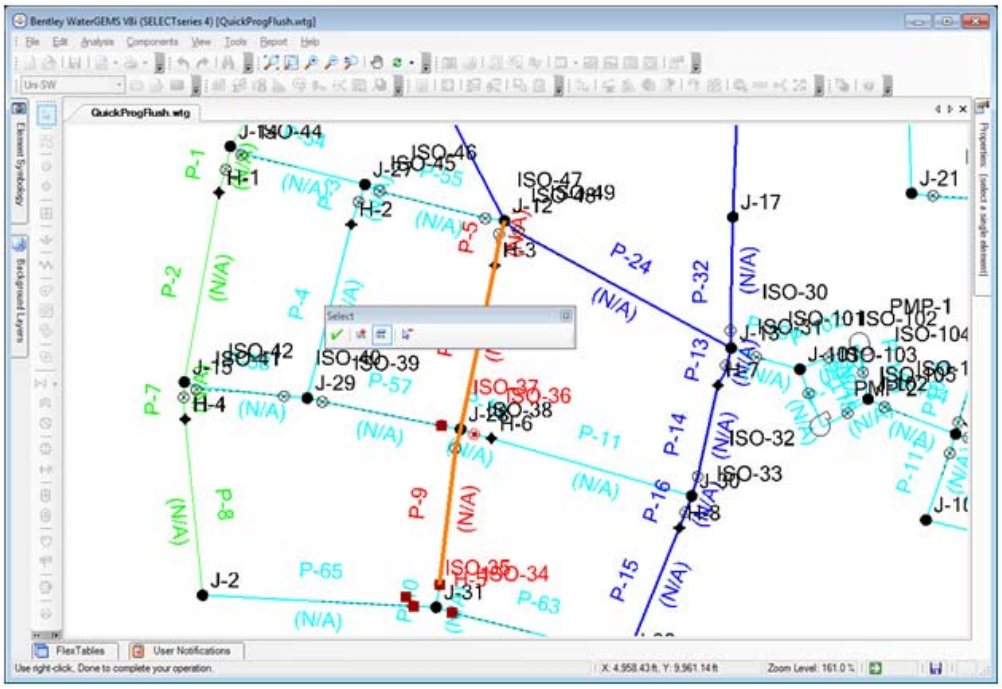
17. Select the Events tab to create a unidirectional event by picking New Unidirectional Event from the top left pane. Then pick the 4 valves shown below to close and hydrant H-5 to flow as shown below.



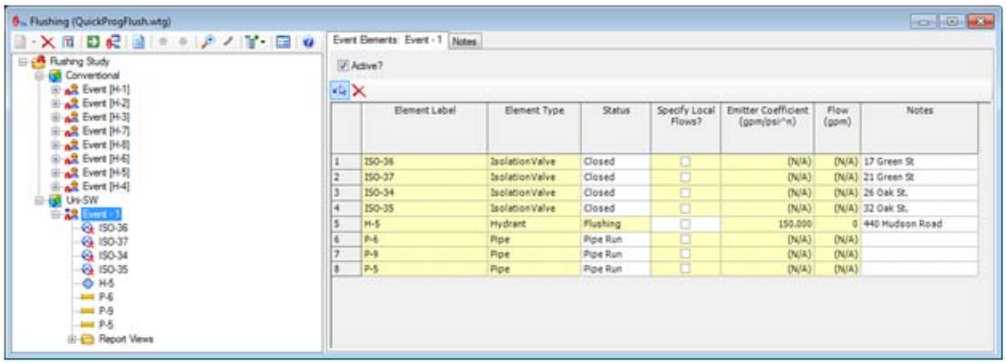
18. After picking the elements and picking the check mark, review the list of elements to be operated. Feel free to add some descriptive notes to the elements to be operated.



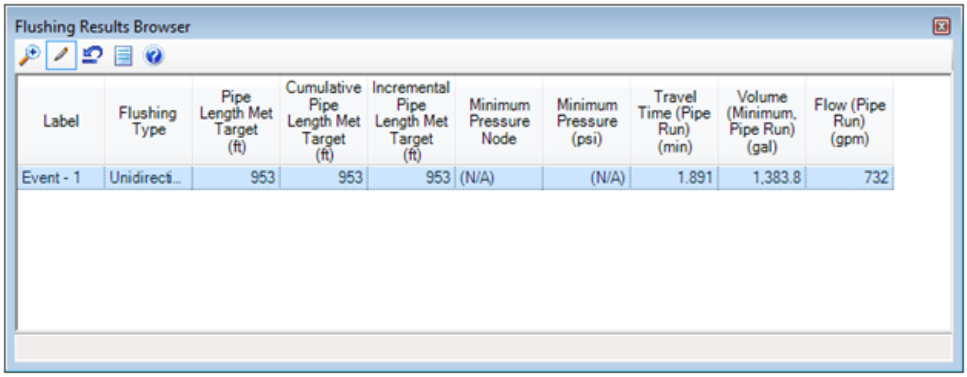
19. Identify the pipes to be part of the Pipe Run, by picking the Select from Drawing button on the right pane. Highlight the third button in the Select dialog (the second button closes pipes). Once again, it may be desirable to use the mouse wheel to zoom in to the shorter pipes.



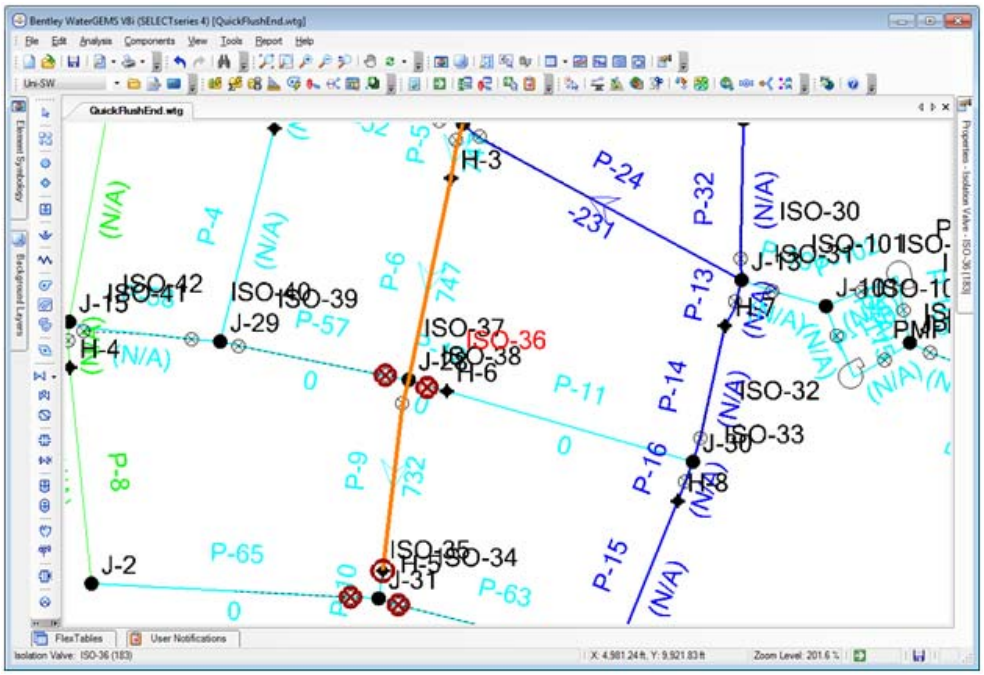
20. With all of the elements identified, select the Compute button (fourth from left in left pane).



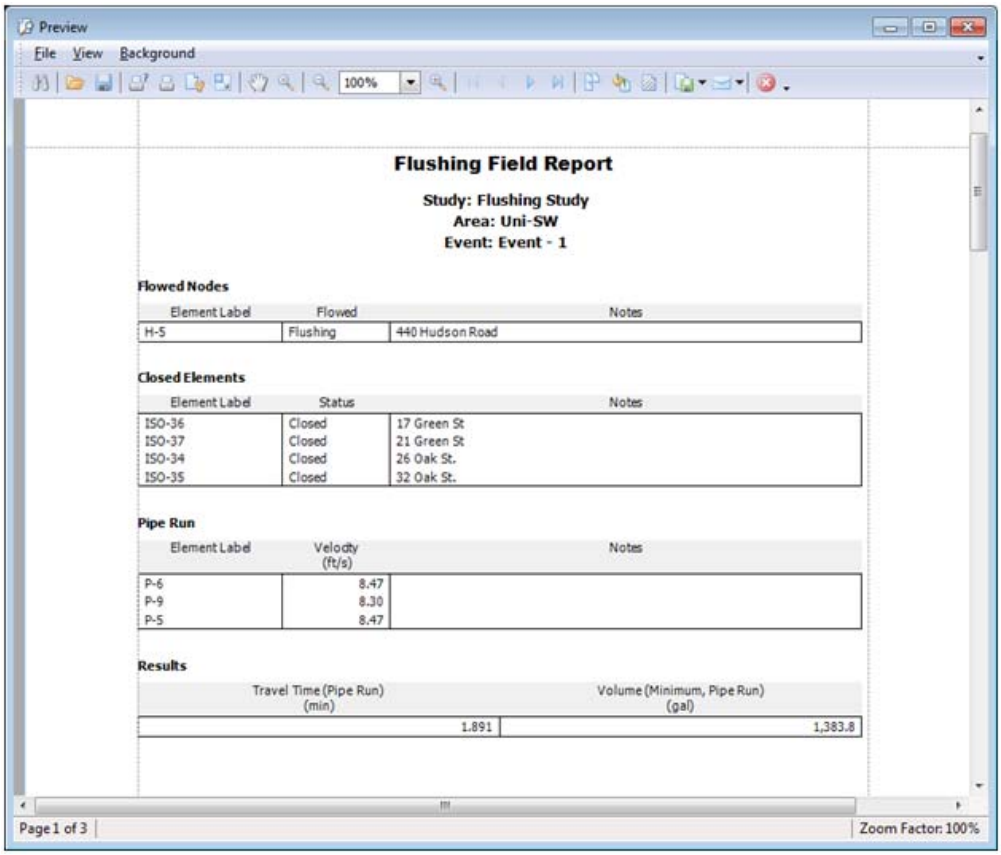
21. Once the run is complete, open the Flushing Browser (fifth button on left pane), and view the results.

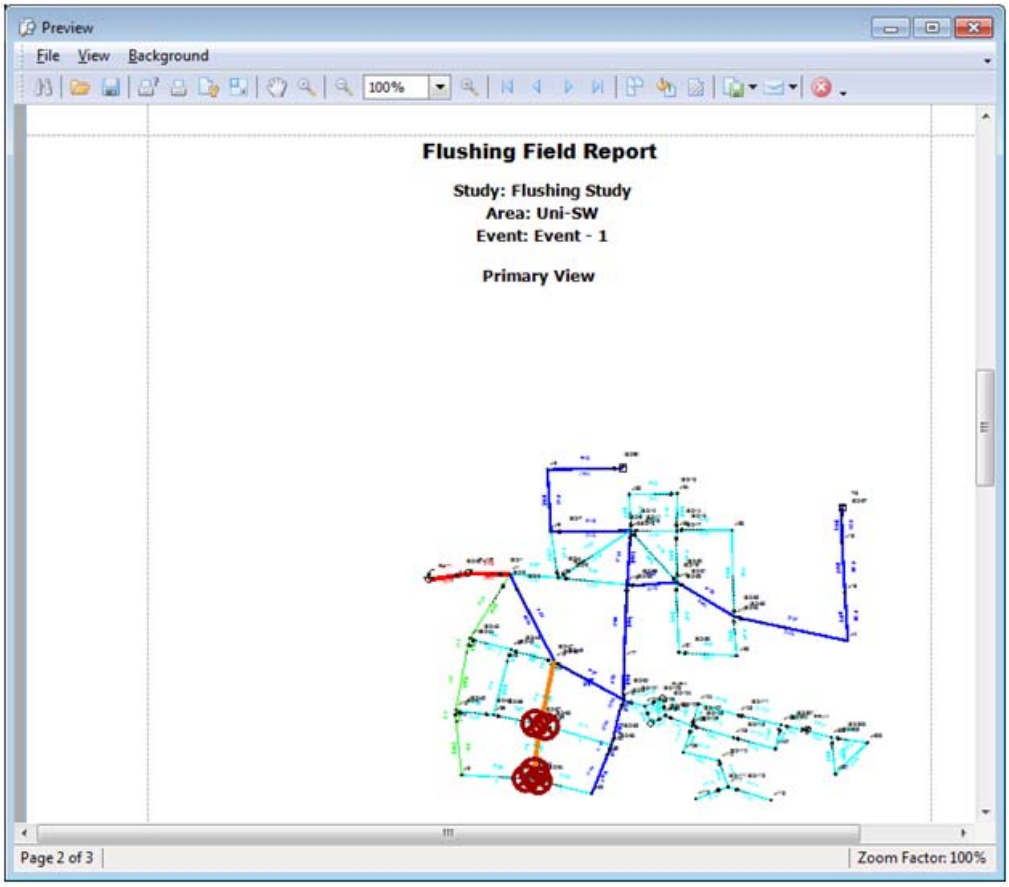


22. Click the highlight button (second from left) and view the event in the drawing.



23. Click on the Operator Report (sixth button on left pane).





24. Close the report to get back to WaterGEMS.

Understanding the Workspace

3

Stand-Alone

MicroStation Environment

Working in AutoCAD

Google Earth Export

Stand-Alone

The Stand-Alone Editor is the workspace that contains the various managers, toolbars, and menus, along with the drawing pane, that make up the Bentley WaterCAD V8*i* interface. The Bentley WaterCAD V8*i* interface uses dockable windows and toolbars, so the position of the various interface elements can be manually adjusted to suit your preference.

The Drawing View

You change the drawing view of your model by using the pan tool or one of the zoom tools:

Panning

Zooming

Drawing Style

Panning

You can change the position of your model in the drawing pane by using the Pan tool.



To use the Pan tool

1. Click the **Pan** button on the Zoom toolbar.

   The mouse cursor changes to the Pan icon.

2. Click anywhere in the drawing, hold down the mouse button and move the mouse to reposition the current view.

   or

   If your mouse is equipped with a mousewheel, you can pan by simply holding down the mousewheel and moving the mouse to reposition the current view.

   or

   Select **View > Pan**, then click anywhere in the drawing, hold down the mouse button and move the mouse to reposition the current view

Zooming

You can enlarge or reduce your model in the drawing pane using one of the following zoom tools:



The current zoom level is displayed in the lower right hand corner of the interface, next to the coordinate display.

Zoom Extents



The Zoom Extents command automatically sets the zoom level such that the entire model is displayed in the drawing pane.

To use Zoom Extents, click Zoom Extents on the Zoom toolbar. The entire model is displayed in the drawing pane.

or

Select **View > Zoom > Zoom Extents**.

Zoom Window



The Zoom Window command is used to zoom in on an area of your model defined by a window that you draw in the drawing pane.

To use Zoom Window, click the Zoom Window button on the Zoom toolbar, then click and drag the mouse inside the drawing pane to draw a rectangle. The area of your model inside the rectangle will appear enlarged.

or

Select **View > Zoom > Zoom Window**, then draw the zoom window in the drawing pane.

Zoom In and Out



The Zoom In and Zoom Out commands allow you to increase or decrease, respectively, the zoom level of the current view by one step per mouse click.

To use Zoom In or Zoom Out, click either one on the Zoom toolbar, or select **View > Zoom > Zoom In** or **View > Zoom > Zoom In**.

If your mouse is equipped with a mousewheel, you zoom in or out by simply moving the mousewheel up or down respectively.

Zoom Realtime



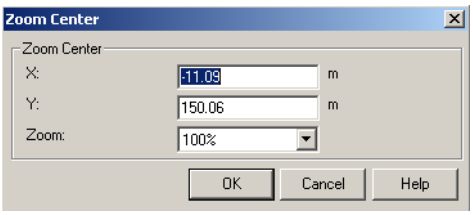
The Zoom Realtime command is used to dynamically scale up and down the zoom level. The zoom level is defined by the magnitude of mouse movement while the tool is active.

Zoom Center



The Zoom Center command is used to enter drawing coordinates that will be centered in the drawing pane.

1. Choose View > Zoom > Zoom Center or click the Zoom Center icon on the Zoom toolbar.. The Zoom Center dialog box opens.



2. The Zoom Center dialog box contains the following:

X	Defines the X coordinate of the point at which the drawing view will be centered.
Y	Defines the Y coordinate of the point at which the drawing view will be centered.
Zoom	Defines the zoom level that will be applied when the zoom center command is initiated. Available zoom levels are listed in percentages of 25, 50, 75, 100, 125, 150, 200 and 400.

3. Enter the X and Y coordinates.
4. Select the percentage of zoom from the Zoom drop-down menu.
5. Click OK.

Zoom to Selection



Enables you to zoom to specific elements in the drawing. You must select the elements to zoom to before you select the tool.

Zoom Previous and Zoom Next

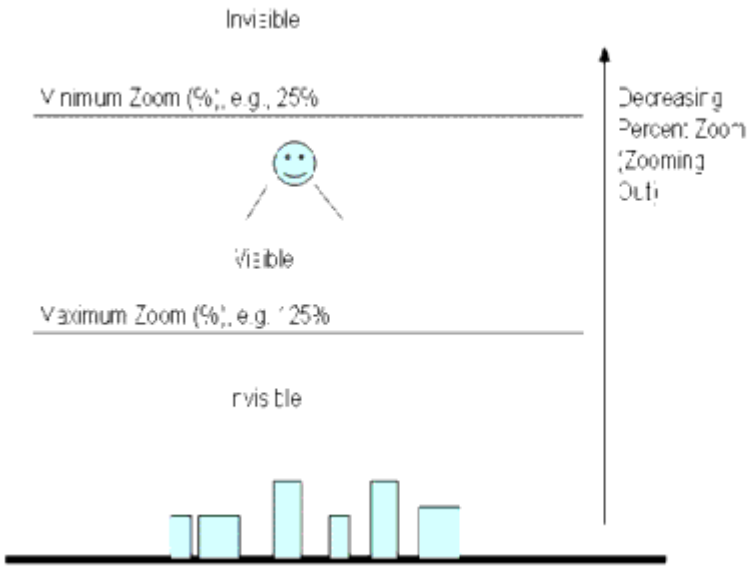


Zoom Previous returns the zoom level to the most recent previous setting. To use Zoom Previous, click View > Zoom > Zoom Previous or click the Zoom Previous icon from the Zoom toolbar.

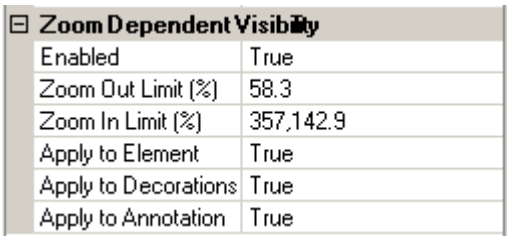
Zoom Next returns the zoom level to the setting that was active before a Zoom Previous command was executed. To use Zoom Previous, click View > Zoom > Zoom Next or click the Zoom Next icon from the Zoom toolbar.

Zoom Dependent Visibility

Available through the Properties dialog box of each layer in the Element Symbology manager, the Zoom Dependent Visibility feature can be used to cause elements, decorations, and annotations to only appear in the drawing pane when the view is within the zoom range specified by the Minimum and Maximum Zoom values.



By default, Zoom Dependent Visibility is turned off. To turn on Zoom Dependent Visibility, highlight a layer in the Element Symbology Manager. In the Properties window, change the Enabled value under Zoom Dependent Visibility to True. The following settings will then be available:



Zoom Dependent Visibility	
Enabled	True
Zoom Out Limit (%)	58.3
Zoom In Limit (%)	357,142.9
Apply to Element	True
Apply to Decorations	True
Apply to Annotation	True

Enabled Set to true to enable and set to false to disable Zoom Dependent Visibility.

Zoom Out Limit (%) The minimum zoom level, as a percent of the default zoom level used when creating the project, at which objects on the layer will appear in the drawing. The current zoom level is displayed in the lower right hand corner of the interface, next to the coordinate display. You can also set the current zoom level as the minimum by right-clicking a layer in the Element Symbology manager and selecting the Set Minimum Zoom command. The zoom out limit is especially important in GIS style symbology because the symbols and text can become very large. (As you zoom out, the Zoom Level as a percent decreases. Once it drops below the zoom out limit, the objects will no longer appear.)

Zoom In Limit (%) The maximum zoom level, as a percent of the default zoom level used when creating the project, at which objects on the layer will appear in the drawing. The current zoom level is displayed in the lower right hand corner of the interface, next to the coordinate display. You can also set the current zoom level as the maximum by right-clicking a layer in the Element Symbology manager and selecting the Set Maximum Zoom command. The zoom in limit is especially important in CAD style symbology because the symbols and text can become very large. (As you zoom in, the Zoom Level as a percent increases. Once it exceeds the zoom in limit, the objects no longer appear.)

Apply to Element Set to true to apply the zoom minimums and maximums to the symbols in the drawing.

Apply to Decorations	Set to true to apply the zoom minimums and maximums to flow arrows, check valves, and constituent sources in the drawing.
Apply to Annotations	Set to true to apply the zoom minimums and maximums to labels in the drawing.

The numerical value for zoom out limit should be smaller than zoom in limit or else the element will not be visible at all.

The current zoom level is displayed at the bottom right of the drawing.

Drawing Style

Elements can be displayed in one of two styles in the Stand-Alone version; GIS style or CAD style.

Under GIS style, the size of element symbols in the drawing pane will remain the same (relative to the screen) regardless of zoom level. Under CAD style, element symbols will appear larger or smaller (relative to the drawing) depending on zoom level.

There is a default Drawing Style that is set on the Global tab of the Options dialog. The drawing style chosen there will be used by all elements by default. Changing the default drawing style will only affect new projects, not existing ones.

You can change the drawing style used by all of the elements in the project, or you can set each element individually to use either drawing style.

To change a single element's drawing style

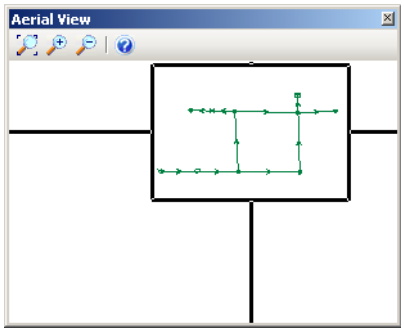
1. Double-click the element in the Element Symbology manager dialog to open the Properties manager.
2. In the Properties manager, change the value in the Display Style field to the desired setting.

To change the drawing style of all elements

Click the Drawing Style button in the Element Symbology manager and select the desired drawing style from the submenu that appears.

Using Aerial View

The Aerial View is a small navigation window that provides a graphical overview of your entire drawing. You can toggle the Aerial View window on or off by selecting View > Aerial View to open the Aerial View window.



A Navigation Rectangle is displayed in the Aerial View window. This Navigation Rectangle provides a you-are-here indicator showing you current zoom location respective of the overall drawing. As you pan and zoom around the drawing, the Navigation Rectangle will automatically update to reflect your current location.

You can also use the Aerial View window to navigate around your drawing. To pan, click the Navigation Rectangle to drag it to a new location. To zoom, click anywhere in the window to specify the first corner of the Navigation Rectangle, and click again to specify the second corner.

In the AutoCAD environment, see the AutoCAD online help for a detailed explanation.

In Stand-Alone environment, with Aerial View window enabled (by selecting the View > Aerial View), click and drag to draw a rectangular view box in the aerial view. The area inside this view box is displayed in the main drawing window. Alternately, any zooming or panning action performed directly in the main window updates the size and location of the view box in the Aerial View window.

The Aerial View window contains the following buttons:

Zoom Extents—Display the entire drawing in the Aerial View window.

Zoom In—Decrease the area displayed in the Aerial View window.

Zoom Out—Increase the area displayed in the Aerial View window.

Help—Opens the online help.

To resize the view box directly from the Aerial View window, click to define the new rectangular view box. To change the location of the view box, hover the mouse cursor over the current view rectangle and click to drag the view box frame to a new location.

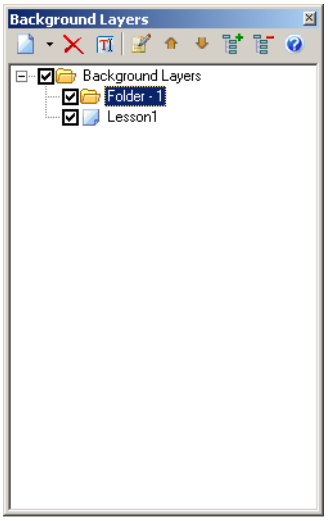
Using Background Layers

Use background layers to display pictures behind your network in order to relate elements in your network to structures and roads depicted in the picture. You can add, delete, edit and rename background layers in the Background Layers Manager. The Background Layers manager is only available in the Stand-Alone version of WaterCAD V8*i*. The MicroStation, ArcGIS, and AutoCAD versions each provide varying degrees of native support for inserting raster and vector files.

You can add multiple pictures to your project for use as background layers, and turn them off and on. Additionally, you can create groups of pictures in folders, so you can hide or show an entire folder or group of pictures at once.

When adding a background layer, it is possible to cause an "out of memory" error if the file is too large. This depends on the size of the background file and the computer. If this type of error occurs, the best solution is to reduce the size of the background file using GIS or CAD tools (e.g. Bentley's Raster manager). It is usually possible to trim or reduce the resolution of the backround without affecting its usefulness. In some instances, it may be possible to run Bentley WaterCAD V8*i* in a CAD or GIS platform which is better able to handle these very large background files.

To add or delete background layers, open the Background Layers manager choose View > Background Layers.



You can use shapefiles, AutoCAD DXF files, and raster (also called bitmap) pictures as background images for your model. The following raster image formats are supported: bmp, jpg, jpeg, jpe, jfif, gif, tif, tiff, png, and sid.

Note: MrSID background files are not supported in x64 version.

Using the Background Layer manager you can add, edit, delete, and manage the background layers that are associated with the project. The dialog box contains a list pane that displays each of the layers currently contained within the project, along with a number of button controls.

When a background layer is added, it opens in the Background Layers list pane, along with an associated check box that is used to control that layer's visibility. Selecting the check box next to a layer causes that layer to become visible in the main drawing pane; clearing it causes it to become invisible. If the layers in the list pane are contained within one or more folders, clearing the check box next to a folder causes all of the layers within that folder to become invisible.

Note: When multiple background layers are overlaid, priority is given to the first one on the list.

You can copy/paste background layers and folders by right-clicking them and selecting Copy/Paste. When a folder is copied in this way all of the contents of that folder are also copied.

The toolbar consists of the following buttons:

New 

Opens a menu containing the following commands:

- **New File**—Opens a Select Background dialog box where you can choose the file to use as a background layer.
- **New Folder**—Creates a folder in the Background Layers list pane.

Delete 

Removes the currently selected background layer.

Rename 

Rrenames the currently selected layer.

Edit 

Opens a Properties dialog box that corresponds with the selected background layer.

Shift Up 

Moves the currently highlighted object up in the list pane.

Shift Down 	Moves the currently highlighted object down in the list pane.
Expand All 	Expands all of the branches in the hierarchy displayed in the list pane.
Collapse All 	Collapses all of the branches in the hierarchy displayed in the list pane.
Help 	Displays online help for the Background Layer Manager.

To add a background layer folder

You can create folders in Background Layers to organize your background layers and create a group of background layers that can be turned off together. You can also create folders within folders. When you start a new project, an empty folder is displayed in the Background Layers manager called Background Layers. New background layer files and folders are added to the Background Layers folder by default.

1. Choose **View > Background Layers** to open the Background Layers manager.
2. In the Background Layers manager, click the **New** button, then click **New Folder** from the shortcut menu.

   Or select the default Background Layers folder, then right-click and select **New > Folder** from the shortcut menu.

   – If you are creating a new folder within an existing folder, select the folder, then click **New > New Folder**. Or right-click, then select **New > Folder** from the shortcut menu.
3. Right-click the new folder and select **Rename** from the shortcut menu.
4. Type the name of the folder, then press <**Enter**>.

To delete a background layer folder

1. Click **View > Background Layers** to open the Background Layers manager.
2. In the Background Layers managers, select the folder you want to delete, then click the **Delete** button.
   - You can also right-click a folder to delete, then select **Delete** from the shortcut menu.

To rename a background layer folder

1. Click **View > Background Layers** to open the Background Layers manager.
2. In the Background Layers managers, select the folder you want to rename, then click the **Rename** button.
   - You can also right-click a folder to rename, then select **Rename** from the shortcut menu.
3. Type the new name of the folder, then press <**Enter**>.
   - You can also rename a background layer folder by selecting the folder, then modifying its label in the Properties Editor.

To add a background layer

In order to add background layers to projects use the Background Layers manager. When you start a new project, an empty folder in the Background Layers manager called Background Layers is displayed. New background layer files and folders are added to the Background Layers folder by default.

1. Click **View > Background Layers** to open the Background Layers manager.
2. In the Background Layers managers, click the **New** button, then click **New File** from the shortcut menu.

   Or right-click on the default Background Layers folder and select **New > File** from the shortcut menu.
   - To add a new background layer file to an existing folder in the Background Layer manager, select the folder, then click **New > New File**. Or right-click, then select **New > File** from the shortcut menu.
3. Navigate to the file you want to add as a background layer and select it.
   - If you select a .dxf file, the DXF Properties dialog box opens.

- If you select a .shp the ShapeFile Properties dialog box opens.
- If you select a .bmp, .jpg, .jpeg, .jpe, .jfif, .gif, .tif, .tiff, .png, or .sid file, the Image Properties dialog box opens.

4. After you add the background layer, you might have to use the Pan button to move the layer within the drawing area; Zoom Extents does not center a background image.

To copy a background layer

1. Right click on the background layer you wish to copy.
2. Right click on the folder you want the background layer copied to and click Paste.

You can also copy an entire folder; the contents of the folder will also be copied.

To delete a background layer

- Select the background layer you want to delete, then click the **Delete** button.
- Or, right-click the background layer, then select **Delete** from the shortcut menu.

To edit the properties of a background layer

You can edit a background layer in two ways: you can edit its properties or its position in a list of background layers displayed in the Background Layers manager.

1. Select the background layer you want to edit.
2. Click the **Edit** button. A Properties dialog box opens.
   - You can also right-click the background layer, then select **Edit** from the shortcut menu.

To change the position of a background layer in the list of background layers

The order of a background layer determines its Z level and what displays if you use more than one background layer. Background layers at the top of the list display on top of the other background layers in the drawing pane; so, background layers that are lower than the top one in the list might be hidden or partially hidden by layers above them in the list.

Select the background layer whose position you want to change in the list of Background Layers manager, then click the **Shift Up** or **Shift Down** buttons to move the selected background layer up or down in the list.

To rename a background layer

Select the background layer you want to rename, then click the **Rename** button.

Or, right-click the background layer that you want to rename, then select **Rename** from the shortcut menu.

Turn background layers on or off

Turn your background layers on or off by using the check box next to the background layer file or folder than contains it in the Background Layers manager.

Image Properties

This dialog box opens when you are adding or editing a background-layer image other than a .dxf or .shp.

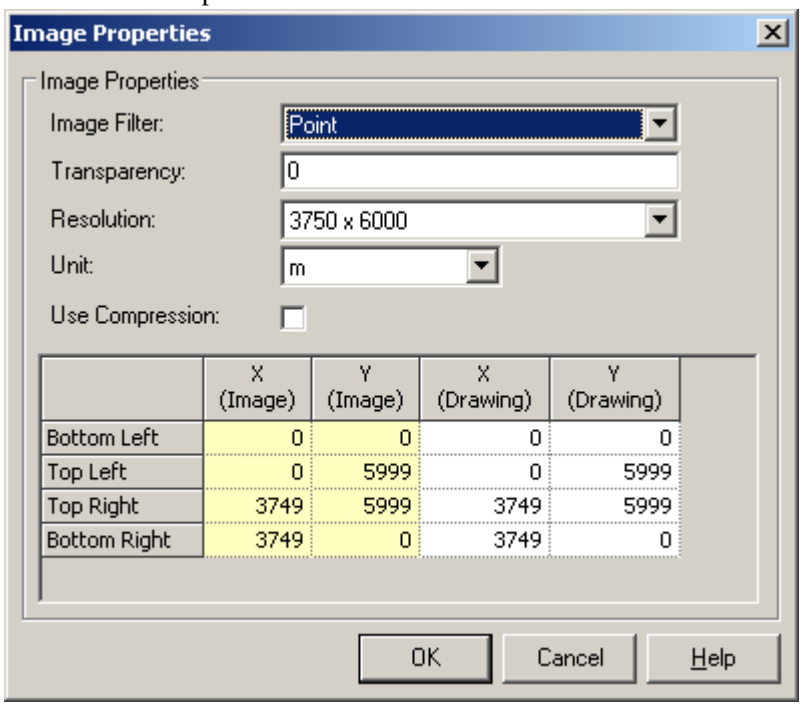


Image Filter

Displays background images that you resize. Set this to **Point**, **Bilinear**, or **Trilinear**. These are methods of displaying your image on-screen.

- Use **Point** when the size of the image in the display, for example,a 500 x 500 pixel image at 100% is the same 500 x 500 pixels on-screen.
- Use **Bilinear** or **Trilinear** when you display your image on-screen using more or fewer pixels than your image contains, for example a 500 x 500 pixel image stretched to 800 x 800 pixels on-screen. Trilinear gives you smoother transitions when you zoom in and out of the image.

Transparency Set the transparency level of the background layer. You can add transparency to any image type you use as a background and it will ignore any transparency that exists in the image before you use it as a background.

Resolution Select the clarity for images that are being used as background images.

Unit Select the unit that should be used.

Use Compression If you check this option you can compress the image in memory so that it takes up less RAM. When checked there may be a slight color distortion in the image.

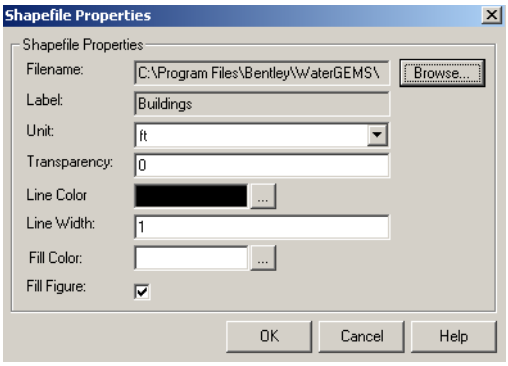
> **Note: The way the image is compressed depends on your computer's video card. Not all video cards support this feature. If you check this option but your computer's video card does not support image compression, the request for compression will be ignored and the image will be loaded uncompressed.**

Image Position Table Position the background layer with respect to your drawing.

- X/Y Image displays the size of the image you are using for a background and sets its position with respect to the origin of your drawing. You cannot change this data.
- X/Y Drawing displays where the corners of the image your are using will be positioned relative to your drawing. By default, no scaling is used. However, you can scale the image you are using by setting different locations for the corners of the image you are importing. The locations you set are relative to the origin of your Bentley WaterCAD V8*i* drawing.

Shapefile Properties

Use the Shapefile Properties dialog box to define a shapefile background layer. In order to access the Shapefile Properties dialog box, click **New File** in the Background Layers manager, then select a .shp file.

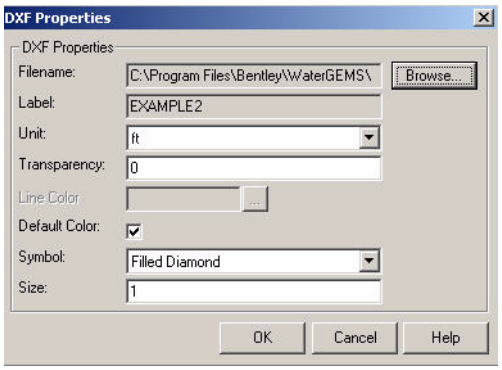


Use the following controls to define the properties of the background layer:

Filename	Lists the path and filename of the shapefile to use as a background layer.
Browse	Opens a browse dialog box, to select the file to be used as a background layer.
Label	Identifies the background layer.
Unit	Select the unit of measurement associated with the spatial data from the menu.
Transparency	Specify the transparency level of the background layer, where 0 has the least and 100 has the most transparency.
Line Color	Sets the color of the layer elements. Click the Ellipsis (...) button to open a Color palette containing more color choices.
Line Width	Sets the thickness of the outline of the layer elements.
Fill Color	Select the fill color.
Fill Figure	Check to fill.

DXF Properties

The DXF Properties dialog box is where you define a .dxf file as the background layer. In order to open the .dxf properties, click **New File** In the Background Layers manager, then select a .dxf file.



Use the following controls to define the properties of the background layer:

Filename	Lists the path and filename of the .dxf file to use as a background layer.
Browse	Click to open a dialog box to select the file to be used as a background layer.
Label	Identifies the background layer.
Unit	Select the unit associated with the spatial data within the shapefile, for example, if the X and Y coordinates of the shapefile represent feet, select ft from the menu.
Transparency	Specify the transparency level of the background layer, where 0 has the least transparency and 100 has the most.
Line Color	Sets the color of the layer elements. Click the Ellipsis (...) button to open a Color palette containing more color choices. Only when Default Color is not selected.
Default Color	Use the default line color included in the .dxf file or select a custom color in the Line Color field by unchecking the box.

Symbol	Choose the symbol that is displayed for each point element in the .dxf.
Size	Sets the size of the symbol for each point element in the .dxf.

Show Flow Arrows (Stand-Alone)

In the Stand-Alone client flow arrows are automatically displayed after a model has been calculated (by default). You can also toggle the display of flow arrows on/off using the Show Flow Arrows control in the Properties dialog when Pipe is highlighted in the Element Symbology manager (see Annotating Your Model).

MicroStation Environment

In the MicroStation environment you can create and model your network directly within your primary drafting environment. This gives you access to all of MicroStation's powerful drafting and presentation tools, while still enabling you to perform Bentley WaterCAD V8*i* modeling tasks like editing, solving, and data management. This relationship between Bentley WaterCAD V8*i* and MicroStation enables extremely detailed and accurate mapping of model features, and provides the full array of output and presentation features available in MicroStation. This facility provides the most flexibility and the highest degree of compatibility with other CAD-based applications and drawing data maintained at your organization.

Bentley WaterCAD V8*i* features support for MicroStation integration. You run Bentley WaterCAD V8*i* in both MicroStation and stand-alone environment.

The MicroStation functionality has been implemented in a way that is the same as the Bentley WaterCAD V8*i* base product. Once you become familiar with the stand-alone environment, you will not have any difficulty using the product in the MicroStation environment.

In the MicroStation environment, you will have access to the full range of functionality available in the MicroStation design and drafting environment. The standard environment is extended and enhanced by using MicroStation's MDL (MicroStation Development Language) client layer that lets you create, view, and edit the native Bentley WaterCAD V8*i* network model while in MicroStation.

MDL is a complete development environment that lets applications take full advantage of the power of MicroStation and MicroStation-based vertical applications. MDL can be used to develop simple utilities, customized commands or sophisticated commercial applications for vertical markets.

Some of the advantages of working in the MicroStation environment include:

- Lay out network links and structures in fully-scaled environment in the same design and drafting environment that you use to develop your engineering plans.
- Have access to any other third party applications that you currently use, along with any custom MDL applications.
- Use native MicroStation insertion snaps to precisely position Bentley WaterCAD V8*i* elements with respect to other entities in the MicroStation drawing.
- Use native MicroStation commands on Bentley WaterCAD V8*i* model entities with automatic update and synchronization with the model database.
- Control destination levels for model elements and associated label text and annotation, giving you control over styles, line types, and visibility of model elements.

Note: Bentley MicroStation V8i is the only MicroStation environment supported by WaterCAD V8*i*.

Additional features of the MicroStation version includes:

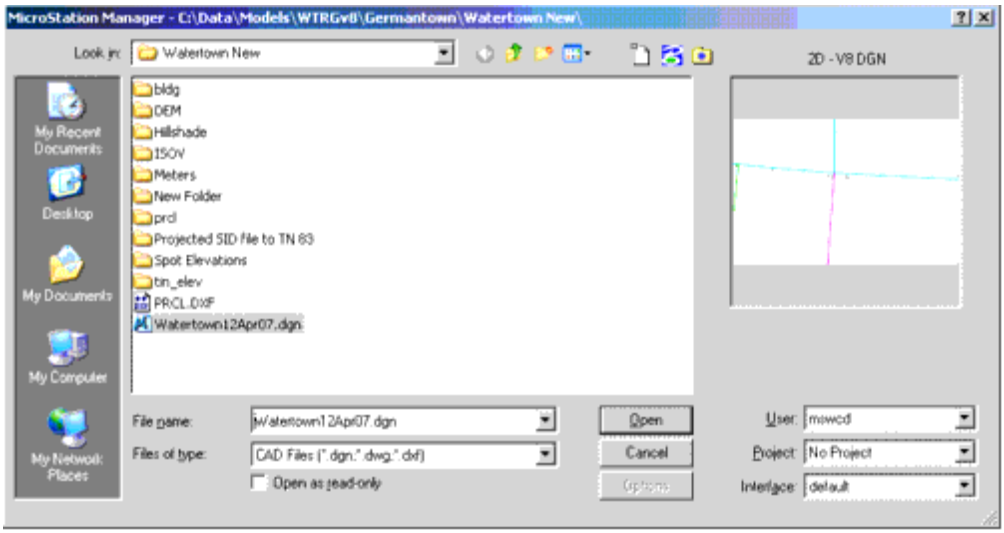
- MicroStation Project Files on page 3-231
- Bentley WaterCAD V8i Element Properties on page 3-232
- Working with Elements on page 3-234
- MicroStation Commands on page 3-236
- Import Bentley WaterCAD V8i on page 3-237

Getting Started in the MicroStation environment

A Bentley MicroStation WaterCAD V8*i* project consists of:

- **Drawing File (.DGN)**—The MicroStation drawing file contains the elements that define the model, in addition to the planimetric base drawing information that serves as the model background.
- **Model File (.**wtg**)**—The model file contains model data specific to WaterCAD V8*i*, including project option settings, color-coding and annotation settings, etc. Note that the MicroStation .dgn that is associated with a particular model may not necessarily have the same filename as the model's .wtg file.
- **Database File (.sqlite)**—The model database file that contains all of the input and output data for the model. Note that the MicroStation .dgn that is associated with a particular model may not have the same filename as the model's .sqlite file.

When you start Bentley WaterCAD V8*i* for MicroStation, you will see the dialog below. You must identify a new or existing MicroStation dgn drawing file to be associated with the model before you can open a Bentley WaterCAD V8*i* model.



Either browse to an existing dgn file or create a new file using the new button on the top toolbar. Once you have selected a file, you can pick the Open button.

Once a drawing is open, you can use the WaterCAD V8*i* Project drop down menu to create a new WaterCAD V8*i* project, attach an existing project, or import a project.

There are a number of options for creating a model in the MicroStation client:

- **Create a model from scratch**—You can create a model in MicroStation. You'll first need to create a new MicroStation .dgn (refer to your MicroStation documentation to learn how to create a new .dgn). Start WaterCAD V8*i* for MicroStation. In the first dialog, pick the New button and assign a name and path to the DGN file. Once the dgn is open, use the New command in the WaterCAD V8*i* Project menu (Project > New). This will create a new WaterCAD V8*i* project file and attach it to the Bentley MicroStation .dgn file. Once the file is created you can start creating WaterCAD V8*i* elements that exist in both the WaterCAD V8*i* database and in the .dgn drawing. See Working with Elements and Working with Elements Using MicroStation Commands for more details.
- **Open a previously created** WaterCAD V8*i* **project**—You can open a previously created WaterCAD V8*i* model and attach it to a .dgn file. To do this, start WaterCAD V8*i* for MicroStation. Open or create a new MicroStation .dgn file (refer to your MicroStation documentation to learn how to create a new .dgn). Use the Project menu on the WaterCAD V8*i* toolbar and click on the Project > "Attach Existing…" command, then select an existing WaterCAD V8*i*.wtg file. The model will now be attached to the .dgn file and you can edit, delete, and modify the WaterCAD V8*i* elements in the model. All MicroStation commands can be used on WaterCAD V8*i* elements.

- **Import a model that was created in another modeling application**—There are four types of files that can be imported into WaterCAD V8*i*:
  - **WaterGEMS / WaterCAD / HAMMER Database**—this can either be a HAMMER V8*i* or V8, WaterGEMS V8*i* or V3, or WaterCAD V8*i* or V7 database. The model will be processed and imported into the active MicroStation .dgn drawing. See Exporting a HAMMER v7 Model for more details.
  - **EPANET**—You can import EPANET input (.inp) files. The file will be processed and the proper elements will be created and added to the MicroStation drawing. See Importing and Exporting EPANET Files for more details.
  - **Submodel**—You can import a WaterCAD V8*i* V8 subenvironmentl into the MicroStation drawing file. See Importing and Exporting Submodel Files for more details.

Bentley Water model—You can import Bentley Water model data into your WaterCAD V8*i* model in MicroStation. See Importing a Bentley Water Model for more details.

If you want to trace the model on top of a dgn or other background file, you would load the background into the dgn first by using either File/Reference or File/Raster Manager Then you start laying out elements over top of the background.

The MicroStation Environment Graphical Layout

In the MicroStation environment, our products provide a set of extended options and functionality beyond those available in stand-alone environment. This additional functionality provides enhanced control over general application settings and options and extends the command set, giving you control over the display of model elements within MicroStation.

It is important to be aware that there are two lists of menu items when running WaterCAD V8*i* in MicroStation:

1. MicroStation menu (File Edit Element Settings …) which contains MicroStation commands. The MicroStation menu contains commands which affect the drawing.
2. WaterCAD V8*i* menu (Project Edit Analysis …) which contains WaterCAD V8*i* commands. The WaterCAD V8*i* menu contains commands which affect the hydraulic analysis.

It is important to be aware of which menu you are using.

Key differences between MicroStation and stand-alone environment include:

- Full element symbol editing functionality is available through the use of custom cells. All elements and graphical decorations (flow arrows, control indicators, etc.) are contained within a WaterCAD V8*i* .cel file.To do this open the .cel file that's in the WTRG install directory in MSTN (at the first, Open dialog), and then using the File>models you can select each of the WTRG symbols and change them using normal MSTN commands. Then when you create a new dgn and start laying out the WTRG elements, the new symbols will be used.
- The more powerful Selection tools are in the MicroStation select menu.
- Element symbols like junction are circles that are not filled. The user must pick the edge of the circle, not inside the circle to pick a junction.
- The MicroStation background color is found in Workspace>Preferences>View Options. It can also be changed in Settings>Color Tab.
- Zooming and panning are controlled by the MicroStation zooming and panning tools.
- Depending on how MicroStation was set up, a single right click will simply clear the last command, while holding down the right mouse button will bring up the context sensitive menu. There are commands in that menu (e.g. rotate) that are not available in WaterCAD V8*i* stand alone.

You can control the appearance and destination of all model elements using the Element Levels command under the View menu. For example, you can assign a specific level for all outlets, as well as assign the label and annotation text style to be applied. Element attributes are either defined by the MicroStation Level Manager, using by-level in the attributes toolbox, or by the active attributes. You can change the element attributes using the change element attributes tool, located in the change attributes toolbox, located on the MicroStation Main menu.

WaterCAD V8*i* toolbars are turned off by default when you start. They are found under View>Toolbars and they can be turned on. By default they will be floating toolbars but they can be docked wherever the user chooses.

Note: Any MicroStation tool that deletes the target element (such as Trim and IntelliTrim) will also remove the connection of that element to WaterCAD V8*i*. After the WaterCAD V8*i* connection is removed, the element is no longer a valid wtg link element and will not show properties on the property grid. The element does not have properties because it is not part of the WTRG model. It's as if the user just used MSTN tools to layout a rectangle in a WTRG dgn. It's just a dgn drawing element but has nothing to do with the water model.

MicroStation Project Files

When using Bentley WaterCAD V8*i* in the MicroStation environment, there are three files that fundamentally define a Bentley WaterCAD V8*i* model project:

- **Drawing File (.DGN)**—The MicroStation drawing file contains the elements that define the model, in addition to the planimetric base drawing information that serves as the model background.
- **Model File (.**wtg**)**—The model file contains model data specific to WaterCAD V8*i*, including project option settings, color-coding and annotation settings, etc. Note that the MicroStation .dgn that is associated with a particular model may not have the same filename as the model's .wtg file.
- **Database File (.sqlite)**—The model database file that contains all of the input and output data for the model. Note that the MicroStation .dgn that is associated with a particular model may not have the same filename as the model's .sqlite file.

To send the model to another user, all three files are required.

It is important to understand that archiving the drawing file is not sufficient to reproduce the model. You must also preserve the associated .wtg and .sqlite files.

Saving Your Project in MicroStation

The WaterCAD V8*i* project data is synchronized with the current MicroStation .dgn. WaterCAD V8*i* project saves are triggered when the .dgn is saved. This is done with the MicroStation File>Save command, which saves the .dgn, .sqlite and .wtg files. If you want to have more control over when the WaterCAD V8*i* project is saved, turn off MicroStation's AutoSave feature; then you will be prompted for the .dgn.

There are two File>Save As commands in MicroStation. SaveAs in MSTN is for the dgn, and allows the user to, for example, change the dgn filename that they're working with .wtg model filenames in this case stay the same. The Project's SaveAs allows the user to change the filename of the .wtg and .sqlite files, but it doesn't change the dgn's filename. Keep in mind that the dgn and model filenames don't have any direct correlation. They can be named the same, but they don't have to be.

Bentley WaterCAD V8*i* Element Properties

Bentley WaterCAD V8*i* element properties includes:

- Element Properties
- Element Levels Dialog
- Text Styles

Element Properties

When working in the MicroStation environment, this feature will display a dialog box containing fields for the currently selected element's associated properties. To modify an attribute, click each associated grid cell. To open the property grid, pick View>Properties from the WaterCAD V8*i* menu.

You can also review or modify MicroStation drawing information about an element(s), such as its type, attributes, and geometry, by using the Element Information dialog. To access the Element Information dialog, click the Element Information button or click the Element menu and select the Information command. This is where the user can change the appearance for individual elements. However, in general, if WaterCAD V8*i* color coding conflicts with MicroStation element symbology, the WaterCAD V8*i* color will show.

To control display of elements in the selected levels, use the Level Display dialog box. To access the Level Display dialog, click the Settings menu and select the Level > Display command.

To move WaterCAD V8*i* elements to levels other than the default (Active) level, select the elements and use the Change Element Attribute command.

If you want to freeze elements in levels, select Global Freeze from the View Display menu in the Level Display dialog.

You can create new Levels in the Level Manager. To access the Level Manager, click the Settings menu and select the Level > Manager command.

To control the display of levels, use level filters. Within MicroStation, you can also create, edit, and save layer filters to DWG files in the Level Manager. To access the Level Manager, click the Settings menu and select the Level > Manager command. Layer filters are loaded when a DWG file is opened, and changes are written back when the file is saved. To create and edit Level Filters,

Element Levels Dialog

This dialog allows you to assign newly created elements and their associated annotations to specific MicroStation levels.

To assign a level, use the pulldown menu next to an element type (under the Element Level column heading) to choose the desired level for that element. You can choose a seperate level for each element and for each element's associated annotation.

You cannot create new levels from this dialog; to create new levels use the MicroStation Level Manager. To access the Level Manager, click the Settings menu and select the Level > Manager command.

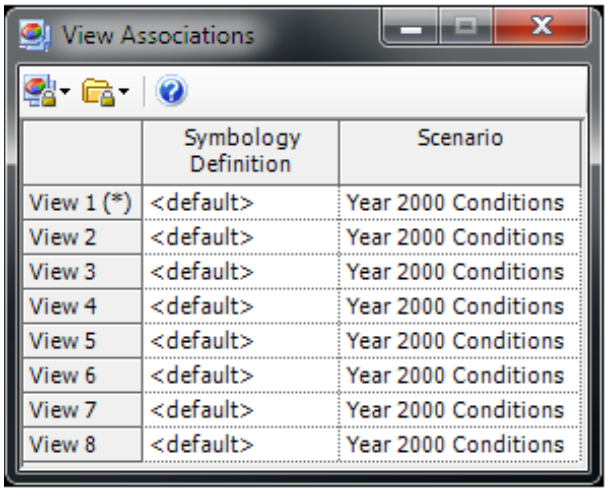
Text Styles

You can view, edit, and create Text Style settings in the MicroStation environment by clicking the MicroStation Element menu and selecting the Text Styles command to open the Text Styles dialog.

View Associations (MicroStation Only)

To open the View Associations dialog, click **View > View Assocations**.

MicroStation has support for opening multiple View windows on the current design drawing. By default, each MicroStation View reflects the current Scenario and the current Symbology Definition. View Associations allows you to control the Scenario and Symbology Definition to display in each MicroStation View.



	Symbology Definition	Scenario
View 1 (*)	<default>	Year 2000 Conditions
View 2	<default>	Year 2000 Conditions
View 3	<default>	Year 2000 Conditions
View 4	<default>	Year 2000 Conditions
View 5	<default>	Year 2000 Conditions
View 6	<default>	Year 2000 Conditions
View 7	<default>	Year 2000 Conditions
View 8	<default>	Year 2000 Conditions

The View Associations window allows you to see (and change) the Symbology Definition and Scenario associated with each MicroStation View.

Located along the top of the window are two toolbars buttons for controlling the view association mode: 

The first toolbar button controls the Symbology Definition mode, and the second controls the Scenario mode.

View Associations provides two modes: **Synchronized** mode and **Independent** mode.

Synchronized mode: In Synchronized mode, all Views reflect the active Scenario and active Symbology-Definition. If you change the active Scenario, all views will update to reflect that change; similar for a change to the active Symbology Definition. A small padlock symbol ( ) will appear on the icon to indicate if Synchronized mode is active.

Independent mode: Independent mode allows you to independently control which Scenario and Symbology definition are shows in each view. You can show one Scenarion\Symbology Definition on one view, and different Scenarios\Symbology Definition combingation in the other views.

Note: The default setting for View Associations (for Scenarios and Symbology-Definitions) is "Synchronized" mode. Scenarios and Symbology definition modes can each be controlled separately.

For convenience, these same mode toolbar buttons are available at the top of the Scenario management Window and the Element Symbology management window. Changes to current Scenario and current Symbology Definition will be applied to the active MicroStation View (for synchronized mode, changes you make will be reflected in all Views).

See also:

Annotating Your Model

Symbology Definitions Manager

Scenarios Manager

Working with Elements

Working with elements includes:

- Edit Elements

- Deleting Elements
- Modifying Elements

Edit Elements

Elements can be edited in one of two ways in the MicroStation environment:

Properties Editor Dialog: To access the Properties Editor dialog, click the WaterCAD V8*i* View menu and select the Properties command. For more information about the Properties Editor dialog, see Property Editor.

FlexTables: To access the FlexTables dialog, click the WaterCAD V8*i* View menu and select the FlexTables command. For more information about the FlexTables dialog, see Viewing and Editing Data in FlexTables.

Deleting Elements

In the MicroStation environment, you can delete elements by clicking on them using the Delete Element tool, or by highlighting the element to be deleted and clicking your keyboard's Delete key.

Note: Any MicroStation tool that deletes the target element (such as Trim and IntelliTrim) will also remove the connection of that element to WaterCAD V8*i*. After the WaterCAD V8*i* connection is removed, the element is no longer a valid wtg link and will not show properties on the property grid.

Modifying Elements

In the MicroStation environment, these commands are selected from the shift-right-click shortcut menu (hold down the Ctrl key while right-clicking). They are used for scaling and rotating model entities.

Context Menu

Certain commands can be activated by using the right-click context menu. To access the context menu, right-click and hold down the mouse button until the menu appears.

Working with Elements Using MicroStation Commands

Working with elements using MicroStation commands includes:

Bentley WaterCAD V8i Custom MicroStation Entities on page 3-236

MicroStation Commands on page 3-236

Moving Elements on page 3-236

Moving Element Labels on page 3-237

Snap Menu on page 3-237

Bentley WaterCAD V8*i* Custom MicroStation Entities

The primary MicroStation-based Bentley WaterCAD V8*i* element entities are all implemented using native MicroStation elements (the drawing symbols are standard MSTN objects).These elements have feature linkages to define them as WaterCAD V8*i* objects.

This means that you can perform standard MicroStation commands (see MicroStation Commands on page 3-236) as you normally would, and the model database will be updated automatically to reflect these changes.

It also means that the model will enforce the integrity of the network topological state, which means that nodes and pipes will remain connected even if individual elements are moved. Therefore, if you delete a nodal element such as a junction, its connecting pipes will also be deleted since their connecting nodes topologically define model pipes.

Using MDL technology ensures the database will be adjusted and maintained during Undo and Redo transactions.

See “The MicroStation Environment Graphical Layout” on page 229.

MicroStation Commands

When running in the MicroStation environment, WaterCAD V8*i* makes use of all the advantages that MicroStation has, such as plotting capabilities and snap features. Additionally, MicroStation commands can be used as you would with any design project. For example, our products’ elements and annotation can be manipulated using common MicroStation commands. To get at the MicroStation command line (called the "Key-In Browser, the user can pick Help>Key-In Browser or hit the Enter key.

Moving Elements

When using the MicroStation environment, the MicroStation commands Move, Scale, Rotate, Mirror, and Array (after right clicking on the label) can be used to move elements.

To move a node, execute the MicroStation command by either typing it at the command prompt or selecting it. Follow the MicroStation prompts, and the node and its associated label will move together. The connecting pipes will shrink or stretch depending on the new location of the node.

Moving Element Labels

When using the MicroStation environment, the MicroStation commands Move, Scale, Rotate, Mirror, and Array can be used to move element text labels.

To move an element text label separately from the element, click the element label you wish to move. The grips will appear for the label. Execute the MicroStation command either by typing it at the command prompt, by selecting it from the tool palette, or by selecting it from the right-click menu. Follow the MicroStation prompt, and the label will be moved without the element.

Snap Menu

When using the MicroStation environment, you can enable the Snaps button bar by clicking the Settings menu and selecting the Snaps > Button Bar command. See the MicroStation documentation for more information about using snaps.

Background Files

Adding MicroStation Background images is different than in stand alone. You need to go to File>References>Tools>Attach. Background files to be attached with this command include .dgn, .dwg and .dxf files. Raster files should be attached using File>Raster Manager. GIS files (e.g. shapefiles) may need to be converted to the appropriate CAD or raster formats using GeoGraphics to be used as background. See MicroStation for details about the steps involved in creating these backgrounds.

Import Bentley WaterCAD V8*i*

When running WaterCAD V8*i* in the MicroStation environment, this command (Project>Import>WaterCAD V8*i* database) imports a selected WaterCAD V8*i* data (.wtg) file for use in the current drawing (.dgn). You will be prompted for the WaterCAD V8*i* filename to save. The new project file will now correspond to the drawing name, such as, CurrentDrawingName.wtg. Whenever you save changes to the network model through WaterCAD V8*i* the associated .wtg data file is updated and can be loaded into WaterCAD V8*i* or higher.

Warning! A WaterCAD V8*i* Project can only be imported to a new, empty MicroStation design model (.dgn file).

Annotation Display

Some fonts do not correctly display the full range of characters used by WaterCAD V8*i*'s annotation feature because of a limited character set. If you are having problems with certain characters displaying improperly or not at all, try using another font.

Multiple models

You can have two or more WaterCAD V8*i* models open in MicroStation. However, you need to open them in MicroStation, not in wtg. In MicroStation choose File > Open and select the .dgn file.

Native Format Contours

WaterCAD V8*i* can export contours as native-format Microstation contours. This feature behaves differently depending on whether or not the original model is 2 or 3 dimensional. Since the native contours are 3-dimensional elements they don't display properly in a 2-d model and reference attachments are created and added to the model.

In a 2-d source model the contours are created in their own 3-d model, which is referenced to the default model. In order to manipulate the contours you'll need to activate the respective model, then make any modifications, then switch back. On the same token, in order to delete the contours you need to delete the model that they're actually a part of.

In a 3-d source model the contours are added directly to the model, and all manipulations can be done directly in the main drawing.

Note: This feature is only available to users of MicroStation SS3 and higher.

Working in AutoCAD

The AutoCAD environment lets you create and model your network directly within your primary drafting environment. This gives you access to all of AutoCAD's drafting and presentation tools, while still enabling you to perform Bentley WaterCAD V8*i* modeling tasks like editing, solving, and data management. This relationship between Bentley WaterCAD V8*i* and AutoCAD enables extremely detailed and accurate mapping of model features, and provides the full array of output and presentation features available in AutoCAD. This facility provides the most flexibility and the highest degree of compatibility with other CAD-based applications and drawing data maintained at your organization.

Bentley WaterCAD V8*i* features support for AutoCAD integration. You can determine if you have purchased AutoCAD functionality for your license of Bentley WaterCAD V8*i* by using the **Help > About** menu option. Click the **Registration** button to view the feature options that have been purchased with your application license. If AutoCAD support is enabled, then you will be able to run your Bentley WaterCAD V8*i* application in both AutoCAD and stand-alone environment.

The AutoCAD functionality has been implemented in a way that is the same as the WaterCAD V8*i* base product. Once you become familiar with the stand-alone environment, you will not have any difficulty using the product in the AutoCAD environment.

Some of the advantages of working in the AutoCAD environment include:

- Layout network links and structures in fully-scaled environment in the same design and drafting environment that you use to develop your engineering plans. You will have access to any other third party applications that you currently use, along with any custom LISP, ARX, or VBA applications that you have developed.
- Use native AutoCAD insertion snaps to precisely position Bentley WaterCAD V8*i* elements with respect to other entities in the AutoCAD drawing.
- Use native AutoCAD commands such as ERASE, MOVE, and ROTATE on Bentley WaterCAD V8*i* model entities with automatic update and synchronization with the model database.
- Control destination layers for model elements and associated label text and annotation, giving you control over styles, line types, and visibility of model elements.

Note: **Bentley WaterGEMSV8i supports the 32-bit and 64-bit versions of AutoCAD 2012, 2013, 2014, 2015 only.**

Caution: **If you previously installed Bentley ProjectWise and turned on AutoCAD integration, you must add the following key to your system registry using the Windows Registry Editor. Before you edit the registry, make a backup copy.**

HKEY_LOCAL_MACHINE\SOFTWARE\Bentley\ProjectWise iDesktop Integration\XX.XX\Configuration\AutoCAD"

String value name: DoNotChangeCommands

Value: 'On'

To access the Registry Editor, click Start > Run, then type regedit. Using the Registry Editor incorrectly can cause serious, system-wide problems that may require you to re-install Windows to correct them. Always make a backup copy of the system registry before modifying it.

The AutoCAD Workspace

In the AutoCAD environment, you will have access to the full range of functionality available in the AutoCAD design and drafting environment. The standard environment is extended and enhanced by an AutoCAD ObjectARX Bentley WaterCAD V8*i* client layer that lets you create, view, and edit the native Bentley WaterCAD V8*i* network model while in AutoCAD.

AutoCAD Integration with WaterCAD V8*i*

When you install WaterCAD V8*i* after you install AutoCAD, integration between the two is automatically configured.

If you install AutoCAD after you install WaterCAD V8*i*, you must manually integrate the two by selecting **Start** > **All Programs** > **Bentley** >WaterCAD V8*i* > **Integrate** WaterCAD V8*i* **with ArcGIS-AutoCAD-MicroStation**. The integration utility runs automatically. You can then run WaterCAD V8*i* in the AutoCAD environment.

The Integrate WaterCAD V8*i* with AutoCAD-ArcGIS command can also be used to fix problems with the AutoCAD configuration file. For example, if you have CivilStorm installed on the same system as Bentley WaterCAD V8*i* and you uninstall or reinstall CivilStorm, the AutoCAD configuration file becomes unusable. To fix this problem, you can delete the configuration file then run the Integrate WaterCAD V8*i* with AutoCAD-ArcGIS command.

Getting Started within AutoCAD

There are a number of options for creating a model in the AutoCAD client:

- **Create a model from scratch**—You can create a model in AutoCAD. Upon opening AutoCAD a Drawing1.dwg file is created and opened. Likewise an untitled new WaterCAD V8*i* project is also created and opened if WaterCAD V8*i* has been loaded. WaterCAD V8*i* has been loaded if the WaterCAD V8*i* menus and docking windows are visible. WaterCAD V8*i* can be loaded in two ways: automatically by using the "WaterCAD V8*i* for AutoCAD" shortcut, or by starting AutoCAD and then using the command: WaterCAD V8*i*Run. Once loaded, you can immediately begin laying out your network and creating your model using the Bentley WaterCAD V8*i* menus and the WaterCAD V8*i* file menu (See Menus). Upon saving and titling your AutoCAD file for the first time, your WaterCAD V8*i* project files will also acquire the same name and file location.
- **Open a previously created** Bentley WaterCAD V8*i* **project**—You can open a previously created Bentley WaterCAD V8*i* model. If the model was created in the Stand Alone version, you must import your WaterCAD V8*i* project while a .dwg file is open. From the WaterCAD V8*i* menu select Project -> Import -> WaterCAD V8*i* Database. Alternatively you can use the command: _wtgImportProject. You will have the choice to import your WaterCAD V8*i* database file (.sqlite) or your WaterCAD V8*i* project file (.wtg).
- **Import a model that was created in another modeling application**—You can import a model that was created in EPANET. See Importing and Exporting Data for further details.

Menus

In the AutoCAD environment, in addition to AutoCAD's menus, the following Bentley WaterCAD V8*i* menus are available:

- Project
- Edit
- Analysis
- Components
- View
- Tools
- Report
- Help

The Bentley WaterCAD V8*i* menu commands work the same way in AutoCAD and the Stand-Alone Editor. For complete descriptions of Bentley WaterCAD V8*i* menu commands, see Menus.

Many commands are available from the right-click context menu. To access the menu, first highlight an element in the drawing pane, then right-click it to open the menu.

Drawing Setup

When working in the AutoCAD environment, you may work with our products in many different AutoCAD scales and settings. However, WaterCAD V8*i* elements can only be created and edited in model space.

Symbol Visibility

In the AutoCAD environment, you can control display of element labels using the check box in the Drawing Options dialog box.

Note: In AutoCAD, it is possible to delete element label text using the ERASE command. You should not use ERASE to control visibility of labels. If you desire to control the visibility of a selected group of element labels, you should move them to another layer that can be frozen or turned off.

AutoCAD Project Files

When using Bentley WaterCAD V8*i* in the AutoCAD environment, there are three files that fundamentally define a Bentley WaterCAD V8*i* model project:

- **Drawing File (.dwg)**—The AutoCAD drawing file contains the custom entities that define the model, in addition to the planimetric base drawing information that serves as the model background.
- **Model File (.**wtg**)**—The native Bentley WaterCAD V8*i* model database file that contains all the element properties, along with other important model data. Bentley WaterCAD V8*i* .etc files can be loaded and run using the Stand-Alone Editor. These files may be copied and sent to other Bentley WaterCAD V8*i* users who are interested in running your project. This is the most important file for the Bentley WaterCAD V8*i* model.
- wtg **Exchange Database (.**wtg**.sqlite)**—The intermediate format for wtg project files. When you import a wtg file into Bentley WaterCAD V8*i* , you first export it from wtg into this format, then import the .wtg.sqlite file into Bentley WaterCAD V8*i* . Note that this works the same in the Stand-Alone Editor and in AutoCAD.

The three files have the same base name. It is important to understand that archiving the drawing file is not sufficient to reproduce the model. You must also preserve the associated .etc and wtg.sqlite file.

Since the .etc file can be run and modified separately from the .dwg file using the Stand-Alone Editor, it is quite possible for the two files to get out of sync. Should you ever modify the model in the Stand-Alone Editor and then later load the AutoCAD .dwg file, the Bentley WaterCAD V8*i* program compares file dates, and automatically use the built-in AutoCAD synchronization routine.

Click one of the following links to learn more about AutoCAD project files and Bentley WaterCAD V8*i* :

- Drawing Synchronization on page 3-243
- Saving the Drawing as Drawing*.dwg on page 3-244

Drawing Synchronization

Whenever you open a Bentley WaterCAD V8*i* -based drawing file in AutoCAD, the Bentley WaterCAD V8*i* model server will start. The first thing that the application will do is load the associated Bentley WaterCAD V8*i* model (.wtg) file. If the time stamps of the drawing and model file are different, Bentley WaterCAD V8*i* will automatically perform a synchronization. This protects against corruption that might otherwise occur from separately editing the Bentley WaterCAD V8*i* model file in stand-alone environment, or editing proxy elements at an AutoCAD station where the Bentley WaterCAD V8*i* application is not loaded.

The synchronization check will occur in two stages:

- First, Bentley WaterCAD V8*i* will compare the drawing model elements with those in the server model. Any differences will be listed. Bentley WaterCAD V8*i* enforces network topological consistency between the server and the drawing state. If model elements have been deleted or added in the .wtg file during a WaterCAD V8*i* session, or if proxy elements have been deleted, Bentley WaterCAD V8*i* will force the drawing to be consistent with the native database by restoring or removing any missing or excess drawing custom entities.
- After network topology has been synchronized, Bentley WaterCAD V8*i* will compare other model and drawing states such as location, labels, and flow directions.

You can run the Synchronization check at any time using the following command:

```
wtgSYNCHRONIZE
```

Or by selecting Tools > Database Utilities > Synchronize Drawing.

Saving the Drawing as Drawing*.dwg

AutoCAD uses Drawing*.dwg as its default drawing name. Saving your drawing as the default AutoCAD drawing name (for instance Drawing1.dwg) should be avoided, as it makes overwriting model data very likely. When you first start AutoCAD, the new empty drawing is titled Drawing*.dwg, regardless of whether one exists in the default directory. Since our modeling products create model databases associated with the AutoCAD drawing, the use of Drawing*.dwg as the saved name puts you at risk of causing synchronization problems between the AutoCAD drawing and the modeling files.

Note: If this situation inadvertently occurs (save on quit for example), restart AutoCAD, use the Open command to open the Drawing*.dwg file from its saved location, and use the Save As command to save the drawing and model data to a different name.

Working with Elements Using AutoCAD Commands

This section describes how to work with elements using AutoCAD commands, including:

- WaterCAD V8i Custom AutoCAD Entities
- Explode Elements
- Moving Elements
- Moving Element Labels
- Snap Menu
- Polygon Element Visibility
- Undo/Redo
- Contour Labeling

WaterCAD V8*i* Custom AutoCAD Entities

The primary AutoCAD-based WaterCAD V8*i* element entities—pipes, junctions, pumps, etc.—are all implemented using ObjectARX custom objects. Thus, they are vested with a specialized model awareness that ensures that any editing actions you perform will result in an appropriate update of the model database.

This means that you can perform standard AutoCAD commands (see Working with Elements Using AutoCAD Commands) as you normally would, and the model database will be updated automatically to reflect these changes.

It also means that the model will enforce the integrity of the network topological state. Therefore, if you delete a nodal element such as a junction, its connecting pipes will also be deleted since their connecting nodes topologically define model pipes.

Using ObjectARX technology ensures the database will be adjusted and maintained during Undo and Redo transactions.

When running in the AutoCAD environment, Bentley Systems' products make use of all the advantages that AutoCAD has, such as plotting capabilities and snap features. Additionally, AutoCAD commands can be used as you would with any design project. For example, our products' elements and annotation can be manipulated using common AutoCAD commands.

Explode Elements

In the AutoCAD environment, running the AutoCAD Explode command will transform all custom entities into equivalent AutoCAD native entities. When a custom entity is exploded, all associated database information is lost. Be certain to save the exploded drawing under a separate filename.

Use Explode to render a drawing for finalizing exhibits and publishing maps of the model network. You can also deliver exploded drawings to clients or other individuals who do not own a Bentley Systems Product license, since a fully exploded drawing will not be comprised of any ObjectARX proxy objects.

Moving Elements

When using the AutoCAD environment, the AutoCAD commands Move, Scale, Rotate, Mirror, and Array can be used to move elements.

To move a node, execute the AutoCAD command by either typing it at the command prompt or selecting it. Follow the AutoCAD prompts, and the node and its associated label will move together. The connecting pipes will shrink or stretch depending on the new location of the node.

Moving Element Labels

When using the AutoCAD environment, the AutoCAD commands Move, Scale, Rotate, Mirror, and Array can be used to move element text labels.

To move an element text label separately from the element, click the element label you wish to move. The grips will appear for the label. Execute the AutoCAD command either by typing it at the command prompt, by selecting it from the tool palette, or by selecting it from the right-click menu. Follow the AutoCAD prompt, and the label will be moved without the element.

Snap Menu

When using the AutoCAD environment, the Snap menu is a standard AutoCAD menu that provides options for picking an exact location of an object. See the Autodesk AutoCAD documentation for more information.

Polygon Element Visibility

By default, polygon elements are sent to the back of the draw order when they are drawn. If the draw order is modified, polygon elements can interfere with the visibility of other elements. This can be remedied using the AutoCAD Draw Order toolbar.

To access the AutoCAD Draw Order toolbar, right-click on the AutoCAD toolbar and click the Draw Order entry in the list of available menus.

By default, polygon elements are filled. You can make them unfilled (just borders visible) using the AutoCAD FILL command. After turning fill environment OFF, you must REGEN to redraw the polygons.

Undo/Redo

The menu-based undo and redo commands operate exclusively on Bentley WaterCAD V8*i* elements by invoking the commands directly on the model server. The main advantage of using the specialized command is that you will have unlimited undo and redo levels. This is an important difference, since in layout or editing it is quite useful to be able to safely undo and redo an arbitrary number of transactions.

Whenever you use a native AutoCAD undo, the server model will be notified when any Bentley WaterCAD V8*i* entities are affected by the operation. Bentley WaterCAD V8*i* will then synchronize the model to the drawing state. Wherever possible, the model will seek to map the undo/redo onto the model server's managed command history. If the drawing's state is not consistent with any pending undo or redo transactions held by the server, Bentley WaterCAD V8*i* will delete the command history. In this case, the model will synchronize the drawing and server models.

Note: If you use the native AutoCAD undo, you are limited to a single redo level. The Bentley WaterCAD V8*i* undo/redo is faster than the native AutoCAD undo/redo. If you are rolling back Bentley WaterCAD V8*i* model edits, it is recommended that you use the menu-based Bentley WaterCAD V8*i* undo/redo.

If you undo using the AutoCAD undo/redo and you restore Bentley WaterCAD V8*i* elements that have been previously deleted, morphed, or split, some model state attributes such as diameters or elevations may be lost, even though the locational and topological state is fully consistent. This will only happen in situations where the Bentley WaterCAD V8*i* command history has been deleted. In such cases, you will be warned to check your data carefully.

Contour Labeling

You can apply contour labels after the contour plot has been exported to the AutoCAD drawing. The labeling commands are accessed from the Tools menu. The following options are available:

- **End**—Allows you to apply labels to one end, both ends, or any number of selected insertion points. After selecting this labeling option, AutoCAD will prompt you to Select Contour to label. After selecting the contour to label, AutoCAD prompts for an Insertion point. Click in the drawing view to place labels at specified points along the contour. When prompted for an Insertion point, clicking the Enter key once will prompt you to select point nearest the contour endpoint. Doing so will apply a label to the end of the contour closest to the area where you clicked. Clicking the Enter key twice when prompted for an Insertion point will apply labels to both ends of the contour.
- **Interior**—This option applies labels to the interior of a contour line. You will be prompted to select the contour to be labeled, then to select the points along the contour line where you want the label to be placed. Any number of labels can be placed inside the contour in this way. Clicking the label grip and dragging will move the label along the contour line.

- **Group End**—Choosing this option opens the Elevation Increment dialog box. The value entered in this dialog box determines which of the contours selected will be labeled. If you enter 2, only contours representing a value that is a multiple of 2 will be labeled, and so on. After clicking OK in this dialog box, you will be prompted to select the Start point for a line. Contours intersected by the line drawn thusly will have a label applied to both ends, as modified by the Elevation Increment that was selected.
- **Group Interior**—Choosing this option opens the Elevation Increment dialog box. The value entered in this dialog box determines which of the contours selected will be labeled. If you enter 2, only contours representing a value that is a multiple of 2 will be labeled, and so on. After clicking OK in this dialog box, you will be prompted to select the Start point for a line.
- **Change Settings**—Allows you to change the Style, Display Precision, and Font Height of the contour labels.
- **Delete Label**—Prompts to select the contour from which labels will be deleted, then prompts to select the labels to be removed.
- **Delete All Labels**—Prompts to select which contours the labels will be removed from, then removes all labels for the specified contours.

Note: Contours are only views unless they are exported to to native format, and only native format contours can be edited.

Note: This option is for the ArcGIS client only.

This dialog allows you to initialize the range. The Layer Symbology dialog is accessed by clicking HAMMER > Tools > Layer Symbology.

By default, elements that fall outside of the defined range will not be displayed. Choose the "Include Undefined?" option to display elements that fall outside the defined range.

Google Earth Export

Google Earth export allows a WaterCAD V8*i* user to display WaterCAD V8*i* spatial data and information (input/results) in a platform that is growing more and more popular with computer users around the world for viewing general spatial data on the earth.

WaterCAD V8*i* supports a limited export of model features and results to Google Earth through the Microstation V8i and ArcGIS 9.3 platforms. The benefits of this functionality include:

- Share data and information with non WaterCAD V8*i* users in a portable open format,
- Leverage the visual presentation of Google Earth to create compelling visual presentations,
- Present data along side other Google Earth data such as satellite imagery and 3D buildings.

Steps for using the export feature in each platform are described below.

In general, the process involves creation of a Google Earth format file (called a KML - Keyhole Markup Language - file). This file can be opened in Google Earth. Google Earth however is not a "platform" as ArcGIS is because it is not possible to edit or run the model in Google Earth. It is simply for display.

Once the KML file has been generated in WaterCAD V8*i* it can be viewed in Google Earth by opening Google Earth (version 3 or later) and selecting File > Open and selecting the KML file that was created.

The layers you open in Google Earth will appear as "Temporary Places" in the Places manager. These can be checked or unchecked to turn the layers on or off.

Google Earth Export from the MicroStation Platform

For the purpose of describing the export process these steps will assume that the model you wish to export has been defined (laid out) in terms of a well-known spatial reference (coordinate system). The model if opened in the WaterCAD V8*i* stand alone interface is in scaled drawing mode (Tools --> Options --> Drawing Tab --> Drawing Mode: Scaled).

Preparing to Export to Google Earth from Microstation

In order to describe how to export WaterCAD V8*i* data to Google Earth we will cover a set of questions to determine which steps need to be performed. Each question will result in either performing some steps or moving on to the next question. Each question is relating to your WaterCAD V8*i* model.

Q1: Do you already have a *.dgn (Microstation drawing file)? If yes go to Q2, else follow steps 1 to 6.

1. Open WaterCAD V8*i* for Microstation V8i.
2. Locate the model folder and create a new dgn file (new file icon at the top right of the File Open dialog) with a name of your choice. e.g., if the model is called "MyModel.wtg" a dgn file called "MyModel.dgn" might be appropriate.
3. Select the newly created *.dgn file and click Open.
4. From the WaterCAD V8*i* menu, select Project --> Attach Existing…
5. Select the *.wtg model file and click Open.
6. After the model has been imported save the *.dgn. in Microstation, File --> Save.

Q2: Do you have a spatial reference defined in the dgn? If yes go to Q3, else follow steps 1 and 2 below.

> **Note: If your model is not modelled in a known coordinate system or you don't know the coordinate system, but the model is to scale you may be able to determine an approximate fit to Google Earth features using Place Mark Monuments. For more information on how to use Place Mark Monuments as an alternative to a Geographic Coordinate System please consult the Microstation help.**

1. In Microstation choose Tools --> Geographic --> Select Geographic Coordinate System.
2. In the dialog that opens, using the toolbar, you may select a Geographic Coordinate System from a library or from an existing *.dgn. Select the projected coordinate system that applies to your model. For further information on Geographic Coordinate Systems please consult the Microstation documentation.

Note: You may be prompted by Microstation saying that your DGN storage units are different from the coordinate system you selected. Assuming your model is already correctly to scale, you should choose not to change the units inside Microstation. Consult the Microstation help should you need more information.

Q3: Have you configured the Google Earth Export settings? If yes go to step Q4, else follow steps 1 and 2 below.

1. In Microstation choose Tools --> Geographic --> Google Earth Settings. Ensure that the Google Earth Version is set to version 3.
2. If you have Google Earth installed on your machine you may find it convenient for the export to open the exported Google Earth file directly. If so, ensure that the "Open File After Export" setting is checked. If you do not have Google Earth installed uncheck this option. Please consult the Microstation documentation for the function of other settings. In most cases the defaults should suffice.

Q4: Have you set up your model as you wish it to be displayed in Google Earth? If yes go to "Exporting to Google Earth from Microstation", else follow step 1 below.

1. Use the WaterCAD V8*i* Element Symbology to define the color coding and annotation that you wish to display in Google Earth.

Exporting to Google Earth from Microstation

1. Once you are ready to export to Google Earth the process is very simple. In Microstation choose File --> Export --> Google Earth…
2. Select a name for your Google Earth file and click Save. If you have Google Earth installed and chose to open the Google Earth file after export (see step 10) then the exported file will open inside Google Earth and you can view the result. The exported file can be used inside Google Earth independently of the original WaterCAD V8*i* or Microstation model.

Google Earth Export from ArcGIS

For the purpose of describing the export process these steps will assume that the model you wish to export has been defined (laid out) in terms of a well-known spatial reference (coordinate system). The model if opened in the WaterCAD V8*i* stand alone interface is in scaled drawing mode (Tools --> Options --> Drawing Tab --> Drawing Mode: Scaled).

Preparing to Export to Google Earth from ArcGIS

In order to describe how to export WaterCAD V8*i* data to Google Earth we will cover a set of questions to determine which steps need to be performed. Each question will result in either performing some steps or moving on to the next question. Each question is relating to your WaterCAD V8*i* model.

Q1: Do you already have a *.mxd (ArcMap map file)? If yes go to Q2, else follow steps 1 to 10.

1. Open ArcMAP 9.3.
2. Start with a new empty map.
3. From the WaterCAD V8*i* toolbar, choose WaterCAD V8*i* --> Project --> Add Existing Project.
4. Locate and select the model *.wtg and click Open.
5. In the Attach Geodatabase dialog select the blue folder at top right and create a new Geodatabase with the name of your choice. e.g., if the model database is called "MyModel.wtg.sqlite" a geodatabase file called "MyModelGeo.sqlite" might be appropriate. Click Save.
6. Select the appropriate spatial reference (projected coordinate system) by clicking the Change --> Select… (or Import… from an existing geodataset).
7. Ensure that the X/Y Domain settings are valid for your model.
8. Make sure the correct Spatial Data Coordinates Unit is selected, then click OK.

   Note: For further assistance on setting spatial references and related settings please consult the ArcMap documentation.

9. Once the model add process is complete save the map file (*.mxd).
10. Go to Q3.

Q2 Do you have a spatial reference defined in the geodatabase? If yes go to Q3, else follow steps 1 to 9 below.

Note: For assistance on setting spatial references and related settings please consult the ArcMap documentation.

1. To add a spatial reference to your model, close ArcMap if already open.
2. Open ArcCatalog.
3. Browse for the geodatabase of interest.
4. Expand the dataset node (cylinder) to show the feature dataset (3 rectangles).
5. Right-click on the feature dataset and choose Properties.
6. Click the XY Coordinate System tab.

7. Either Select… or Import… the appropriate projected coordinate system.
8. Close ArcCatalog.
9. Open ArcMap and re-open the *.mxd.

Q3: Have you set up your model as you wish it to be displayed in Google Earth? If yes go to Exporting to a KML File from ArcGIS**, else follow steps 1 to 8 below.**

1. Prior to exporting to Google Earth you should configure the layers that you wish to export. Many of the layer properties supported in ArcMap presentation can be used with Google Earth export. Please consult the ArcGIS documentation for detailed instructions on layer properties. Some basic examples are provided.
2. Right click on a layer, for example the Pipes layer, and choose Properties.
3. Select the Fields tab.
4. Change the Primary Display Field to Label. (If this field is not available, you need to make sure the WaterCAD V8*i* project is open. See details below.)
5. Click on the HTML Popup tab.
6. Check "Show content for this layer using the HTML Popup tool."
7. Click "Verify" to see the fields. (These can be customized by editing your WaterCAD V8*i* GeoTables). This table will be viewable inside Google Earth after exporting.
8. Repeat steps 1 through 6 above for each layer you wish to export.

Exporting to a KML File from ArcGIS

1. In ArcMap, Window --> ArcToolbox.
2. ArcToolbox --> Conversion Tools --> To KML --> Layer to KML.
3. In the dialog that opens, select the layer you wish to export to Google Earth, e.g., Pipe.
4. Specify the Google Earth file name, e.g., Pipe.kmz.
5. Pick a layer output scale that makes sense for your layer. (See the ArcGIS help topic on the effect of this value). Assuming you have no zoom dependent scaling or are not exporting any symbology, a value of 1 should work fine.
6. Click OK to commence the export. (This may take some time.)
7. If you have Google Earth installed you may now open the exported *.kmz file and view it in Google Earth.
8. Repeat steps 2 to 7 for each layer you wish to export.

Note: You can export all layers at once using the Map to KML tool.

Using a Google Earth View as a Background Layer to Draw a Model

Google Earth images generally do not possess the accuracy of engineering drawings. However, in some cases, a user can create a background image (as a jpg or bmp file) and draw a model on that image. In general this model will not be to scale and the user must then enter pipe lengths using user defined lengths.

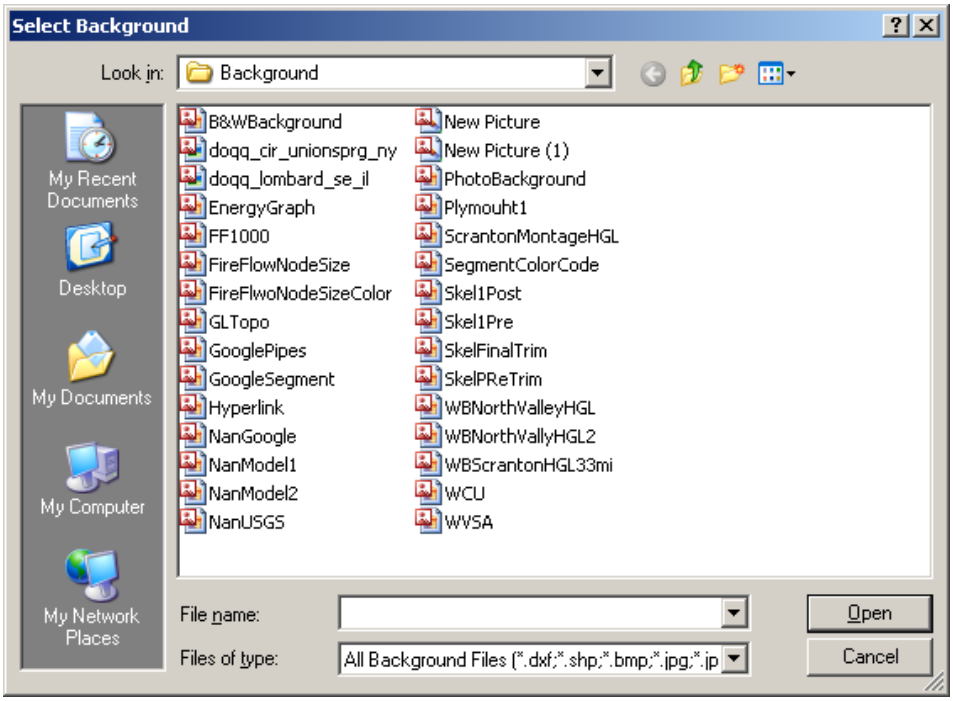
There is an approach that can be used to draw a roughly scaled model in the stand alone platform without the need to employ user define lengths which can be fairly time consuming. The steps are given below:

1. Open the Google Earth image and zoom to the extents that will be used for the model. Make certain that the view is vertical straight down (not tilted). Using Tools > Ruler, draw a straight line with a known length (in an inconspicuous part of the image). Usually a 1000 ft is a good length as shown below:

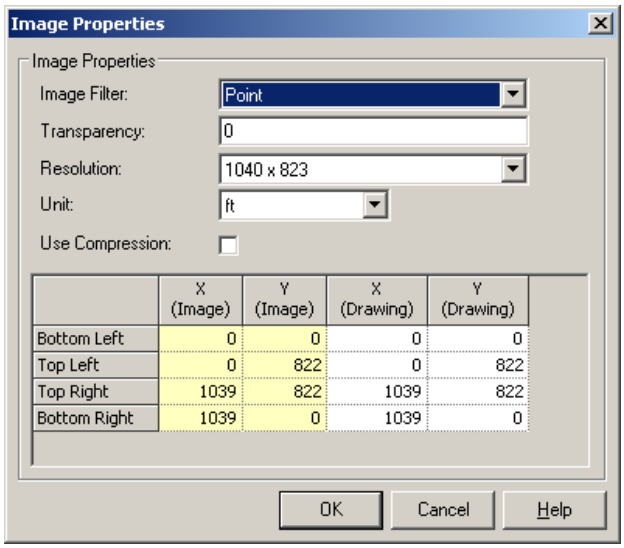


2. Save the image using File > Save > Save Image and assign the image a file name.

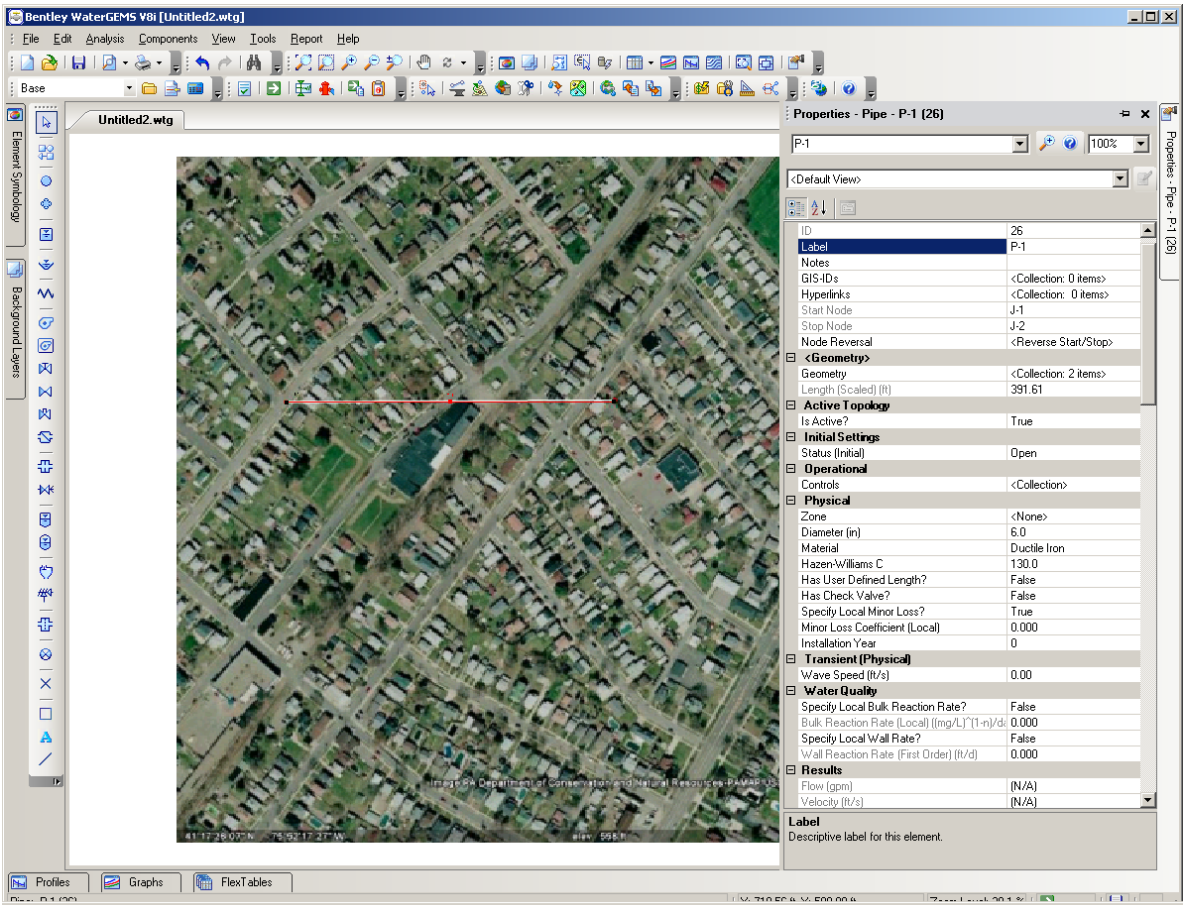
3. Open WaterCAD V8*i* and create a new project.
4. Import the file as a background using View > Background > New > New File. Browse to the image file and pick Open.



5. You will see the default image properties for this drawing. Write down the values in the first two columns of the lower pane and Select OK.



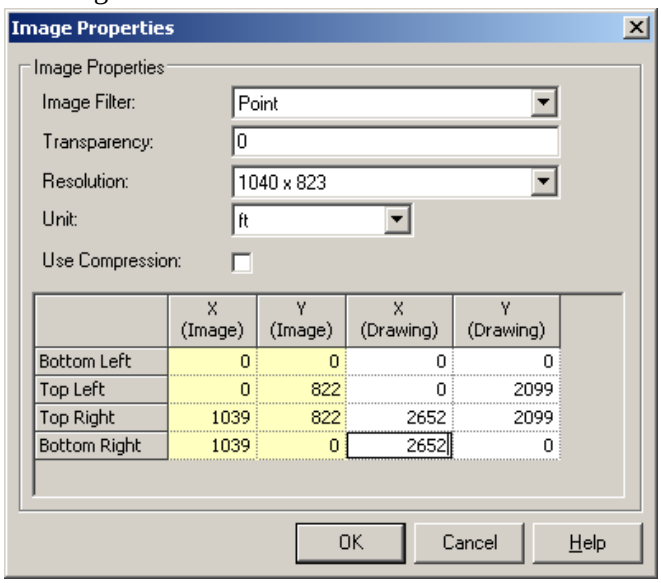
6. The background file will open in the model with the scale line showing. Zoom to that scaled line. Draw a pipe as close the exact length as the scale line as possible. Look at the Length (scaled) property of that line. (In this example it is 391.61 ft.) This means that the background needs to be scaled by a factor of 1000/391.61 = 2.553.



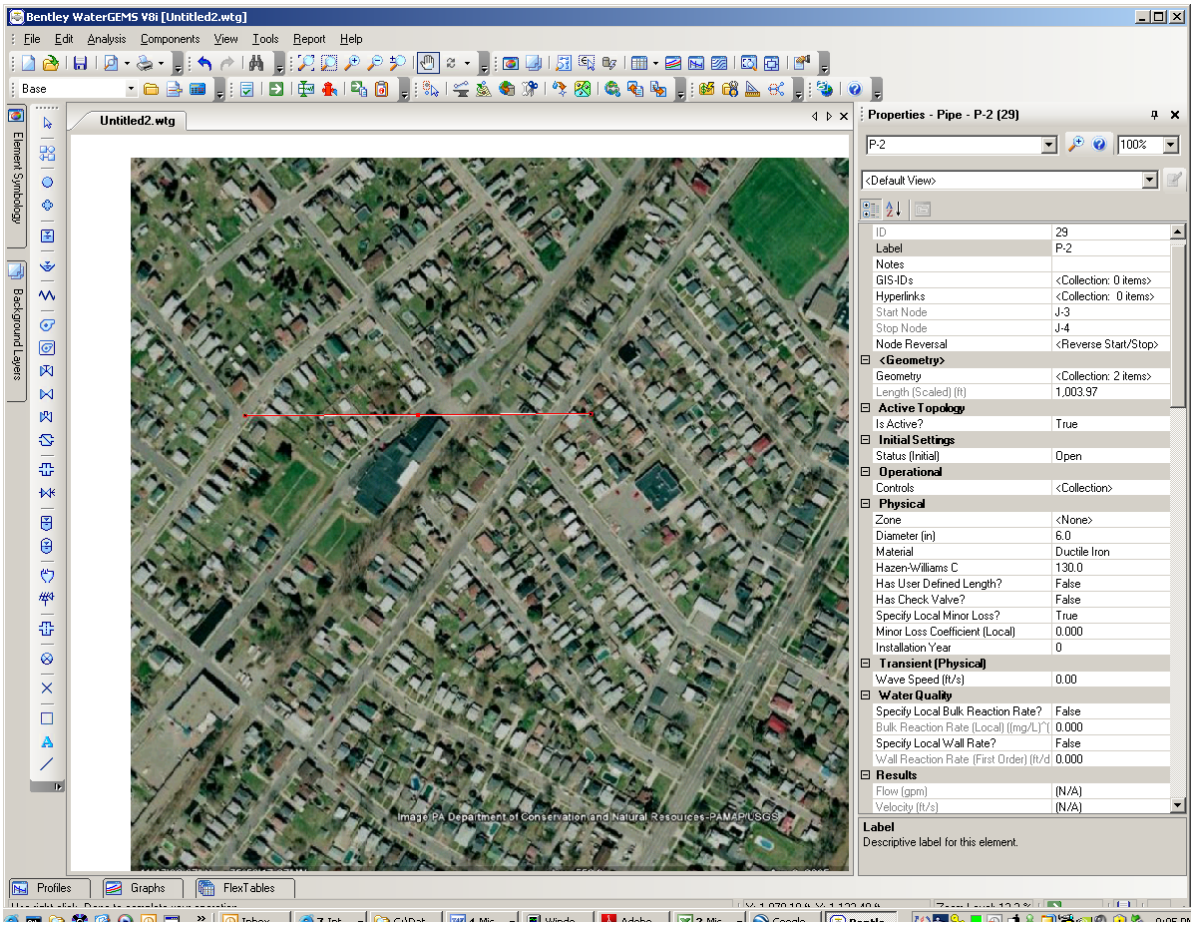
7. Close the background image by selecting View > Background > Delete and Yes. Delete the pipe and any end nodes.

8. Reopen the background image using View > Background > New > New File. This time do not accept the default scale. Instead multiply the values in the two right-most (image) columns by the scale factor determined in step 6 to obtain the values

in the two leftmost columns (drawing). For example, the scale factor was (2.553) to the Y value for the top left corner becomes 822 x 2.553 = 2099. Fill in all the image values.



9. The image will appear at the correct (approximate) scale. This can be checked by drawing a pipe on top of the scale line in the background image. The Length (scaled) of the pipe should be nearly the same as the length of the scale line. Delete than line and any nodes at the end points.



10. The model is now roughly scaled. Remember that the lengths determined this way are not survey accuracy and are as accurate as the care involved in measuring lengths. They may be off by a few percent which may be acceptable for some applications.

Creating Models

4

Starting a Project

Elements and Element Attributes

Adding Elements to Your Model

Manipulating Elements

Editing Element Attributes

Using Named Views

Using Selection Sets

Using the Network Navigator

Using Prototypes

Zones

Engineering Libraries

Hyperlinks

Using Queries

User Data Extensions

Starting a Project

When you first start Bentley WaterCAD V8*i* , the Welcome dialog box opens.

The Welcome dialog box contains the following controls:

Quick Start Lessons	Opens the online help to the Quick Start Lessons Overview topic.
Create New Project	Creates a new WaterCAD V8*i* project. When you click this button, an untitled Bentley WaterCAD V8*i* project is created.
Open Existing Project	Opens an existing project. When you click this button, a Windows browse dialog box opens allowing you to browse to the project to be opened. If you have ProjectWise installed and integrated with WaterCAD V8*i*, you are prompted to log into a ProjectWise datasource if you are not already logged in.
Show This Dialog at Start	When selected, the Welcome dialog box opens whenever you start Bentley WaterCAD V8*i* . Turn off this box if you do not want the Welcome dialog box to open whenever you start Bentley WaterCAD V8*i* .

To Access the Welcome Dialog During Program Operation

Click the Help menu and select the Welcome Dialog command.

To Disable the Automatic Display of the Welcome Dialog Upon Startup

In the Welcome dialog, turn off the box labeled Show This Dialog at Start.

To Enable the Automatic Display of the Welcome Dialog Upon Startup

In the Welcome dialog, turn on the box labeled Show This Dialog at Start.

Bentley WaterCAD V8*i* Projects

All data for a model are stored in WaterCAD V8*i* as a project. WaterCAD V8*i* project files have the file name extension **.**wtg. You can assign a title, date, notes and other identifying information about each project using the Project Properties dialog box. You can have up to five WaterCAD V8*i* projects open at one time.

To Start a New Project

To start a new project, choose File > New or press <Ctrl+N>. An untitled project is opened in the drawing pane.

To Open an Existing Project

To open an existing project, choose File > Open or press <Ctrl+O>. A dialog box opens allowing you to browse for the project you want to open.

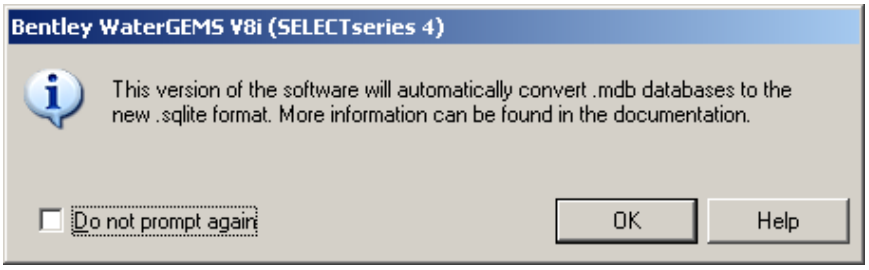
To Switch Between Multiple Projects

To switch between multiple open projects, select the appropriate tab at the top of the drawing pane. The file name of the project is displayed on the tab.

Database Format Conversion

This version of the software includes a change in the database format used to store modeling data. Microsoft Access .sqlite files will be automatically converted to the new .sqlite format when they are opened. Existing .sqlite files will be left untouched after the conversion. New files will be only created in this new format.

Upon program startup the following prompt is displayed:



The new .sqlite database format brings the following benefits:

- Smaller database file-size (50% reduction in average).
- Greatly increased file-size limit (2 TBs).
- Better overall performance.
- No conflicts with Microsoft Office.

Keep in mind that:

- Older versions of this software are not able to read .sqlite files.
- After conversion, .sqlite files will not be accessed/needed for the usage of this software. It is still a good practice to keep existing .sqlites as data back-ups/ history tracking.
- .sqlite files will be added automatically to existing and new ProjectWise sets.

Setting Project Properties

The Project Properties dialog box allows you to enter project-specific information to help identify the project. Project properties are stored with the project.

The dialog box contains the following text fields and controls:

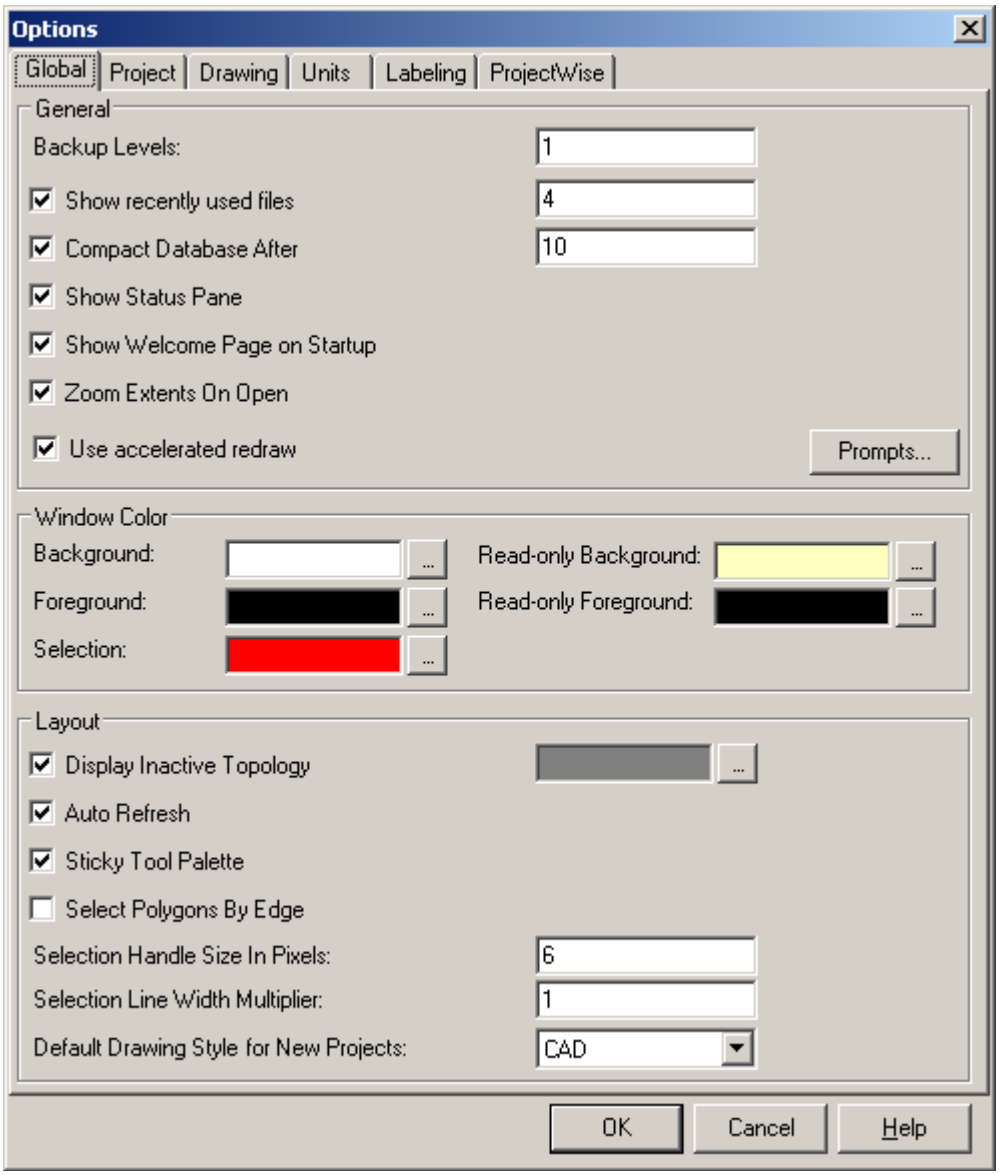
Title	Enter a title for the project.
File Name	Displays the file name for the current project. If you have not saved the project yet, the file name is listed as “Untitled*x*.wtg.”, where *x* is a number between 1 and 5 chosen by the program based on the number of untitled projects that are currently open.
Engineer	Enter the name of the project engineer.
Company	Enter the name of your company.
Date	Click this field to display a calendar, which is used to set a date for the project.
Notes	Enter additional information about the project.

To set project properties

1. Choose File > Project Properties and the Project Properties dialog box opens.
2. Enter the information in the Project Properties dialog box and click **OK**.

Setting Options

You can change global settings for WaterCAD V8*i* in the Options dialog box. Choose Tools > Options. The Options dialog box contains different tabs where you can change settings.



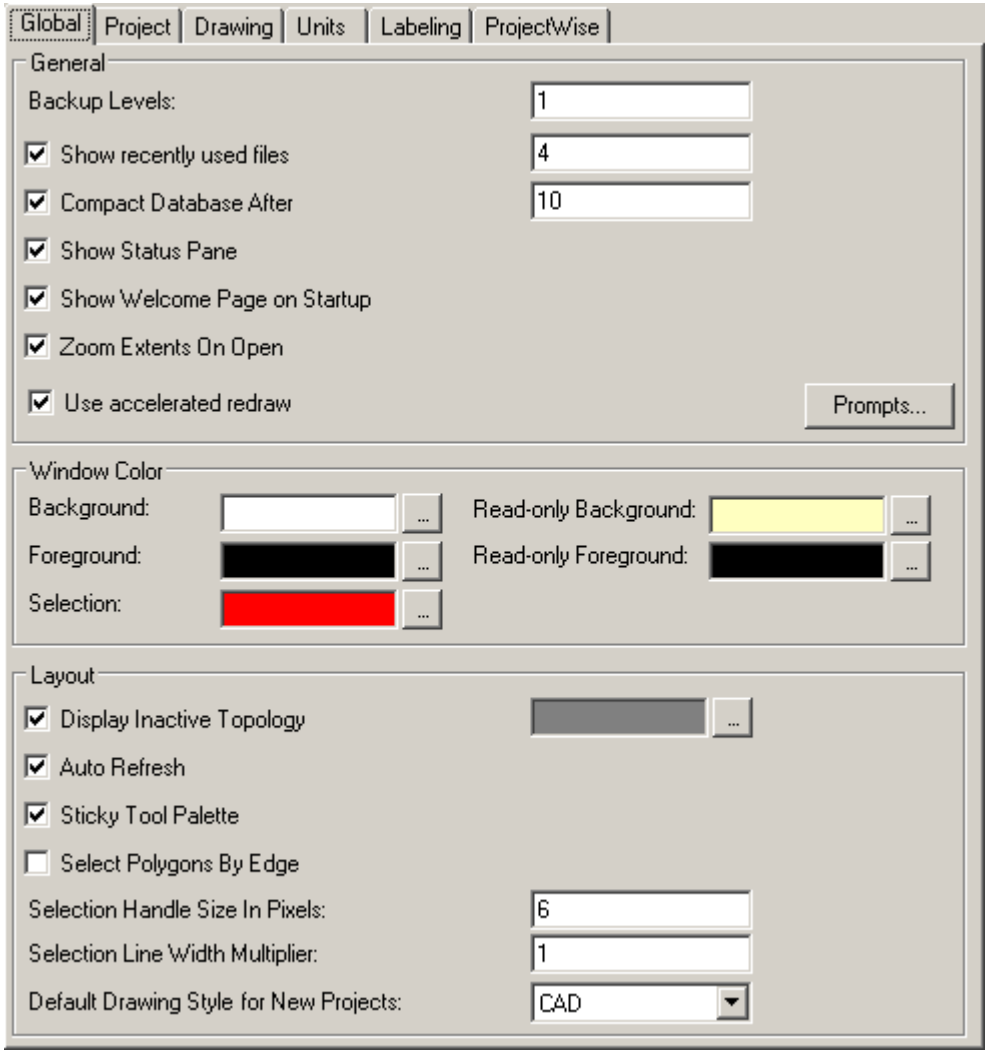
Click one of the following links to learn more about the Options dialog box:

- Options Dialog Box - Global Tab
- Options Dialog Box - Project Tab
- Options Dialog Box - Drawing Tab

- Options Dialog Box - Units Tab
- Options Dialog Box - Labeling Tab
- Options Dialog Box - ProjectWise Tab

Options Dialog Box - Global Tab

The Global tab changes general program settings for the WaterCAD V8*i* stand-alone editor, including whether or not to display the status pane, as well as window color and layout settings.



The Global tab contains the following controls:

General Settings

Backup Levels

Indicates the number of backup copies that are retained when a project is saved. The default value is 1.

Note: The higher this number, the more .BAK files (backup files) are created, thereby using more hard disk space on your computer.

Show Recently Used Files

When selected, activates the recently opened files display at the bottom of the File menu. This check box is turned on by default. The number of recently used files that are displayed depends on the number specified here.

Show Status Pane

When turned on, activates the Status Pane display at the bottom of the WaterCAD V8*i* stand-alone editor. This check box is turned on by default.

Show Welcome Page on Startup

When turned on, activates the Welcome dialog that opens when you first start WaterCAD V8*i*. This check box is turned on by default.

Zoom Extents On Open

When turned on, a Zoom Extents is performed automatically in the drawing pane.

Use accelerated redraw

Some video cards use "triple buffering", which we do not support at this time. If you see anomalies in the drawing (such as trails being left behind from the selection rectangle), then you can shut this option off to attempt to fix the problem. However, when this option is off, you could see some performance degradation in the drawing.

Prompts

Opens the Stored Prompt Responses dialog, which allows you to change the behavior of the default prompts (messages that appear allowing you to confirm or cancel certain operations).

Window Color

Background Color — Displays the color that is currently assigned to the drawing pane background. You can change the color by clicking the ellipsis (...) to open the Color dialog box.

Foreground Color — Displays the color that is currently assigned to elements and labels in the drawing pane. You can change the color by clicking the ellipsis (...) to open the Color dialog box.

Read Only Background Color — Displays the color that is currently assigned to read-only data field backgrounds. You can change the color by clicking the ellipsis (...) to open the Color dialog box.

Read Only Foreground Color — Displays the color that is currently assigned to read-only data field text. You can change the color by clicking the ellipsis (...) to open the Color dialog box.

Selection Color — Displays the color that is currently applied to highlighted elements in the drawing pane. You can change the color by clicking the ellipsis (...) to open the Color dialog box.

Layout

Display Inactive Topology — When turned on, activates the display of inactive elements in the drawing pane in the color defined in Inactive Topology Line Color. When turned off, inactive elements will not be visible in the drawing pane. This check box is turned on by default.

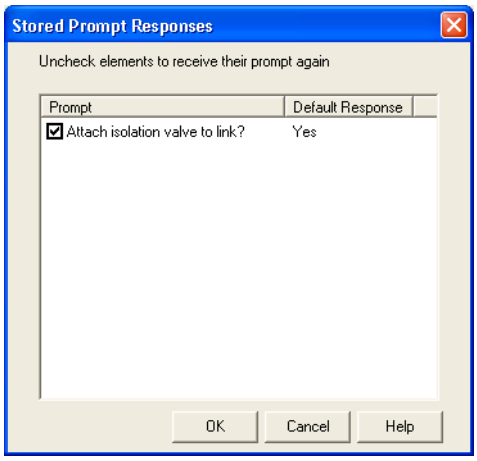
Inactive Topology Line Color — Displays the color currently assigned to inactive elements. You can change the color by clicking the ellipsis (...) to open the Color dialog box.

Auto Refresh — Activates Auto Refresh. When Auto Refresh is turned on, the drawing pane automatically updates whenever changes are made to the WaterCAD V8*i* datastore. This check box is turned off by default.

Sticky Tool Palette	When turned on, activates the Sticky Tools feature. When Sticky Tools is turned on, the drawing pane cursor does not reset to the Select tool after you create a node or finish a pipe run in your model, allowing you to continue dropping new elements into the drawing without re-selecting the tool. When Sticky Tools is turned off, the drawing pane cursor resets to the Select tool after you create a node. This check box is selected by default.
Select Polygons By Edge	When this box is checked, polygon elements (catchments) can only be selected in the drawing pane by clicking on their bordering line, in other words you cannot select polygons by clicking their interior when this option is turned on.
Selection Handle Size In Pixels	Specifies, in pixels, the size of the handles that appear on selected elements. Enter a number from 1 to 10.
Selection Line Width Multiplier	Increases or decreases the line width of currently selected link elements by the factor indicated. For example, a multiplier of 2 would result in the width of a selected link being doubled.
Default Drawing Style	Allows you to select GIS or CAD drawing styles. Under GIS style, the size of element symbols in the drawing pane will remain the same regardless of zoom level. Under CAD style, element symbols will appear larger or smaller depending on zoom level.

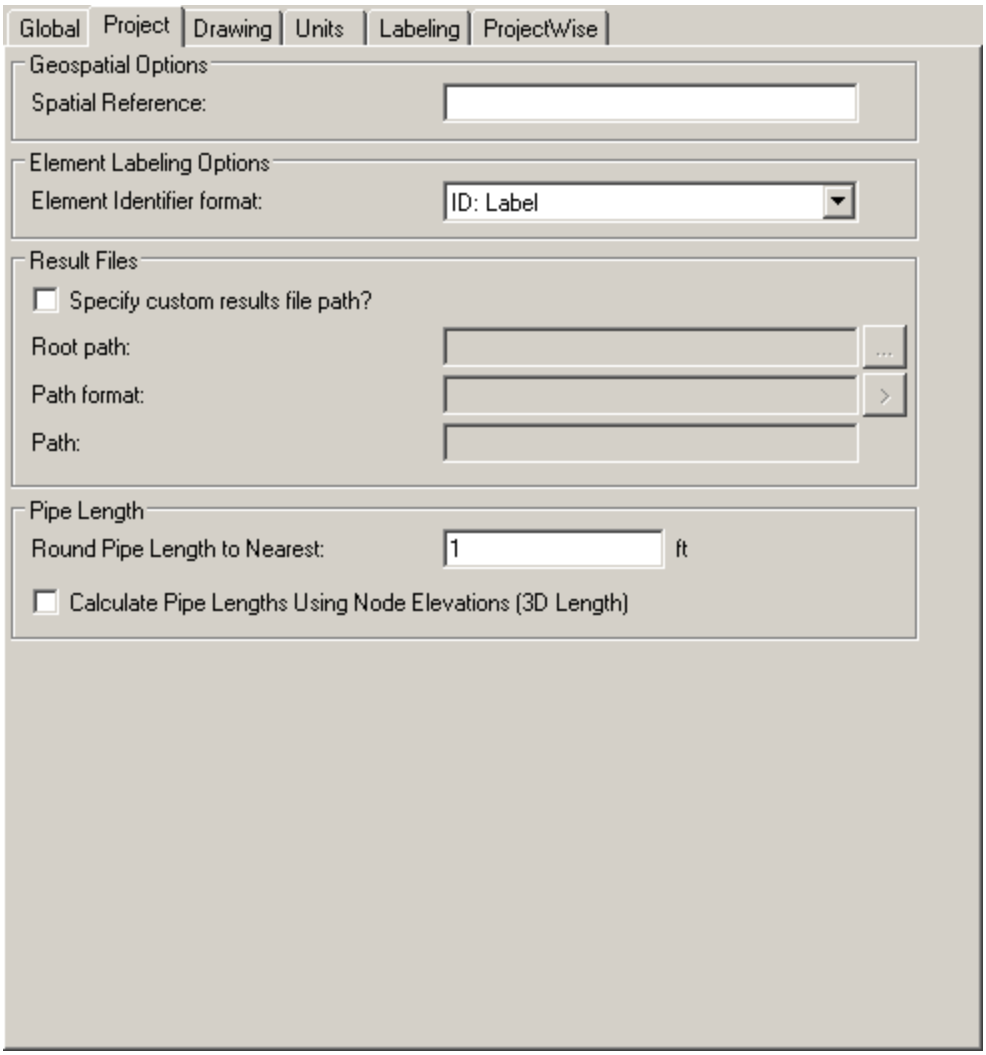
Stored Prompt Responses Dialog Box

This dialog allows you to change the behavior of command prompts back to their default settings. Som,e commands trigger a command prompt that can be suppressed by using the Do Not Prompt Again check box. You can turn the prompt back on by accessing this dialog and unchecking the box for that prompt type.



Options Dialog Box - Project Tab

This tab contains miscellaneous settings. You can set pipe length calculation, spatial reference, label display, and results file options in this tab.



The Project tab contains the following controls:

Geospatial Options

Spatial Reference	Used for integration with Projectwise. Can leave the field blank if there is no spatial information.

Element Identifier Options

Element Identifier Format	Specifies the format in which reference fields are used. Reference fields are fields that link to another element or support object (pump definitions, patterns, controls, zones, etc.).

Result Files

Specify Custom Results File Path?	When checked, allows you to edit the results file path and format by enabling the other controls in this section.
Root Path	Allows you to specify the root path where results files are stored. You can type the path manually or choose the path from a Browse dialog by clicking the ellipsis (...) button.
Path Format	Allows you to specify the complete path that you wish to use for storing your result files for the current project. You can type the path manually and/or use predefined attributes from the menu accessed with the [>] button. One of the predefined choices is the Root Path. It is recommended that you start building your Path Format with this Root Path choice. Then optionally extend this path with the other predefined choices.
Path	Displays a dynamically updated view of the custom result file path based on the settings in the Root Path and Path Format fields

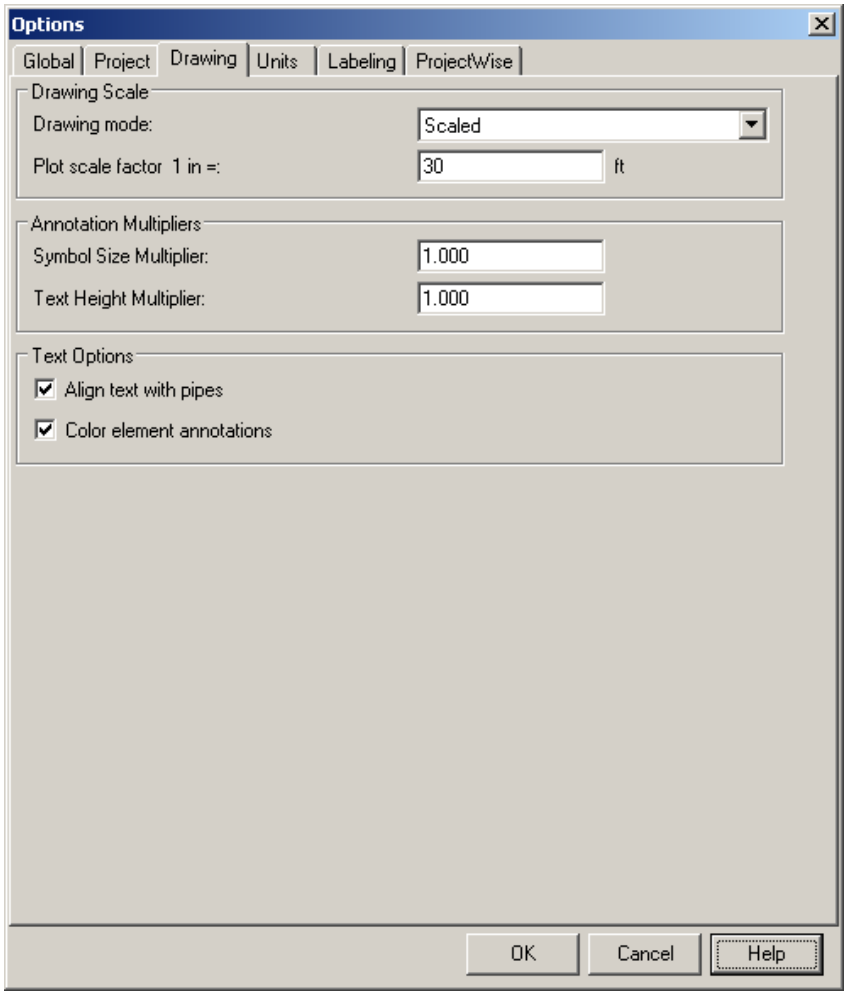
Pipe Length

Round Pipe Length to Nearest	The program will round to the nearest unit specified in this field when calculating scaled pipe length

Calculate Pipe Lengths Using Node Elevations (3D Length)	When checked, includes differences in Z (elevation) between pipe ends when calculating pipe length.

Options Dialog Box - Drawing Tab

This tab contains drawing layout and display settings. You can set the scale that you want to use as the finished drawing scale for the plan view output. Drawing scale is based upon engineering judgment and the destination sheet sizes to be used in the final presentation.



The Drawing tab contains the following controls:

Drawing Scale

Drawing Mode	Selects either Scaled or Schematic mode for models in the drawing pane.
Horizontal Scale Factor 1 in. =:	Controls the scale of the plan view.

Annotation Multipliers

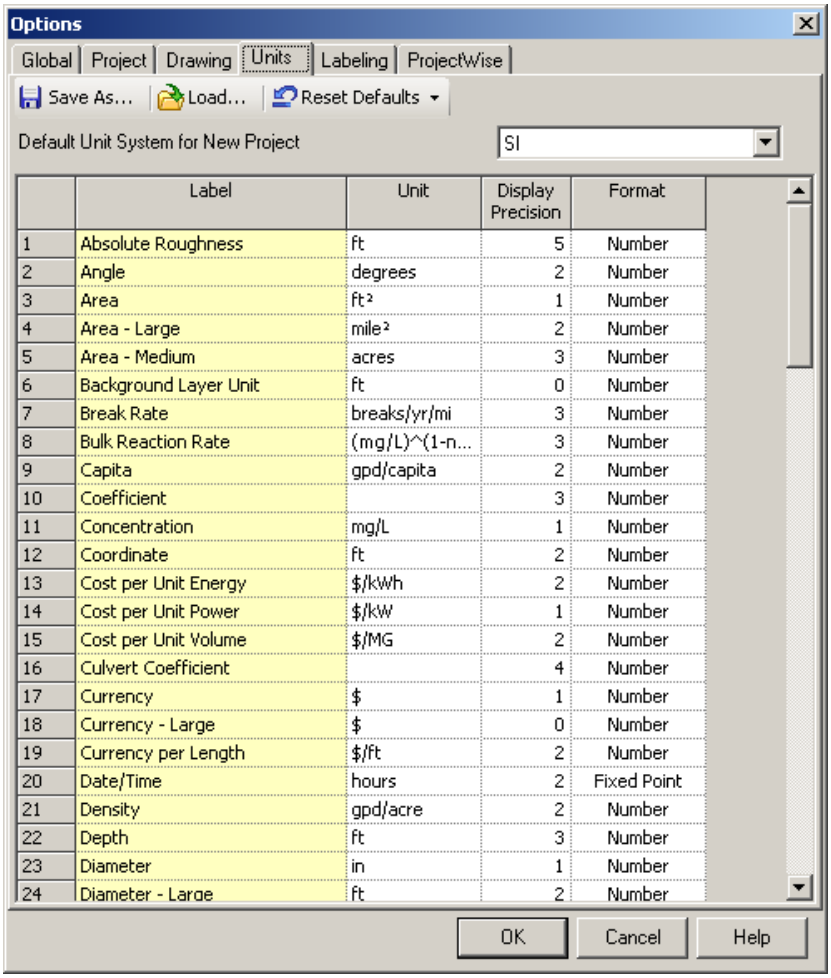
Symbol Size Mulitplier	Increases or decreases the size of your symbols by the factor indicated. For example, a multiplier of 2 would result in the symbol size being doubled. The program selects a default symbol height that corresponds to 4.0 ft. (approximately 1.2 m) in actual-world units, regardless of scale.
Text Height Multiplier	Increases or decreases the default size of the text associated with element labeling by the factor indicated. The program automatically selects a default text height that displays at approximately 2.5 mm (0.1 in) high at the user-defined drawing scale. A scale of 1.0 mm = 0.5 m, for example, results in a text height of approximately 1.25 m. Likewise, a 1 in. = 40 ft. scale equates to a text height of around 4.0 ft.

Text Options

Align Text with Pipes	Turns text alignment on and off. When it is turned on, labels are aligned to their associated pipes. When it is turned off, labels are displayed horizontally near the center of the associated pipe.
Color Element Annotations	When this box is checked, color coding settings are applied to the element annotation.

Options Dialog Box - Units Tab

The Units tab modifies the unit settings for the current project.



	Label	Unit	Display Precision	Format
1	Absolute Roughness	ft	5	Number
2	Angle	degrees	2	Number
3	Area	ft²	1	Number
4	Area - Large	mile²	2	Number
5	Area - Medium	acres	3	Number
6	Background Layer Unit	ft	0	Number
7	Break Rate	breaks/yr/mi	3	Number
8	Bulk Reaction Rate	(mg/L)^(1-n...	3	Number
9	Capita	gpd/capita	2	Number
10	Coefficient		3	Number
11	Concentration	mg/L	1	Number
12	Coordinate	ft	2	Number
13	Cost per Unit Energy	$/kWh	2	Number
14	Cost per Unit Power	$/kW	1	Number
15	Cost per Unit Volume	$/MG	2	Number
16	Culvert Coefficient		4	Number
17	Currency	$	1	Number
18	Currency - Large	$	0	Number
19	Currency per Length	$/ft	2	Number
20	Date/Time	hours	2	Fixed Point
21	Density	gpd/acre	2	Number
22	Depth	ft	3	Number
23	Diameter	in	1	Number
24	Diameter - Large	ft	2	Number

The Units tab contains the following controls:

Save As	Saves the current unit settings as a separate .xml file. This file allows you to reuse your Units settings in another project. When the button is clicked, a Windows Save As dialog box opens, allowing you to enter a name and specify the directory location of the .xml file.
Load	Loads a previously created Units project .xml file, thereby transferring the unit and format settings that were defined in the previous project. When the button is clicked, a Windows Load dialog box opens, allowing you to browse to the location of the desired .xml file.
Reset Defaults - SI	Resets the unit and formatting settings to the original factory defaults for the System International (Metric) system.
Reset Defaults - US	Resets the unit and formatting settings to the original factory defaults for the Imperial (U.S.) system.
Default Unit System for New Project	Specifies the unit system that is used globally across the project. Note that you can locally change any number of attributes to the unit system other than the ones specified here.

Units Table

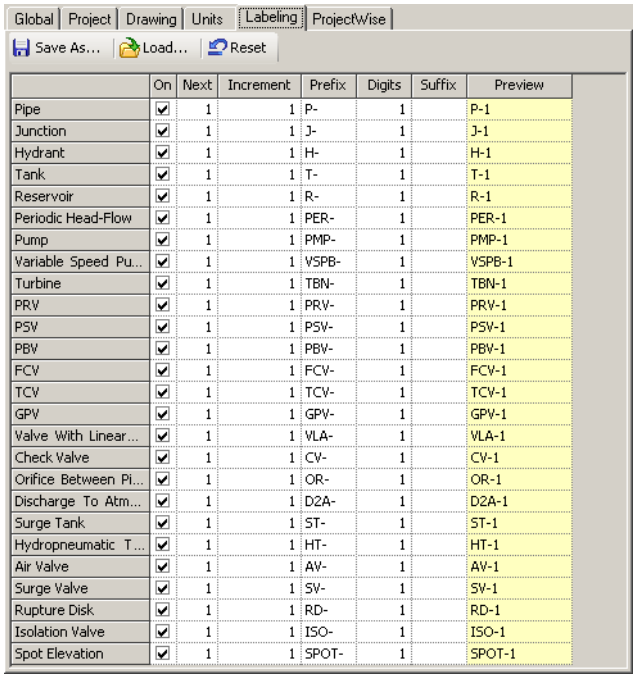
The units table contains the following columns:

- **Label**—Displays the parameter measured by the unit.
- **Unit**—Displays the type of measurement. To change the unit of an attribute type, click the choice list and click the unit you want. This option also allows you to use both U.S. customary and SI units in the same worksheet.
- **Display Precision**—Sets the rounding of numbers and number of digits displayed after the decimal point. Enter a number from 0 to 15 to indicate the number of digits after the decimal point.
- **Format Menu**—Selects the display format used by the current field. Choices include:
  - **Scientific**—Converts the entered value to a string of the form "-d.ddd...E+ddd" or "-d.ddd...e+ddd", where each 'd' indicates a digit (0-9). The string starts with a minus sign if the number is negative.
  - **Fixed Point**—Abides by the display precision setting and automatically enters zeros after the decimal place to do so. With a display precision of 3, an entered value of 3.5 displays as 3.500.
  - **General**—Truncates any zeros after the decimal point, regardless of the display precision value. With a display precision of 3, the value that would appear as 5.200 in Fixed Point format displays as 5.2 when using General format. The number is also rounded. So, an entered value of 5.35 displays as 5.4, regardless of the display precision.
  - **Number**—Converts the entered value to a string of the form "-d,ddd,ddd.ddd...", where each 'd' indicates a digit (0-9). The string starts with a minus sign if the number is negative. Thousand separators are inserted between each group of three digits to the left of the decimal point.

Note: The conversion for pressure to ft. (or m) H20 uses the specific gravity of water at 4C (39F), or a specific gravity of 1. Hence, if the fluid being used in the simulation uses a specific gravity other than 1, the sum of the pressure in ft. (or m) H20 and the node elevation will not be exactly equal to the calculated hydraulic grade line (HGL).

Options Dialog Box - Labeling Tab

The Element Labeling tab is used to specify the automatic numbering format of new elements as they are added to the network. You can save your settings to an .xml file for later use.



	On	Next	Increment	Prefix	Digits	Suffix	Preview
Pipe	☑	1	1	P-	1		P-1
Junction	☑	1	1	J-	1		J-1
Hydrant	☑	1	1	H-	1		H-1
Tank	☑	1	1	T-	1		T-1
Reservoir	☑	1	1	R-	1		R-1
Periodic Head-Flow	☑	1	1	PER-	1		PER-1
Pump	☑	1	1	PMP-	1		PMP-1
Variable Speed Pu...	☑	1	1	VSPB-	1		VSPB-1
Turbine	☑	1	1	TBN-	1		TBN-1
PRV	☑	1	1	PRV-	1		PRV-1
PSV	☑	1	1	PSV-	1		PSV-1
PBV	☑	1	1	PBV-	1		PBV-1
FCV	☑	1	1	FCV-	1		FCV-1
TCV	☑	1	1	TCV-	1		TCV-1
GPV	☑	1	1	GPV-	1		GPV-1
Valve With Linear...	☑	1	1	VLA-	1		VLA-1
Check Valve	☑	1	1	CV-	1		CV-1
Orifice Between Pi...	☑	1	1	OR-	1		OR-1
Discharge To Atm...	☑	1	1	D2A-	1		D2A-1
Surge Tank	☑	1	1	ST-	1		ST-1
Hydropneumatic T...	☑	1	1	HT-	1		HT-1
Air Valve	☑	1	1	AV-	1		AV-1
Surge Valve	☑	1	1	SV-	1		SV-1
Rupture Disk	☑	1	1	RD-	1		RD-1
Isolation Valve	☑	1	1	ISO-	1		ISO-1
Spot Elevation	☑	1	1	SPOT-	1		SPOT-1

The Element Labeling tab contains the following controls:

Save As	Saves your element labeling settings to an element label project file, which is an. xml file.
Load	Opens an existing element label project file.
Reset	Assigns the correct Next value for all elements based on the elements currently in the drawing and the user-defined values set in the Increment, Prefix, Digits, and Suffix fields of the Labeling table.

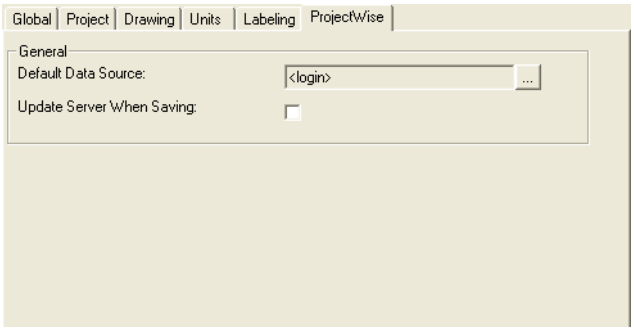
Labeling Table

The labeling table contains the following columns:

- **Element**—Shows the type of element to which the label applies.
- **On**—Turns automatic element labeling on and off for the associated element type.
- **Next**—Type the integer you want to use as the starting value for the ID number portion of the label. Bentley WaterCAD V8*i* generates labels beginning with this number and chooses the first available unique label.
- **Increment**—Type the integer that is added to the ID number after each element is created to yield the number for the next element.
- **Prefix**—Type the letters or numbers that appear in front of the ID number for the elements in your network.
- **Digits**—Type the minimum number of digits that the ID number has. For instance, 1, 10, and 100 with a digit setting of two would be 01, 10, and 100.
- **Suffix**—Type the letters or numbers that appear after the ID number for the elements in your network.
- **Preview**—Displays what the label looks like based on the information you have entered in the previous fields.

Options Dialog Box - ProjectWise Tab

The ProjectWise tab contains options for using WaterCAD V8*i* with ProjectWise.



This tab contains the following controls:

Default Datasource	Displays the current ProjectWise datasource. If you have not yet logged into a datasource, this field will display <login>. To change the datasource, click the **Ellipses (...)** to open the Change Datasource dialog box. If you click **Cancel** after you have changed the default datasource, the new default datasource is retained.
Update server on Save	When this is turned on, any time you save your WaterCAD V8*i* project locally using the File > Save menu command, the files on your ProjectWise server will also be updated and all changes to the files will immediately become visible to other ProjectWise users. This option is turned off by default. **Note: This option, when turned on, can significantly affect performance, especially for large, complex projects.**

Note: These settings affect ProjectWise users only.

For more information about ProjectWise, see the Working with ProjectWise topic.

Working with ProjectWise

Bentley ProjectWise provides managed access to WaterCAD V8*i* content within a workgroup, across a distributed organization, or among collaborating professionals. Among other things, this means that only one person is allowed to edit the file at a time, and document history is tracked. When a WaterCAD V8*i* project is stored using ProjectWise, project files can be accessed quickly, checked out for use, and checked back in directly from within WaterCAD V8*i*. With ProjectWise Explorer, it is possible to read the file's audit trail to determine who edited the file and when that occurred.

If ProjectWise Explorer is installed on your computer, WaterCAD V8*i* automatically installs all the components necessary for you to use ProjectWise to store and share your WaterCAD V8*i* projects. A WaterCAD V8*i* project consists of a *.wtg file, a *.wtg.sqlite file, and in the case of a standalone model a *.dwh file.

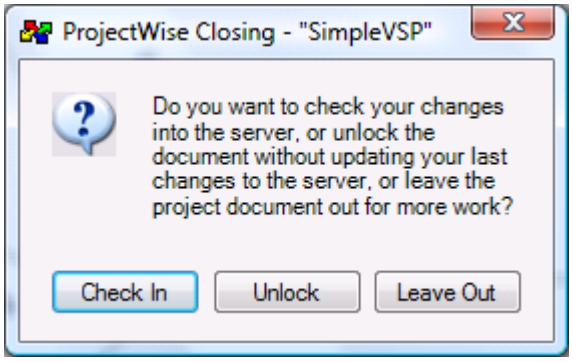
To learn more about ProjectWise, refer to the ProjectWise online help.

ProjectWise and Bentley WaterCAD V8*i*

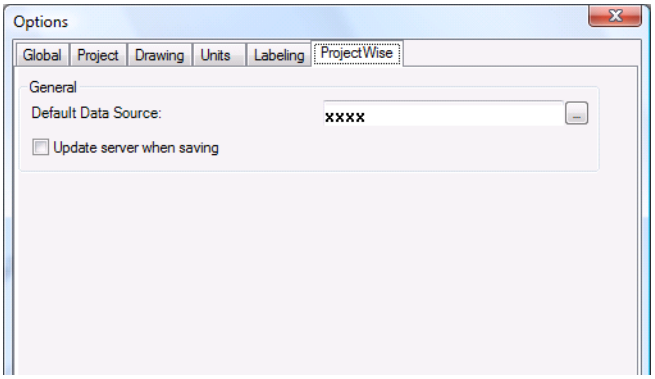
Follow these guidelines when using WaterCAD V8*i* with ProjectWise:

- ProjectWise integration must be enabled before WaterCAD V8*i* can directly interact with ProjectWise. Refer to the "Setting up ProjectWise Integration" section for more details.
- Once ProjectWise integration is enabled, use the normal Open/Save commands to access the ProjectWise datasources. A Datasource refers to a collection of folders and documents set up by the ProjectWise Administrator. The File > Open operation, for example, will first show the ProjectWise file browser, where you can open a project that is already saved into ProjectWise. File > SaveAs can be used to save any project into ProjectWise, whether it exists in ProjectWise or locally on your system's disk.
- The first time the ProjectWise prompt is opened in your current WaterCAD V8*i* session, you are prompted to log into a ProjectWise datasource. The datasource you log into remains the current datasource until you change it via the ProjectWise tab of the Global Options in WaterCAD V8*i* Tools. The user needs to know the name of the Datasource, a user name and a password.
- If a project is opened from ProjectWise, then all subsequent open/save operations will prompt to open/save the file to ProjectWise first. At the ProjectWise prompt you can click the Cancel button to get a Windows file browse prompt if you want to pick a file on your local system or network. This applies to cases like import/export, as well as any other file selection operation such as picking a file for ModelBuilder to use, or referencing a file with Hyperlinks. If the current project is not opened from ProjectWise however, you will only be allowed to choose files on your local system or network.
- Use the WaterCAD V8*i* File > New command to create a new project. The project is not stored in ProjectWise until you perform a File > Save As operation.
- Use the WaterCAD V8*i* File > Save command to save a copy of the current project to your local computer.
- When you Close a project already stored in ProjectWise using File > Close, you are prompted to select one of the following options:
  - **Check In**—Updates the project files in ProjectWise with your latest changes and unlocks the project so other ProjectWise users can edit it.
  - **Unlock**—Unlocks the project files so other ProjectWise users can edit it but does not update the project in ProjectWise. Note that this will abandon any changes you have made since the last Check-in command.

– **Leave Out**—Leaves the project checked out so others cannot edit it and retains any changes you have made since the last server update to the files on your local computer. Select this option if you want to exit Bentley WaterCAD V8*i* but continue working on the project later. The project files may be synchronized when the files are checked in later.



- In the WaterCAD V8*i* Options dialog box, there is a ProjectWise tab with a **Update server on Save** check box. This option, when turned on, can significantly affect performance, especially for large, complex projects. When this is checked, any time you save your WaterCAD V8*i* project locally using the File > Save menu command, the files on your ProjectWise server will also be updated and all changes to the files will immediately become visible to other ProjectWise users. This option is turned off by default, which means the ProjectWise server version of the project will not be updated until the files are checked in.



- Use the File > Update Server Copy command to update the files on your ProjectWise server with all changes made to the files, which will immediately become visible to other ProjectWise users. Note that this command saves the project and any edits that have been made before it updates the ProjectWise files.
- In the SS2 release of WaterCAD V8*i*, calculation result files are not managed inside ProjectWise. A local copy of results is maintained on the user's computer, but to ensure accurate results the user should recalculate desired scenarios for projects when the user first opens them from ProjectWise.

- WaterCAD V8*i* projects associated with ProjectWise appear in the Most Recently Used Files list (at the bottom of the File menu) in the following format:

  pw://PointServer:_TestDatasource/Documents/TestFolder/Test1

Performing ProjectWise Operations from within WaterCAD V8*i*

You can quickly tell whether or not the current WaterCAD V8*i* project is in ProjectWise or not by looking at the title bar and the status bar of the WaterCAD V8*i* window. If the current project is in ProjectWise, “pw://” will appear in front of the file name in the title bar, and a ProjectWise icon will appear on the far right side of the status bar, as shown below.



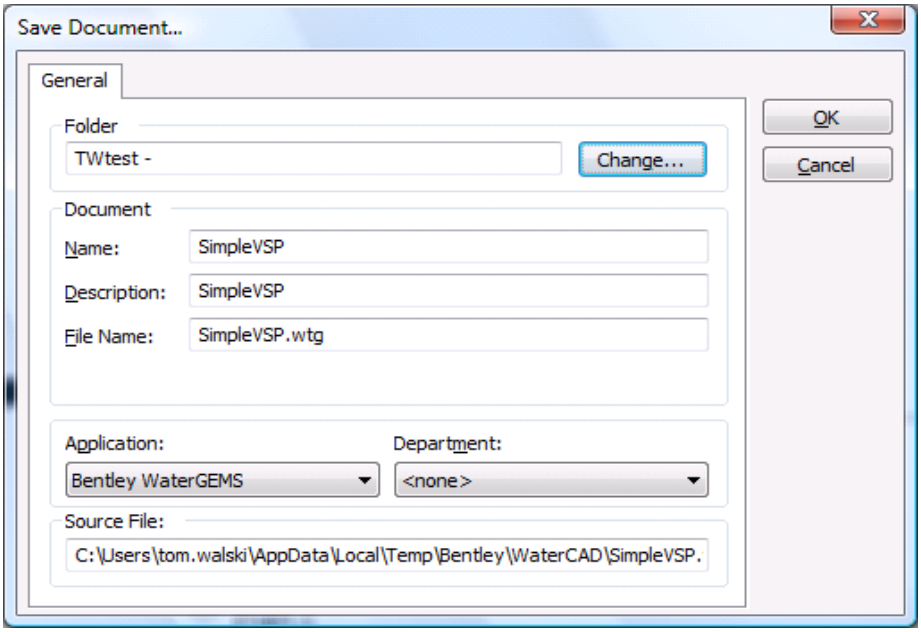
If you have enabled ProjectWise integration, you can perform the following ProjectWise operations from within WaterCAD V8*i*:

To save an open WaterCAD V8*i* project to ProjectWise

1. In WaterCAD V8*i*, select **File > Save As**.
2. If you haven’t already logged into ProjectWise, you are prompted to do so. Select a ProjectWise datasource, type your ProjectWise user name and password, then click **Log in**.
3. In the ProjectWise Save Document dialog box, enter the following information:
   a. Click **Change** next to the Folder field, then select a folder in the current ProjectWise datasource in which to store your project.
   b. Type the name of your WaterCAD V8*i* project in the Name field. It is best to keep the ProjectWise name the same as or as close to the WaterCAD V8*i* project name as possible.

c. Keep the default entries for the rest of the fields in the dialog box.

d. Click **OK**. There will be two new files in ProjectWise; a *.wtg and a *.wtg.sqlite.

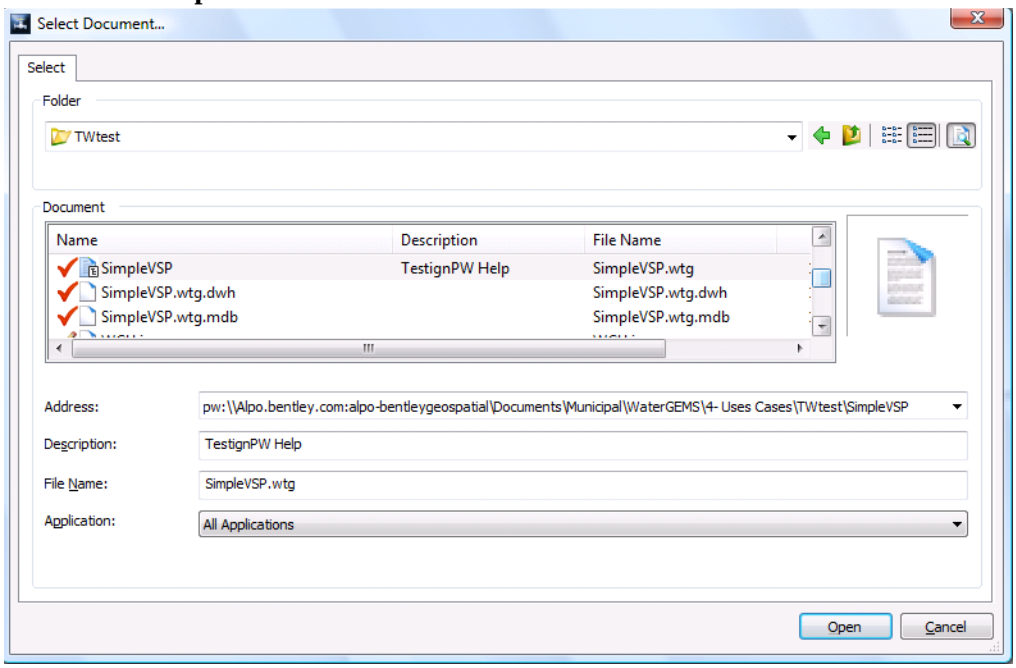


To open a WaterCAD V8*i* project from a ProjectWise datasource from within WaterCAD V8*i*

1. Select **File > Open**.
2. If you haven't already logged into ProjectWise, you are prompted to do so. Select a ProjectWise datasource, type your ProjectWise user name and password, then click **Log in**.
3. In the ProjectWise Select Document dialog box, perform these steps:
   a. From the Folder drop-down menu, select a folder that contains WaterCAD V8*i* projects.
   b. In the Document list box, select a WaterCAD V8*i* project.

c. Keep the default entries for the rest of the fields in the dialog box.

d. Click **Open**.



To open a WaterCAD V8*i* project from ProjectWise, it is also possible to double click on the project in ProjectWise.

To copy an open WaterCAD V8*i* project from one ProjectWise datasource to another

1. Select File > Open to open a project stored in ProjectWise.
2. Go to Tools > Options, and on the ProjectWise tab click to change the default datasource.
3. In the ProjectWise Log in dialog box, select a different ProjectWise datasource, then click Log in.
4. Select File > Save As.
5. In the ProjectWise Save Document dialog box, change information about the project as required, then click OK.

To make a local copy of a WaterCAD V8*i* project stored in a ProjectWise datasource

1. Select File > Open.
2. If you haven't already logged into ProjectWise, you are prompted to do so. Select a ProjectWise datasource, type your ProjectWise user name and password, then click Log in.

3. Select File > Save As.

At the ProjectWise save prompt click Cancel.

4. Save the WaterCAD V8*i* project to a folder on your local computer.

To change the default ProjectWise datasource

1. Start WaterCAD V8*i*.
2. Select **Tools > Options> ProjectWise** tab**.**
3. Change the Default Datasource to the one you want to log into.

To use background layer files with ProjectWise

- Using File > Save As—If there are background files assigned to the model, the user is prompted with two options: copy the background layer files to the project folder for use by the project, or remove the background references and manually reassign them once the project is in ProjectWise to other existing ProjectWise documents.
- Using File > Open—Using this method, background layer files are not locked in ProjectWise for the current user to edit. The files are intended to be shared with other users at the same time.

To add a background layer file reference to a project that exists in ProjectWise:

Using File > Save As—When you use File > Save As on a project that is already in ProjectWise and there are background layer files, you are prompted with two options: you can copy all the files to the local project folder for use by the project, or you can remove the background references and manually reassign them after you have saved the project locally.

Note: When you remove a background layer file reference from a project that exists in ProjectWise, the reference to the file is removed but the file itself is not deleted from ProjectWise.

Setting Up ProjectWise Integration

Before you may interact with ProjectWise from inside the WaterCAD V8*i* application, you must integrate it to work with ProjectWise. This step varies depending on the platform under which you wish to integrate. Until you set up this ProjectWise integration the file prompts in the application will not allow interaction with ProjectWise datasources.

For the Standalone platform, you must edit the ProjectWiseIntegrationLocalOptions.xml file using a text editor. The file is located in the All User documents directory:

In Windows XP: C:\Documents and Settings\All Users\Application Data\Bentley\\8

In Windows Vista/Windows 7/Windows 8: C:\ProgramData\Bentley\\8

Find the line that sets the PWDIR variable

```
PWDIR=""
```

and change it so that it refers to the directory where a supported version of the ProjectWise Explorer is installed, such as

```
PWDIR="C:\Program Files\Bentley\ProjectWise\"
```

For the MicroStation platform, you must enable the ProjectWise iDesktop integration for Microstation when installing the ProjectWise Explorer client software. You can also Change the ProjectWise Explorer installation to enable this from the Windows Control Panel.

About ProjectWise Geospatial

ProjectWise Geospatial gives spatial context to Municipal Products Group product projects in their original form. An interactive map-based interface allows users to navigate and retrieve content based upon location. The environment includes integrated map management, dynamic coordinate system support, and spatial indexing tools.

ProjectWise Geospatial supports the creation of named spatial reference systems (SRSs) for 2D or 3D cartesian coordinate systems, automatic transformations between SRSs, creation of Open GIS format geometries, definition of spatial locations, association of documents and folders with spatial locations, and the definition of spatial criteria for document searching.

A *spatial location* is the combination of a geometry for a project plus a designated SRS. It provides a universal mechanism for graphically relating ProjectWise documents and folders.

The ProjectWise administrator can assign background maps to folders, against which the contained documents or projects will be registered and displayed. For documents such as Municipal Products Group product projects, ProjectWise Geospatial can automatically retrieve the embedded spatial location. For documents that are nonspatial, the document can simply inherit the location of the folder into which it is inserted, or users can explicitly assign a location, either by typing in coordinates, or by drawing them.

Each document is indexed to a universal coordinate system or SRS, however, the originating coordinate system of each document is also preserved. This enables search of documents across the boundary of different geographic, coordinate, or engineering coordinate systems.

Custom geospatial views can be defined to display documents with symbology mapped to arbitrary document properties such as author, time, and workflow state.

For a complete description of how to work with ProjectWise Geospatial, for example how to add background maps and coordinate systems, see the *ProjectWise Geospatial Explorer Guide* and the *ProjectWise Geospatial Administrator Guide*.

Maintaining Project Geometry

A spatial location is comprised of an OpenGIS-format geometry plus a Spatial Reference System (SRS). For Municipal Products Group product projects, the product attempts to automatically calculate and maintained this geometry, as the user interacts with the model. Most transformations such as additions, moves, and deletes result in the bounding box or drawing extents being automatically updated.

Whenever the project is saved and the ProjectWise server is updated, the stored spatial location on the server, which is used for registration against any background map, will be updated also. (Note the timing of this update will be affected by the "Update Server When Saving" option on the Tools-Options-ProjectWise tab.)

Most of the time the bounding box stored in the project will be correct. However, for performance reasons, there are some rare situations (e.g., moving the entire model) where the geometry can become out of date with respect to the model. To guarantee the highest accuracy, the user can always manually update the geometry by using "Compact Database" or "Update Database Cache" as necessary, before saving to ProjectWise.

Setting the Project Spatial Reference System

The Spatial Reference System (SRS) for a project is viewed and assigned on the Tools-Options-Project tab in the Geospatial group.

The SRS is a standard textual name for a coordinate system or a projection, designated by various national and international standards bodies. The SRS is assumed to define the origin for the coordinates of all modeling elements in the project. It is the user's responsibility to set the correct SRS for the project, and then use the correct coordinates for the contained modeling elements. This will result in the extents of the

modeling features being correct with respect to the spatial reference system chosen. The SRS is stored at the project database level. Therefore, a single SRS is maintained across all geometry alternatives. The product does not manipulate or transform geometries or SRS's - it simply stores them.

The primary use of the project's SRS is to create correct spatial locations when a managing a project in the ProjectWise Integration Server's spatial management system.

The SRS name comes from the internal list of spatial reference systems that ProjectWise Spatial maintains on the ProjectWise server and is also known as the "key name." To determine the SRS key name, the administrator should browse the coordinate system dictionary in the ProjectWise administrator tool (under the Coordinate Systems node of the datasource), and add the desired coordinate system to the datasource. For example, the key name for an SRS for latitude/longitude is LL84, and the key name for the Maryland State Plane NAD 83 Feet SRS is MD83F.

ProjectWise Spatial uses the SRS to re-project the project's spatial location to the coordinate system of any spatial view or background map assigned by the administrator.

If the project's SRS is left blank, then ProjectWise will simply not be updated with a spatial location for that project.

If the project's SRS is not recognized, an error message will be shown, and ProjectWise will simply not be updated with a spatial location for that project.

Interaction with ProjectWise Explorer

Geospatial Administrators can control whether users can edit spatial locations through the ProjectWise Explorer. This is governed by the checkbox labeled "This user is a Geospatial Administrator" on the Geospatial tab of the User properties in the ProjectWise Administrator.

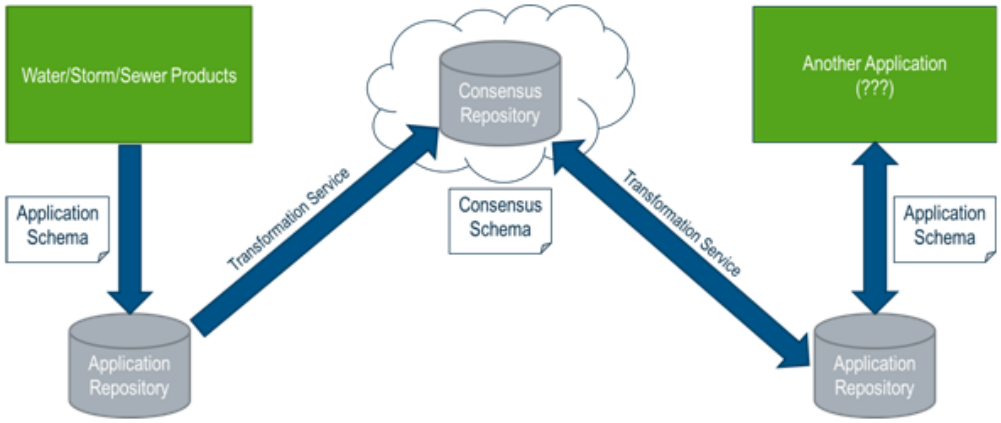
Users should decide to edit spatial locations either through the ProjectWise Explorer, or through the Municipal application, but not both at the same time. The application will update and overwrite the spatial location (coordinate system and geometry) in ProjectWise as a project is saved, if the user has added a spatial reference system to the project. This mechanism is simple and flexible for users - allowing them to choose when and where spatial locations will be updated.

Note: **If the spatial reference system referenced by the project does not exist in the ProjectWise datasource, the user will receive a warning and the spatial location will not be saved. The user may then add the spatial reference system to the datasource, through the Geospatial Administrator, before re-saving.**

ProjectWise Cross-Discipline Coordination Services Support

ProjectWise Cross-discipline Coordination Services (henceforth referred to as PWXDCS) refers to a shared library of code and tools used to facilitate the communication of model engineering data between 2 (or more) separate applications. For example, suppose building construction software wants to communicate relevant information about the model with software being used to design the parking lot for the building. PWXDCS allows this communication through a separate store of information called a consensus repository. This consensus repository has a schema called the consensus schema. The consensus schema only contains those fields/attributes that are common/relevant to software using it to sync data (in this example, the common fields/attributes between the building software and the parking lot software).

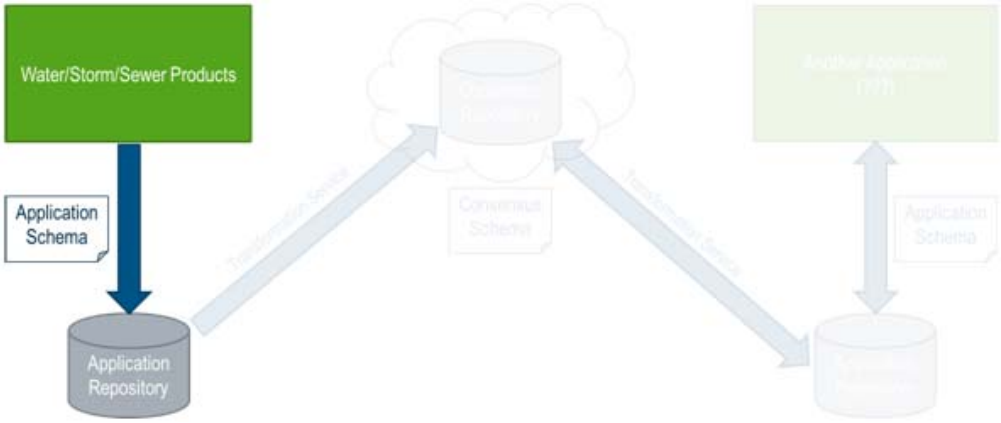
This is the basic workflow:



Water/Storm/Sewer Products publish their changes to an application repository. An application schema is adhered so that only relevant properties are published.

A consensus repository exists in some shared location (perhaps on a server of some sort) and may be in a completely different (consensus) schema. If the schema is incompatible with the schema of the applications using it, transformation services need to be written to transform data between the two schemas.

Bentley Water/Storm/Sewer products only write our data out to the application repository, so the part of the process handled by those products looks like this:



Workflow Walkthrough

Initial creation of a consensus repository:

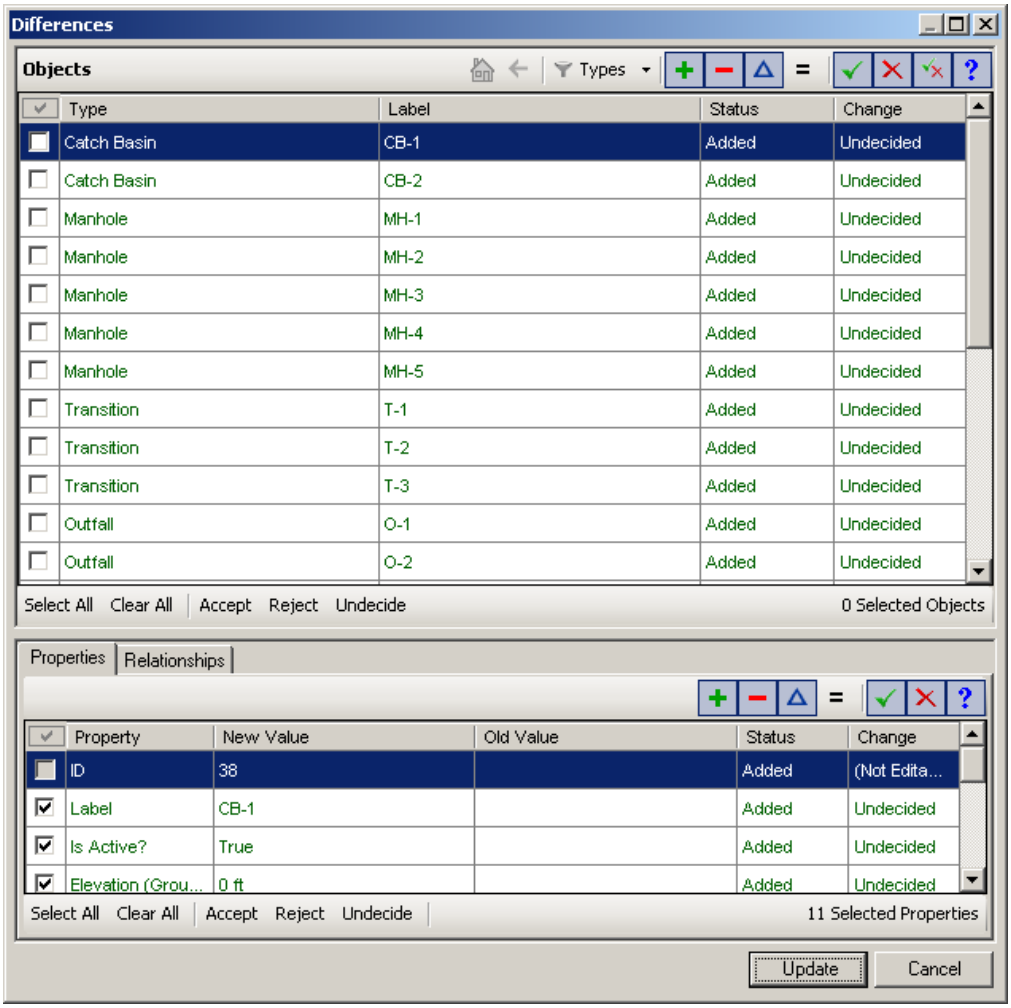
1. Open a model you want to sync out.
2. Click File menu, select Repository Management -> Create Repository.
3. Select the name and location of the consensus repository.
4. Progress dialogs appear.
5. After the process is complete, the repository file (*.dgn) should be on the disk where you indicated.

Sync out changes to existing consensus repository:

1. Open the model you want to sync out.
2. Click File menu, select Repository Management -> Update Repository.
3. Pick the consensus repository you want to update.
4. Progress dialogs appear.
5. A dialog appears displaying what has changed since the last time you synced out.
6. Accept/reject the changes you want/don't want.
7. The consensus repository is updated.

Differences Dialog Box

The Differences dialog appears when you update a repository. It shows the differences between the previous head revision and the new about-to-be-created revision. The user can select which changes they want to accept (keep) and which they would like to reject (ignore).



Going from left-to-right across the top toolbar of the upper section of the dialog, the buttons are as follows:

- **Home**: Restores the grid view back to its original state after following any relationships.
- **Back**: Goes back a step after following any relationships.
- **Filter**: Filters on an elements of the chosen types.
- **Show Added**: Toggles the showing of newly added elements in the grid view.

- **Show Deleted**: Toggles the showing of the newly deleted elements in the grid view.
- **Show Modified**: Toggles the showing of the newly modified elements in the grid view.
- **Show Unchanged**: Toggles the showing of the elements that haven't changed since the latest repository revision.
- **Show Accepted**: Toggles the showing of elements whose changes have all been accepted.
- **Show Rejected**: Toggles the showing of elements whose changes have all been rejected.
- **Show Partial**: Toggles the showing of elements whose changes are a mixture of accepted, rejected, and undecided.
- **Show Undecided**: Toggles the showing of elements whose changes are all undecided.

The grid view lists the elements (filtered as described above):

- **Check Box**: Selects/deselects the element as part of the set of elements affected by the bottom toolbar (described below).
- **Type**: The element type.
- **Label**: The element's label.
- **Status**: The status (added, deleted, modified, etc.) of the element.
- **Change**: The current state of the decision to include the changes or not (accepted, rejected, etc.).

Going from left-to-right across the bottom toolbar of the upper section of the dialog, the buttons are as follows:

- **Select All**: Checks all of the check boxes for the elements listed in the grid view above it.
- **Clear All**: Unchecks all of the check boxes for the elements listed in the grid view above it.
- **Accept**: Sets the change state of all of the checked elements in the grid view above it to accepted.
- **Reject**: Sets the change state of all of the checked elements in the grid view above it to rejected.
- **Undecide**: Sets the change state of all of the checked elements in the grid view above it to undecided.
- **Selected Objects**: Gives the count of elements in the grid view above it that are checked.

In the lower section of the dialog, the Properties tab shows the properties of the currently selected elements in the grid view of the upper section of the dialog.

Going from left-to-right across the top toolbar of the lower section of the dialog, the buttons are as follows:

- **Show Added**: Toggles the showing of newly added properties in the grid view.
- **Show Deleted**: Toggles the showing of the newly deleted properties in the grid view.
- **Show Modified**: Toggles the showing of the newly modified properties in the grid view.
- **Show Unchanged**: Toggles the showing of the properties that haven't changed since the latest repository revision.
- **Show Accepted**: Toggles the showing of properties that have been accepted.
- **Show Rejected**: Toggles the showing of properties that have been rejected.
- **Show Undecided**: Toggles the showing of properties that are still undecided.

The grid view lists the elements (filtered as described above):

- **Check Box**: Selects/deselects the property as part of the set of properties affected by the bottom toolbar (described below).
- **Property**: The name of the property.
- **New Value**: The new (changed) value of the property.
- **Old Value**: The previous value of the property.
- **Status**: The status (added, deleted, modified, etc.) of the property.
- **Change**: The current state of the decision to include the change or not (accepted, rejected, etc.).

Going from left-to-right across the bottom toolbar of the lower section, the buttons are as follows:

- **Select All**: Checks all of the check boxes for the properties listed in the grid view above it.
- **Clear All**: Unchecks all of the check boxes for the properties listed in the grid view above it.
- **Accept**: Sets the change state of all of the checked properties in the grid view above it to accepted.
- **Reject**: Sets the change state of all of the checked properties in the grid view above it to rejected.

- **Undecide**: Sets the change state of all of the checked properties in the grid view above it to undecided.
- **Selected Properties**: Gives the count of properties in the grid view above it that are checked.

At the bottom of the dialog are the following buttons:

- **Update**: commits the decisions on the changes you've made in this dialog to the repository.
- **Cancel**: Cancels out of the dialog and the entire update operation. The repository is left as it was unchanged.

Elements and Element Attributes

Pipes

Junctions

Hydrants

Tanks

Reservoirs

Pumps

Variable Speed Pump Battery

Valves

Spot Elevations

Turbines

Periodic Head-Flow Elements

Air Valves

Hydropneumatic Tanks

Surge Valves

Check Valves

Rupture Disks

Discharge to Atmosphere Elements

Orifice Between Pipes Elements

Valve with Linear Area Change Elements

Surge Tanks

Other Tools

Pipes

Pipes are link elements that connect junction nodes, pumps, valves, tanks, and reservoirs. Each pipe element must terminate in two end node elements.

Note: When laying out a pipe, you can add bends by holding the Ctrl key and clicking.

Applying a Zone to a Pipe

You can group elements together by any desired criteria through the use of zones. A Zone can contain any number of elements and can include a combination of any or all element types. For more information on zones and their use, see Zones.

To Apply a Previously Created Zone to a Pipe

1. Click the pipe in the **Drawing View.**
2. In the **Properties** window, click the menu in the **Zone** field and choose the zone from the drop-down list.



Choosing a Pipe Material

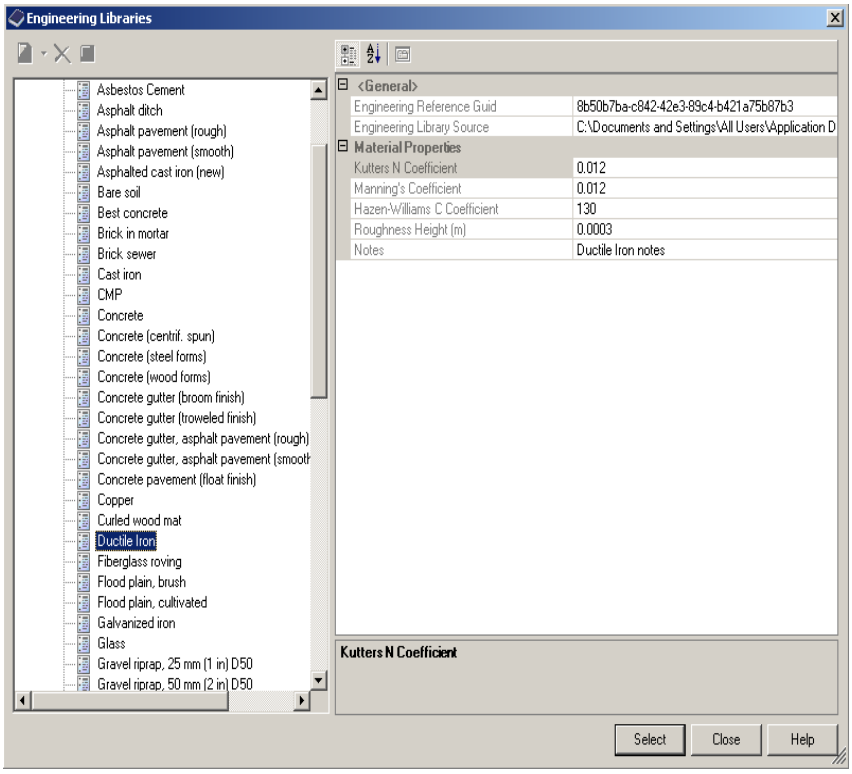
Pipes can be assigned a material type chosen from an engineering library. Each material type is associated with various pipe properties, such as roughness coefficient and roughness height. When a material is selected, these properties are automatically assigned to the pipe.

To Select a Material for a Pipe From the Standard Material Library

1. Select the pipe in the Drawing View.
2. In the Properties window, click the ellipsis (...) in the Material field.



3. The Engineering Libraries dialog box opens.



4. Choose Material Libraries > MaterialLibraries.xml.
5. Select the material and click Select.

Adding a Minor Loss Collection to a Pipe

Pressure pipes can have an unlimited number of minor loss elements associated with them. Bentley WaterCAD V8*i* provides an easy-to-use table for editing these minor loss collections in the Minor Loss Collection dialog box.

To add a minor loss collection to a pressure pipe

1. Click a pressure pipe in your model to display the Property Editor, or right-click a pressure pipe and select **Properties** from the shortcut menu.
2. In the Physical: Minor Losses section of the Property Editor, set the **Specify Local Minor Loss?** value to **False**.
3. Click the **Ellipses (...)** button next to the **Minor Losses** field.
4. In the **Minor Loses** dialog box, each row in the table represents a single minor loss type and its associated headloss coefficient. For each row in the table, perform the following steps:

a. Type the number of minor losses of the same type to be added to the composite minor loss for the pipe in the **Quantity** column, then press the Tab key to move to the **Minor Loss Coefficent** column.

b. Click the arrow button to select a previously defined Minor Loss, or click the **Ellipses (...)** button to display the Minor Loss Coefficients to define a new Minor Loss.

5. When you are finished adding minor losses to the table, click **Close**. The composite minor loss coefficient for the minor loss collection appears in the Property Editor.

6. Perform the following optional steps:

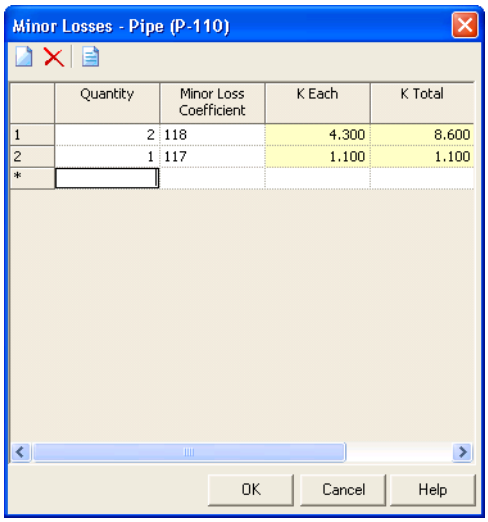
   – To delete a row from the table, select the row label then click **Delete**.

   – To view a report on the minor loss collection, click **Report**.

Minor Losses Dialog Box

The Minor Loss Collection dialog box contains buttons and a minor loss table. The dialog box contains the following controls:

	**New**	This button creates a new row in the table.
	**Delete**	This button deletes the currently highlighted row from the table. You can hold down the Ctrl key while clicking on items in the list to select multiple entries at once.
	**Report**	Opens a print preview window containing a report that details the input data for this dialog box.

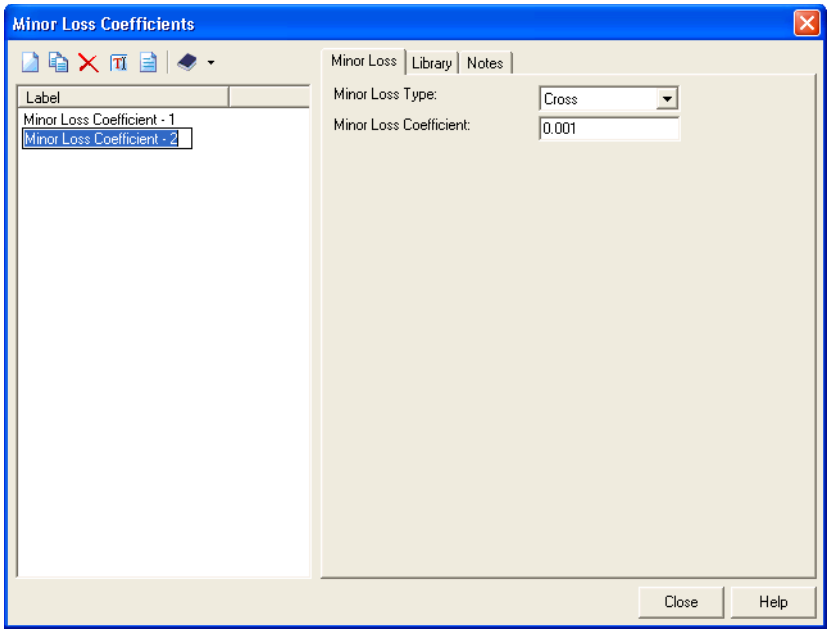


The table contains the following columns:

Column	Description
Quantity	The number of minor losses of the same type to be added to the composite minor loss for the pipe.
Minor Loss Coefficient	The type of minor loss element. Clicking the arrow button allows you to select from a list of previously defined minor loss coefficients. Clicking the Ellipses button next to this field displays the Minor Loss Coefficients manager where you can define new minor loss coefficients.
K Each	The calculated headloss coefficient for a single minor loss element of the specified type.
K Total	The total calculated headloss coefficient for all of the minor loss elements of the specified type.

Minor Loss Coefficients Dialog Box

The Minor Loss Coefficients dialog box allows you to create, edit, and manage minor loss coefficient definitions.



The following management controls are located above the minor loss coefficient list pane:

New



Creates a new Minor Loss Coefficient.

Duplicate



Creates a copy of the currently highlighted minor loss coefficient.

Delete 
Deletes the minor loss coefficient that is currently highlighted in the list pane.

Rename 
Renames the minor loss coefficient that is currently highlighted in the list pane.

Report 
Opens a report of the data associated with the minor loss coefficient that is currently highlighted in the list pane.

Synchronization Options 
Browses the Engineering Library, synchronizes to or from the library, imports from the library or exports to the library.

The tab section is used to define the settings for the minor loss that is currently highlighted in the minor loss list pane. The following controls are available:

Minor Loss Tab
This tab consists of input data fields that allow you to define the minor loss.

Minor Loss Type
General type of fitting or loss element. This field is used to limit the number of minor loss elements available in choice lists. For example, the minor loss choice list on the valve dialog box only includes minor losses of the valve type. You cannot add or delete types.

Minor Loss Coefficient
Headloss coefficient for the minor loss. This unitless number represents the ratio of the headloss across the minor loss element to the velocity head of the flow through the element.

Library Tab

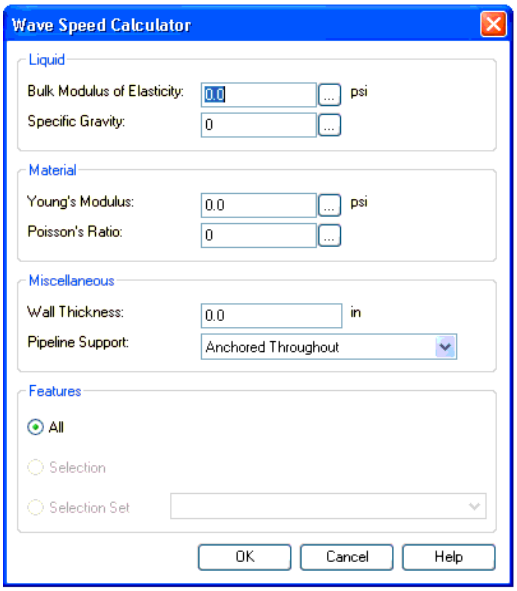
This tab displays information about the minor loss that is currently highlighted in the minor loss list pane. If the minor loss is derived from an engineering library, the synchronization details can be found here. If the minor loss was created manually for this project, the synchronization details will display the message Orphan (local), indicating that the minor loss was not derived from a library entry.

Notes Tab

This tab contains a text field that is used to type descriptive notes that will be associated with the minor loss that is currently highlighted in the minor loss list pane.

Wave Speed Calculator

The wave speed calculator allows you to determine the wave speed for a pipe or set of pipes.



The dialog consists of the following controls:

Bulk Modulus of Elasticity	The bulk modulus of elasticity of the liquid. Click the ellipsis button to choose a liquid from the Liquid Engineering Library. Choosing a liquid from the library will populate both this field and the Specific Gravity field with the values for the chosen liquid.
Specific Gravity	The specific gravity of the liquid. Click the ellipsis button to choose a liquid from the Liquid Engineering Library. Choosing a liquid from the library will populate both this field and the Bulk Modulus of Elasticity field with the values for the chosen liquid.
Young's Modulus	The Young's modulus of the elasticity of the pipe material. Click the ellipsis button to choose a material from the Material Engineering Library. Choosing a material from the library will populate both this field and the Poisson's Ratio field with the values for the chosen material.
Poisson's Ratio	The Poisson's ratio of the pipe material. Click the ellipsis button to choose a material from the Material Engineering Library. Choosing a material from the library will populate both this field and the Young's Modulus field with the values for the chosen material.
Wall Thickness	The thickness of the pipe wall.

Pipeline Support	Select the method of pipeline support.
All	When this button is selected, the calculated Wave Speed value will be applied to all pipes in the model.
Selection	When this button is selected, the calculated Wave Speed value will be applied to all of the pipes that are currently selected in the model.
Selection Set	When this button is selected, the calculated Wave Speed value will be applied to all of the pipes contained within the specified selection set.

Junctions

Junctions are non-storage nodes where water can leave the network to satisfy consumer demands or enter the network as an inflow. Junctions are also where chemical constituents can enter the network. Pipes are link elements that connect junction nodes, pumps, valves, tanks, and reservoirs. Each pipe element must terminate in two end node elements.

Assigning Demands to a Junction

Junctions can have an unlimited number of demands associated with them. Demands are assigned to junctions using the Demands table to define Demand Collections. Demand Collections consists of a Base Flow and a Demand Pattern. If the demand doesn't vary over time, the Pattern is set to Fixed.

To Assign a Demand to a Junction

1. Select the Junction in the Drawing View.
2. In the Properties window, click the ellipsis (...) button in the Demand Collection field under the Demands heading.
3. In the Demands dialog that opens, enter the base demand in the Flow column.
4. Click the arrow button to assign a previously created Pattern, click the ellipsis button to create a new Pattern in the Patterns dialog, or leave the value at Fixed (Fixed means the demand doesn't vary over time).

Applying a Zone to a Junction

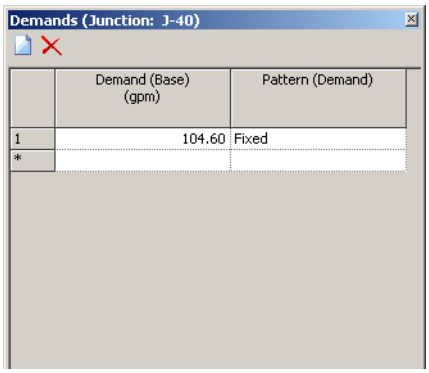
You can group elements together by any desired criteria through the use of zones. A Zone can contain any number of elements and can include a combination of any or all element types. For more information on zones and their use, see Zones.

To Apply a Previously Created Zone to a Junction

1. Select the junction in the **Drawing View**.
2. In the **Properties** window, click the menu in the **Zone** field and select the zone you want.

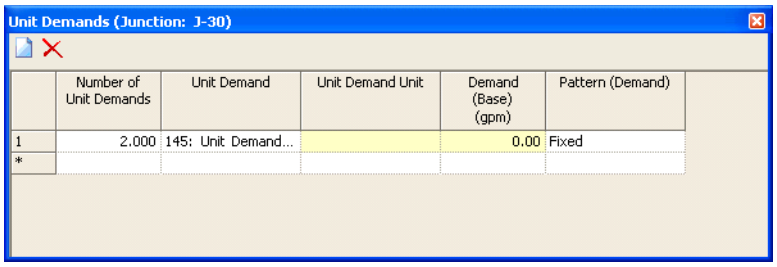
Demand Collection Dialog Box

The Demand collection dialog box allows you to assign single or composite demands and demand patterns to the elements in the model.



Unit Demand Collection Dialog Box

The Unit Demand Collection dialog box allows you to assign single or composite unit demands to the elements in the model.



To assign one or more unit demands

1. Specify the Unit Demand count.
2. Select a previously created Unit Demand from the list or click the ellipsis button to open the Unit Demands Dialog Box, allowing you to create a new one.
3. Select a previously created Demand Pattern from the list or click the ellipsis button to open the Pattern Manager, allowing you to create a new one.

Hydrants

Hydrants are non-storage nodes where water can leave the network to satisfy consumer demands or enter the network as an inflow. Hydrants are also where chemical constituents can enter the network.

Applying a Zone to a Hydrant

You can group elements together by any desired criteria through the use of zones. A Zone can contain any number of elements and can include a combination of any or all element types. For more information on zones and their use, see Zones.

To Apply a Previously Created Zone to a Hydrant

1. Select the hydrant in the **Drawing View**.
2. In the **Properties** window, click the menu in the **Zone** field and select the zone you want.

Hydrant Flow Curves

Hydrant curves allow you to find the flow the distribution system can deliver at the specified residual pressure, helping you identify the system's capacity to deliver water that node in the network. See following topics for more information about Hydrant Flow Curves:

Hydrant Flow Curve Manager

Hydrant Flow Curve Editor

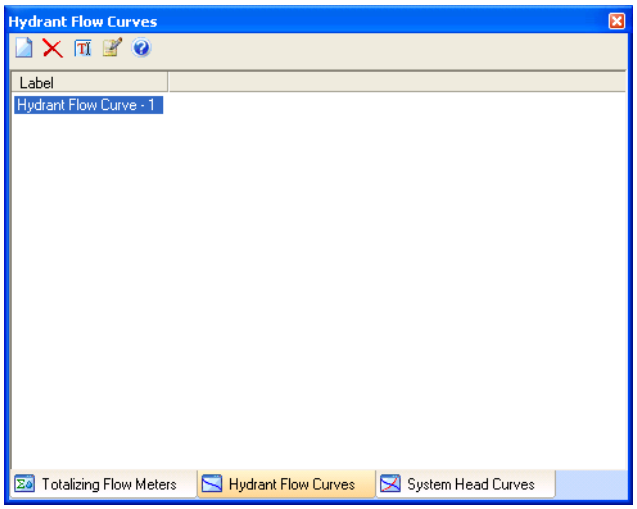
Also, see Hydrant Lateral Loss.

Hydrant Flow Curve Manager

The Hydrant Flow Curve Manager consists of the following controls:

New	Creates a new hydrant flow curve definition.

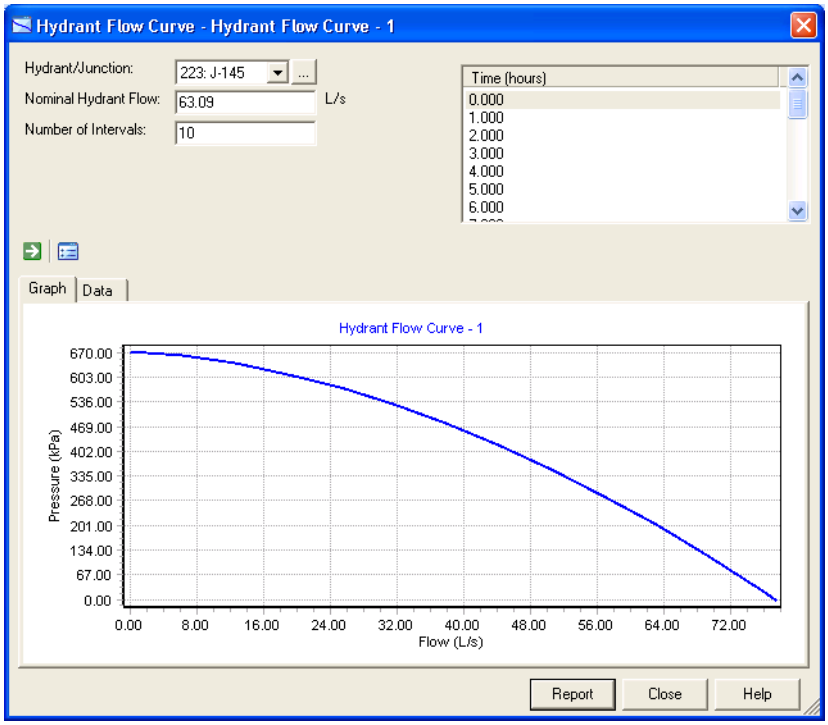
Delete	Deletes the selected hydrant flow curve definition.
Rename	Renames the label for the current hydrant flow curve definition.
Edit	Opens the hydrant flow curve definition editor for the currently selected definition.
Refresh	Recomputes the results of the currently selected hydrant flow curve definition.
Help	Opens the online help for the hydrant flow curve manager.



Hydrant Flow Curve Editor

Hydrant curves allow you to find the flow the distribution system can deliver at the specified residual pressure, helping you identify the system's capacity to deliver water that node in the network. Hydrant curves are useful when you are trying to balance the flows entering a part of the network, the flows being demanded by that part of the network, and the flows being stored by that part of the network.

The Hydrant Flow Curve Editor dialog displays the flow vs pressure table, which is computed by the program; the table is in part based on the Nominal Hydrant Flow and Number of Intervals values you define, which are used for formatting of the curve.



- **Nominal Hydrant Flow**: This value should be the expected nominal flow for the hydrant (i.e., the expected flow or desired flow when the hydrant is in use). The value for nominal flow is used together with the number of intervals value to determine a reasonable flow step to use when calculating the hydrant curve. A higher nominal flow value results in a larger flow step and better performance of the calculation. Note that if you choose a nominal hydrant flow that is too small and not representative of the hydrant then the high flow results on the resultant curve may not be correct since the calculation will not calculate more than 1000 points on the curve, for performance reasons.
- **Number of Intervals**: This value is used with the nominal flow value to determine the flow step to be used with the hydrant calculation. For example, a nominal hydrant flow of 1000gpm and number of intervals set to 10 will result in a flow step of 1000/10 = 100gpm. This results in points on the hydrant curve

being calculated from 0 flow to the zero pressure point in steps of 100gpm. Note that if you have a number of intervals value that is too high then high flow results on the resultant curve may not be correct since the calculation will not calculate more than 1000 points on the curve, for performance reasons.

- **Time**: Choosing the time of the hydrant curve can affect the results of the curve. Choose the time at which you wish to run your hydrant curve and the corresponding pattern multipliers will be used for that time. This behaves the same way as an EPS snapshot calculation. You may also select multiple times in order to generate multiple hydrant curves for comparison

To define a Hydrant Flow Curve

- Choose the junction or hydrant element that will be used for the hydrant flow curve from the **Hydrant/Junction** pull-down menu or click the **ellipsis** button to select the element from the drawing pane.
- Enter values for **Nominal Hydrant Flow** and **Number of Intervals** in the corresponding fields.
- Choose a time step from the **Time** list pane.
- Click the **Compute** button to calculate the hydrant flow curve.

Hydrant Lateral Loss

Hydrant lateral losses are calculated by the pressure engine the same as any pipe (the lateral pipe is actually loaded into the model), using the supplied lateral diameter, minor loss coefficient and length. Additionally, the engine assumes the following values.

Darcy Weisbach e: 0.0009

Hazen Williams C: 130.0

Mannings n: 0.012

Tanks

Tanks are a type of Storage Node. A Storage Node is a special type of node where a free water surface exists, and the hydraulic head is the elevation of the water surface above some datum (usually sea level). The water surface elevation of a tank will change as water flows into or out of it during an extended period simulation.

Water Level/Elevation

The user can choose either Elevation or Level as the Operating Range Type. The water level in a tank can be described based on either the hydraulic grade line elevation (Elevation) or the water level above the base elevation (Level).

Applying a Zone to a Tank

You can optionally group elements together by any desired criteria through the use of zones. A Zone can contain any number of elements and can include a combination of any or all element types. For more information on zones and their use, see Zones on page 4-450.

To Apply a Previously Created Zone to a Tank

1. Select the tank in the **Drawing View**.
2. In the **Properties** window, click the menu in the **Zone** field and select the zone you want.

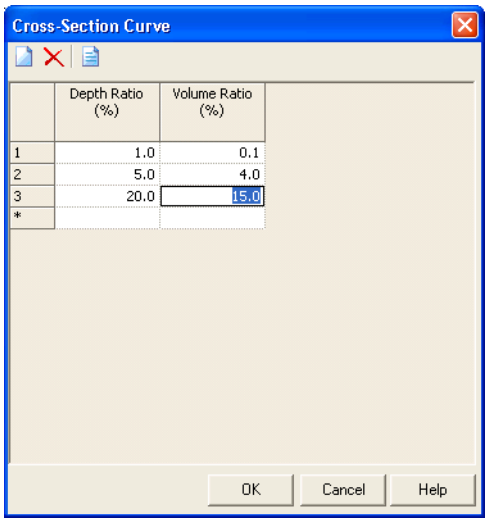
Active Topology

By default a tank is active in a model. A tank can be made inactive (not used in calculations) by changing the **Is active?** property to **False**. If a tank is made inactive, any connective pipes should also be made inactive as otherwise this will give an error.

Defining the Cross Section of a Variable Area Tank

By default, tanks are treated as having a circular shape with a constant cross section described by its diameter. If the tank has a constant cross section that is not circular, the user can select Non-circular and specify the cross sectional area. If the user selects Variable Area, it is necessary to provide a depth to volume table.

In a variable area tank, the cross-sectional geometry varies between the minimum and maximum operating elevations. A depth-to-volume ratio table is used to define the cross sectional geometry of the tank.



To Define the Cross Section of a Variable Area Tank

1. Select the tank in the **Drawing View**.
2. In the **Properties** window, click the **Section** menu and select the **Variable Area** section type.
3. Click the **ellipsis** button (...) in the **Cross-Section Curve** field.
4. In the **Cross-Section Curve** dialog that appears, enter a series of points describing the storage characteristics of the tank. For example, at 0.1 of the total depth (depth ratio = 0.1) the tank stores 0.028 of the total active volume (volume ratio = 0.028). At 0.2 of the total depth the tank stores 0. 014 of the total active volume (0.2, 0.014), and so on.

Setting High and Low Level Alarms

You can specify upper and lower tank levels at which user notification messages will be generated during calculation.

To set a High Level Alarm

1. Double-click a tank element to open the associated Properties editor.
2. In the Operating Range section, change the Use High Alarm? value to True.
3. In the Elevation (High Alarm) field, enter the high alarm elevation value. A high alarm user notification message will be generated for each time step during which the tank elevation exceeds this value.

To set a Low Level Alarm

1. Double-click a tank element to open the associated Properties editor.
2. In the Operating Range section, change the Use Low Alarm? value to True.
3. In the Elevation (Low Alarm) field, enter the low alarm elevation value. A low alarm user notification message will be generated for each time step during which the tank elevation goes below this value.

Inlet Type

In general, tank inlet and outlet piping are treated as being connected to the tank at the bottom and have only a single altitude valve that shuts the tank off from the rest of the system when the tank reaches its maximum level or elevation. However, some tanks are filled from the top or have altitude valves (sometimes called a "Float Valve") that gradually throttle before they shut. This can be controlled by setting the Has Separate Inlet? Property to True. The user must pick which of the pipes connected to the tank is the inlet pipe which is controlled or top fill. (If there is a valve vault at the tank with a altitude valve on the fill line and a check valve on the outlet, these should be treated as two pipes from the tank even if there is a single pipe from the tank to the vault.)

If the tank is a top filled tank (which may refer to a side inflow tank above the bottom but below the top), the user should set Tank Fills From Top? To true and set the invert level (relative to the base) of the inflow pipe at its highest point. Water will not flow into the tank through that pipe unless the hydraulic grade is above that elevation.

If the inlet valve throttles the flow as it nears full, the user should set "Inlet Valve Throttles?" to True. The user must then enter the discharge coefficient for the valve when it is fully open, the level at which the valve begins to close and the level at which it is fully closed. These levels must be below the top level and any pumps controlled by the valve should not be set to operate at levels above the fully closed level. The closure characteristics are determined by the Valve Type which the user selects from a drop down menu.

When the tank is described as having a separate inlet, additional results properties are calculated beyond the usual values of tank levels (elevations) and flow. The user can also obtain the relative closure of the inlet valve, the calculated discharge coefficient, the head loss across the valve, and the inlet and outlet hydraulic grade of the valve and finally the inlet valve status.

Water Quality (Tanks)

If the user is performing a water quality analysis, it is necessary to specify the initial value for Age, Concentration or Trace depending on the type of run. If the tank is a source for some water quality constituent concentration, the user should set "Is Constituent Source?" to True and specify the constituent source type. See the Constituent Alternatives help topic.

If this analysis is a constituent analysis, the user may specify the bulk reaction rate in the tank by setting "Specify local bulk rate?" to True and setting the "Bulk reaction rate (Local)" value.

Tank Mixing Models

Real water distribution tanks cannot be exactly described as plug flow or completely mixed but these are reasonable approximations to fluid behavior in tanks. WaterCAD V8*i* supports four types of tank mixing models which the user selects in the drop down menu of Tank Mixing Models.

The Complete Mixing model assumes that all water that enters a tank is instantaneously and completely mixed with the water already in the tank. It applies well to a large number of facilities that operate in filland-draw fashion with the exception of tall standpipes.

The Two-Compartment Mixing model divides the available storage volume in a tank into two compartments, both of which are assumed completely mixed. The inlet/outlet pipes of the tank are assumed to be located in the first compartment. New water that enters the tank mixes with the water in the first compartment. If this compartment is full, then it sends its overflow to the second ompartment where it completely mixes with the water already stored there. When water leaves the tank, it exits from the first compartment, which if full, receives an equivalent amount of water from the second compartment to make up the difference. The first compartment is capable of simulating short-circuiting between inflow and outflow while the second compartment can represent dead zones. The user must supply a single parameter, which is the fraction of the total tank volume devoted to the first compartment. This value canbe determined during calibration if this model is selected.

The FIFO Plug Flow model assumes that there is no mixing of water at all during its residence time in a tank. Water parcels move through the tank in a segregated fashion where the first parcel to enter is also the first to leave. Physically speaking, this model is most appropriate for baffled tanks that operate with simultaneous inflow and outflow such as ideal clear wells at water treatment plants. There are no additional parameters needed to describe this mixing model.

The LIFO Plug Flow model also assumes that there is no mixing between parcels of water that enter a tank. However in contrast to FIFO Plug Flow, the water parcels stack up one on top of another, where water enters and leaves the tank on the bottom. This type of model might apply to a tall, narrow standpipe with an inlet/outlet pipe at the bottom and a low momentum inflow. It requires no additional parameters be provided.

Reservoirs

Reservoirs are a type of storage node. A Storage Node is a special type of node where a free water surface exists, and the hydraulic head is the elevation of the water surface above sea level. The water surface elevation of a reservoir does not change as water flows into or out of it during an extended period simulation.

Applying a Zone to a Reservoir

You can group elements together by any desired criteria through the use of zones. A Zone can contain any number of elements, and can include a combination of any or all element types. For more information on zones and their use, see Zones on page 4-450.

To Apply a Previously Created Zone to a Reservoir

1. Select the reservoir in the **Drawing View**.
2. In the **Properties** window, click the menu in the **Zone** field and select the zone you want.

Applying an HGL Pattern to a Reservoir

You can apply a pattern to reservoir elements to describe changes in hydraulic grade line (HGL) over time, such as that caused by tidal activity or when the reservoir represents a connection to another system where the pressure changes over time.

To Apply a Previously Created HGL Pattern to a Reservoir

1. Select the reservoir in the **Drawing View**.
2. In the **Properties** window, click the menu in the **HGL Pattern** field and select the desired pattern. To create a new pattern, select **Edit Pattern...** from the list to open the **Patterns** dialog.

For more information about Patterns, see Patterns.

Customer Meter Elements

Customer meter elements provide a way for users to maintain customer water demand data within WaterGEMS/CAD which provides the user access to features such as element symbology and the ability to visualize customer location and assignment of demand to node elements.

There are several main steps in using customer meter elements.

- Creating element

- Entering demands for the element
- Assigning customer metering element to hydraulic model element

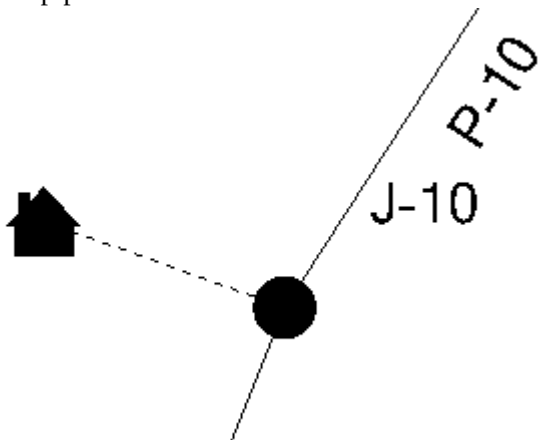
Customer meter elements are not directly used in hydraulic calculations but are used to load demands to elements that are used in hydraulic calculations.

Creating Customer Meter Elements

Customer meter elements can be created by:

1. Selecting the customer meter icon from the layout toolbar and placing the customer meter in the correct position in the drawing.
2. Importing the customer element from an external data source using ModelBuilder (see ModelBuilder help). The data source should contain a label, the x-y coordinate and some demand data for the new element.

The customer meter element symbol is shown below. The association of the element with a node or pipe is shown as a dashed line.



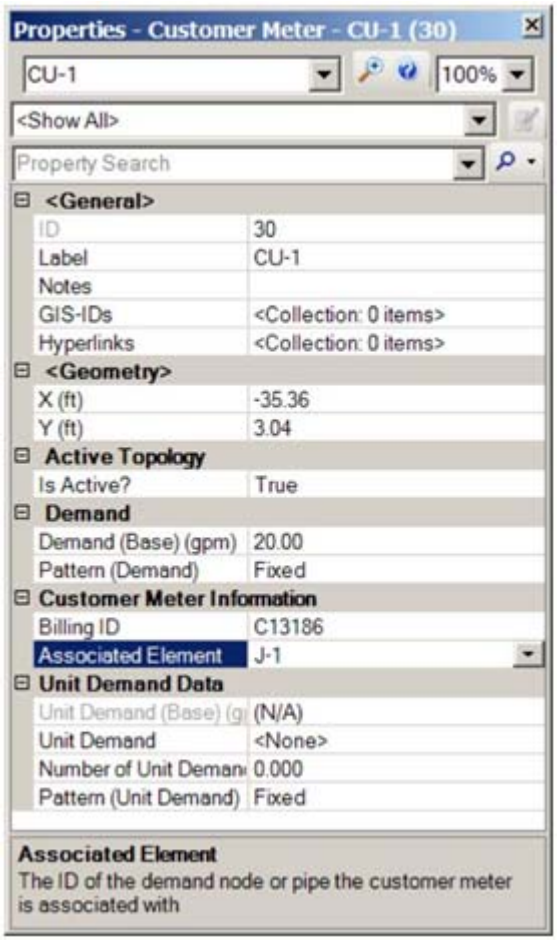
Entering Demands for Customer Meter Elements

Demand data for customer meters can be entered

1. Manually by entering values in the element property grid, the customer meter element flex table or Demand Alternative under the Customer Meter tab.
2. Using ModelBuilder. This could be done while the elements are being created or as a separate import.

The demand data can consist of demand, unit demand, pattern (demand) and pattern (unit demand). The Demand Control Center is not used for Customer Meter elements because there can only be a single demand and unit demand for a customer meter.

The Property Grid for a Customer Meter element is shown below:



Assigning Demands to Modeling Elements

The demands from the customer meter element must be assigned to a hydraulic modeling element in order to be used in hydraulic calculations. This can be done by:

1. Manually by picking the property "Associated Element" in the property grid or flex table for the customer meter element, then choosing "Select Associated Element" from the drop down and picking the hydraulic element to associate with the customer meter element.
2. Using ModelBuilder if the assignment of the customer meter element to a model element is available in the data source.
3. Using LoadBuilder to assign customer meter elements to hydraulic model elements using one of LoadBuilder's allocation methods under Customer load data such as nearest node or nearest pipe (see LoadBuilder help). In using LoadBuilder, the "Model Node Layer" will usually be Junction\All Elements but it can be any selection set of node elements that have Demand (Base) as a property. The

"Customer Data" is usually Customer\All Elements although it can be any selection set of Customer Meter elements. If the customers are being assigned based on nearest pipe method, in addition to specifying the Model Node Layer and Customer Data, the user must also specify the Model Pipe Data which identifies which pipes are to be considered. This enables the user to use a selection set which can ignore large transmission mains with no customers.

There are no pressure, HGL or water quality results for customer meter elements as they are maintained with the associated hydraulic element.

Displaying Customer Meter Assignments

Once the customer meter elements have been assigned to hydraulic model elements, these assignments can be viewed as lines connecting the elements. The display of these lines can be controlled in Element Symbology > Customer meter > Show attached Line Decorations set to either True of False.

Customer Element Behavior

During a simulation, the appropriate pattern and global multipliers are applied to the demand entered or calculated using unit demands and this value is passed on to the associated node to be used in the hydraulic simulation. Demand data for inactive customers are ignored.

Customer Meter Element Results

There are no hydraulic results for customer meter elements. The user should find results in the node to which the customer element is associated. The pressures may differ from actual pressure slightly due to differences in elevation between the junction node and the customer meter node and head losses in service lines.

Viewing Customer Meter Demands/Unit Demands from Hydraulic Element

Customer meter demands can also be viewed from the element (usually junction) to which the demand is assigned. Pick Customer Demand Collection or Customer Unit Demand Collection in the node property grid and the demand collection will open.

Customer Meter Element Notifications

Customer meter elements must be associated with active nodes at the time of a run. If the associated node is inactive, the run fails and the user receives a user notification "Reference to inactive or deleted associated element." The user can also execute a predefined query, "Customer associated with inactive element." Customer meter elements with no association will also be detected in an "Orphaned nodes" query.

The demand pattern assigned to a customer meter must exist and be valid. If a pattern assigned to a customer meter is later deleted and the user attempts to run the model, the run will fail and the user receives the notification "Reference to a deleted demand pattern."

The unit demand referenced by a customer meter element must exist. If one is assigned to an element and is later deleted, the run fails and the user receives the user notification, 'Reference to a deleted unit demand."

If the user does not associate a customer meter element with a hydraulic node, the run will complete with a warning message, "At least one customer node without an associated element." The user can find the customer meter without an association by looking in flex tables or executing the query for "Orphaned Customer Meters."

Customer Meter Element Predefined Queries

A user can determine the characteristics of Customer Meter element using one of the predefined queries that address this element. In addition to the standard queries such as "All Customer Meters" and "Elements with GIS-ID", there are some special queries for Customer Meter Elements. They include:

"Orphaned Customer Meters" which displays which customers are not connected to the network.

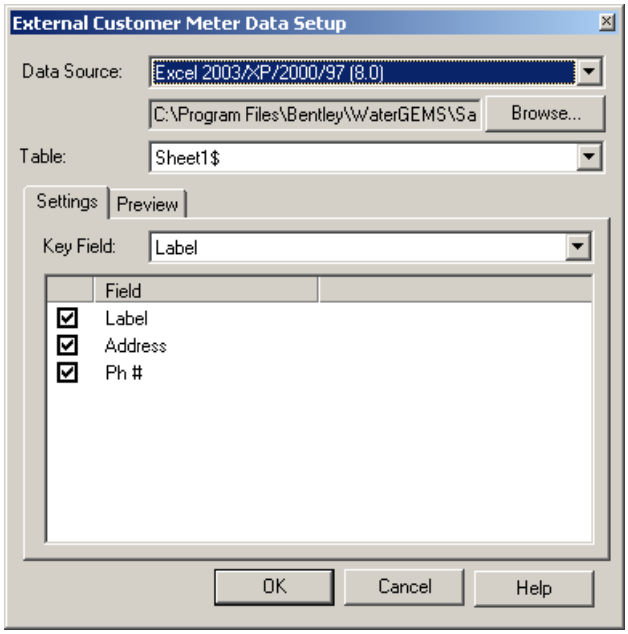
"Find Associated Customer Meters" which displays the customer elements associated with the hydraulic model elements in the current selection.

"Find Elements Associated with Customer Meters" which displays the hydraulic model elements associated with the customer meter elements in the current selection.

"Customer Meters Associated with Inactive Elements" which displays customer meter elements which have been associated with elements that have "Is active" equals "False".

External Customer Meter Data Setup

This dialog allows the user to setup a connection to the external data source.



The data source dropdown contains all the available data source types (based on platform this is the same as modelbuilder)

The browse button allows the user to specify the file they would like to connect to. The file type in the open dialog is determined by the file type that is selected in the Data Source drop down.

Once the file is selected using browse, the file path is shown. The Table dropdown is populated based on the external data source.

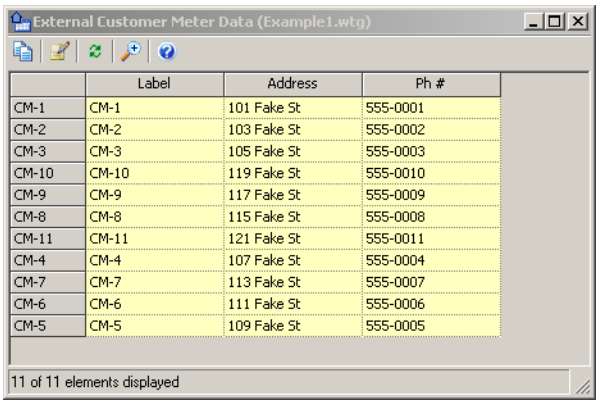
In the Settings tab, select the Key Field - the user needs to select the field in the data source that contains label data (this is what is used to match the data in the model). Check the box next to the items you want to use.

Click the Preview tab to see the data as it will be imported.

Click OK to import the data.

External Customer Meter Data

This dialog displays the table containing the external customer meter data.



Click the Copy button to copy the contents of the table to the Windows clipboard.

Click the Edit button to return to the External Customer Meter Data Setup dialog.

Click the Refresh button to update the table according to changes made in the linked datasource.

Highlight an element row in the table and click the Zoom To button to zoom the drawing pane view to the highlighted element.

Pumps

Pumps are node elements that add head to the system as water passes through.

Applying a Zone to a Pump

You can group elements together by any desired criteria through the use of zones. A Zone can contain any number of elements and can include a combination of any or all element types. For more information on zones and their use, see Zones on page 4-450.

To Apply a Previously Created Zone to a Pump

1. Select the pump in the **Drawing View**.
2. In the **Properties** window, click the menu in the **Zone** field and select the zone you want.

Defining Pump Settings

You define the settings for each pump in your model in the Pump Definitions dialog box. You can define a collection of pump settings for each pump.

To define pump settings

1. Click a pump in your model to display the Property Editor, or right-click a pump and select **Properties** from the shortcut menu.
2. In the Physical section of the Property Editor, click the **Ellipses (...)** button next to the Pump Definitions field. The Pump Definitions dialog box opens.
3. In the Pump Definitions dialog box, each item in the list represents a separate pump definition. Click the **New** button to add a new definition to the list.
4. For each definition in the list, perform these steps:
   a. Type a unique label for the pump definition.
   b. Define a new pump definition by entering Head, Efficiency, and Motor data.
5. Click **OK** to close the Pump Definitions dialog box and save your data in the Property Editor.

For more information about pump definitions, see the following topics:

Pump Definitions Dialog Box

Pump Curve Dialog Box

Flow-Efficiency Curve Dialog Box

Pump Definitions Dialog Box

This dialog box is used to create pump definitions. There are two sections: the pump definition pane on the left and the tab section on the right. The pump definition pane is used to create, edit, and delete pump definitions.

The following controls are available in the pump definitions dialog box:

	**New**	Creates a new entry in the pump definition Pane.
	**Duplicate**	Creates a copy of the currently highlighted pump definition.
	**Delete**	Deletes the currently highlighted entry in the pump definition Pane. You can hold down the Ctrl key while clicking on items in the list to select multiple entries at once.
	**Rename**	Renames the currently highlighted entry in the pump definition Pane.
	**Report**	Generates a pre-formatted report that contains the input data associated with the currently highlighted entry in the pump definition Pane.
	**Synchronization Options**	Clicking this button opens a submenu containing the following commands: • **Browse Engineering Library**—Opens the Engineering Library manager dialog, allowing you to browse the Pump Definition Libraries. • **Synchronize From Library**—Updates a set of pump definition entries previously imported from a Pump Definition Engineering Library. The updates reflect changes that have been made to the library since it was imported. • **Synchronize To Library**—Updates an existing Pump Definition Engineering Library using current pump definition entries that were initially imported but have since been modified. • **Import From Library**—Imports pump definition entries from an existing Pump Definition Engineering Library. • **Export To Library**—Exports the current pump definition entries to an existing Pump Definition Engineering Library.

The tab section includes the following controls:

Head Tab This tab consists of input data fields that allow you to define the pump head curve. The specific fields vary depending on which type of pump is selected in the Pump Definition type field.

Pump Definition Type

A pump is an element that adds head to the system as water passes through it. This software can currently be used to model six different pump types:

- **Constant Power**—When selecting a Constant Power pump, the following attribute must be defined:
  - **Pump Power**—Represents the water horsepower, or horsepower that is actually transferred from the pump to the water. Depending on the pump's efficiency, the actual power consumed (brake horsepower) may vary.
- **Design Point (One-Point)**—When selecting a Design Point pump, the following flow vs. head points must be defined:
  - **Shutoff**—Point at which the pump will have zero discharge. It is typically the maximum head point on a pump curve. This value is automatically calculated for Design Point pumps.
  - **Design**—Point at which the pump was originally intended to operate. It is typically the best efficiency point (BEP) of the pump. At discharges above or below this point, the pump is not operating under optimum conditions.
  - **Max Operating**—Highest discharge for which the pump is actually intended to run. At discharges above this point, the pump may behave unpredictably, or its performance may decline rapidly. This value is automatically calculated for Design Point pumps.
- **Standard (Three-Point)**—When selecting a Standard Three-Point pump, the following flow vs. head points must be defined:
  - **Shutoff**—Point at which the pump will have zero discharge. It is typically the maximum head point on a pump curve.
  - **Design**—Point at which the pump was originally intended to operate. It is typically the best efficiency point (BEP) of the pump. At discharges above or below this point, the pump is not operating under optimum conditions.
  - **Max Operating**—Highest discharge for which the pump is actually intended to run. At discharges above this point, the pump may behave unpredictably, or its performance may decline rapidly.

Pump Definition Type (cont'd)

- **Standard Extended**—When selecting a Standard Extended pump, the following flow vs. head points must be defined:
  - **Shutoff**—Point at which the pump will have zero discharge. It is typically the maximum head point on a pump curve.
  - **Design**—Point at which the pump was originally intended to operate. It is typically the best efficiency point (BEP) of the pump. At discharges above or below this point, the pump is not operating under optimum conditions.
  - **Max Operating**—Highest discharge for which the pump is actually intended to run. At discharges above this point, the pump may behave unpredictably, or its performance may decline rapidly.
  - **Max Extended**—Absolute maximum discharge at which the pump can operate, adding zero head to the system. This value may be computed by the program, or entered as a custom extended point. This value is automatically calculated for Standard Extended pumps.
- **Custom Extended**—When selecting a Custom Extended pump, the following attributes must be defined:
  - **Shutoff**—Point at which the pump will have zero discharge. It is typically the maximum head point on a pump curve.
  - **Design**—Point at which the pump was originally intended to operate. It is typically the best efficiency point (BEP) of the pump. At discharges above or below this point, the pump is not operating under optimum conditions.
  - **Max Operating**—Highest discharge for which the pump is actually intended to run. At discharges above this point, the pump may behave unpredictably, or its performance may decline rapidly.
  - **Max Extended**—Absolute maximum discharge at which the pump can operate, adding zero head to the system. This value may be computed by the program, or entered as a custom extended point.
- **Multiple Point**—When selecting a Multiple Point pump, an unlimited number of Flow vs. Head points may be defined.

Efficiency Tab This tab allows you to specify efficiency settings for the pump that is being edited.

Pump Efficiency Allows you to specify the pump efficiency type for the pump that is being edited. The following efficiency types are available:

- **Constant Efficiency**—This efficiency type maintains the efficiency determined by the input value regardless of changes in discharge. When the Constant Efficiency type is selected, the input field is as follows:
  - **Pump Efficiency**—The Pump Efficiency value is representative of the ability of the pump to transfer the mechanical energy generated by the motor to Water Power.
- **Best Efficiency Point**—This efficiency type generates a parabolic efficiency curve using the input value as the best efficiency point. When the Best Efficiency Point type is selected, the input fields are as follows:
  - **BEP Flow**—The flow delivered when the pump is operating at its Best Efficiency point.
  - **BEP Efficiency**—The efficiency of the pump when it is operating at its Best Efficiency Point.
  - **Define BEP Max Flow**—When this box is checked the User Defined BEP Max Flow field is enabled, allowing you to enter a maximum flow for the Best Efficiency Point. The user defined BEP Max Flow value will be the highest flow value on the parabolic efficiency curve.
  - **User Defined BEP Max Flow**—Allows you to enter a maximum flow value for the Best Efficiency Point. The user defined BEP Max Flow value will be the highest flow value on the parabolic efficiency curve.
- **Multiple Efficiency Points**—This efficiency type generates an efficiency curve based upon two or more user-defined efficiency points. These points are linearly interpolated to form the curve. When the Multiple Efficiency Points type is selected, the input field is as follows:
  - **Efficiency Points Table**—This table allows you to enter the pump's efficiency at various discharge rates.

Motor Tab

This tab allows you to define the pump's motor efficiency settings. It contains the following controls:

Motor Efficiency

The Motor Efficiency value is representative of the ability of the motor to transform electrical energy to rotary mechanical energy.

Is Variable Speed Drive?

This check box allows you to specify whether or not the pump is a Variable Speed Pump. Toggling this check box On allows you to input points on the Efficiency Points table.

Efficiency Points Table

This table allows you to enter efficiency points for variable speed pumps. This table is activated by toggling the "Variable Speed Drive" check box On. See Efficiency Points Table for more information.

Transient Tab

This tab allows you to define the pump's WaterCAD V8*i*-specific transient settings. It contains the following controls:

Inertia (Pump and Motor)

Inertia is proportional to the amount of stored rotational energy available to keep the pump rotating (and transferring energy to the fluid), even after the power is switched off. You can obtain this parameter from manufacturer's catalogs, or from pump curves, or by using the Pump and Motor Inertia Calculator. To access the calculator, click the ellipsis button.

Speed (Full)

Speed denotes thenumber of rotations of the pump impeller per unit time, generally in revolutions per minute or rpm. This is typically shown prominently on pump curves and stamped on the name plate on the pump itself.

Specific Speed

Specific speed provides four-quadrant characteristic curves to represent typical pumps for each of the most common types, including but not limited to: 1280, 4850, or 7500 (U.S. customary units) and 25, 94, or 145 (SI metric units).

Reverse Spin Allowed?

Indicates whether the pump is equipped with a ratchet or other device to prevent the pump impeller from spinning in reverse.

Library Tab — This tab displays information about the pump that is currently highlighted in the Pump Curves Definition Pane. If the pump is derived from an engineering library, the synchronization details can be found here. If the pump was created manually for this project, the synchronization details will display the message Orphan (local), indicating that the pump was not derived from a library entry.

Notes Tab — This tab contains a text field that is used to type descriptive notes that will be associated with the pump that is currently highlighted in the Pump Curves Definition Pane.

To create a pump definition

1. Select **Components > Pump Definitions**.
2. Click **New** to create a new pump definition.
3. For each pump definition, perform these steps:
   a. Select the type of pump definition in the Pump Definition Type menu.
   b. Type values for Pump Power, Shutoff, Design point, Max Operating, and/or Max Extended as required. The available table columns or fields change depending on which definition type you choose.
   c. For Multiple Point pumps, click the **New** button above the curve table to add a new row to the table, or press the **Tab** key to move to the next column in the table. Click the **Delete** button above the curve table to delete the currently highlighted row from the table.
   d. Define efficiency and motor settings in the Efficiency and Motor tabs.
4. You can save your new pump definition in WaterCAD V8*i*' Engineering Libraries for future use. To do this, perform these steps:
   a. Click the **Synchronization Options** button, then select **Export to Library**. The Engineering Libraries dialog box opens.
   b. Use the plus and minus signs to expand and collapse the list of available libraries, then select the library into which you want to export your new unit sanitary load.
   c. Click **Close** to close the Engineering Libraries dialog box.
5. Perform the following optional steps:
   – To delete a pump definition, select the curve label then click **Delete**.

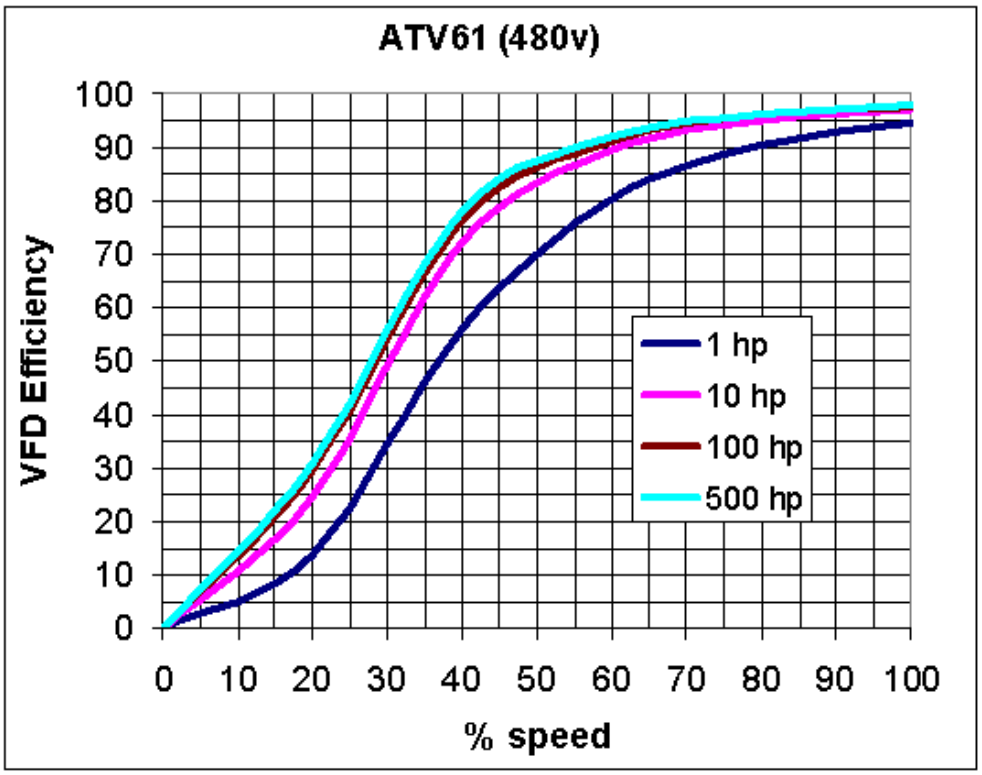
– To rename a pump definition, select the label of the pump definition you want to rename, click **Rename**, then type the new name.

– To view a report on a pump definition, select the label for the pump definition, then click **Report**.

6. Click **Close** to close the dialog box.

Efficiency Points Table

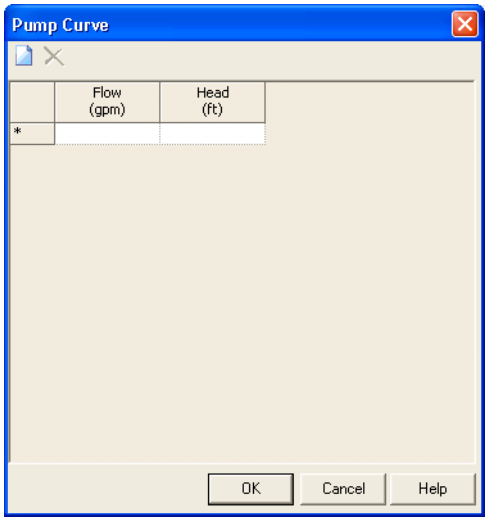
A variable speed drive introduces some inefficiency into the pumping system. The user needs to supply a curve relating variable speed drive efficiency to pump speed. This data should be obtained from the variable speed drive manufacturer but is often difficult to find. Variable frequency drives (VFD) are the most common type of variable speed drive used. The graph below shows the efficiency vs. speed curves for a typical VFD: Square D (Schneider Electric) model ATV61:



Pump Curve Dialog Box

This dialog is used to define the points that make up the pump curve that is associated with the Pump Curve Library entry that is currently highlighted in the Engineering Library Manager explorer pane.

The Pump Curve dialog is only available for Multiple Point pump type. The pump is defined by entering points in the Flow vs. Head table. Click the New button to add a new row and click the Delete button to delete the currently highlighted row.

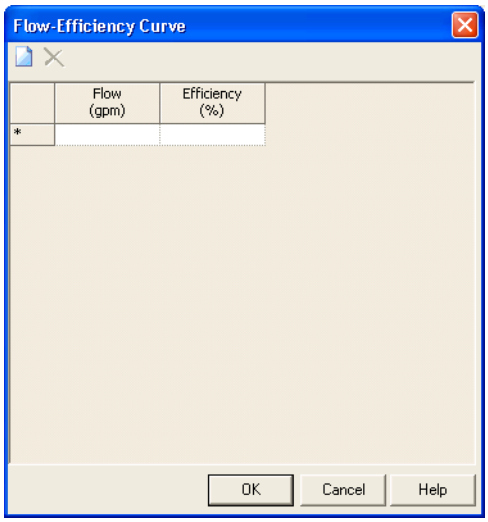


For more information about Engineering Libraries, see Engineering Libraries.

Flow-Efficiency Curve Dialog Box

This dialog is used to define the points that make up the flow-efficiency curve that is associated with the Pump Curve Library entry that is currently highlighted in the Engineering Library Manager explorer pane.

The Flow-Efficiency Curve dialog is only available for the Multiple Efficiency Points efficiency curve type. The curve is defined by entering points in the Flow vs. Efficiency table. Click the New button to add a new row and click the Delete button to delete the currently highlighted row.

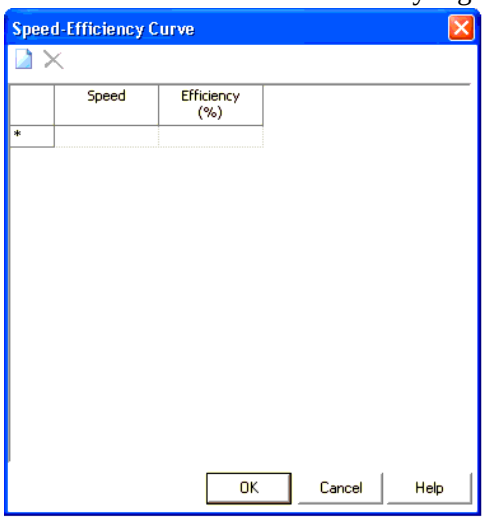


For more information about Engineering Libraries, see Engineering Libraries.

Speed-Efficiency Curve Dialog Box

This dialog is used to define the points that make up the speed-efficiency curve that is associated with the Pump Curve Library entry that is currently highlighted in the Engineering Library Manager explorer pane

The Speed-Efficiency Curve dialog is only available for Variable Speed Drive pumps (Is Variable Speed Drive? is set to True). The curve is defined by entering points in the Speed vs. Efficiency table. Click the New button to add a new row and click the Delete button to delete the currently highlighted row.

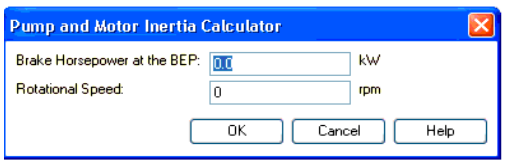


For more information about Engineering Libraries, see Engineering Libraries.

Pump and Motor Inertia Calculator

If the motor and pump inertia values are not available, you can use this calculator to determine an estimate by entering values for the following attributes:

- **Brake Horsepower at the BEP:** The brake horsepower in kilowatts at the pump's BEP (best efficiency point).
- **Rotational Speed:** The rotational speed of the pump in rpm.



When you click the OK button, the calculated inertia value will be automatically populated in the Inertia (Pump and Motor) field on the WaterCAD V8*i* tab of the Pump Definition dialog.

The calculator uses the following empirical relation developed by Thorley

$$I_{motor} = 118 \times (P/N)^{1.48} kgm^2$$

$$:I_{pump} = 1.5 \times 10^7 \times (P/N^3)^{0.9556} kgm^2$$

where: P is the brake horsepower in kilowatts at the BEP

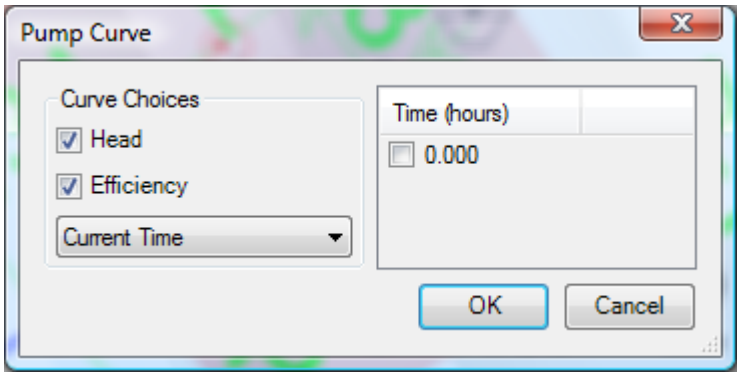
N is the rotational speed in rpm

If uncertainty in this parameter is a concern, several simulations should be run to assess the sensitivity of the results to changes in inertia.

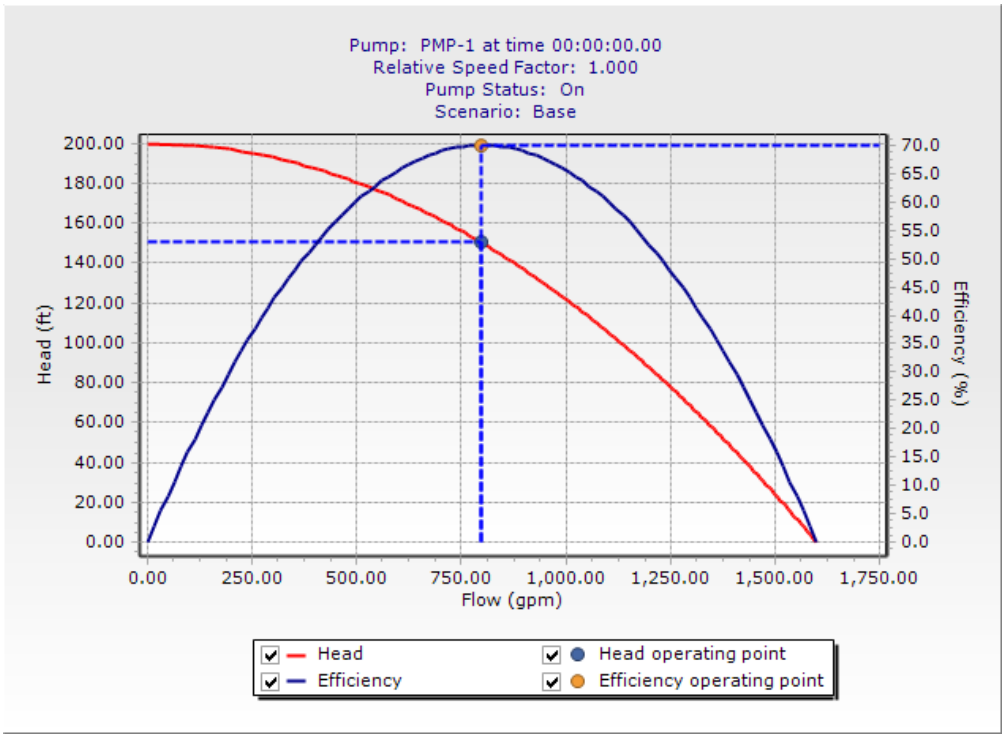
$$I_{pump} = 1.5 \times 10^7 \times (P/N^3)^{0.9556} kgm^2$$

Pump Curve Display

The user can obtain a display of pump curves (after a run) by right clicking on the pump and selecting Pump Curve. The user then sees a dialog where the type of curve and time steps, for which the curve is plotted, are controlled.



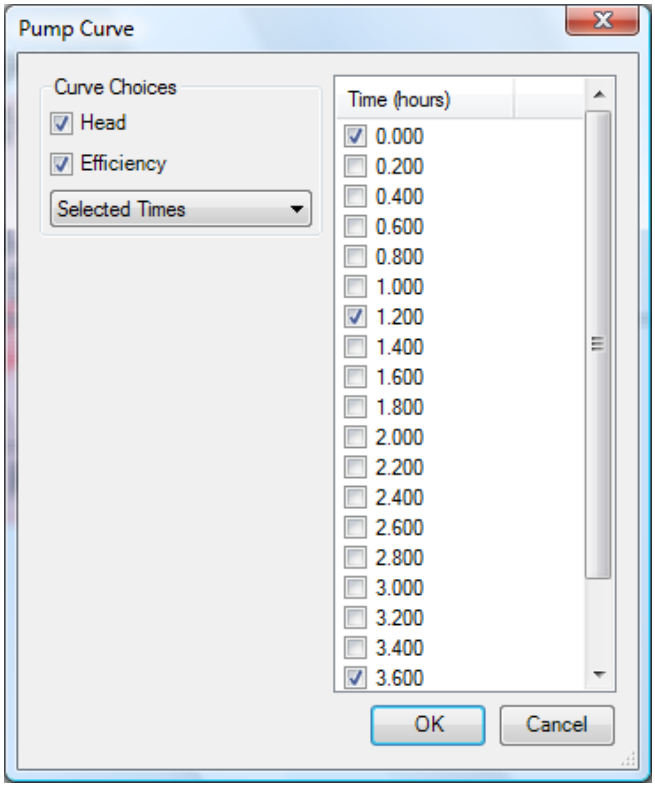
The default options are to plot both the head and efficiency curve at the current time. The types of curves can be turned off by unchecking the boxes. A plot for a single time step look like the graph below.



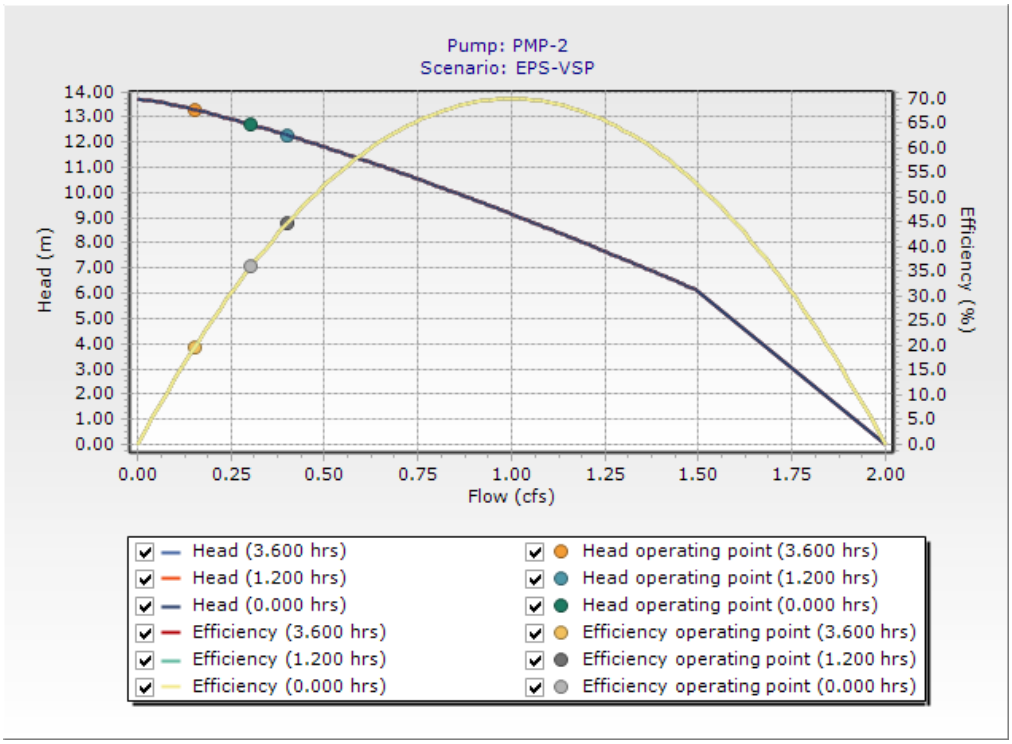
The graph shows both the head and efficiency curve and highlights the operating point for the current time step. If the pump is Off, the operating point is plotted at the origin.

The buttons on top of the drawing control the display. The first button enables the user to modify the look of the graph by changing colors, fonts, legends, etc. The second button prints the graph while the third is a print preview. The fourth copies the graph to the clipboard.

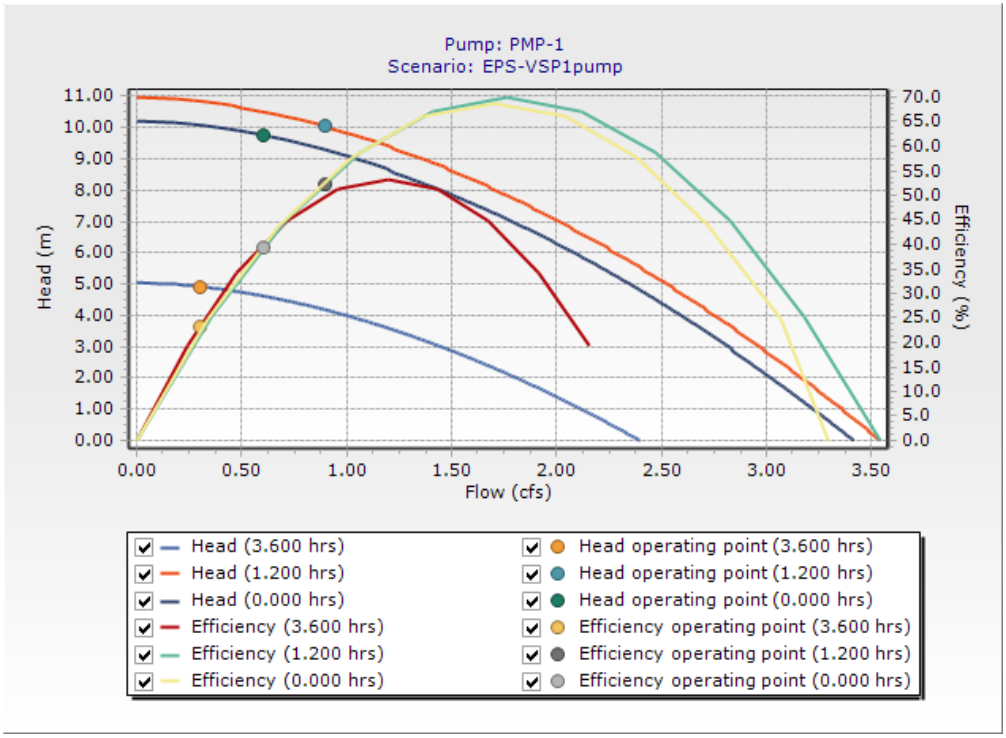
In the case of an EPS run, if the user wants to view more than the current time step, he should pick Selected Times from the drop down.



If the pump is a constant speed pump, then a single head and efficiency curve are shown with multiple points showing each selected time.



If a variable speed pump is selected, then a separate head and efficiency curve are generated for each time step.



If the user picks Current Time for an EPS run, it is possible to user the Time Browser to animate the pump curve and operating points moving over time.

Pump Curve Combinations

WaterCAD V8*i* provides a number of ways to view pump curves including Components > Pump Definition which shows all available pump curves, and right clicking on a pump and selecting Pump Curves once a run is complete. Users also need to view the performance of multiple pumps running together in parallel in a pump station. To do this it is first necessary to include the pumps in a Pump Station element. This can be done by opening the property grid for the pump, picking the Pump Station property and selecting the pump station in which this pump is located. It is usually advisable to draw the pump station polygon to include the pumps within the polygon.

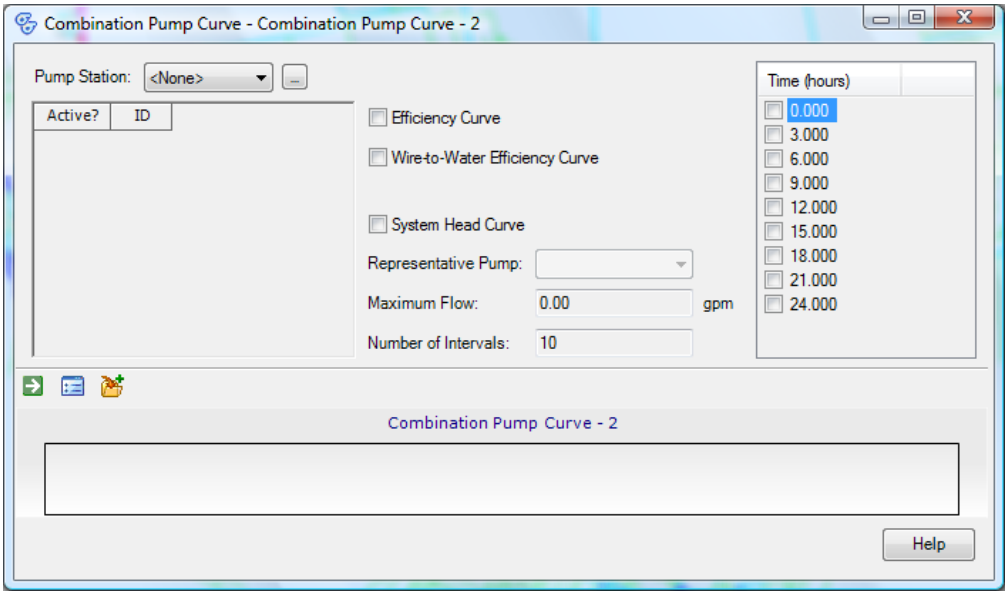
The pump head and efficiency characteristic curves are properties solely of the pump and can be displayed even if the model only consists of the pump station with the pumps. If the user wants to display system head curves, then the pump station must be part of a valid hydraulic model.

To start the Combination Pump Curve feature to view the curves either

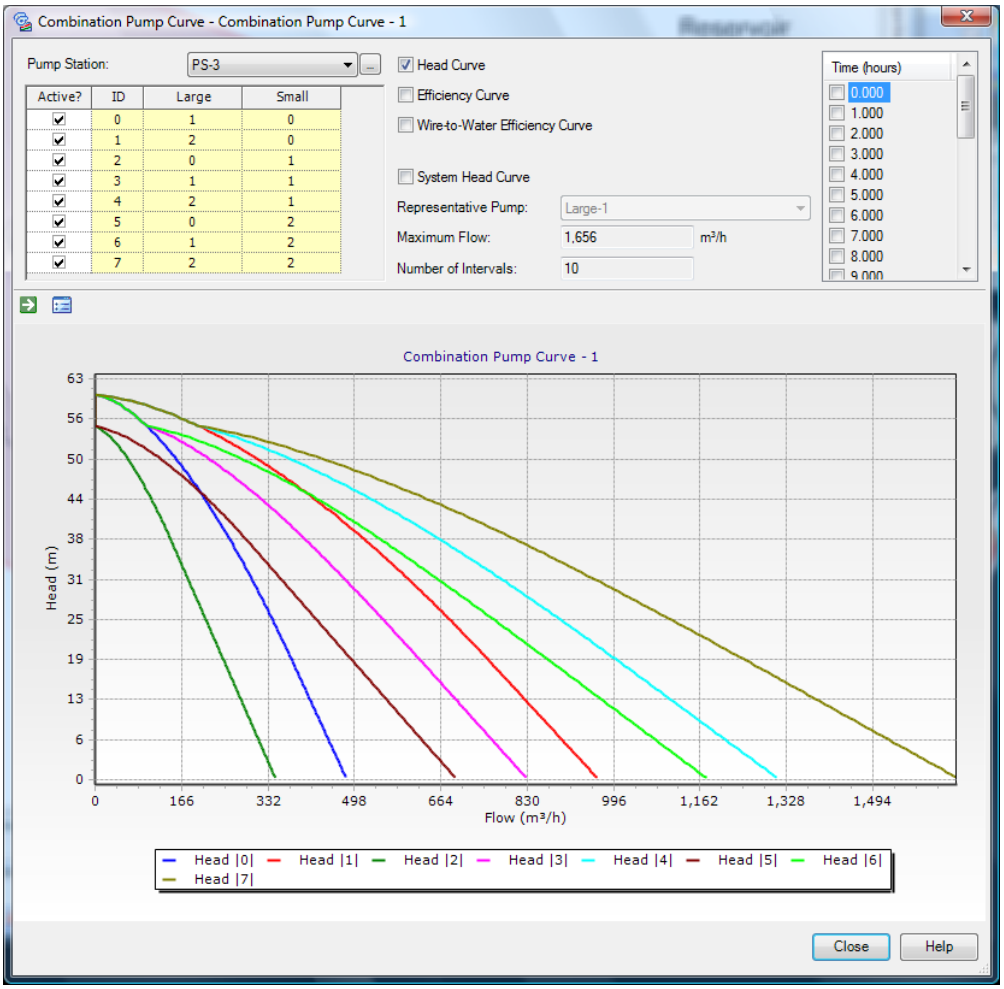
1. Select Analysis > Combination Pump Curve
2. Right click on the Pump Station and select Combination Pump Curves

Pump Curve Combination Editor

Upon opening a Combination Pump Curve dialog, the user must first select which pump station is to be analyzed by either selecting one of the previously used pump stations from the drop down or picking the ellipse (…) button and selecting the station from the drawing.



Once the pump station has been selected, the dialog displays the possible pump combinations in the top left pane and the head curves in the bottom pane.



The column marked "Active" is checked if the user wants that combination displayed in the graph.

The column "ID" displays the index on the curve in the graph (e.g. Head[1] is the curve corresponding to the head of the pump combination with ID = 1).

There is one column in the table for each pump definition referenced in that pump station. The number in the cell indicates the number of pumps of that definition that are running for the combination corresponding to that row. If there is a zero in a cell, the pump is off for that combination.

The top middle pane determines which type of pump or system curve is displayed. By default, only the Head characteristic curve is displayed. The user can also turn on the (pump) efficiency or wire-to-water (overall) efficiency curves.

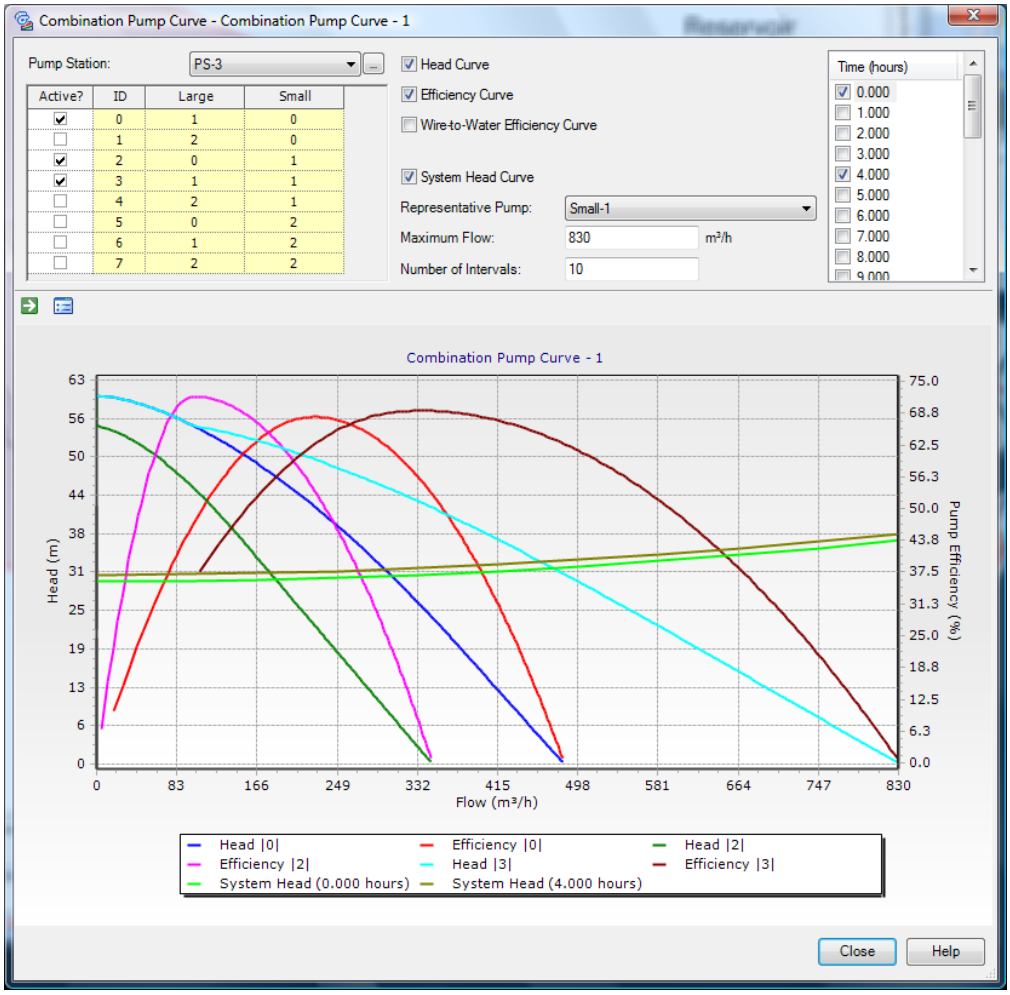
The system head curves are a property of the system calculated from the perspective of a pump. When the System Head Curve box is checked, the user must specify which pump is the Representative Pump which means which path through the station is head loss calculated. Usually the results don't vary significantly depending on which pump is selected.

The Maximum flow and Number of Intervals entries determine the horizontal extent of the system head curve and the number of points along the curve that will be calculated.

The top right pane is used to account for the fact that the system head curve will depend somewhat on the time of day. The user must select at least one time step to use in determining the system head curve. If the user selects a time step in which the pump is discharging into a closed system with no pressure dependent demands, the system head curve may show very high or low values for head. Do not select time steps where this occurs.

In order to run or rerun the pump combination graph, select the green Compute button at the top left of the bottom pane.

The graph below shows an example with three different combinations for two time steps (system head curves).



If the user wants to change the look of the graph such as the range of head values, use the second button in the bottom pane. That opens the graphing manager. To change the axis range, pick Chart > Axes > Left Axis > Maximum > Change and enter a new value. See Graphs for more details.

Variable Speed Pump Battery

A Variable Speed Pump Battery element represents multiple variable speed pumps that meet the following criteria:

1. the VSPs are parallel with each other (not in-line)
2. the VSPs are sharing common upstream (inflow) and downstream (outflow) nodes
3. the VSPs are identical (have the same pump definition)
4. the VSPs are controlled by the same target node and the same target head.

Parallel variable speed pumps (VSPs) are operated as one group and led by a single VSP, the so-called **lead** VSP, while the other VSPs at the same battery are referred as to as **lag** VSPs. A lag VSP turns on and operates at the same speed as the lead VSP when the lead VSP is not able to meet the target head and turns off when the lead VSP is able to deliver the target head or flow.

From the standpoint of input data, Variable Speed Pump Batteries are treated exactly the same as single pump elements that are defined as variable speed pumps of the Fixed Head Type with one exception; number of Lag Pumps must be defined in the Lag Pump Count field.

When simulating a Pump Battery in a transient analysis, the pump battery is converted to an equivalent pump using the following conversion rules:

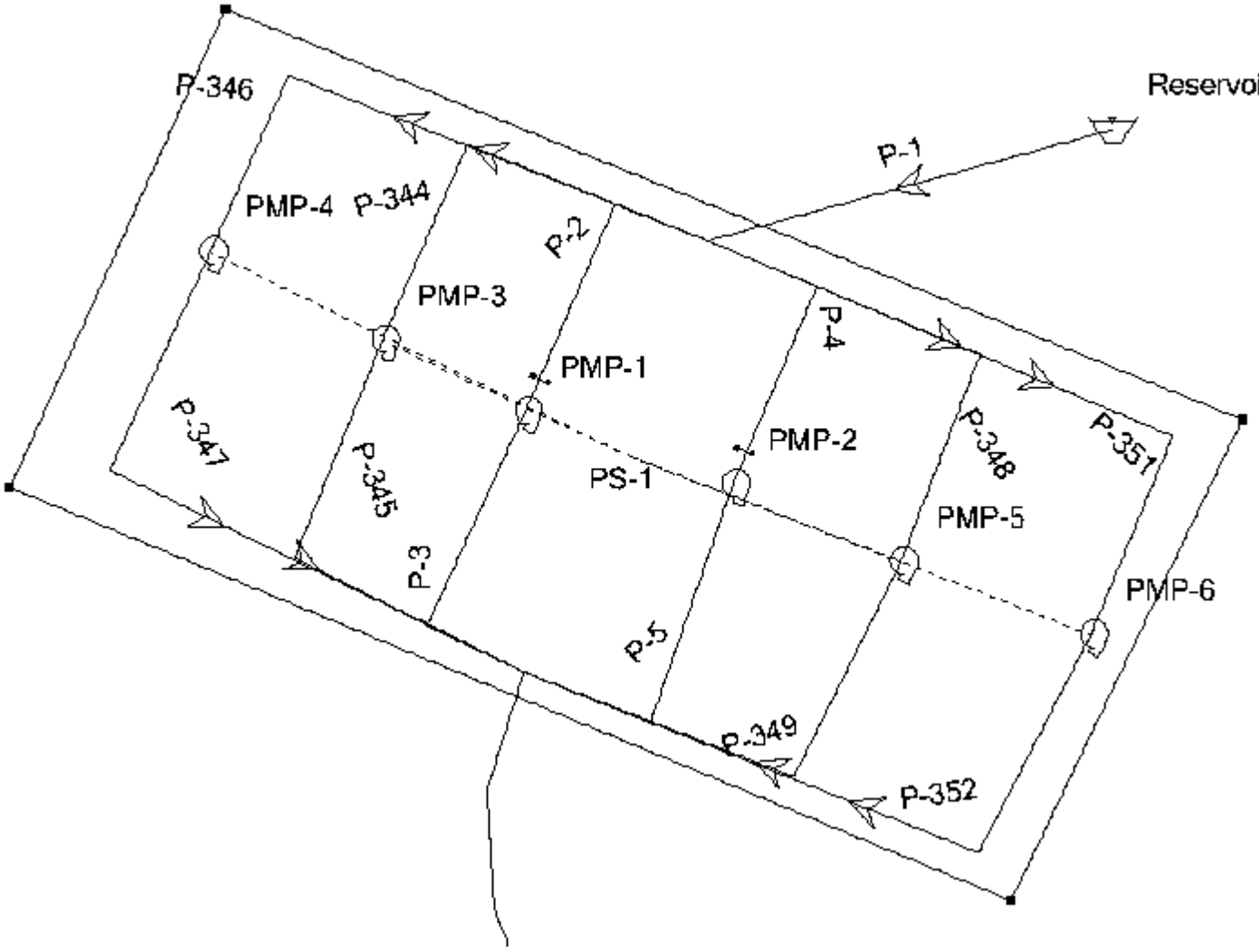
1. The Flow (Initial) of the equivalent pump is the total flow of all the running pumps in the pump battery.
2. The Inertia of the Pump and Motor of the equivalent pump is the sum of all the inertia values for all the running pumps.
3. The Specific Speed of the equivalent pump is the Specific Speed value that is closest to the result of the following equation:

   sqrt(number of running pumps) * Specific Speed of pump battery

Pump Stations

A pump station element provides a way for a user to indicate which pumps are in the same structure, serving the same pressure zone. It provides a graphical way to display the pumps associated with the station. A pump station is not a hydraulic element in that it is not directly used in a hydraulic analysis but rather it is a collection of pumps which are the hydraulic elements.

A pump station is a polygon element which displays which pumps are in the station by dashed lines connecting the pumps with the station polygon centroid. A pump does not need to be inside the polygon to be a pump assigned to the station and pumps inside the polygon still need to be assigned to the station. The only information saved with a pump station is the geometry of the station and the list of pumps assigned to the station.



A pump station element is useful in calculating and displaying an analysis of pump combinations (see Pump Curve Combinations).

Usually the pumps and associated piping are laid out before the station is drawn. However, the station polygon can be drawn first. The station element is created by picking the pump station element icon  from the layout menu and drawing a polygon around the extents of the station. When the polygon is complete, the user right clicks and selects "Done".

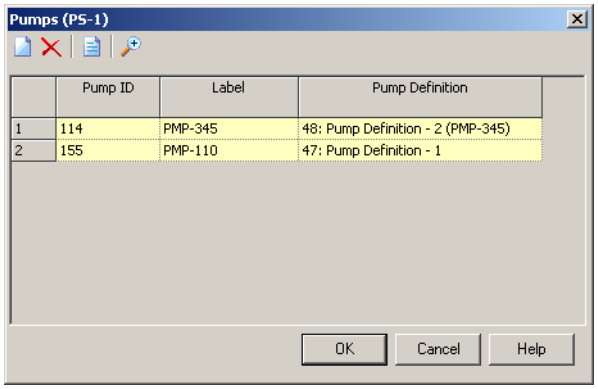
Individual pump elements are assigned to a station by selecting the pump element and in the Pump Station property, picking the pump station which the pump is associated. A dashed line is drawn from the pump to the station. This also can be done in the physical alternative for pumps. To assign several pumps at once, a global edit can be used provided that at least one pump has already been assigned to that station.

Sometimes a pump station structure can house pumps pumping to more than one pressure zone (e.g. medium service and high service). For the purposes of WaterCAD V8*i*, this would be two (or more) pump station polygon elements, one for each pressure zone.

The property grid contains a Controls collection field that opens a filtered controls editor that only displays the controls associated with the pumps in the selected pump station.

Pumps Dialog Box

This dialog allows you to view the collection of pumps assigned to a pump station element.



Click the **New** button to select a pump from the drawing view to be added to the pump station. Click **Delete** to remove the currently highlighted pump from the pump station. Click the **Report** button to generate a report containing the list of pumps included in the pump station as well as their associated pump definitions. Click the **Zoom To** button to focus the drawing view on the pump that is highlighted in the list.

Polygon Vertices Dialog Box

This dialog box lets you define X vs. Y points that plot the shape of the polygon that represents the selected element. The dialog box contains the X vs. Y table that allows you to define any number of points and the following buttons:

New—Creates a new row in the table.

Delete—Deletes the currently highlighted row from the table.

SCADA Elements



SCADA Element

Define the SCADA element using the following properties:

Target Element: The domain element that the ASCADA Signal targets.

Real-Time Signal: The signal returning realtime values for the selected attribute.

Historical Signal: The signal returning historical value(s) for the selected attributes.

Target Element (Storage Unit): Displays the storage unit used by the target element.

Field: The attribute of the target element that the SCADA signal relates to.

Valves

A valve is a node element that opens, throttles, or closes to satisfy a condition you specify. The following valve types are available in Bentley WaterCAD V8*i* :

Valve Type	Description
Pressure Reducing Valve (PRV)	PRVs throttle to prevent the downstream hydraulic grade from exceeding a set value. If the downstream grade rises above the set value, the PRV will close. If the head upstream is lower than the valve setting, the valve will open fully.
Pressure Sustaining Valve (PSV)	A Pressure Sustaining Valve (PSV) is used to maintain a set pressure at a specific point in the pipe network. The valve can be in one of three states: • partially opened (i.e., active) to maintain its pressure setting on its upstream side when the downstream pressure is below this value • fully open if the downstream pressure is above the setting • closed if the pressure on the downstream side exceeds that on the upstream side (i.e., reverse flow is not allowed).
Pressure Breaker Valve (PBV)	PBVs are used to force a specified pressure (head) drop across the valve. These valves do not automatically check flow and will actually boost the pressure in the direction of reverse flow to achieve a downstream grade that is lower than the upstream grade by a set amount.
Flow Control Valve (FCV)	FCVs are used to limit the maximum flow rate through the valve from upstream to downstream. FCVs do not limit the minimum flow rate or negative flow rate (flow from the To Pipe to the From Pipe).
Throttle Control Valve (TCV)	TCVs are used as controlled minor losses. A TCV is a valve that has a minor loss associated with it where the minor loss can change in magnitude according to the controls that are implemented for the valve. If you don't know the headloss coefficient, you can also use the discharge coefficient, which will be automatically converted to an equivalent headloss coefficient in the program. To specify a discharge coefficient, change the Coefficient Type to Discharge Coefficient.

Valve Type	Description
General Purpose Valve (GPV)	GPVs are used to model situations and devices where the flow-to-headloss relationship is specified by you rather than using the standard hydraulic formulas. GPVs can be used to represent reduced pressure backflow prevention (RPBP) valves, well draw-down behavior, and turbines.
Isolation Valves	Isolation Valves are used to model devices that can be set to allow or disallow flow through a pipe. Note that for Isolation valves, “Left” as referred to by the **Is offset to the left of referenced link?** property is “left” relative to the pipe's coordinate system (which is the alignment of the pipe), and not the absolute or world coordinate system. When an isolation valve is placed, a pipe bend is added at the location of the valve; that way if the pipe’s end node(s) are moved later the valve will remain attached to the pipe. If an isolation valve is closed, it will report N/A for HGL and Pressure results.

Applying a Zone to a Valve

You can group elements together by any desired criteria through the use of zones. A Zone can contain any number of elements and can include a combination of any or all element types. For more information on zones and their use, see Zones on page 4-450.

To Apply a Previously Created Zone to a Valve:

1. Select the valve in the Drawing View.
2. In the **Properties** window, click the menu in the **Zone** field and select the zone you want.

Applying Minor Losses to a Valve

Valves can have an unlimited number of minor loss elements associated with them. Minor losses are used on pressure pipes and valves to model headlosses due to pipe fittings or obstructions to the flow.

If you have a single minor loss value for a valve, you can type it in the Minor Loss field of the Properties window. If you have multiple minor loss elements for a valve and would like to define a composite minor loss, or would like to use a predefined minor loss from the Minor Loss Engineering Library, access the Minor Losses dialog by clicking the ellipsis button in the Minor Losses field of the Properties window.

To Apply a Minor Loss to a Valve

1. Select the valve in the **Drawing View**.
2. In the **Properties** window, type the minor loss value in the **Minor Loss** field.

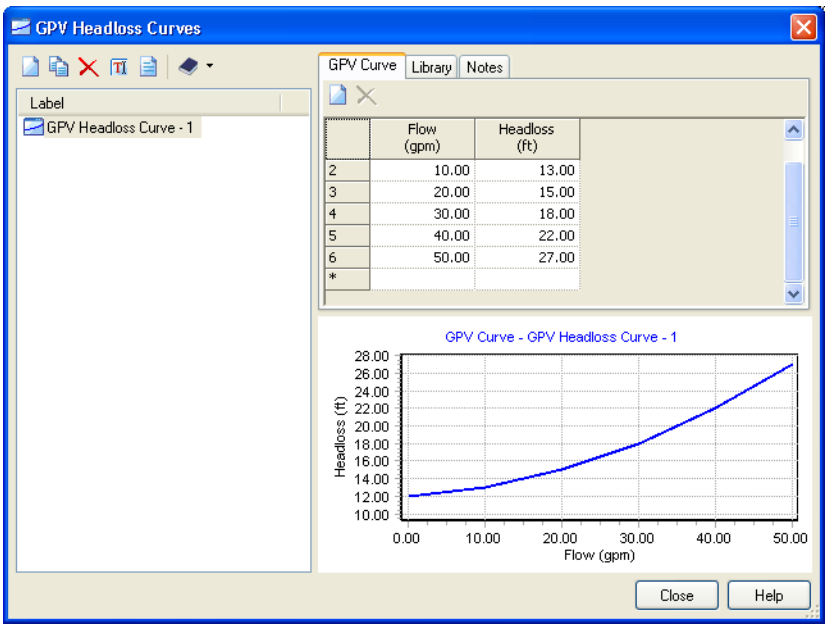
To Apply Composite Minor Losses to a Valve

1. Click a valve in your model to display the Property Editor, or right-click a valve and select **Properties** from the shortcut menu.
2. In the Physical: Minor Losses section of the Property Editor, set the **Specify Local Minor Loss?** value to **False**.
3. Click the **Ellipses (...)** button next to the **Minor Losses** field.
4. In the **Minor Losses** dialog box, each row in the table represents a single minor loss type and its associated headloss coefficient. For each row in the table, perform the following steps:
   a. Type the number of minor losses of the same type to be added to the composite minor loss for the valve in the **Quantity** column, then press the Tab key to move to the **Minor Loss Coefficent** column.
   b. Click the arrow button to select a previously defined Minor Loss, or click the **Ellipses (...)** button to display the Minor Loss Coefficients to define a new Minor Loss.
5. When you are finished adding minor losses to the table, click **Close**. The composite minor loss coefficient for the minor loss collection appears in the Property Editor.
6. Perform the following optional steps:
   – To delete a row from the table, select the row label then click **Delete**.
   – To view a report on the minor loss collection, click **Report**.

Defining Headloss Curves for GPVs

A General Purpose Valve (GPV) element can be used to model head loss vs. flow for devices that cannot be adequately modeled using either minor losses or one of the other control valve elements. Some examples of this would included reduced pressure backflow preventers (RPBP), compound meters, well draw down, turbines, heat exchangers, and in-line granular media or membrane filters.

To model a GPV, the user must define a head loss vs. flow curve. This is done by picking Component > GPV Head Loss Curve > New. The user would then fill in a table with points from the curve.



The user can create a library of these curve or read them from a library. Because there is so much variability in the equipment that can be modeled using GPVs, there is no default library.

Once the GPV head loss curve has been created, the user can place GPV elements like any other element. Once placed, the user assigns a head loss curve to the specific GPV using "General Purpose Head Loss Curve" in the property grid.

A GPV can also have an additional minor loss. To specify that, the user must provide a minor loss coefficient and the (effective) diameter of the valve.

A GPV does not act as a check valve. Flow can move in either direction through the valve. Therefore, when modeling a device like a RPBP, it may be necessary to place a check valve on one of the adjacent pipes to account for that behavior.

Note that minor losses do not apply to the following valve types: **General Purpose Valve** and **Valve With Linear Area Change**. These two valve types do not support a (fully) open status and always apply the head/flow relationship defined by their head-loss curve and discharge coefficient respectively.

To Define a Headloss Curve

1. Select the GPV in the **Drawing View.**
2. In the **Properties** window, click the menu in the **GPV Headloss Curve** field and select **Edit GPV Headloss Curves**.

3. In the **GPV Headloss Curves** dialog that appears, click the **New** button. Enter a name for the curve, or accept the default name.
4. Define at least two points to describe a headloss curve. A point consists of a flow value for each headloss value in the Flow vs. Headloss table. The curve will be plotted in the curve display panel below the table.
5. Click the **Close** button.

To Import a Predefined Headloss Curve From an Engineering Library

1. Select the GPV in the **Drawing View**.
2. In the **Properties** window, click the menu in the **GPV Headloss Curve** field and select **Edit GPV Headloss Curves**.
3. In the **GPV Headloss Curves** dialog that appears, click the **New** button. Enter a name for the curve, or accept the default name.
4. Click the **Synchronization Options** button and select **Import From Library**.
5. In the **Engineering Libraries** dialog that appears, click the **plus** button to expand the **GPV Headloss Curves Libraries** node, then click the **plus** button to expand the node for the library you want to browse.
6. Select the headloss curve entry you want to use and click the **Select** button.
7. Click the **Close** button.

Defining Valve Characteristics

You can apply user-defined valve characteristics to any of the following valve types:

- PRV
- PSV
- PBV
- FCV
- TCV
- GPV

To create a valve with user-defined valve characteristics:

1. Place a PRV, PSV, PBV, FCV, TCV, or GPV valve element.
2. Double-click the new valve to open the Properties editor.
3. In the WaterCAD V8*i* Data section, change the Valve Type to User Defined.
4. In the Valve Characteristics field, select Edit Valve Characteristics.

5. Define the valve characteristics in the Valve Charateristics dialog that opens.
6. In the Valve Characteristics field, select the valve characteristic definition that the valve should use.

Note: If the Valve Characteristic Curve is not defined then a default curve will be used. The default curve will have (Relative Closure, Relative Discharge Coefficient) points of (0,1) and (1,0).

Valve Characteristics Dialog Box

The following management controls are located above the valve characteristic list pane:

New 	Creates a new valve characteristic definition.
Duplicate 	Creates a copy of the currently highlighted valve characteristic definition.
Delete 	Deletes the valve characteristic definition that is currently highlighted in the list pane. You can hold down the Ctrl key while clicking on items in the list to select multiple entries at once.
Rename 	Renames the valve characteristic definition that is currently highlighted in the list pane.
Report 	Opens a report of the data associated with the valve characteristic definition that is currently highlighted in the list pane.
Synchronization Options 	Browses the Engineering Library, synchronizes to or from the library, imports from the library or exports to the library.

The tab section is used to define the settings for the minor loss that is currently highlighted in the valve characteristic list pane. The following controls are available:

Valve Characteristic Tab

This tab consists of input data fields that allow you to define the valve characteristic.

Relative Closure

The ratio of valve stroke/travel to the total stroke/travel required to close the valve. A Relative Closure of 100% represents a fully closed valve.

Relative Discharge Coefficient

The discharge coefficient of the valve relative to the fully open discharge coefficient. A Relative Discharge Coefficient of 100% represents a fully open valve (exactly equal to the fully open discharge coefficient) and 0% represents a discharge coefficient of zero (fully closed).

Library Tab

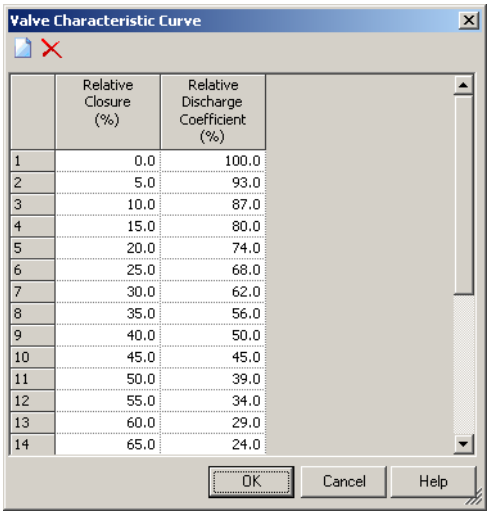
This tab displays information about the valve characteristic that is currently highlighted in the valve characteristic list pane. If the valve characteristic is derived from an engineering library, the synchronization details can be found here. If the valve characteristic was created manually for this project, the synchronization details will display the message Orphan (local), indicating that the valve characteristic was not derived from a library entry.

Notes Tab

This tab contains a text field that is used to type descriptive notes that will be associated with the valve characteristic that is currently highlighted in the valve characteristic list pane.

Valve Characteristic Curve Dialog Box

This dialog is used to define a valve characteristic entry in the Valve Characteristics Engineering Library.



The dialog consists of a table containing the following attribute columns:

- **Relative Closure**: Percent opening of the valve (100% = fully closed, 0% = fully open).
- **Relative Discharge Coefficient**:The discharge coefficient of the valve relative to the fully open discharge coefficient. A Relative Discharge Coefficient of 100% represents a fully open valve (exactly equal to the fully open discharge coefficient) and 0% represents a discharge coefficient of zero (fully closed).

Click New to add a new row to the table. Click Delete to remove the currently highlighted row from the table. You can hold down the Ctrl key while clicking on items in the list to select multiple entries at once.

General Note About Loss Coefficients on Valves

Valves are modeled as links (like pipes) in the steady state / EPS engine and as such the engine supports the notion of minor losses in fully open links. This is to account for such things as bends and fittings, or just the physical nature of the link (element). However, note that the minor loss for a valve only applies when the valve is fully open (inactive) and not restricting flow. For example, a flow control valve (FCV) that has a higher set flow than the hydraulics provide for, is fully open and not limiting the flow passing through. In this case the computation will use any minor loss on the FCV and

calculate the corresponding head loss. If on the other hand the set flow of the FCV was low enough for the valve to be required to operate, the head loss across the valve is determined by the function of the valve. In this case the head loss would be the value corresponding to the function of reducing the flow to the set value of the FCV.

The purpose of several of the valve types included in WaterCAD V8*i* is simply to impart a head loss in the system, similar in some ways to a minor loss. One example here is the Throttle Control Valve (TCV). The TCV supports a head loss coefficient (or discharge coefficient) that is used to determine the head loss across the valve. It is important to note, however, that the head loss coefficient on the TCV is actually different from a minor loss in the way it is used by the computation. The minor loss applies when the valve is fully open (inactive) and the head loss coefficient applies when the valve is active. This same principle applies to other valve types such as General Purpose Valves (GPVs), Pressure Breaker Valves (PBVs) and Valves with a Linear Area Change (VLAs), the only difference being that GPVs use a headloss/flow curve, PBVs use a headloss value and VLAs use a discharge coefficient, instead of a head loss coefficient, to define the valve's behavior when it is in the active state.

In some cases a minor loss coefficient sounds like it could be a duplicate of another input value, but the way in which it is used in the computation is not the same.

Modulating Control Valve

Control valves, such as pressure reducing valves (PRV), modify their opening to control pressure or flow in the system. For example, PRV's adjust valve position to reduce inlet pressure meet a target outlet pressure.

Through HAMMER V8i SELECT series 3, HAMMER maintained a constant valve position throughout a transient analysis. In many cases that opening is correct, but there are instances where the valve position will modulate significantly in response to the transient and must be accounted for. In some instances, valve modulation can contribute to transient problems.

With SELECT series 4, there is a new PRV property "Modulate Valve during Transient" which, when set to True, enables HAMMER to adjust the valve opening during a transient run. The default value for this property is False. This property is saved in the Transient alternative.

When "Modulate Valve during Transient" is set to True, the user must set the "Opening rate coefficient" and Closure rate coefficient". The units for these properties are % change in opening/second/foot of HGL difference between the control valve setting and the calculated pressure at the previous time step (xxx %/sec/ft or yyy %/sec/m). These values are highly valve specific. The default values are for both rates. The closing and opening rates for a given valve may be different. Values will be lower for larger valves and will be much higher for direct acting valves than pilot controlled valves. The values should be calibrated using high speed pressure loggers. A reasonable initial estimate may be on the order of 0.1.

The valve position is calculated in HAMMER as

$V(t+1) = V(t) + c_r (H(t) - H_s) dt$, if $H(t) > Hs$

$V(t+1) = V(t) + c_o (H(t) - H_s) dt$, if $H(t) < Hs$

Where:

V= valve position (% closed)

c_r = closing rate (%/s/ft)

c_r = opening rate (%/s/ft)

H_s = target outlet hydraulic grade (ft)

H(t) = outlet hydraulic grade at time t (ft)

dt = time step size, s

If the opening or closing rates are set too high, it is possible to create numerical instability in HAMMER.

When using modulating control valves, it is necessary to specify either a non-zero fully open minor loss coefficient or discharge coefficient. This value is set in the property "Valve coefficient type".

While modulation is possible in any type of control valve, HAMMER SELECT series 4 only supports this behavior in PRV's.

Inaccurate results may occur if the valve becomes fully open or fully closed during a run or the pressure drops below vapor pressure at the valve. The percent closure for the valve can be found in temporary file C:\Users\FirstName.LastName\AppData\Local\Temp\Bentley\HAMMER\ PRVCLOSURE.TXT.

If the user selects False for "Modulate Valve during Transient", it is still possible to adjust valve opening during a transient run by changing the default value for "Operating Rule" from Fixed to an Operational (Transient Valve) pattern that the user has established under Patterns. In these patterns, the relative closure is a function of time. (See help topic Pattern Manager.)

Spot Elevations

Spot elevations can be placed to better define the terrain surface throughout the drawing. They have no effect on the calculations of the network model. Using spot elevations, elevation contours and enhanced pressure contours can be generated with more detail. The only input required for spot elevation elements is the elevation value.

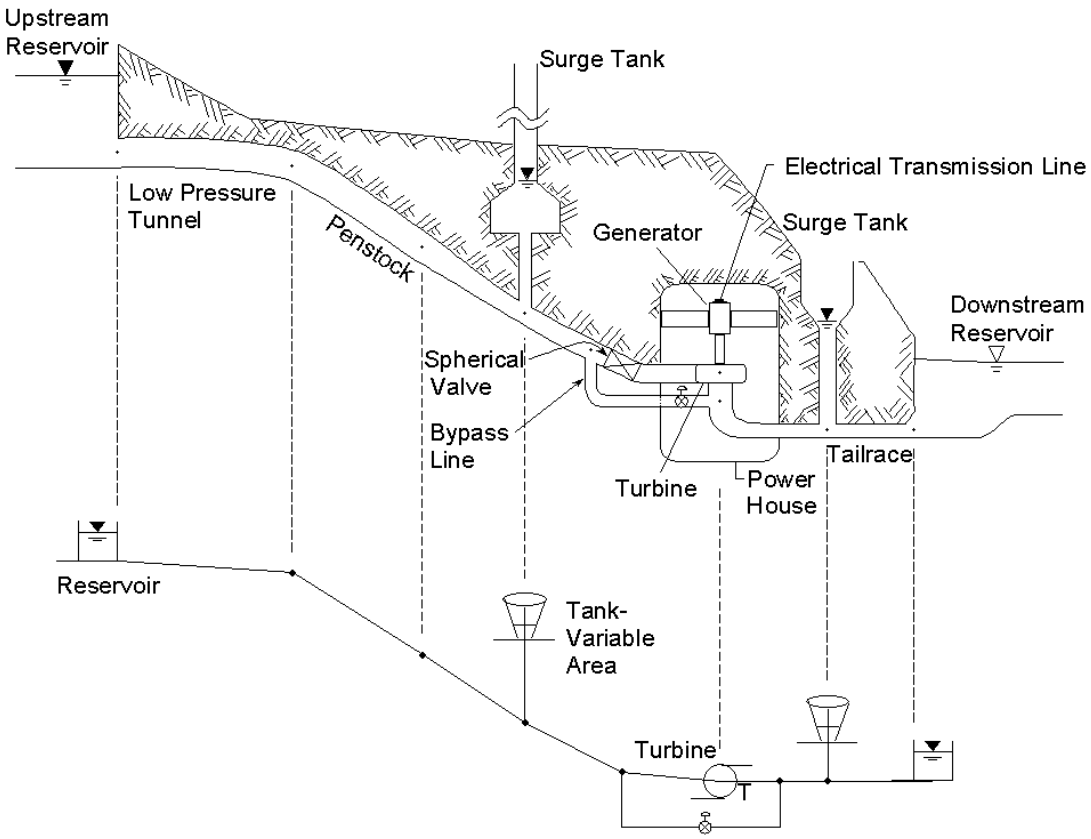
Turbines

A turbine is a type of rotating equipment designed to remove energy from a fluid. For a given flow rate, turbines remove a specific amount of the fluid's energy head.

In a hydroelectric power plant, turbines convert the moving water’s kinetic energy to mechanical (rotational) energy. Each turbine is mechanically coupled with a generator that converts rotational energy to electrical energy. Each generator's output terminal transmits electricity to the distribution grid. At steady state, the electricity produced by the turbine-generator system is equal to the electrical grid load on the generator.

The figure below is a generalized schematic of a hydroelectric power generation plant. A reservoir (usually elevated) supplies a low pressure tunnel and a penstock. Water flows through the penstock under increasingly higher pressure (and velocity if diameter decreases) as it approaches the turbine. Most of the turbine's rotational energy drives a generator to produce electricity. Water emerges from the turbine through the

draft tube and tailrace and flows into the downstream reservoir. Surge tanks can be connected to the penstock and/or tailrace to limit the magnitude of transient pressures, especially if the length of the upstream conduit/penstock or if (rarely) the tailrace is relatively long.



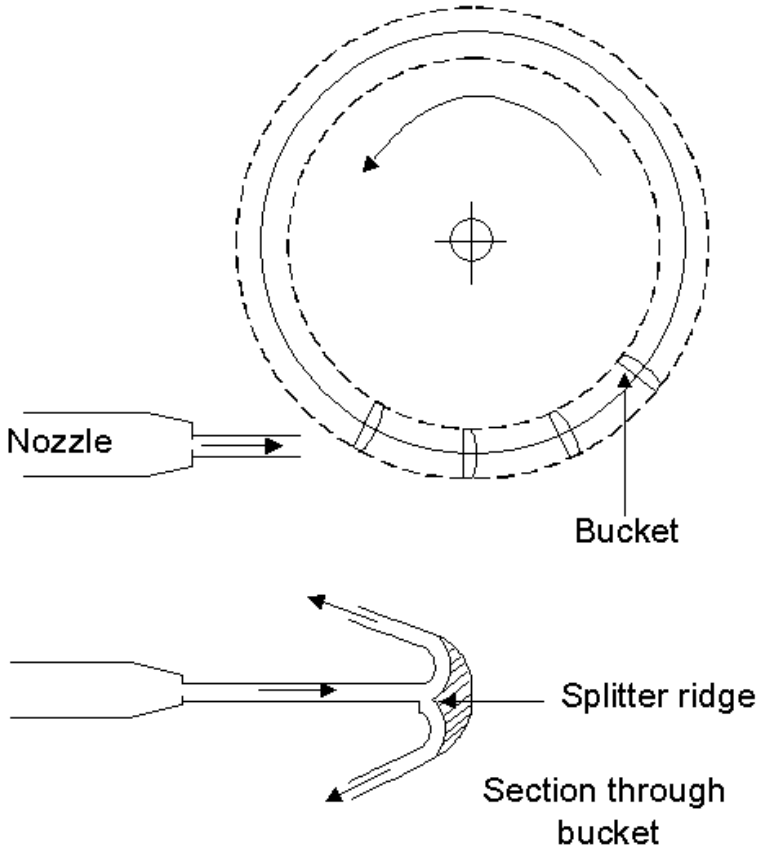
Hydraulic turbines and penstocks often operate under high pressure at steady-state. Rapid changes such as electrical load rejection, load acceptance or other emergency operations can result in very high transient pressures that can damage the penstock or equipment. During load rejection, for example, the wicket gates must close quickly enough to control the rapid rise in rotational speed while keeping pressure variations in the penstock and tailrace within established tolerances. Using Hammer, designers can verify whether the conduits and flow control equipment are likely to withstand transient pressures that may occur during an emergency.

Electrical load varies with time due to gradual variations in electricity demand in the distribution grid. Depending on the type of turbine, different valves are used to control flow and match the electrical load. Turbines can be classified into two broad categories: a) impulse turbine, and b) reaction turbine.

Impulse Turbine

An impulse turbine has one or more fixed nozzles through which pressure is converted to kinetic energy as a liquid jet(s) – typically the liquid is water. The jet(s) impinge on the moving plates of the turbine runner that absorbs virtually all of the moving water's kinetic energy. Impulse turbines are best suited to high-head applications. One definition of an impulse turbine is that there is no change in pressure across the runner.

In practice, the most common impulse turbine is the **Pelton wheel** shown in the figure below. Its rotor consists of a circular disc with several "buckets" evenly spaced around its periphery. The splitter ridge in the centre of each bucket divides the incoming jet(s) into two equal parts that flow around the inner surface of the bucket. Flow partly fills the buckets and water remains in contact with the air at ambient (or atmospheric) pressure.



Once the free jet has been produced, the water is at atmospheric pressure throughout the turbine. This results in two isolated hydraulic systems: the runner and everything upstream of the nozzle (including the valve, penstock and conduit). Model the penstock independently using regular pipe(s), valve(s) and a valve to atmosphere for the nozzle. Transients occur whenever the valve opens or closes and the penstock must withstand the resulting pressures.

Note: The turbine element in HAMMER is not used to represent impulse turbines. Transients caused by impulse turbines can be approximated in HAMMER by using a Throttle Control Valve (TCV) or Discharge to Atmosphere element to represent the turbine nozzle.

Reaction Turbines

The figure below is a schematic of a typical reaction turbine. A volute casing and a ring of guide vanes (or wicket gate around the circumference) deliver water to the turbine runner. The wicket gate controls the flow passing through the turbine and the power it generates. A mechanical and/or electrical governor senses gradual load variations on the generator and opens or closes the wicket gates to stabilize the system (by matching electrical output to grid load).

Transient Tip: Hammer currently models hydraulic transients that result from changes in variables controlled by the governor: it does not explicitly model the governor's internal operation or dynamics. Depending on the Operating Case being simulated, HAMMER either assumes the governor is 'disconnected' or 'perfect'.

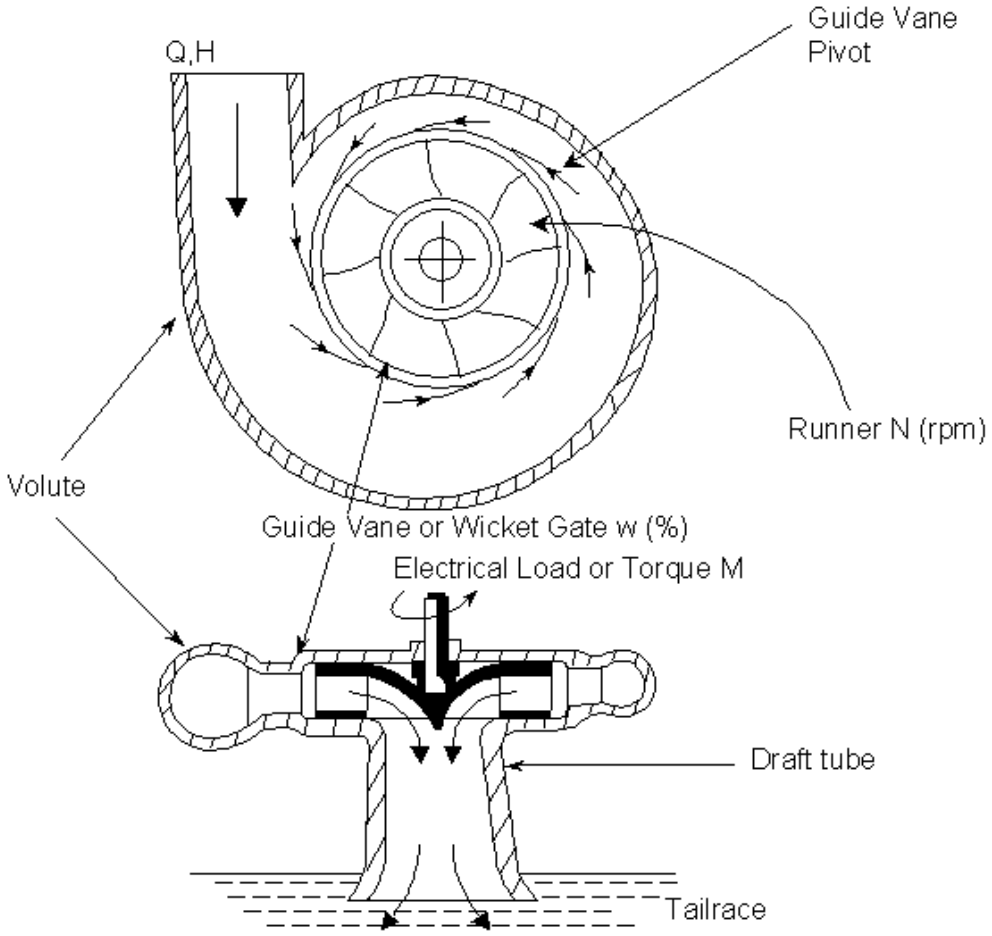
The governor is an electro or mechanical control system that may not be active – or may not react fast enough – during the emergency conditions of primary interest to modelers: instant load rejection or (rapid) load rejection. Instant load rejection assumes the governor is disconnected.

At other times, the governor will strive to match electrical output at the synchronous or 'no-load' speed: e.g. during load acceptance or load variation. Given the fact that no two governors are the same, it is useful to assume the governor is 'perfect' in those cases and that it can match the synchronous speed exactly.

The runner must always be full to keep losses to a minimum, in contrast to an impulse turbine where only a few of the runner blades are in use at any moment. Therefore, reaction turbines can handle a larger flow for a given runner size. The number of runner blades varies with the hydraulic head–the higher the head the more bladesReaction turbines are classified according to the direction of flow through the runner. In a radial-flow turbine, the flow path is mainly in the plane of rotation: water enters the rotator at one radius and leaves at a different radius–the **Francis turbine** being an example of this type. In an axial-flow turbine, the main flow direction is parallel to the axis of rotation – the **Kaplan turbine** being an example of this type. The term: **mixed flow turbine** is used when flow is partly radial and partly axial.

Each of these categories corresponds to a range of specific speeds that can be calculated from the turbine's rated power, rotational (synchronous) speed and head.

Note that there is no option in HAMMER to change the runner blade angle of a Kaplan turbine, so it is assumed the runner blade angle is constant during the transient analysis. Engineering judgment should be used to determine if this approximation is satisfactory in each case.



The primary hydraulic variables used to describe a turbine in the above schematic are:

Q = Flow
H = Head
N = Rotational speed
I = Rotational Inertia
w = Wicket gate position (% open)
M = Electrical load or torque

Modeling Hydraulic Transients in Hydropower Plants

In a hydropower generation plant, it is essential to predict the transient pressures that could occur and to implement an adequate surge control strategy to ensure the safety and reliability of the unit. The impact of gradual or diurnal load variations on the turbine-generator may be of interest during normal operations but an electric or mechanical governor can control moderate transients.

The primary purpose of hydraulic transient simulations is therefore to protect the system against rapid changes in the electrical and/or hydraulic components of the hydroelectric system. In each case, hydraulic transients result from changes in the variables controlled by the governor.

Electrical Load or Torque on the turbine-generator system varies with the electrical load in the distribution grid. In steady-state operation, the electrical torque and the hydraulic torque are in dynamic equilibrium. From a hydraulic perspective, electrical torque is an external load on the turbine-generator unit.

Speed is another possible control variable for numerical simulations. For turbines, however, the governor strives to keep the turbine at synchronous speed by varying the wicket gate position during load variation and acceptance (assuming a perfect governor). If field data were available, the speed could be used to determine whether the model simulates the correct flow and pressures.

Once the time-varying electrical torque and wicket gate positions are known, the turbine equations (Numerical Representation of Hydroelectric Turbines), HAMMER solves flow, Q, and rotational speed, N, in conjunction with the characteristic curves for the turbine unit(s). This yields the transient pressures for the load rejection, load acceptance, emergency shutdown, operator error or equipment failure. The possible emergency or transient conditions are discussed separately in the sections that follow.

Load Rejection

Load rejection occurs when the distribution grid fails to accept electrical load from the turbine-generator system. After the load is rejected by the grid, there is no external load on the turbine-generator unit and the speed of the runner increases rapidly. This can be catastrophic if immediate steps are not taken to slow and stop the system. To keep the speed rise within an acceptable limit, the wicket gates must close quickly and this may result in high (followed by low) hydraulic transient pressures in the penstock. Since load rejection usually results in the most severe transient pressures, it typically governs the design of surge control equipment.

During load rejection, the generation of electrical power by the turbine-generator unit should decrease to zero as quickly as possible to limit the speed rise of the unit. To accomplish this, the wicket gates close gradually in order to reduce flow. The table below shows an example of electrical load and wicket gate position versus time to simulate load rejection. In a real turbine a governor would control the wicket gate closure rate, however the turbine governor is not modeled explicitly in HAMMER and the user controls the rate of wicket gate closure.

If the power generated by the water flowing through the turbine is greater than the electrical load, then the turbine will speed up; if the electrical load is greater, the turbine will slow down.

Note: Load and gate position are entered in different parameter tables in HAMMER because they may not use the same time intervals. HAMMER interpolates automatically as required.

Table 4-1: Load and Wicket Gate Changes for Load Rejection

Time (s)	Electrical Load (MW)	Wicket Gate Position (%)
0	350	100
1	100	50
2	0	0

Instant Load Rejection

Instant Load Rejection is similar to the Load Rejection case, except the electrical load on the turbine drops instantaneously to zero (i.e. the turbine is disconnected from the generator).

During instant load rejection, the generation of electrical power by the turbine-generator unit should decrease to zero as quickly as possible to limit the speed rise of the unit. To accomplish this, the wicket gates close gradually in order to reduce flow. The table below shows an example of wicket gate position versus time to simulate Instant Load Rejection. In a real turbine a governor would control the wicket gate closure rate, however the turbine governor is not modeled explicitly in HAMMER and the user controls the rate of wicket gate closure..

Table 4-2: Wicket Gate Changes for Instant Load Rejection

Time (s)	Wicket Gate Position (%)
0	100
1	50
2	0

Load Acceptance

Full load acceptance occurs when the turbine-generator unit is connected to the electrical grid. Transient pressures generated during full load acceptance can be significant but they are usually less severe than those resulting from full load rejection.

HAMMER assumes the turbine initially operates at no-load speed (NLS), and the turbine generates no electrical power. When the transient simulation begins, HAMMER assumes the electrical grid is connected to the output terminal of the generator and wicket gates have to be open as quickly as possible to meet the power demand - all without causing excessive pressure in the penstock.

Note that in this case, HAMMER assumes the turbine governor is 'perfect' - in other words the power produced by the turbine always equals the electrical load. Therefore the user doesn't need to enter an electrical load; just a curve of wicket gate position versus time, and the turbine's rated flow and head. Under the Load Acceptance case the turbine will always operate at its rated (or synchronous) speed. .

Table 4-3: Wicket Gate Changes for Full Load Acceptance

Time (s)	Wicket Gate Position (%)
0	0
1	50
2	100

Load Variation

Load variation on the turbine-generator unit can occur due to the diurnal changes in electricity demand in the distribution grid. During load variation, the governor controls the wicket gate opening to adjust flow through the turbine so that the unit can match the electrical demand. The water column in the penstock and conduit system accelerates or decelerates, resulting in pressure fluctuations.

The transient pressures that occur during general load variation may not be significant from a hydraulic design perspective since they are often lower than the pressure generated during a full load rejection or emergency shutdown.

At steady-state, the turbine-generator system usually runs at full load with the wicket gates 100% open. The amount of electricity produced by the system depends on the flow through the wicket gates. A decrease in electrical load requires a reduction in the wicket gate opening to adjust the flow.the table below shows an example of typical user input to simulate transient pressures for load variation.

Note that in this case, HAMMER assumes the turbine governor is 'perfect' - in other words the power produced by the turbine always equals the electrical load. Therefore the user doesn't need to enter an electrical load; just a curve of wicket gate position versus time. Under the Load Variation case the turbine will always operates at its rated (or synchronous) speed..

Table 4-4: Wicket Gate Changes for General Load Variation

Time (s)	Wicket Gate Position (%)
0	100
5	85
10	70
15	57
20	43
30	30
35	35
42	42

Table 4-4: Wicket Gate Changes for General Load Variation

Time (s)	Wicket Gate Position (%)
55	57
65	70
80	85
90	100

Turbine Parameters in HAMMER

Note: These attributes are used by HAMMER only.

Fundamentally, a turbine is a type of rotating equipment designed to remove energy from a fluid. For a given flow rate, turbines remove a specific amount of the fluid's energy head. Bentley WaterCAD V8*i* provides a single but very powerful turbine representation:

- **Turbine between 2 Pipes**—A turbine that undergoes electrical load rejection at time zero, requiring it to be shut down rapidly. The four-quadrant characteristics of generic units with certain specific speeds are built into Bentley WaterCAD V8*i* . The turbine element allows nonlinear closure of the wicket gates and is equipped with a spherical valve that can be closed after a time lag. It has the following parameters:
  - **Time (Delay until Valve Operates)** is a period of time that must elapse before the spherical valve of the turbine activates.
  - **Time for Valve to Operate** is the time required to operate the spherical valve. By default, it is set equal to one time step.
  - **Pattern (Gate Opening)** describes the percentage of wicket gate opening with time.
  - **Operating Case** allows you to choose among the four possible cases: instantaneous load rejection, load rejection (requires torque/load vs time table), load acceptance and load variation.
  - **Diameter (Spherical Valve)** is the diameter of the spherical valve.
  - **Efficiency** represents the efficiency of the turbine as a percentage. This is typically shown on the curves provided by the manufacturer. A typical range is 85 to 95%, but values outside this range are possible.
  - **Moment of Inertia** The moment of inertia must account for the turbine, generator, and entrained water.

- **Speed (Rotational)** denotes the rotation of the turbine blades per unit time, typically as rotations per minute or rpm. The power generated by the turbine depends on it.
- **Specific Speed** enables you to select from four-quadrant characteristic curves to represent typical turbines for three common types: 30, 45, or 60 (U.S. customary units) and 115, 170, or 230 (SI metric units).

  The equation to estimate specific speed for a turbine is as follows:

  $$n_s = n \times p^{0.5} / H^{\frac{5}{4}}$$

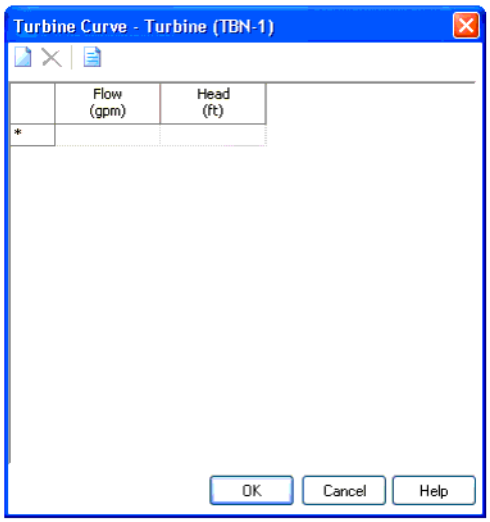
  In US units n is in rpm, P is in hp, and H is in ft.

  In SI units n is in rpm, P is in kW, and H is in m.
- **Turbine Curve** For a transient run, HAMMER uses a 4-quadrant curve based on Specific Speed, Rated Head, and rated Flow. This is only used for steady state computations.
- **Flow (Rated)** denotes the flow for which the turbine is rated.
- **Head (Rated)** denotes the head for which the turbine is rated.
- **Electrical Torque Curve** defines the time vs torque response for the turbine. Only applies to the Load Rejection operating case.

Turbine Curve Dialog Box

This dialog is used to define the points that make up the flow-head curve that is associated with the turbine curve for the associated turbine element. The turbine curve represents the head-discharge relationship of the turbine at its rated speed.

The **New** button adds a new row to the table; the **Delete** button removes the currently selected row from the table, and the **Report** button generates a preformatted report displaying the Head vs. Flow data points for the current turbine curve.



Periodic Head-Flow Elements

The Periodic Head-Flow element represents a versatile hydraulic boundary condition which allows you to specify a constant head (pressure), flow, or any time-dependent variation, including periodic changes that repeat indefinitely until the end of the simulation.

Note: The Periodic Head/Flow element supports a single branch connection only. If there is more than one branch connected to it, the transient run will fail and an error message may appear, such as:

"Only one active pipe may be connected to this type of node in its current configuration."

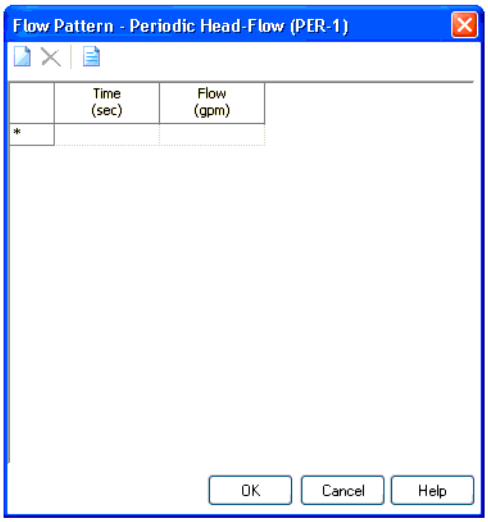
This element is used to prescribe a boundary condition at a hydraulic element where flow can either enter or leave the system as a function of time. It can be defined either in terms of Head (for example, the water level of a clear well or process tank) or Flow (for example, a time-varying industrial demand). The periodic nature of variation of head/flow can be of sinusoidal or of any other shape that can be approximated as a series of straight lines.

Note: During a Steady State of EPS run (used to determine the initial conditions for a transient analysis), the head/flow for this element is held constant at the initial head/flow value on the sinusoidal or user-defined pattern. The head/flow only varies during a transient analysis.

Periodic Head-Flow Pattern Dialog Box

This dialog is used to define the points that make up the head or flow pattern that is associated with a non-sinusoidal periodic head-flow element. The pattern is defined by creating **Head** or **Flow** vs **Time** points.

The **New** button adds a new row to the table; the **Delete** button removes the currently selected row from the table, and the **Report** button generates a preformatted report displaying the Time vs. Flow (or Head) data points for the Periodic Head-Flow curve.



Air Valves

Air valves are installed at local high points to allow air to come into the system during periods when the head drops below the pipe elevation and expels air from the system when fluid columns begin to rejoin. The presence of air in the line limits subatmospheric pressures in the vicinity of the valve and for some distance to either side, as seen in profiles. Air can also reduce high transient pressures if it is compressed enough to slow the fluid columns prior to impact.

There are essentially two ways in which an active air valve can behave during the transient simulation:

1. **Pressure below atmospheric** - air valve is open and acts to maintain pressure to 0 on the upstream end and maintains the same flow on the upstream and downstream side.
2. **Pressure above atmospheric** - air valve is closed and acts as any junction node.

If an air valve becomes open during the initial conditions calculation (steady state or EPS), the hydraulic grade on the downstream side may be less than the pipe elevation. This can be displayed as the hydraulic grade line drawn below the pipe. This should be interpreted as a pressure pipe that is not flowing full. Full flow resumes at the point where the hydraulic grade line crosses back above the pipe.

Because air valves have the possibility to switch status during a steady state or EPS, they can lead to instability in the model especially if there are many air valves in the system. To improve the stability of the model, it is desirable to force some of the valves closed. This can be done by setting the property "Treat air valve as junction" to True for those valves that are expected to be closed anyway.

If all of the pumps upstream of an air valve are off during a steady state or EPS, the pressure subnetwork is disconnected in that area and the model will issue warning messages for all nodes in that vicinity indicating that they are disconnected.

Note: In the rare event that you need to model an air valve that is open during the initial conditions, the initial air volume will need to be entered. The friction factors in the adjacent pipes may also need to be checked, as the head loss computed by the initial conditions calculation may not be a true head loss. It may be necessary to specify the initial conditions manually (by setting the 'Specify Initial Conditions?' Transient Solver calculation option to True - see Calculation Options for details - then manually typing in values for the fields grouped under Transient Initial in the Property Editor.

The following attributes describe the air valve behavior:

Note: The following are HAMMER attributes.

- **Slow Closing Air Valve Type**:
  - **Time to Close**: For an air valve, adiabatic compression (i.e., gas law exponent = 1.4) is assumed.The valve starts to close linearly with respect to area only when air begins to exit from the pipe. If air subsequently re-enters, then the valve opens fully again. It is possible for liquid to be discharged through this valve for a period after the air has been expelled.
  - **Diameter (Air Outflow Orifice)**: Diameter of the air outflow orifice (the orifice through which air is expelled from the pipeline).

- **Double Acting Air Valve Type**:
  - **Air Volume (Initial)**: Volume of air near the valve at the start of the simulation. The default is zero. If volume is nonzero, the pressure must be zero.
  - **Diameter (Air Inflow Orifice)**: Diameter of the air inflow orifice (the orifice through which air enters the pipeline when the pipe internal pressure is less than atmospheric pressure). This diameter should be large enough to allow the free entry of air into the pipeline. By default, this diameter is considered infinite (i.e. there is no restriction to air inflow).
  - **Diameter (Air Outflow Orifice)**: Diameter of the air outflow orifice (the orifice through which air is expelled from the pipeline). By default, this diameter is considered infinite.

- **Triple Acting Air Valve Type**:
  - **Air Volume (Initial)**: Volume of air near the valve at the start of the simulation. The default is zero. If volume is nonzero, the pressure must be zero.
  - **Trigger to Switch Outflow Orifice Size**: Select whether the transient solver switches from the large air outflow orifice to the small air outflow orifice based on Transition Volume or Transition Pressure.
  - **Transition Pressure:** The local internal system air pressure at the air valve above which the transient solver switches from using the large air orifice to the small air orifice (in order to minimize transients).
  - **Transition Volume**: The local volume of air at the air valve below which the transient solver switches from using the large air orifice to the small air orifice (in order to minimize transients). This volume often corresponds to the volume of the body of the air valve.
  - **Diameter (Small Air Outflow Orifice)**:): Diameter of the air outflow orifice (the orifice through which air is expelled from the pipeline) when the local air volume is less than the transition volume (TV), or the air pressure is greater than the transition pressure (TP) (depending on which trigger is used to switch the outflow orifice size). This diameter is typically small enough for the injected air to be compressed, which can help prevent severe transient pressures. Generally air flows out the large air outflow orifice for some time before switching to the small air outflow orifice for the final stages of air release.

  - **Diameter (Large Air Outflow Orifice)**: Refers to the discharge of air when the local air volume is greater than or equal to the transition volume (TV), or the air pressure is less than or equal to the transition pressure (TP) (depending on which trigger is used to switch the outflow orifice size). This diameter is typically large enough that there is little or no restriction to air outflow. Generally air flows out the large air outflow orifice for some time before switching to the small air outflow orifice for the final stages or air release.
  - **Diameter (Air Inflow Orifice)**: Diameter of the air inflow orifice (the orifice through which air enters the pipeline when the pipe internal pressure is less than atmospheric pressure). This diameter should be large enough to allow the free entry of air into the pipeline. By default, this diameter is considered infinite (i.e. there is no restriction to air inflow).

- **Vacuum Breaker Air Valve Type**:
  - **Diameter (Air Inflow Orifice)**: Diameter of the air inflow orifice (the orifice through which air enters the pipeline when the pipe internal pressure is less than atmospheric pressure). This diameter should be large enough to allow the free entry of air into the pipeline. By default, this diameter is considered infinite (i.e. there is no restriction to air inflow).

Determining the Type of Air Valve to Use

When modeling an air valve, it must conform to one of the four available types: (selected from the "Air Valve Type" attribute) Double Acting, Triple Acting, Vacuum Breaker and Slow Closing. Industry terminology is sometimes not consistent with HAMMER's definition of these types, so it is important to understand their behavior and assumptions. Below describes each air valve type and when it should be used.

> **Note: If you cannot approximate the size of your openings with a circular orifice diameter or if you need to enter a specific relationship between pressure and air flow rate, select "Air Flow Curve" as the "Air Flow Calculation Method" in the properties of the air valve.**

Double Acting - This type of air valve has two actions:

1. Air inflow through an inflow orifice diameter
2. Air outflow through an outflow orifice diameter

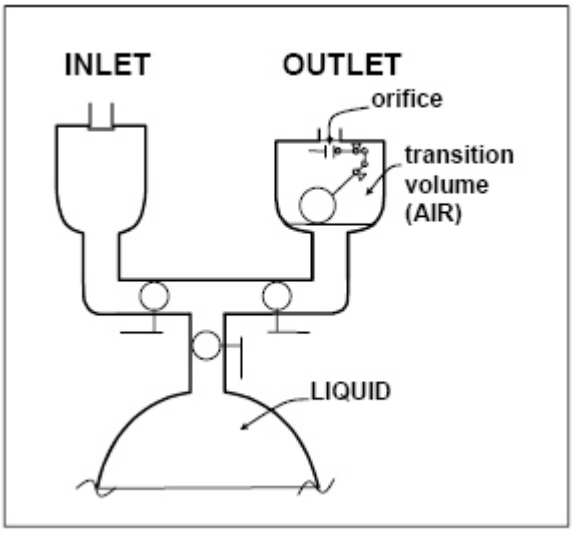
The diameters of these orifices don't change during the transient simulation. This type of air valve should be used when air enters the valve through a specific size opening, and leaves the system through another specific size opening, without any transition. The opening that allows air outflow is typically smaller, in order to control air release. Here are some examples of when the Double Acting air valve type would be used:

- An air valve with an "anti-slam", spring loaded disc with perforations, which opens under vacuum conditions. When pressure returns, the spring closes the disc and air is forced to exit through the small perforations. The air inflow orifice would be the size of the opening through which air flows when the disc rises off the seat. The air outflow orifice would be the equivalent orifice size of the perforations in the disc.
- An air valve with a spring loaded orifice that admits air on vacuum conditions and a separate, smaller opening that expels air. The spring loaded orifice would be the air inflow orifice and the smaller opening would be the air outflow orifice.

Triple Acting - This type of air valve has three actions:

1. Air Inflow
2. Air Outflow through a large orifice
3. Air Outflow through a small orifice

Air inflow passes through an opening with a fixed size. Air outflow first passes through a large-sized opening, which switches to a smaller sized opening just before all of the air has escaped. This cushions the air pocket collapse and subsequent collision of the water columns. This type of air valve should be used when the opening through which air is expelled changes based on some condition. The condition to trigger the reduction in size of the outflow orifice can either be based on a pressure differential or an air volume. Typically a float is used to decrease the opening size, but not always.



Here are some examples of when the Triple Acting air valve type would be used:

- An air valve similar to the one seen in the above diagram, consisting of two openings and a float. When the volume of air in the system becomes less than the "transition volume", the float rises, which partially closes the outlet opening. The air inflow orifice would be the size of the "inlet" opening. The "large air outflow orifice" would be the full size of the outlet opening. The "small air outflow orifice" would be the size of the outlet opening after the float has risen.
- An air valve with a float that closes off the outlet opening completely, forcing air out of a separate, smaller opening. The "large air outflow orifice" would be a diameter equivalent to the size of the main outlet opening plus the small opening. The "small air outflow orifice" would be the size of the separate, smaller opening alone.
- An "anti-slam" air valve with a disc or float that first allows air outflow to freely pass out of a large opening. As air velocity increases, the float is "blown" into position by the pressure differential it creates, forcing air out of a smaller opening. The "large air outflow orifice" would be the large size opening (before the float rises) and the "small air outflow orifice" would be the smaller sized opening (after the float rises). "Transition Pressure" would be selected as the outflow orifice trigger type.

Vacuum Breaker - This type of air valve has only one operation: air inflow. During subatmospheric pressure, air enters through the air inflow orifice diameter. The outflow orifice diameter is assumed to be very small (effectively zero) so it doesn't let air out. When looking at the detailed report, you may notice the air volume change as the air pocket is compressed, but the mass of air in the pipe doesn't reduce. There are probably a limited number of applications for this type valve, but it may be used for a draining pipeline.

> **Note:** **Any air pocket left in the system due to a vacuum breaker valve is assumed to be expelled out of the system by some other means. HAMMER currently cannot track the behavior of these trapped air pockets (the underlying assumption is that the air must exit the system where it came in)**

Slow Closing - This type of air valve has two actions:

- Free air inflow upon subatmospheric pressure
- Linear closure of the air outflow orifice when air begins to exit

Although similar to the other air valve types, the slow-closing air valve only has a single orifice involved; for the expulsion of air and liquid. An air inflow orifice is not required because HAMMER assumes that air will be freely allowed into the system (no throttling) when the head drops below the air valve elevation. The valve starts to close linearly with respect to area only when air begins to exit from the pipeline (after the head begins to rise).

It is possible for liquid to be discharged through this valve for a period after the air has been expelled, unlike the other air valve types, which closes when all the air has been evacuated from the pipeline. Typically you will want the valve to be fully closed after all air has been expelled, but before too much water has been expelled.

Air Flow Curves Dialog Box

The following management controls are located above the air flow curve list pane:

New 	Creates a new air flow curve.
Delete 	Deletes the air flow curve that is currently highlighted in the list pane. You can hold down the Ctrl key while clicking on items in the list to select multiple entries at once.
Duplicate 	Creates a copy of the currently highlighted air flow curve.
Rename 	Renames the air flow curve that is currently highlighted in the list pane.
Report 	Opens a report of the data associated with the air flow curve that is currently highlighted in the list pane.
Synchronization Options 	Browses the Engineering Library, synchronizes to or from the library, imports from the library or exports to the library.

The tab section is used to define the settings for the air flow curve that is currently highlighted in the air flow curve list pane. The following controls are available:

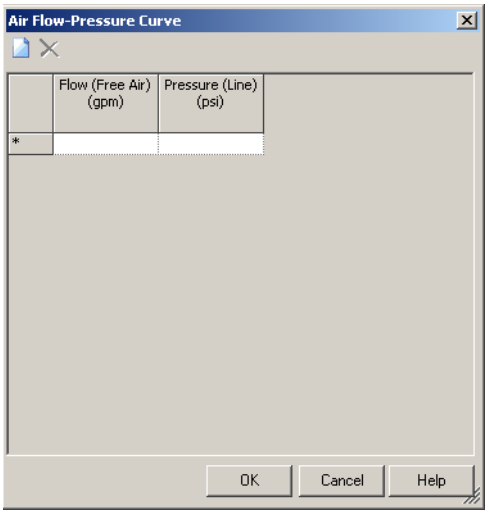
Air Flow Curve Tab	This tab consists of input data fields that allow you to define the air flow curve.
Flow (Free Air)	The volume of air flow at the associated pressure.

Pressure (Line)	The pressure at the air flow curve point. Note that only gauge pressure values are supported, not absolute pressure.
Library Tab	This tab displays information about the air flow curve that is currently highlighted in the air flow curve list pane. If the curve is derived from an engineering library, the synchronization details can be found here. If the curve was created manually for this project, the synchronization details will display the message Orphan (local), indicating that the curve was not derived from a library entry.
Notes Tab	This tab contains a text field that is used to type descriptive notes that will be associated with the air flow curve that is currently highlighted in the air flow curve list pane.

Note: The Air Flow result attribute shown in the detailed report shows the volumetric flow rate of air at the conditions present inside the pipeline.

Air Flow-Pressure Curve

This dialog allows you to define pattern curves for the Air Flow Curve Engineering Library.



The following buttons are located above the curve points table on the left:

-  **New**—Creates a new row in the curve points table.
-  **Delete**—Deletes the currently highlighted row from the curve points table.

The curve points table contains the following columns:

- **Flow (Free Air)**—The volume of air flow at the associated pressure.
- **Pressure (Line)**—The pressure at the air flow curve point. Note that only gauge pressure values are supported, not absolute pressure.

Hydropneumatic Tanks

A pressure vessel connected to the system and containing fluid in its lower portion and a pressurized gas, usually air, in the top portion. A flexible and expandable bladder is sometimes used to keep the gas and fluid separate. When the tank is being filled (usually from a pump), the water volume increases and the air is compressed. When the pump is turned off, the compressed air maintains pressure in the system until the water drains and the pressure drops.

In WaterCAD V8*i* there are two ways of modeling water fluctuations in hydropneumatic tanks during Steady State / EPS (initial conditions) simulations:

1. As an equivalent constant cross section area tank (Constant Area Approximation)
2. Using the ideal gas law (Gas Law Model)

The data requirements for each method differ. Both methods require:

1. Total tank volume
2. Initial HGL
3. Initial water volume
4. Controls set up for any pumps controlled by the tank HGL

The Constant area tank method also requires:

1. Effective tank volume
2. HGL on level
3. HGL off level

The Gas law method requires

1. Atmospheric pressure (if differs from default)

When using the Constant Area Approximation method, you will need to know the effective volume of the tank (usually between 30 and 50% of the total volume), and the hydraulic grade line elevation corresponding to the maximum and minimum water volumes. The values are referred to as the HGL on and HGL off values because the feed pump turns off when the maximum effective volume is reached and turns on when the minimum effective volume is reached. The effective cross sectional area of an equivalent tank is given by

$$\text{Area} = \text{Effective volume}/(HGL_{off} - HGL_{on})$$

Note: Specifying these on and off HGL levels does not mean that logical controls have been established. You must still set up logical controls for the pumps feeding the tank and these control levels should not be significantly different from the HGL on and off levels.

The results from a steady state run are the flows in and out of the tanks. These results should be the same for both the constant area and gas law tanks. The results of an EPS run are the flow plus the HGL and pressure in the tank over time. These results will be slightly different for each type of tank especially at very high and very low pressures, provided that the effective volume is close to the actual effective volume that is physically possible given the control settings, gas volume and tank volume.

When using the Gas Law method, the tank is modeled using a form of the ideal gas law for an isothermal fluid:

$$(P + P_{atm})\ V_{air} = K$$

Where:

P = gauge pressure

P_{atm} = atmospheric pressure

V_{air} = volume of air in tank.

When using this method, you must specify the volume of liquid in the tank, the total volume of the tanks and the initial pressure (or HGL). You can also override the default atmospheric pressure of 32 ft.

Over the narrow range of pressures normally found in hydropneumatic tanks, the constant area tank approximation and the gas law model give comparable results although the gas law model is more theoretically correct. As the range of pressures increases, the gas law model diverges from the constant area tank at high pressures.

Note: Hydropneumatic tanks have a very short cycle time compared with large tanks. Therefore, when hydropneumatic tanks are used in a model, a very short hydraulic time step may be needed or the tank may overshoot its on and off levels. If this occurs, the hydraulic time step in the calculation options should be reduced.

During a transient simulation there are two basic types of tank: (a) direct interface between the liquid and gas, and (b) gas contained in a bladder. Both utilize the expansion/contraction of a gas according to the gas law: $P V_k$ = constant, where P is the absolute pressure, V is the volume and the exponent k lies between 1.0 and 1.2. In the case of (b), the initial volume is determined from the isothermal gas law, PV = constant, for given values of preset pressure, tank volume and initial (gauge) pipe pressure. At the mouth of the vessel, there is a differential orifice with head loss $\Delta H = H_l - H_g = b\ d\ Q^2 / (2g\ A_{or}^2)$, where the subscripts l, g and or refer to the liquid, gas and orifice, respectively, b is the head loss coefficient and $d = d_i$ for inflow ($Q > 0$) and -1 for outflow ($Q < 0$). By definition, d asserts that head losses are d_i times greater for inflow than for outflow - typical value of d_i is 2.5.

With respect to a bladder vessel, the pre-set pressure can range from zero gauge (atmospheric pressure) to some higher pressure. Prior to and during a transient computation:

- HAMMER assumes the bladder is at the pre-set pressure but isolated from the system.
- HAMMER assumes a (virtual) isolation valve is opened, such that the (typically higher) system pressure is now felt by the bladder. HAMMER computes the new (typically smaller) volume of the air inside the bladder.
- When the transient occurs, HAMMER expands or contracts the volume inside the bladder accordingly.
- After the simulation is complete, you can look in the .RPT and/or .OUT text file(s) to see what the preset pressure, pre-transient volume (at system pressure) and subsequent variations in pressure and volume have occurred.

The tank type with a direct interface between the liquid and gas can be classified as one of three different types: 'sealed', 'vented' or 'dipping tube'

A sealed hydropneumatic tank is simply a closed pressure vessel.

A vented hydropneumatic tank is effectively a sealed tank with the addition of an air valve at the top. This allows air at atmospheric pressure to enter the tank during a downsurge so that the device behaves like a one-way surge tank. During an upsurge, the air valve typically throttles the air outflow so that the gas within the tank is

compressed and acts as a 'cushion' against transients (just like a sealed hydropneumatic tank). This device offers several practical benefits - for example since the tank typically has no gas inside, there is no need for compressors or a bladder to ensure a required gas volume is maintained.

A dipping tube hydropneumatic tank has a dipping (or ventilation) tube inside with an air valve at the top. During normal operation the air valve is closed, the water level is above the bottom of the dipping tube, and gas is compressed in the 'compression chamber'. If the hydraulic grade line drops (e.g. after a pump stop) the dipping tube tank acts like a regular (sealed) hydropneumatic tank until the water surface drops below the bottom of the dipping tube, after which the air valve opens and allows air to enter at atmospheric pressure. At this point the tank is acting like a surge tank that is

open to atmosphere. If the hydraulic grade line increases again (e.g. if pumps come on), air will be expelled until the hydraulic grade line rise enough to close the air valve. At this point the water surface will be above the bottom of the dipping tube and the tank will act like a regular sealed hydropneumatic tank once again.

Figure 4-1: Sealed Hydropneumatic Tank

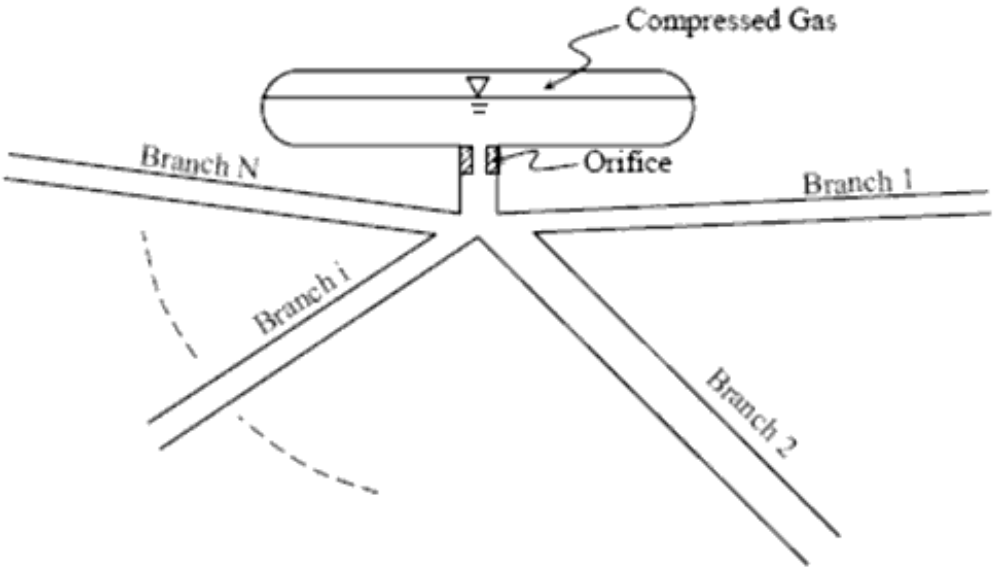


Figure 4-2: Vented Hydropneumatic Tank

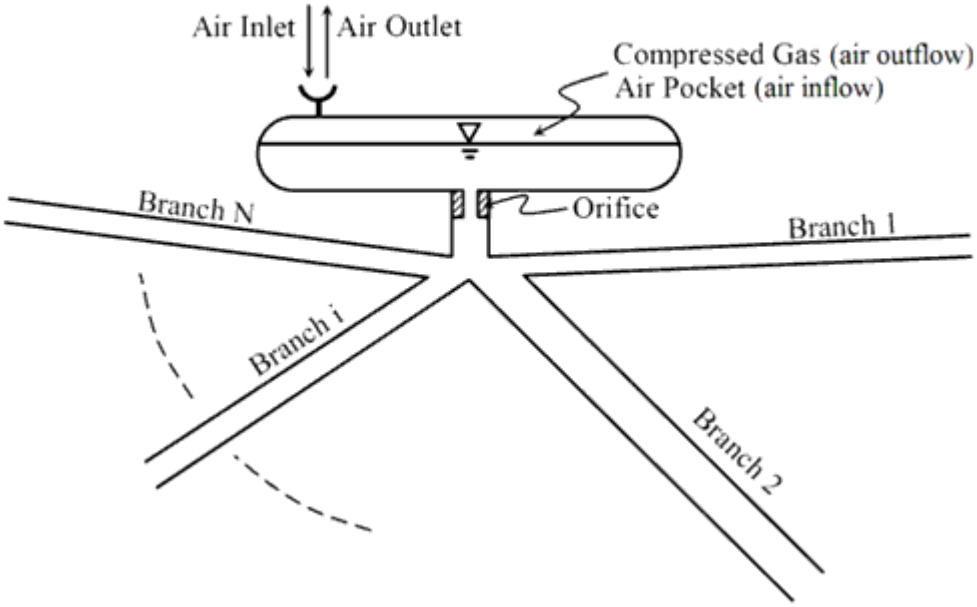
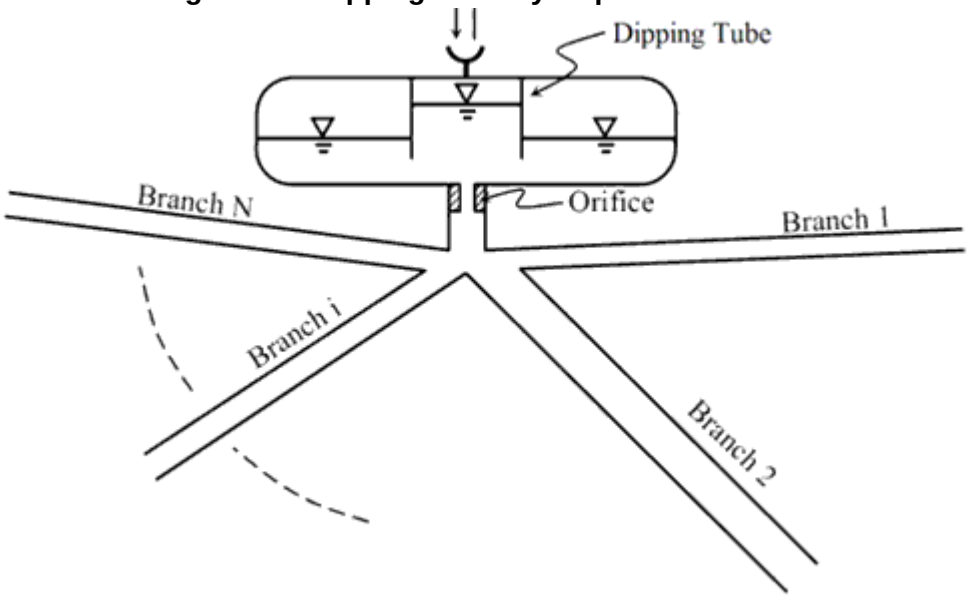


Figure 4-3: Dipping Tube Hydropneumatic Tank



Initial Conditions Attributes

The following attributes of the hydropneumatic tank influence the initial conditions calculation (steady state or EPS). You'll notice that they are all within the "Operating Range" or "Physical" section of the hydropneumatic tank properties.

- **Elevation (base)** - The elevation of the base of the tank. It is used as a reference when entering initial hydraulic grade in terms of "level" (i.e., if the "elevation (base)" is set to 20m and the operating range is set to "level", a "level (initial)" value of 1.0 represents an elevation of 21m).
- **Operating Range Type** - Specify whether the initial hydraulic grade of the tank is based on levels measured from the base elevation or as elevations measured from the global datum (zero). For example, if the base elevation is 20m, you want the initial hydraulic grade to be 70m., and you want to use levels, then select "level" for this field and enter 50m as the initial level.
- **HGL (Initial) or Level (Initial)** - Depending on the operating range type selected, this represents the known boundary hydraulic grade at the tank during steady state. It is the water surface elevation plus the pressure head of the compressed gas in the hydropneumatic tank. The transient simulation will begin with this head. However, if you've selected "true" for the "Treat as Junction" attribute, the transient simulation will ignore this value and instead use the computed steady state hydraulic grade
- **Liquid Volume (Initial)** - This represents the volume of liquid in the tank at the start of the initial conditions, corresponding to the initial HGL. This includes the inactive volume below the affective volume, when using the "constant area approximation" tank calculation model.

- **Elevation** - The elevation from which to calculate pressure in the hydropneumatic tank (typically the bottom of the tank.) It could be set to the estimated water surface, since the air pressure (used in the gas law equation) is above that point. However, the bottom elevation and water surface are typically very close, so this likely will not make a noticeable difference.
- **Volume (Tank)** - This represents the total volume of the tank. This is only used in an EPS simulation (to find the gas volume so that the gas law equation can be used) or when using the bladder option ("Has Bladder?" = "True") during a transient simulation. When using a bladder tank, WaterCAD V8*i* assumes the bladder occupies this full tank volume at its "preset pressure,".
- **Treat as Junction?** - Selects whether or not the hydropneumatic tank is treated as a junction in steady state and EPS simulations. Note that if you wish to use the steady state/EPS results as input for a HAMMER transient analysis and you set this field to **True**, you will need to manually enter the **Volume of Gas (Initial)** for the tank for HAMMER
- **Volume of Gas (Initial)** - The initial volume of gas in the pressure vessel at the start of the simulation. During the transient event, the gas volume expands or compresses, depending on the transient pressures in the system. This value is not used in steady state or EPS analyses.
- **Tank Calculation Model** - Specifies whether to use the gas law or a constant area approximation method during steady state or EPS initial condition calculations. The constant area approximation uses a linear relationship; the user must specify minimum/maximum HGL and the corresponding volume between. The gas law model is non-linear and follows the gas law--as gas is compressed, it becomes harder to compress it further.
- **Atmospheric Pressure Head** - When using the gas law tank calculation model, this field represents atmospheric pressure at the location being modeled. This is required because the gas law equation works in absolute pressure, as opposed to gauge pressure.

  Note: The "atmospheric pressure head" field is not used during the transient simulation. The transient calculation engine assumes an atmospheric pressure head of 1 atm or 10.33 m.

- **HGL on/HGL off** - Exposed when using the constant area approximation method. The "HGL on" field is the lowest operational hydraulic grade desired, and the "HGL off" is the highest operational hydraulic grade desired. Corresponding controls should be entered to turn the pump on and off during an EPS simulation. Note that typically a transient simulation will use steady state initial conditions, so these fields are not considered; only the steady state HGL and user-entered gas volume are used to define the initial volume and head for the transient simulation.
- **Volume (effective)** - Exposed when using the constant area approximation method. Represents the volume between the HGL on and HGL off fields.

Gas Law vs. Constant Area Approximation

For the initial conditions, you must select either "gas law" or "constant area approximation" for the "Tank calculation model" attribute of the hydropneumatic tank. The constant area approximation selection exposes the "Volume (effective)," "HGL on," and "HGL off" fields. The gas law selection exposes the "Atmospheric pressure" field. These fields are primarily there to support the WaterCAD and WaterGEMS products, which can directly open a HAMMER model. They are only used to track the change in HGL/volume for EPS simulations, which typically aren't used in HAMMER. A transient analysis typically begins with a steady state simulation, which only considers the "HGL (Initial)" and "volume of gas (initial)". This is because a steady state simulation is a snapshot in time, so the head/volume are not changing. So in most cases, it does not matter which tank calculation method you choose. You will likely want to select "gas law" for simplicity, but additional information on both approaches is provided below.

- **Constant area approximation**: This method approximates a hydropneumatic tank by using a tall, thin tank whose water surface elevation approximates the HGL in a hydropneumatic tank. The HGL on and HGL off fields represent the maximum and minimum hydraulic grade lines within the hydropneumatic tank (i.e. when an associated booster pump would turn on or off). An approximate diameter is computed based on the effective volume of the hydropneumatic tank so that the tank cross sectional area multiplied by the distance between HGL on and HGL off gives the same volume as the hydropneumatic tank.
- **Gas Law**: This method uses the ideal gas law, PV=nRT, to compute new hydraulic grades as liquid volume changes in the EPS simulation (nRT is assumed to be constant). The initial liquid volume is subtracted from the total tank volume to find the gas volume. The physical "elevation" is subtracted from the initial HGL to find the gauge pressure. The atmospheric pressure is added to the gauge pressure to get absolute pressure, which is used in the ideal gas law equation.

Both methods typically yield similar results within the "effective" control range, but the gas law is technically more accurate.

Transient Simulation Attributes

The following hydropnematic tank attributes influence the transient simulation:

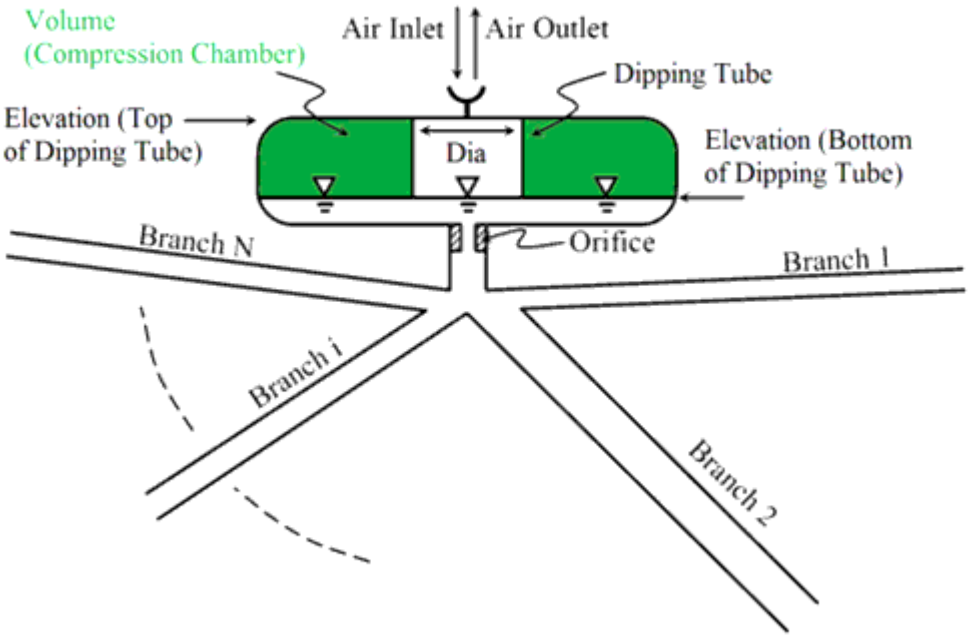
- **Hydropneumatic Tank Type -** Specify the type of Hydropneumatic Tank that this model element represents. Sealed means the tank is a fully sealed pressure vessel. Vented means the tank has an air valve attached. Dipping Tube means the tank has an internal dipping or ventilation tube.

- **Diameter (Tank Inlet Orifice)** - This is the size of the opening between the gas vessel and the main pipe line. It is typically smaller than the main pipe size. It is used to compute the correct velocity through the tank inlet, so the correct headloss is computed based on the minor loss coefficient (the standard head loss equation is used: $Hl = K*V^2/2g$.)
- **Diameter (Dipping Tube)** - The diameter of the dipping or ventilation tube within the hydropneumatic tank (only applicable for the Dipping Tube tank type).
- **Volume (Compression Chamber)** - The volume of the air around the dipping tube that is compressed once the water level elevation exceeds the bottom of the dipping tube.
- **Air Flow Calculation Method** - Specify whether the air valve air flow rate is determined by user-entered curves of pressure vs. air flow rate, or whether it is calculated based on a user-entered orifice diameter (not applicable for a sealed hydropneumatic tank). The calculated Air Flow result attribute shown in the detailed report shows the volumetric flow rate of air at the conditions present inside the pipeline.
- **Diameter (Air Inflow Orifice)** - This is the equivalent orifice size of the opening that allows air to enter the tank.
- **Diameter (Air Outflow Orifice**) - This is the equivalent orifice [1]size of the opening that allows air to leave the tank.
- **Air Flow Curve (Air Inflow Orifice)** - The curve that defines the rate of air inflow (a 'free air' rate, measured at atmospheric pressure) into the tank versus the differential pressure across the air valve.
- **Air Flow Curve (Air Outflow Orifice)** - The curve that defines the rate of air outflow (a 'free air' rate, measured at atmospheric pressure) out of the tank versus the differential pressure across the air valve.
- **Elevation (Top of Dipping Tube)** - The elevation of the top of the dipping tube and the dipping tube-type hydropneumatic tank.

1.

- **Elevation (Bottom of Dipping Tube)** - The elevation of the bottom of the dipping tube.

Figure 4-4: Dipping Tube Hydropneumatic Tank Parameters



- **Minor Loss Coefficient (Outflow)** - This is the 'k' coefficient for computing headlosses using the standard headloss equation, $H = kV^2/2g$. It represents the headlosses for tank outflow. If you lump other minor losses through the tank assembly (bends, fittings, contractions, etc) into this coefficient, keep in mind that the velocity is calculated using the area of the "diameter (tank inlet orifice)" that you entered.
- **Ratio of Losses** - This is the ratio of inflow to outflow headloss. For flows into the tank (inflows), the "minor loss coefficient" is multiplied by this value and the losses are computed using that. For flows out of the tank, HAMMER only uses the "Minor Loss coefficient". So, if you enter a minor loss coefficient of 1.5 and a ratio of losses of 2.5, the headloss coefficient used when the tank is filling would be 1.5 X 2.5 = 3.75.
- **Gas Law Exponent** - refers to the exponent to be used in the gas law equation. (the 'k' in PV^k = constant) The usual range is 1.0 to 1.4. The default is 1.2.
- **Volume of Gas (Initial)** - When not using a bladder, the initial volume of gas is an important attribute. This is a required input field, representing the volume of gas inside the tank at the steady state pressure (initial conditions hydraulic grade minus tank physical elevation). During the transient simulation, this gas volume expands or compresses, depending on the transient pressures in the system. For

example, consider a 500 L tank with base elevation of 20 m and initial hydraulic grade of 70 m. This means that the pressure head is ~50 m. So, the user needs to decide how much space (volume) the entrapped gas pocket would take up, at this pressure.

Note: **If you are not specifying initial conditions and not treating the tank as a junction, then the initial gas volume is not required and the field will not show up. This is because it is either computed from the initial conditions gas volume (which is the full tank volume minus the initial liquid volume for a steady state) or based on the preset pressure (if using the bladder option)**

In some cases, you may want to analyze a range of different initial conditions, which could potentially change the starting hydraulic grade of your hydropneumatic tank. The gas law can be employed in this case. For example, if you know the initial gas volume is 300 L at a steady state pressure head of 50 m, you can compute the 'K' constant using the gas law, $PV^k=K$: (50 m + 10.33 m)(0.3m3) = 18.099. (gas law exponent assumed to be 1.0) So, if your new steady state pressure head is 30 m, the new initial gas volume (which you must enter) is computed as V = (18.099)/(30 m+10.33 m) = 0.449 m^3 = 449 L.

The transient calculation engine always uses an atmospheric pressure head of 1 atm or 10.33 m when solving the gas law equation.

- **Has Bladder?** - Denotes whether the gas is contained within a bladder. If it is set to "True", WaterCAD V8*i* automatically assumes that the bladder occupied the full-tank volume at the preset pressure at some time and that the air volume was compressed to a smaller size by the steady-state pressure in the system. The "Volume of gas (initial)" is not used in this case, since it is calculated based on the full tank size, preset pressure and steady state pressure.
- **Pressure (Gas-Preset)** - This is the pressure (not a hydraulic grade) in the gas bladder before it is exposed to pipeline pressure; the pressure when it fills the entire tank volume. Often called the "precharge" pressure; it is only exposed when selecting "true" for "Has bladder?"
- **Report Period** - used to report extended results in the Transient Analysis Detailed Report. Represents a timestep increment. For example, entering '10' would cause extended results to be reported every 10 timesteps.
- **Elevation Type** - This allows you to specify the type of approach used in tracking the gas-liquid interface (a new feature as of version 08.11.01.32). By default, the liquid surface elevation is not tracked and is essentially assumed to be fixed, at the tank physical bottom elevation. For more information on how this option is used for tracking the liquid elevation, see Tracking the Air-Liquid Interface.

Tracking the Air-Liquid Interface

The "Elevation Type" field in the Hydropneumatic tank properties allows you to control how the air-liquid interface (water surface elevation) is tracked. This field presents 3 options, Fixed, Mean Elevation and Variable Elevation.

Fixed

This is the default option for the "Elevation Type" field and is consistent with the behavior of previous versions (prior to 08.11.01.32). The liquid elevation is assumed to be at a fixed location during the transient simulation, equal to the bottom of the tank. The gas pressure used in the gas law equation is then equal to the hydraulic grade line within the tank, plus the atmospheric pressure, minus the tank's base elevation.

This is acceptable for most cases, mainly because the elevation difference between the range of possible liquid levels is typically quite small. So, it does not account for much of a pressure difference. This can be observed by adjusting the "Elevation" attribute in the tank properties.

Mean Elevation

Selecting "Mean Elevation" exposes the "Liquid Elevation (Mean)" field, which allows you to specify a custom liquid (water surface) elevation, instead of assuming it is equal to the tank bottom (as is with the "fixed" option). It represents the average elevation of the liquid/gas interface throughout a transient simulation. This is useful in cases where the liquid elevation is significantly higher than the tank bottom, but doesn't move significantly during a transient simulation. So, although no tracking of changes in liquid elevation occurs, it allows you to get a more accurate calculation in some cases. The absolute gas pressure used in the gas law equation during the calculations based on the mean elevation that you enter.

Variable Elevation

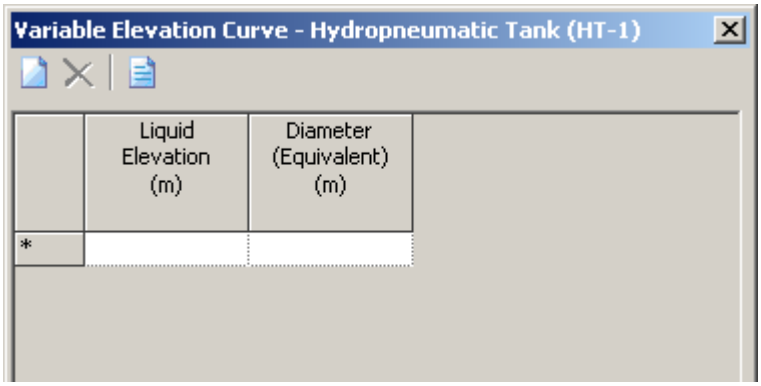
Selecting "Variable Elevation" exposes the "Variable Elevation Curve" field, which allows you to enter a table of liquid elevation versus equivalent diameter. The variable level hydropneumatic tank type is for users who have detailed information about the tank's geometry and want to perform as accurate a simulation as possible. Typically, this type of representation would be selected in the detailed design stage. It would also be appropriate in the case of low-pressure systems and/or relatively tall tanks with large movements of the interface relative to the HGL of the gas. The initial liquid level is determined from the initial gas volume which is an input parameter. The tank cross-sectional area at any elevation is interpolated from an input table of the vessel's geometry spanning the range from the pipe connection at the bottom to the top of the tank.

Reporting

After computing the transient simulation with a variable elevation hydropneumatic tank, you can view the liquid level over time by looking at the Transient Analysis Detailed Report. This report is found under Report > Transient Analysis Reports and will show this extended, tabular data for the tank when you've entered a value for the "report period" property of that tank.

Variable Elevation Curve Dialog Box

This dialog allows you to define the variable elevation curve for hydropneumatic tanks.



The variable level hydropneumatic tank type is for users who have detailed information about the tank's geometry and want to perform as accurate a simulation as possible. Typically, this type of representation would be selected in the detailed design stage. It would also be apropos in the case of low-pressure systems and/or relatively tall tanks with large movements of the interface relative to the HGL of the gas. The initial liquid level is determined from the initial gas volume which is an input parameter. The tank cross-sectional area at any elevation is interpolated from an input table of the vessel's geometry spanning the range from the pipe connection at the bottom to the top of the tank.

The **New** button adds a new row to the table; the **Delete** button removes the currently selected row from the table, and the **Report** button generates a preformatted report displaying the Liquid Elevation vs. Diameter (Equivalent) data points for the current elevation curve.

Acces this dialog by setting the hydropneumatic tank's Elevation Type to Variable Elevation and by clicking the ellipsis button in the Variable Elelvation Curve field.

Surge Valves

Surge Valve elements represent a surge-anticipator valve (SAV), a surge relief valve (SRV), or both of them combined. A SAV opens on low pressure in anticipation of a subsequent high pressure. A SRV opens when pressure exceeds a threshold value.

The following attributes describe the surge-anticipator valve behavior:

- **Threshold Pressure (SAV)**: Pressure below which the SAV opens.
- **SAV Closure Trigger**: The closure of an open/opening SAV is initiated either by time (Time SAV Stays Fully Open attribute) or the threshold pressure (Threshold Pressure attribute), but not both. When based on pressure, the SAV will begin to close when the pressure rises back above the specified Threshold Pressure (SAV) value, which may occur before the SAV has fully opened.
- **Time for SAV to Open**: Amount of time that the SAV takes to fully open after being triggered.
- **Time SAV Stays Fully Open**: Amount of time that the SAV remains fully open (i.e., the time between the end of opening phase and the start of the closing phase).
- **Time for SAV to Close**: Amount of time for the SAV to close fully, measured from the time that it was completely open.

There are three optional valve configurations as defined by the attribute SAV/SRV type: (1) Surge Anticipator Valve, (2) Surge Relief Valve, and (3) Surge Anticipator & Relief Valve.

For the SAV, at full opening it's capacity is represented by the discharge coefficient Cv, while the valve characteristics at partial openings are provided by the valve curves discussed in Closing Characteristics of Valves (note that there is no user-specified valve currently provided for the SAV).

The SRV is modelled as being comprised of a vertical-lift plate which is resisted by a compressed spring. At the threshold pressure, there is an equilibrium between the compressive force exerted by the valve's spring on the movable plate and the counter force applied by the pressure of the liquid. For a linear spring, the lift x is given by the equation: $A (P - P_0) = k x$, where A is the pipe area, P is the instantaneous pressure, P_0 is the threshold pressure, and k is the spring constant. In this formulation, the acceleration of the spring and plate system is ignored. As the plate lifts away from the pipe due to the excess pressure, more flow can be vented to atmosphere to a maximum value at 0.937 times the pipe diameter.

Check Valves

There are several types of check valves available for the prevention of reverse flow in a hydraulic system. The simplest and often most reliable are the ubiquitous swing check valves, which should be carefully selected to ensure that their operational characteristics (such as closing time) are sufficient for the transient flow reversals that can occur in the system. Some transient flow reversal conditions can occur very rapidly; thus, if a check valve cannot respond quickly enough, it may slam closed and cause the valve or piping to fail.

Check valves that have moving discs and parts of significant mass have a higher inertia and therefore tend to close more slowly upon flow reversal. Check valves with lighter checking mechanisms have less inertia and therefore close more quickly. External counterweights present on some check valves (such as swing check valves) assist the valve closing following stoppage of flow. However, for systems that experience very rapid transient flow reversal, the additional inertia of the counterweight can slow the closing time of the valve. Spring-loaded check valves can be used to reduce closing time, but these valves have higher head loss characteristics and can induce an oscillatory phenomenon during some flow conditions.

It is important that the modeler understand the closing characteristics of the check valves being used. For example, ball check valves tend to close slowly, swing check valves close somewhat faster (unless they are adjusted otherwise), and nozzle check valves have the shortest closing times. Modeling the transient event with closing times corresponding to different types of check valves can indicate if a more expensive nozzle-type valve is worthwhile.

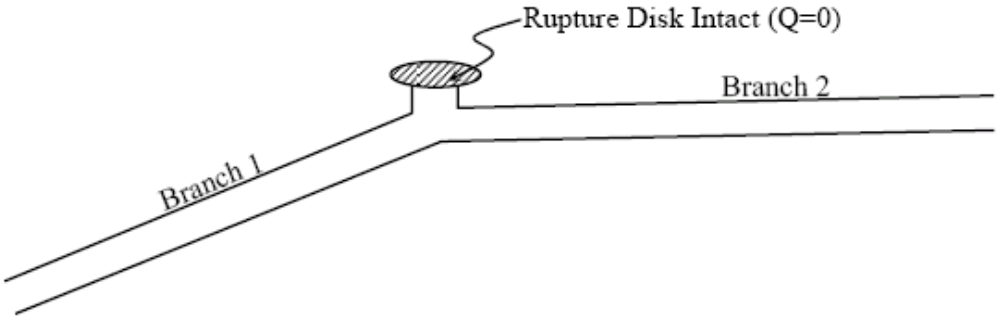
The following attributes describe the check valve behavior:

- **Open Time**: Amount of time to open the valve, from the fully closed position, after the specified Pressure (Threshold) value is exceeded. This establishes the rate of opening if the valve's closure is partial.
- **Closure Time**: Amount of time to close the valve, from the fully open position, after reverse flow is sensed. This establishes the rate of opening if the valve's closure is partial.
- **Allow Disruption of Operation?**: Allows you to define whether an operation (opening or closing) can be terminated prematurely due to a signal to reverse.
- **Pressure (Threshold)**: The pressure difference between the upstream and downstream side that triggers the valve to (re)open the (closed) valve. If 0 is entered, the valve (re)opens when the upstream pressure esceeds the downstream pressure.

Rupture Disks

A rupture disk node is located between two pipes. It is designed to fail when a specified threshold pressure is reached. This creates an opening in the pipe through which flow can exit the system to atmosphere.

If the disk is intact, then this node is represented as a typical Junction. After the threshold pressure is exceeded, it is presumed that the disk has blown off and the liquid rushes out of the newly-created orifice discharging to atmosphere.



Rupture Disk between 2 Pipes

Discharge to Atmosphere Elements

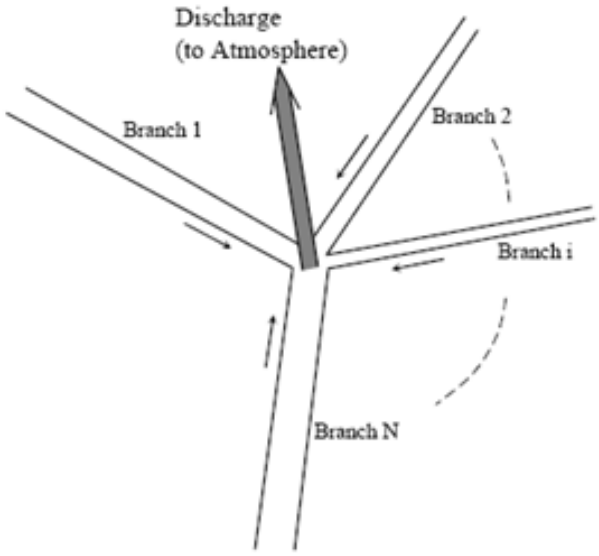
Models a point where flow leaves the pipe network and discharges to atmosphere. There are three choices for the Discharge Element Type:

- **Orifice** - represents an opening to atmosphere at a junction of two or more pipes or the end of a single pipe. The initial pressure is typically positive and there is usually an outflow from the system at time zero. If the pressure P is positive, then the outflow/demand is $Q = \Sigma\ Q_i$. summed over all the Branches, i. P varies quadratically with Q. When the pressure drops to zero, this element allows air to enter the pipeline freely on the assumption that the opening for the liquid is infinite for air. In this case, the air pocket respectively expands or contracts accordingly as the liquid flows away from or towards the node, but the air remains at the branch end point(s) located at the orifice.
- **Valve** - discharges water from the system at a pipe end open to atmospheric pressure. It is essentially an Orifice to Atmosphere with a variable diameter which could become zero; optionally, the valve can start the simulation in the closed position and proceed to open after a time delay. As long as the diameter is positive, either outflow for positive pressure or injection of air for zero pressure are possible. In the latter case, the rate of change of the air volume X_i in each branch

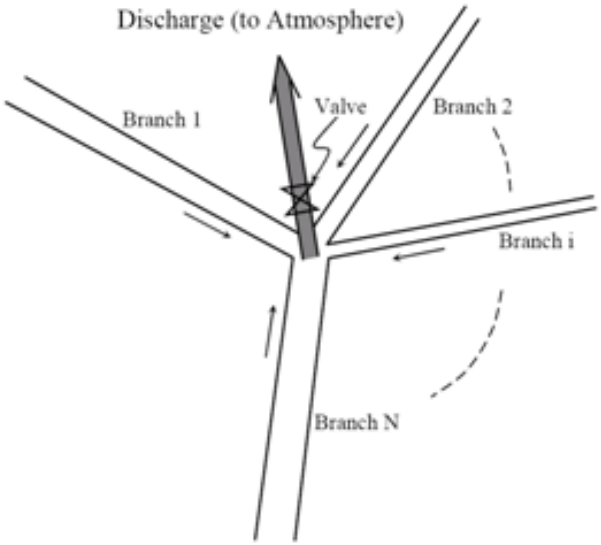
is described by the relation $dX_i / dt = - Q_i$, with the total volume X being the summation over all branch volumes X_i. After the valve closes, it behaves like a Junction element (and as a dead end junction if there is only a single branch connected).

- **Rating Curve** - releases water from the system to atmosphere based on a customizable rating curve relating head and flow. Below a certain value of head, the discharge is zero; in stage-discharge relations, head is equivalent to level for which the discharge increases with increasing level.

.



Orifice to Atmosphere



Valve to Atmosphere

Orifice Between Pipes Elements

This element represents a fixed-diameter orifice which breaks pressure, useful for representing choke stations on high-head pipelines.

Valve with Linear Area Change Elements

This element functions either as a check valve that closes instantaneously and remains closed when reverse flow occurs, or as a positive-acting leaf valve closing linearly over the prescribed time. An ideal valve useful for verifying best-case assumptions or representing motorized valves.

The head loss/discharge coefficient accounts for the vena contracta by means of a formula for two-dimensional flow solved with the Schwartz-Christoffel transformation.

If the check valve closes, it remains shut independent of the pressure difference across it. When the valve is closed, independent vapor pockets can exist on both sides of the valve.

Surge Tanks

A surge tank (also known as a stand pipe) typically has a relatively small volume and is located such that its normal water level is typically equal to the hydraulic grade line at steady state. When low transient pressures occur, the tank feeds water into the system by gravity to avoid subatmospheric pressure at the tank connection and vicinity.

There are two different surge tank types, as defined in the attribute called Surge Tank Type.

Simple Surge Tanks

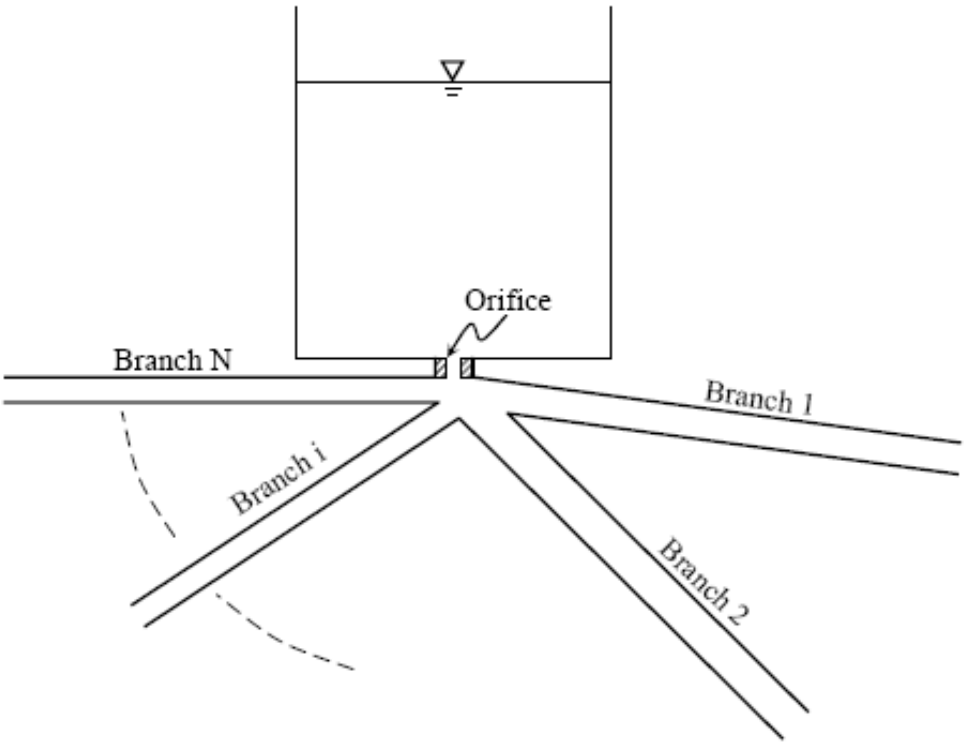
This node can operate in three distinct modes during a transient analysis: normal (level between the top and the connecting pipe(s) at the bottom); weir overflow (level at the top) with the cumulative volume being tracked and printed in the output log; and drainage (level at the elevation of the connecting branch(es)).

If equipped with an optional check valve, it becomes a one-way surge tank which supplies the pipeline with liquid whenever the adjacent head is sufficiently low (the refilling operation is a slow process which is not represented in HAMMER). During normal operation, the continuity equation applied to this node is $dH_T / dt = Q / A$, where H_T is the tank level, A is the tank's cross-sectional area and $Q = \Sigma Q_i$ is the net inflow to the tank. At the mouth of the tank, there is a differential orifice with head loss $\Delta H = H \angle H_T = bdQ^2 / (2gA_{or^2})$, where the subscripts T and or

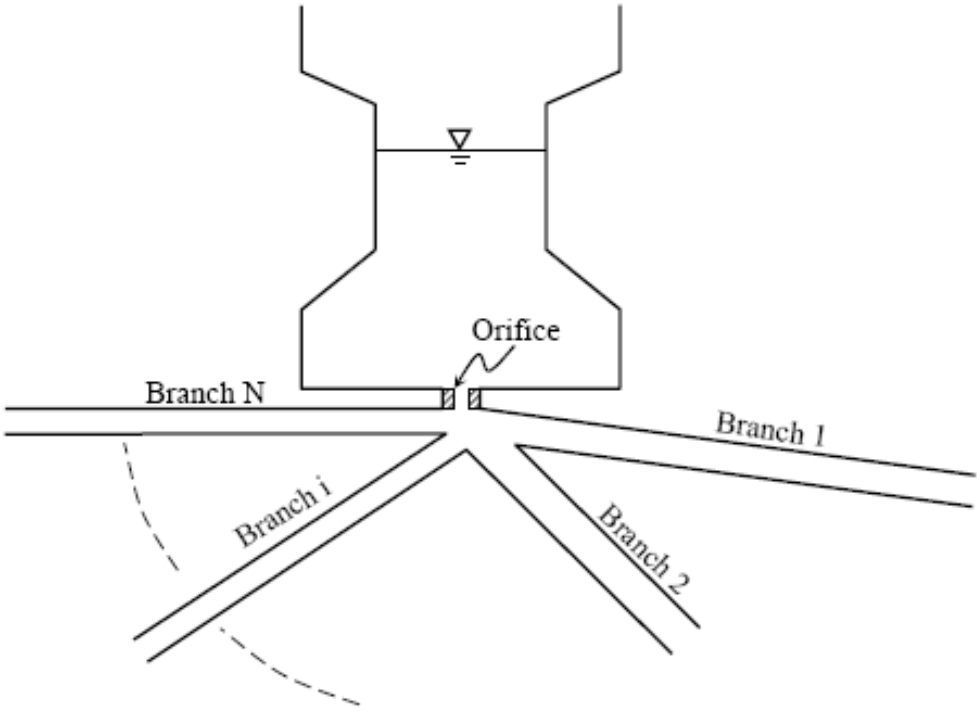
refer to the tank and orifice, respectively, b is the head loss coefficient and d = d_i for inflow (Q > 0) and -1 for outflow (Q < 0). By definition, d (known as the Ratio of Losses in HAMMER) asserts that head losses are d_i times greater for inflow than for outflow. A typical value of d_i is 2.5.

A user can optionally choose a Section type for the Simple Surge Tank. The choices are: a). Circular - so a tank diameter is required; b). non-circular - so an equivalent cross-sectional area is required; or c). variable area - where the cross-sectional area is provided in a table as a function of elevation. Note that for variable area tanks there is

no facility for a check valve to preclude inflow to the tank.



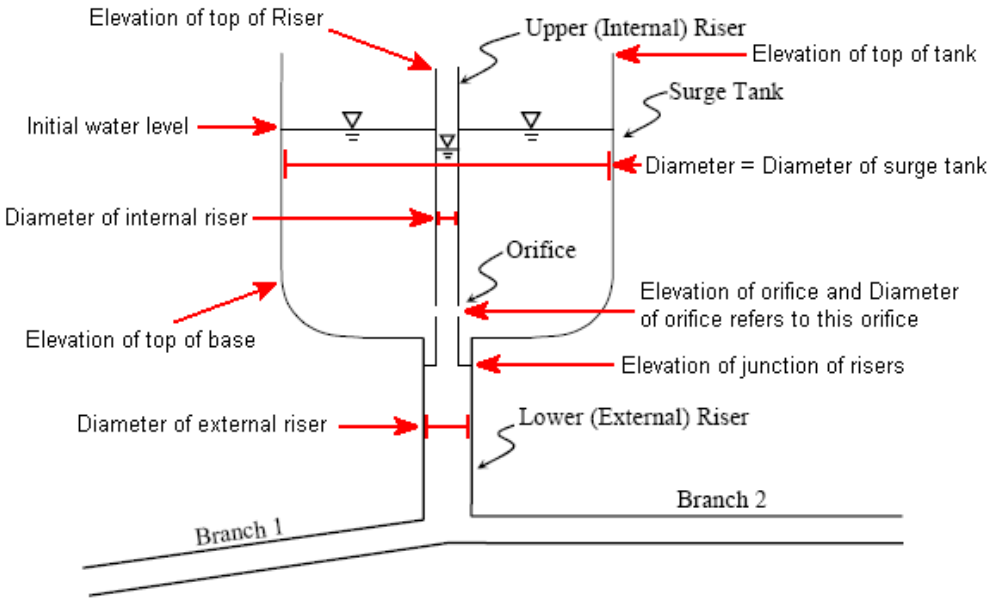
Surge Tank (Simple)



Surge Tank (Variable Area)

Differential Surge Tanks

There are numerous modes of operation for differential surge tanks ranging from drainage, with the entry of air into the pipeline, to overflow from the tank. Other modes are distinguished by the riser level relative to the orifice elevation and the tank level versus the top of the riser. For "normal" operation, the tank level is between the orifice and the top of the riser. During a powerful upsurge, the upper riser will overflow into the tank to complement the orifice flow.



Surge Tank (Differential) between 2 Pipes

Other Tools

Although WaterCAD V8*i* is primarily a modeling application, some additional drafting tools can be helpful for intermediate calculations and drawing annotation. MicroStation and AutoCAD provide a tremendous number of drafting tools. Bentley WaterCAD V8*i* itself (including Stand-Alone) provides the following graphical annotation tools:

- Border tool
- Text tool
- Line tool.

You can add, move, and delete graphical annotations as you would with any network element (see Manipulating Elements on page 4-403).

Border Tool

The Border tool adds rectangles to the drawing pane. Examples of ways to use the Border tool include drawing property lines and defining drawing boundaries.

To Draw a Border in the Drawing View

1. Click the **Border** tool in the **Layout** toolbox.
2. Click in the drawing to define one corner of the border.
3. Drag the mouse cursor until the border is the shape and size you want, then click.

Text Tool

The text tool adds text to the drawing pane. Examples of ways to use the Text tool include adding explanatory notes, titles, or labels for non-network elements. The size of the text in the drawing view is the same as the size of labels and annotations. You can define the size of text, labels, and annotation in the **Drawing** tab of the **Tools** > **Options** dialog.

To Add Text to the Drawing View

1. Click the **Text** tool in the **Layout** toolbox.
2. Click in the drawing to define where the text should appear.
3. In the **Text Editor** dialog, type the text as it should appear in the drawing view, then click **OK**. Note that text will be in a single line (no carriage returns allowed). To add multiple lines of text, add each line separately with the Text tool.

To Rotate Existing Text in the Drawing View

1. Click the **Select** tool in the **Layout** toolbox.
2. Right-click the text and select the **Rotate** command.
3. Move the mouse up or down to define the angle of the text, then click when done.

To Edit Existing Text in the Drawing View

1. Click the **Select** tool in the **Layout** toolbox.
2. Right-click the text and select the **Edit Text** command.
3. Make the desired changes in the **Text Editor** dialog that appears, then click **OK**.

Line Tool

The Line tool is used to add lines and polylines (multi segmented lines) to the drawing pane. Bentley WaterCAD V8*i* can calculate the area inside a closed polyline. Examples of ways to use the Line tool include drawing roads or catchment outlines.

To Draw a Line or Polyline in the Drawing View

1. Click the **Line** tool in the **Layout** toolbox.
2. Click in the drawing to define where the line should begin.
3. Drag the mouse cursor and click to place the line, or to place a bend if you are drawing a polyline.
4. Continue placing bends until the line is complete, then right-click and select **Done**.

To Close an Existing Polyline in the Drawing View

1. Click the **Select** tool in the **Layout** toolbox.
2. Right-click the polyline and select the **Close** command.

To Calculate the Area of a Closed Polyline

1. Click the **Select** tool in the **Layout** toolbox.
2. Right-click the polyline and select the **Enclosed Area** command.

To Add a Bend to an Existing Line or Polyline

1. Click the **Select** tool in the **Layout** toolbox.
2. Right-click at the location along the line or polyline where the bend should be placed and select the **Bend** > **Add Bend** command.

To Remove Bends from an Existing Line or Polyline

1. Click the **Select** tool in the **Layout** toolbox.
2. Right-click the bend to be removed and select the **Bend** > **Remove Bend** command. To remove all of the bends from a polyline (not a closed polyline), right-click the polyline and select the **Bend** > **Remove All Bends** command.

   Note: For a 64-bit installation of HAMMER, the folder location is C:\Program Files\Bentley\HAMMER8\x64.

3.

How The Pressure Engine Loads Bentley HAMMER Elements

The pressure engine models the various HAMMER elements as follows:

- Periodic Head/Flow Element using Head: A reservoir with the HGL determined from the sinusoidal wave properties, or from the head pattern. Only the initial (time zero) HGL is applied so that the steady state analysis will correspond to the transient initial conditions.
- Periodic Head/Flow Element using Flow: A junction with demand determined from the sinusoidal wave properties, or from the flow pattern. Only the initial (time zero) flow is applied so that the steady state analysis will correspond to the transient initial conditions.
- Air Valve: If the "Treat Air Valve as Junction" property is set to True the Air Valve is loaded as a junction with no demand. If the "Treat Air Valve as Junction" property is set to False, the air valve is loaded such that it opens the system to atmosphere. This is most commonly used to simulate high points in pumped sewer systems, so the default behavior is to treat the air valve as a junction.
- Hydropneumatic Tank: A hydropneumatic tank is loaded as a normal tank with the properties of the tank being dictated by the tank calculation model that is used.
- Surge Valve: Junction with no Demand.
- Check Valve: Short Pipe with a Check Valve in line with the direction of flow.
- Rupture Disk: Junction with no demand.
- Discharge to Atmosphere: For the Orifice and Valve types this element is loaded as a junction with emitter coefficient determined by the flow and pressure drop properties. If either of these properties are invalid (<= 0) then no emitter coefficient is loaded. Furthermore, for the valve type if the valve is initially closed, no emitter coefficient is loaded. For the rating curve type this element is loaded as a reservoir connected to a GPV with rating curve used as the GPV headloss curve.
- Valve with linear area change: GPV with a headloss curve based on the valve's discharge coefficient.
- Turbine: GPV using the turbine’s headloss curve.
- Orifice: GPV with a headloss curve calculated from the nominal head/flow loss using the orifice equation.
- Surge Tank: Without a check valve, this element is loaded as a tank. With a check valve this element is loaded as a Junction.

Adding Elements to Your Model

WaterCAD V8*i* provides several ways to add elements to your model. They include:

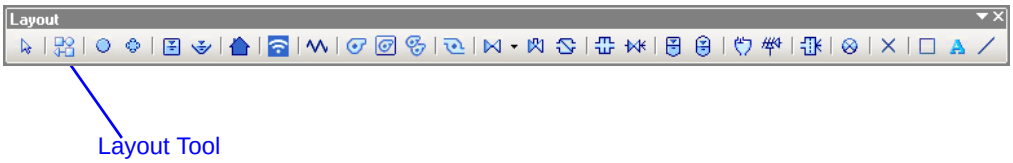
- Adding individual elements
- Adding elements using the layout tool
- Replacing an element with another element.

To add individual elements to your model

1. Click an element symbol on the Layout toolbar. The mouse cursor changes to the element symbol you selected.
2. Click in the drawing pane to add the element to your model.
3. Click again to add another element of the same type to your model.
4. To add a different element, click on the desired element symbol in the Layout toolbar, then click in the drawing pane.
5. To stop adding elements, right-click in the drawing pane to display a shortcut menu, then click **Done**.

To add elements using the layout tool

The layout tool is used to quickly add new elements to your model without having to select a new element button on the Layout toolbar. When the layout tool is active, you can right-click in the drawing pane to select different elements and pipes to add to the model.



1. Click the **Layout** tool on the **Layout** toolbar.
2. Right-click in the drawing pane, then select the type of element you want to add from the shortcut menu. The shortcut menu displays only those element types that are compatible with your pipe selection.
3. Click in the drawing pane to add the element.
4. Click again to add another of the same element type. The elements you add will automatically be connected by pipes.

5. To change the element, right-click and select a different element from the shortcut menu.
6. To stop adding elements using the Layout tool, right-click anywhere in the drawing pane and click **Done**.

Manipulating Elements

You can manipulate elements in your model in any one of the following ways:

- Select elements—Manually select individual elements, manually select multiple elements, select all elements, or select all elements of a single element type. See Select, Move, and Delete Elements.
- Move elements—Move elements in the drawing pane. See Select, Move, and Delete Elements.
- Delete elements—Remove elements from the model. See Select, Move, and Delete Elements.
- Split pipes—Split an existing pipe into two new pipes by adding a new node element along the existing pipe. See Splitting Pipes.
- Reconnect pipes—Disconnect an exisiting pipe from an existing node element and attach it to another existing node element. See Reconnect Pipes.
- Model curved pipes—You can lay out curved pipes. See Modeling Curved Pipes.
- Assign isolation valves to pipes—This tool finds the nearest pipe for each of the specified isolation valves and assigns the valve to that pipe. See Assign Isolation Valves to Pipes Dialog Box.
- Batch split pipes—This tool allows you to split pipes with neighboring nodes that are found within the specified tolerance. See Batch Pipe Split Dialog Box.
- Batch morph nodes—This tool allows you to morph a selected node type into another type of node element as a batch operation. See Batch Morph.
- Merge nodes in close proximity—allows you to merge together nodes that fall within a specified tolerance of one another. See Merge Nodes in Close Proximity.
- Select links adjacent to one or more nodes—This command lets you select all link elements attached to one or more nodes. See Select Adjacent Links.

Select, Move, and Delete Elements

The following element selection options are available:

To manually select an element

Click the element. Selected elements appear in red.

Note: You can change the selection color in the Options dialog box, which is accessible by selecting Tools > Options.

To manually select multiple elements

Click the first element, then click additional elements while holding down **Shift** or **Ctrl**.

To select elements by drawing a polygon

1. Select **Edit > Select By Polygon**.
2. Click in the drawing pane near the elements you want to select, then drag the mouse to draw the first side of the polygon.
3. Click again to finish drawing the first side of the polygon and drag the mouse to begin drawing the next side of the polygon.
4. Repeat step 3 until the polygon is complete, then right-click and select **Done**.

To select all elements

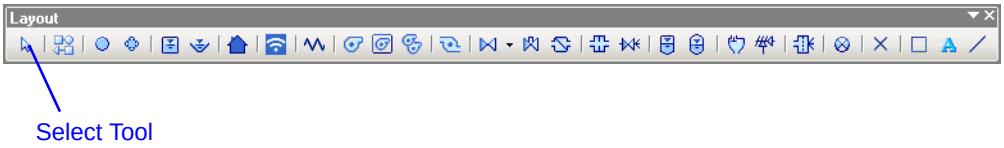
To select all of the elements in your model, select **Edit > Select All**.

To select all elements of the same type

To select all elements of the same type (for example, all junction chambers), select **Edit > Select by Element**, then click the desired element type.

All elements of the selected type appear in red, including connecting pipes.

To clear selected elements



Click the **Select** tool then click any blank space in the drawing pane.

or

Click **Edit** > **Clear Selection**.

or

Press the **Esc** key.

You can also clear a selected element by clicking a different element.

To move an element in the model

1. Click the Select tool on the Layout toolbar.
2. Select the element(s) you want to move, then drag it to its new location. Pipe connections move with the element.

To delete an element

Select the element, then press **Delete**.

or

Select **Edit** > **Delete**.

Splitting Pipes

You may encounter a situation in which you need to add a new element in the middle of an existing pipe.

To split an existing pipe

1. Select the desired element symbol on the Layout toolbar.
2. In the drawing pane, place the cursor over the pipe you want to split and click.
3. You are prompted to confirm that you want to split the pipe.

- If you choose to split the pipe, the element will be inserted and two new pipes will be created with the same characteristics as the original pipe (lengths are split proportionally).
- If you choose not to split the pipe, the new element will be placed on top of the pipe without connecting to anything.

If you accidentally split a pipe, this action can be undone by selecting **Edit > Undo**.

You can also split an existing pipe with an existing element. To do this, drag the element into position along the pipe to be split, then right-click the node and select **Split <Pipe Label>** from the shortcut menu (where **<Pipe Label>** is the name of the pipe to be split).

Reconnect Pipes

In certain circumstances, you may wish to disconnect a pipe from a node without deleting and redrawing the pipe in question. For example, if the model was built from a database and the Establish By Spatial Data option was used to determine pipe connectivity, pipes may have been connected to the wrong nodes.

To disconnect and reconnect a pipe:

1. Right-click the pipe to be disconnected close to the end of the pipe nearest the end that you want disconnected.
2. The pipe is now connected to the junction that it will remain connected to and your mouse cursor. Hover the mouse cursor over the junction to which you would like to connect the pipe and click the left mouse button. The pipe will now be connected to this junction.

Modeling Curved Pipes

You can model curved pipes in WaterCAD V8*i* by using the Bend command, which is available by right-clicking in the Drawing Pane when placing a link element.

WaterCAD V8*i* does not account for any additional head loss due to the curvature because in most cases the increased head loss is negligible. If you feel the extra head loss is significant, it is possible to increase the Manning's n value to account for such losses.

To model a curved pipe

1. Select the desired link element using the **Layout** button on the Layout toolbar.



2. Place the first segment of the curved pipe in your model, then right click and select **Bend** from the shortcut menu.
3. Repeat Step 2 for each segment in the curved pipe. Be sure to insert bends to clearly show the curved alignment.
4. When the curved pipe is complete, right click and select the next downstream element.

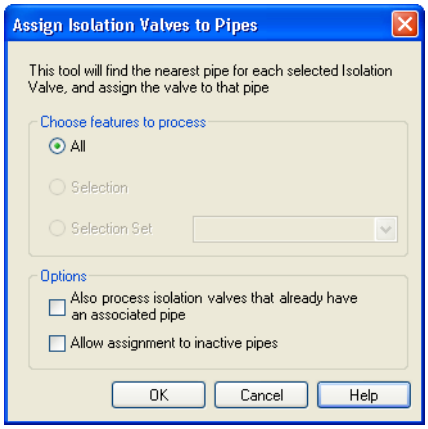
Polyline Vertices Dialog Box

This dialog box contains the *X* vs. *Y* table that allows you to define any number of points that plot the shape of the polyline representing the selected link element. The dialog box contains the following controls:

New	This button creates a new row in the table.
Delete	This button deletes the currently highlighted row from the table.

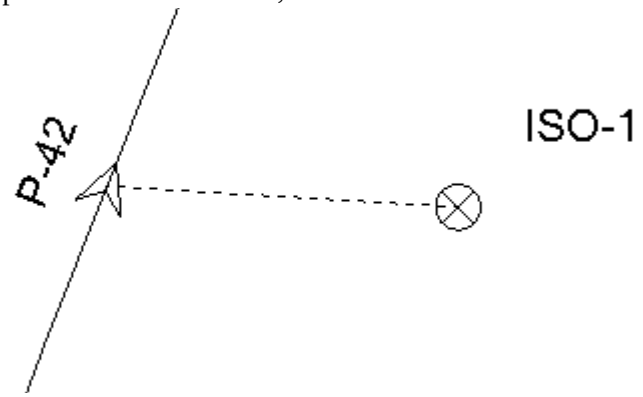
Assign Isolation Valves to Pipes Dialog Box

The Assign Isolation Valves to Pipes tool finds the nearest pipe for each of the specified isolation valves and assigns the valve to that pipe.



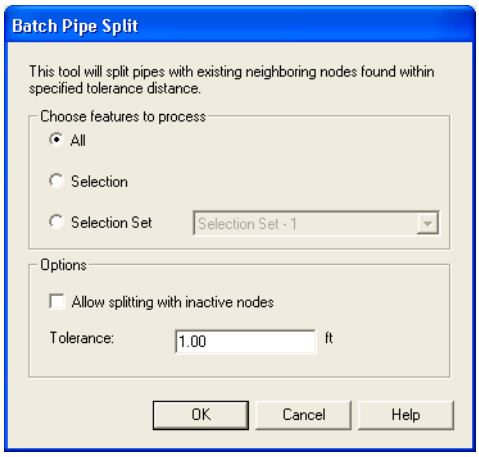
Choose Features to Process	Allows you to specify which isolation valves to include in the assignment operation. The following options are available: • **All**: All isolation valves within the model will be assigned to their nearest pipe. • **Selection**: Only the isolation valves that are currently selected in the drawing pane will be assigned to their nearest pipe. • **Selection Set**: Only those isolation valves that are contained within the selection set specified in the drop down list will be assigned to their nearest pipe.
Also process isolation valves that already have an associated pipe	When this box is checked, the assign operation will also assign to the nearest pipe those valves that are already assigned to a pipe.
Allow assignment to inactive pipes	When this box is checked, pipes that are marked Inactive will not be ignored during the assignment operation.

The relationship between an isolation valve and their referenced pipe is displayed in the drawing pane with a dashed line, like this:



Batch Pipe Split Dialog Box

The Batch Pipe Split dialog allows you to split pipes with neighboring nodes that are found within the specified tolerance.



Choose Features to Process

Allows you to specify which pipes to include in the split operation. The following options are available:

- **All**: All pipes in the model that have a neighboring node within the specified tolerance will be split by that junction.
- **Selection**: Only the pipes that are currently selected in the drawing pane will be split by a neighboring junction that lies within the specified tolerance.
- **Selection Set**: Only those pipes that are contained within the selection set specified in the drop down list will be split by a neighboring junction that lies within the specified tolerance.

Allow splitting with inactive nodes

When this box is checked, nodes that are marked Inactive will not be ignored during the split operation.

Tolerance

This value is used to determine how close a pipe must be to a node in order for the pipe to be split by that junction.

Pipes will be split by every junction that falls within the specified tolerance. To prevent unwanted pipe splits, first use the Network Navigator's "Network Review > Pipe Split Candidates" query to verify that the tolerance you intend to use for the Batch Split operation will not include nodes that you do not want involved in the pipe split operation.

To use the Network Navigator to assist in Batch Pipe Split operations

1. Open the **Network Navigator**.
2. Click the **[>]** button and select the **Network Review...Pipe Split Candidates** query.
3. In the **Query Parameters** dialog box, type the tolerance you will be using in the pipe split operation and click **OK**.
4. In the **Network Navigator**, highlight nodes in the list that you do not want to be included in the pipe split operation and click the **Remove** button.
5. Open the **Batch Pipe Split** dialog.
6. Click the **Selection** button.
7. Type the tolerance you used in the Network Review query and click **OK**.

Batch Pipe Split Workflow

We recommend that you thoroughly review and clean up your model to ensure that the results of the batch pipe split operation are as expected.

> **Note:** **Cleaning up your model is something that needs to be done with great care. It is best performed by someone who has good familiarity with the model, and/or access to additional maps/ personnel/information that will allow you to make the model match the real world system as accurately as possible.**

We provide a number of Network Navigator queries that will help you find "potential" problems (see Using the Network Navigator).

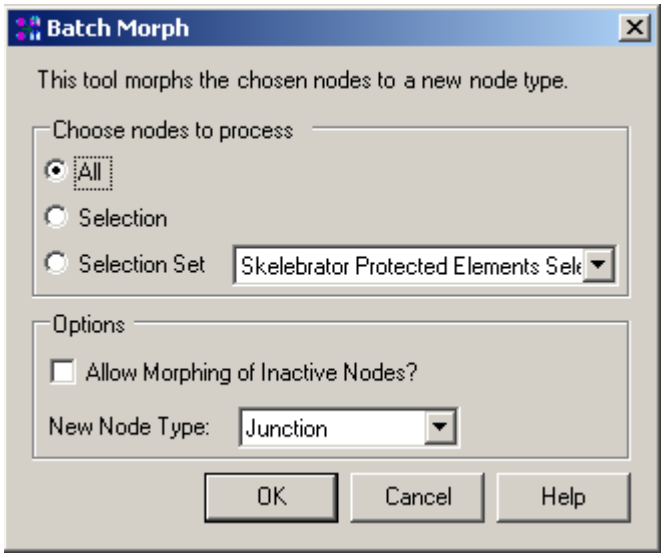
1. Review and clean up your model as much as possible prior to running the "batch split" operation. Run the "duplicate pipes" and "nodes in close proximity" queries first. (Click the View menu and select Queries. In the Queries dialog expand the Queries-Predefined tree. The Duplicate Pipes and Nodes in Close Proximity queries are found under the Network Review folder.)
2. Next, use the network navigator tool to review "pipe split candidates" prior to running batch split.
   a. Using the network navigator tool, run the "pipe split candidates" query to get the list of potential batch split candidate nodes. Take care to choose an appropriate tolerance (feel free to run the query multiple times to settle on a tolerance that works best; jot down the tolerance that you settle on, you will want to use that same tolerance value later when you perform the batch split operation).
   b. Manually navigate to and review each candidate node and use the "network navigator" remove tool to remove any nodes that you do not want to process from the list.

c. After reviewing the entire list, use the network navigator "select in drawing" tool to select the elements you would like to process.

d. Run the batch split tool. Choose the "Selection" radio button to only process the nodes that are selected in the drawing. Specify the desired tolerance, and press OK to proceed.

Batch Morph

This tool allows you to morph a selected node type into another type of node element as a batch operation.



First, select the nodes to be morphed from the following choices:

- **All**: All nodes in the model will be morphed to the specified **Target Element Type**.
- **Selection**: Only the nodes that are currently selected in the drawing pane will be morphed to the specified **Target Element Type**.
- **Selection Set**: Only those nodes that are contained within the selection set specified in the drop down list will be morphed to the specified **Target Element Type**.

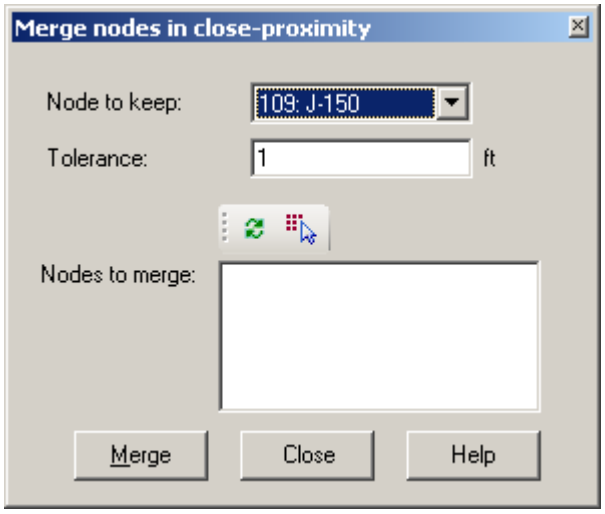
Check the **Allow Morphing of Inactive Nodes?** box to include nodes set as Inactive in the batch operation.

Finally, select the **Target Element Type** that the selected nodes will be morphed into.

Note: **Users can morph junction elements into Isolation Valves using two steps: First, morph the desired junctions into TCV's, GPV's, or PBV's. Then use the Skelebrator "Inline Isolation Valve Replacement" operation.**

Merge Nodes in Close Proximity

This dialog allows you to merge together nodes that fall within a specified tolerance of one another.



To access the dialog, right-click one of the nodes to be merged and select the Merge nodes in close proximity command.

The dialog consists of the following controls:

Node to keep: Displays the node that will be retained after the merge operation.

Tolerance: Allows you to define the tolerance for the merge operation. Nodes that fall within this distance from the "Node to keep" will be available in the "Nodes to merge" pane.

Refresh: Refreshes the nodes displayed in the "Nodes to merge" pane. Click this button after making a change to the tolerance value to update the list of nodes available for the merge operation.

Select nodes to merge: Toggle this button on to select the nodes that are selected in the "Nodes to merge" pane in the drawing pane.

Nodes to merge: This pane lists the nodes that fall within the specified tolerance of the "Node to keep". Nodes whose associated boxes are checked will be merged with the Node to keep when the Merge operation is initiated.

Merge: Performs the merge operation using the nodes whose boxes are checked in the "Nodes to merge" list.

Close: Closes the dialog without performing the merge operation.

Select Adjacent Links

This command allows you to select all link elements attached to one or more nodes.

To find all links adjacent to a single node, right-click the node and click the Select Adjacent Links command.

You can also find all links adjacent to a group of selected nodes; with multiple nodes selected in the drawing view, right-click one of them and click the Select Adjacent Links command.

Editing Element Attributes

You edit element properties in the Property Editor, one of the dock-able managers in WaterCAD V8*i*.

To edit element properties:

Double-click the element in the drawing pane. The Property Editor displays the attributes of the selected element.

or

Select the element whose properties you want to edit, then select **View > Properties** or click the Properties button on the Analysis toolbar.

Property Editor

The Property Editor is a contextual dialog box that changes depending on the status of other dialog boxes. For example, when a network element is highlighted in the drawing pane, the Property Editor displays the attributes and values associated with that element. When one of the manager dialog boxes is active, the Property Editor displays the properties pertaining to the currently highlighted manager element.

Attributes displayed in the Property Editor are grouped into categories. An expanded category can be collapsed by clicking the minus (-) button next to the category heading. A collapsed category can be expanded by clicking the plus (+) button next to the category heading.

You can change the sorting to alphabetical by clicking the Search button and selecting “Arrange Alphabetically”.

For the most efficient data entry in Text Box style fields, instead of clicking on the Field, click on the label to the left of the field you want to edit, and start typing. Press Enter to commit the value, then use the Up/Down keyboard arrows to navigate to the next field you want to edit. You can then edit the field data without clicking the label first; when you are finished editing the field data, press the Enter key, and proceed to the next field using the arrow keys, and so on.

Find Element

The top section of the Property Editor contains the Find Element tool. The Find Element tool is used to:

- Quickly find a recently-created or added element in your model. The Element menu contains a list of the most recently-created and added elements. Click an element in the Element menu to center the drawing pane around that element and highlight it.
- Find an element in your model by typing the element label or ID in the Element menu then clicking the Find button or pressing Enter. The drawing pane centers around the highlighted element.
- Find all elements of a certain type by using a percent sign (%) as a wild-card character. For example, if you want to find all of the pipes in your model, you type co% (this is not case-sensitive) then click the Find button. The drawing pane centers around and highlights the first instance of a pipe in your model, and lists all pipes in your model in the Element menu. For more information about using wildcards, see Using the Like Operator.
- % and _ are wildcard characters. If the element(s) you are looking for contains one or more of those characters, you will need to prefix each one of those characters in the search term with \. E.g. J\%1
- If Find returns multiple results then Network Navigator automatically opens.

The following controls are included:

Element — Type an element label or ID in this field then click the Find button to quickly locate it in your model. The element selected in this menu will be centered in the drawing pane when the Zoom To command is initiated, at the magnification level specified by the Zoom Level menu. The drop-down menu lists recently-created or added elements, elements that are part of a selection set, and that are part of the results from a recent Find operation.

Find Previous  — This button allows you to find the previous element in the list of results from a recent Find operation.

Find  — Zooms the drawing pane view to the element typed or selected in the Element menu at the magnification level specified in the Zoom Level menu.

Find Next  — This button allows you to find the next element in the list of results from a recent Find operation.

Help  — Displays online help for the Property Editor.

Zoom Level — Allows you to specify the magnification level at which elements are displayed in the drawing pane when the Zoom To command is initiated.

Alphabetic  — Displays the attribute fields in the Property Editor in alphabetical order.

Categorized  — Displays the attribute fields in the Property Editor in categories. This is the default.

Property Search

You can search for a specific attribute by typing the name of the attribute into the search box and clicking the Search button  .

When you have entered one or more search terms, only those properties containing the search term will be displayed in the property editor.

When the box contains search terms the Search button turns to a Clear button  . Click this button to clear the terms from the search box.

To match multiple items, enter the desired list of terms separated by semicolon without spaces in between.

A maximum of 12 search terms are stored in the search box. Click the down arrow to view the last 12 search terms that were used; clicking an entry in this list will make that search term active.

Labeling Elements

When elements are placed, they are assigned a default label. You can define the default label using the **Labeling** tab of the **Tools > Options** dialog.

You can also relabel elements that have already been placed using the Relabel command in the element FlexTables.

Relabeling Elements

You can relabel elements from within the Property Editor.

To relabel an element

1. Select the element in the Drawing Pane then, if the Property Editor is not already displayed, select **View > Properties**.
2. In the General section of the Property Editor, click in the Label field, then type a new label for the element.

Set Field Options Dialog Box

The Set Field Options dialog box is used to set the units for a specific attribute without affecting the units used by other attributes or globally.

To use the Set Field Options dialog box, right-click any numerical field that has units, then select **Units and Formatting**.

Value	Displays the value of the currently selected item.
Unit	Displays the type of measurement. To change the unit, select the unit you want to use from the drop-down list. With this option you can use both U.S. customary and S.I. units in the same worksheet.
Display Precision	Sets the rounding of numbers and number of digits displayed after the decimal point. Enter a number from 0 to 15 to indicate the number of digits after the decimal point.
Format	Selects the display format used by the current field. Choices include: • **Scientific**—Converts the entered value to a string of the form "-d.ddd...E+ddd" or "-d.ddd...e+ddd", where each 'd' indicates a digit (0-9). The string starts with a minus sign if the number is negative. • **Fixed Point**—Abides by the display precision setting and automatically enters zeros after the decimal place to do so. With a display precision of 3, an entered value of 3.5 displays as 3.500. • **General**—Truncates any zeros after the decimal point, regardless of the display precision value. With a display precision of 3, the value that would appear as 5.200 in Fixed Point format displays as 5.2 when using General format. The number is also rounded. So, an entered value of 5.35 displays as 5.4 regardless of the display precision. • **Number**—Converts the entered value to a string of the form "-d,ddd,ddd.ddd...", where each 'd' indicates a digit (0-9). The string starts with a minus sign if the number is negative. Thousand separators are inserted between each group of three digits to the left of the decimal point.

Date/Time Formats

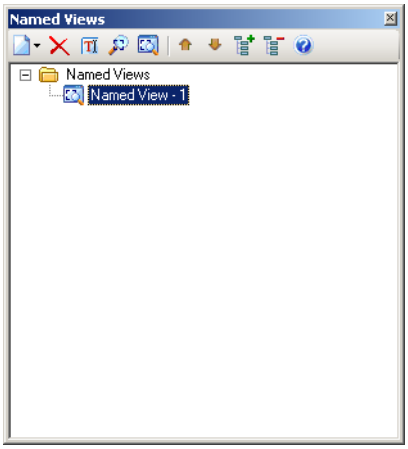
You can pick from various predetermined date/time formats. The following is a list of supported formats, and a sample of what the format will look like for 1 year, 1 month, 1 day, 1 hour, 1 minute, and one second into the simulation.

- **Elapsed Time Short**: 9504.04 (hours)
- **Elapsed Time Long**: 396:01:01:01
- **Short Time**: 1:01 AM
- **Long Time**: 1:01:01 AM
- **Short Date**: 2/01/2009
- **Long Date**: Monday, Feb 01, 2009
- **Short Date & Short Time**: 2/01/2009 1:01 AM
- **Short Date & Long Time**: 6/15/2009 1:01:01 AM
- **Long Date & Short Time**: Monday, Feb 01, 2009 1:01 AM
- **Long Date & Long Time**: Monday, Feb 01, 2009 1:01:01 AM
- **Sortable Date & Time**: 2009-01-01T01:01:01
- **Universal Sortable Date & Time**: 2009-01-01 01:01:01Z
- **Universal Full Date & Time**: Monday, Feb 01, 2009 01:01:01 AM

Using Named Views

The Named View dialog box is where you can store the current views X and Y coordinates. When you set a view in the drawing pane and add a named view, the current view is saved as the named view. You can then center the drawing pane on the named view with the **Go To View** command.

Choose View > Named Views to open the Named View dialog box.



The toolbar contains the following controls:

New 	Contains the following commands: • **Named View**—Opens a Named View Properties box to create a new named view. • **Folder**—Opens a Named Views Folder Properties box to enter a label for the new folder.
Delete 	Deletes the named view or folder that is currently selected.
Rename 	Rename the currently selected named view or folder.
Go to View 	Centers the drawing pane on the named view.
Update Named View 	Updates the currently highlighted view using the current view in the drawing pane.
Shift Up and Shift Down 	Moves the selected named view or folder up or down.
Expand All or Collapse All 	Expands or collapses the named views and folders.
Help 	Displays online help for Named Views.

Using Selection Sets

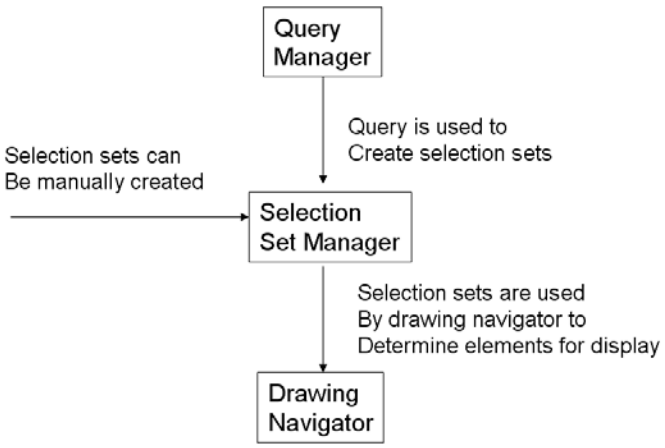
Selection sets are user-defined groups of network elements. They allow you to predefine a group of network elements that you want to manipulate together. You manage selection sets in the Selection Sets Manager.

WaterCAD V8*i* contains powerful features that let you view or analyze subsets of your entire model. You can find these elements using the Network Navigator (see Using the Network Navigator). The Network Navigator is used to choose a selection set, then view the list of elements in the selection set or find individual elements from the selection set in the drawing.

In order to use the Network Navigator, you must first create a selection set. There are two ways to create a selection set:

- From a selection of elements—You create a new selection set in the Selection Sets Manager, then use your mouse to select the desired elements in the drawing pane.
- From a query—Create a query in the Query Manager, then use the named query to find elements in your model and place them in the selection set.

The following illustration shows the overall process.



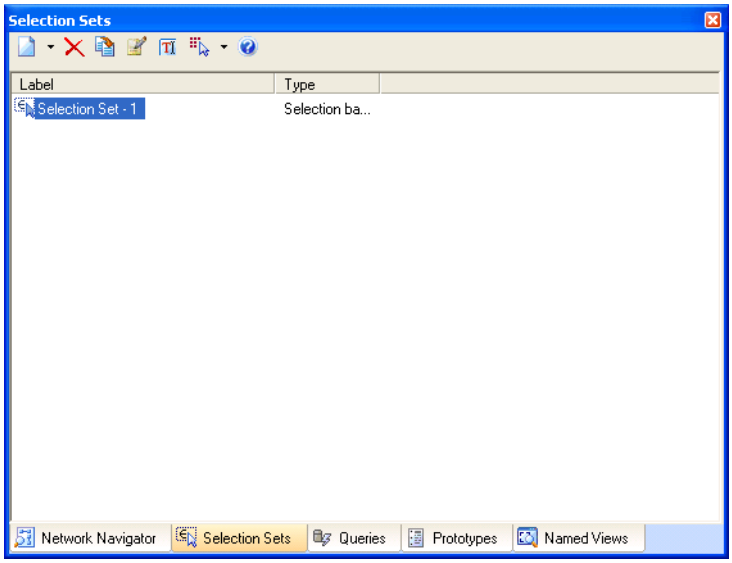
You can perform the following operations with selection sets:

- To view elements in a Selection Set on page 4-423
- To Create a Selection Set from a Selection on page 4-424
- To create a Selection Set from a Query on page 4-424
- To add elements to a Selection Set on page 4-425
- To remove elements from a Selection Set on page 4-426

Selection Sets Manager

The Selection Sets Manager is used to create, edit, and navigate to selection sets. The Selection Sets Manager consists of a toolbar and a list pane, which displays all of the selection sets that are associated with the current project.

To open Selection Sets, click the View menu and select the Selection Sets command, press <Ctrl+4>, or click the Selection Sets button  on the View toolbar.



The toolbar contains the following buttons:

New



Contains the following commands:

- **Create from Selection**—Creates a new static selection set from elements you select in your model.
- **Create from Query**—Creates a new dynamic selection set from existing queries.

Delete



Deletes the selection set that is currently highlighted in the list pane. This command is also available from the short-cut menu, which you can access by right-clicking an item in the list pane. You can hold down the Ctrl key while clicking on items in the list to select multiple entries at once.

Duplicate



Copies the Selection Set that is selected.

Edit 	• When a selection-based selection set is highlighted and you click this button, it opens the Selection Set Element Removal dialog box, which edits the selection set. This command is also available from the short-cut menu, which you can access by right-clicking an item in the list pane. • When a query-based selection set is highlighted and you click this button, it opens the Selection By Query dialog box, which adds or removes queries from the selection set. This command is also available from the short-cut menu, which you can access by right-clicking an item in the list pane.
Rename 	Renames the selection set that is currently highlighted in the list pane. This command is also available from the short-cut menu, which you can access by right-clicking an item in the list pane.
Select In Drawing 	Selects all the elements in the drawing pane that are part of the currently selected selection sets. This command is also available from the short-cut menu, which you can access by right-clicking an item in the list pane.
Help 	Displays online help for the Selection Sets Manager.

You can view the properties of a selection in the Property Editor by right-clicking the selection set in the list pane and selecting **Properties** from the shortcut menu.

To view elements in a Selection Set

You use the Network Navigator to view the elements that make up a selection set.

1. Open the Network Navigator by selecting View > Network Navigator or clicking the Network Navigator button on the View toolbar.
2. Select a selection set from the Selection Set drop-down list. The elements in the selection set appear in the Network Navigator.

Tip: You can double-click an element in the Network Navigator to select and center it in the Drawing Pane.

To Create a Selection Set from a Selection

You create a new selection set by selecting elements in your model.

1. Select all of the elements you want in the selection set by either drawing a selection box around them or by holding down the **Ctrl** key while clicking each one in turn.
2. When all of the desired elements are highlighted, right-click and select **Create Selection Set**.
3. Type the name of the selection set you want to create, then click **OK** to create the new selection set. Click **Cancel** to close the dialog box without creating the selection set.
4. Alternatively, you can open the Selection Set manager and click the **New** button and select **Create from Selection**. Bentley WaterCAD V8*i* prompts you to select one or more elements.

Create Selection Set Dialog Box

This dialog box opens when you create a new selection set. It contains the following field:

New selection set name Type the name of the new selection set.

To create a Selection Set from a Query

You create a dynamic selection set by creating a query-based selection set. A query-based selection set can contain one or more queries, which are valid SQL expressions.

1. In the Selection Sets Manager, click the **New** button and select **Create from Query**. The Selection by Query dialog box opens.
2. Available queries appear in the list pane on the left; queries selected to be part of the selection set appear in the list pane on the right. Use the arrow buttons in the middle of the dialog to add one or all queries from the Available Queries list to the Selected Queries list, or to remove queries from the Selected list.
   – You can also double-click queries on either side of the dialog box to add them to or remove them from the selection set.

Selection by Query Dialog Box

The Selection by Query dialog box is used to create selection sets from available queries. The dialog box contains the following controls:

Available Queries	Contains all the queries that are available for your selection set. The Available Columns list is located on the left side of the dialog box.
Selected Queries	Contains queries that are part of the selection set. To add queries to the Selected Queries list, select one or more queries in the Available Queries list, then click the **Add** button [>].
Query Manipulation Buttons	Select or clear queries to be used in the selection set: • [>] Adds the selected items from the Available Queries list to the Selected Queries list. • [>>] Adds all of the items in the Available Queries list to the Selected Queries list. • [<] Removes the selected items from the Selected Queries list. • [<<] Removes all items from the Selected Queries list. **Note: You can select multiple queries in the Available Queries list by holding down the Shift key or the Control key while clicking with the mouse. Holding down the Shift key provides group selection behavior. Holding down the Control key provides single element selection behavior.**

To add elements to a Selection Set

You can add a single or multiple elements to a static selection set.

1. Right-click the element to be added, then select **Add to Selection Set** from the shortcut menu.
2. In the Add to Selection Set dialog box, select the selection set to which you want to add the element.
3. Click **OK** to close the dialog box and add the element to the selected selection set. Click **Cancel** to close the dialog box without creating the selection set.

To add a group of elements to a static selection set all at once

1. Select all of the elements to be added by either drawing a selection box around them, or by holding down the **Ctrl** key while clicking each one in turn.
2. When all of the desired elements are highlighted, right-click and select **Add to Selection Set**.
3. In the Add to Selection Set dialog box, select the selection set to which you want to add the element.
4. Click **OK** to close the dialog box and add the element to the selected selection set. Click **Cancel** to close the dialog box without creating the selection set.

To Add To Selection Set Dialog Box

This dialog box opens when you select the Add to Selection Set command. It contains the following field:

Add to:	Selects the selection set to which the currently highlighted element or elements will be added.

To remove elements from a Selection Set

You can easily remove elements from a static selection set in the Selection Set Element Removal dialog box.

1. Display the Selection Sets Manager by selecting **View** > **Selection Sets** or clicking the **Selection Sets** button on the View toolbar.
2. In the Selection Sets Manager, select the desired selection set then click the **Edit** button.
3. In the Selection Set Element Removal dialog box, find the element you want to remove in the table. Select the element label or the entire table row, then click the **Delete** button.
4. Click **OK**.

Selection Set Element Removal Dialog Box

This dialog opens when you click the edit button from the Selection Sets manager. It is used to remove elements from the selection set that is highlighted in the **Selection Sets Manager** when the **Edit** button is clicked.

Group-Level Operations on Selection Sets

You can perform group-level deletions and reporting on elements in a selection set by using the Select In Drawing button in the Selection Sets Manager.

Note: While it is not possible to directly edit groups of elements in a selection set, you can use the Next button in the Network Navigator to quickly navigate through each element in the selection set and edit its properties in the Property Editor.

To delete multiple elements from a selection set

1. Open the Selection Sets Manager by selecting **View > Selection Sets** or clicking the **Selection Sets** button on the View toolbar.
2. In the Selection Sets Manager, highlight the selection set that contains elements you want to delete.
3. Click the **Select In Drawing** button in the Selection Sets Manager to highlight all of the selection set's elements in the drawing pane.
   - If there is only one selection set listed in the Selection Sets manager, you don't have to highlight it before clicking the Select In Drawing button.
4. Shift-click (hold down the Shift key and click the left mouse button) any selected elements that you do not want to delete.
5. Right-click and select **Delete**. The highlighted elements in the selection set are deleted from your model.

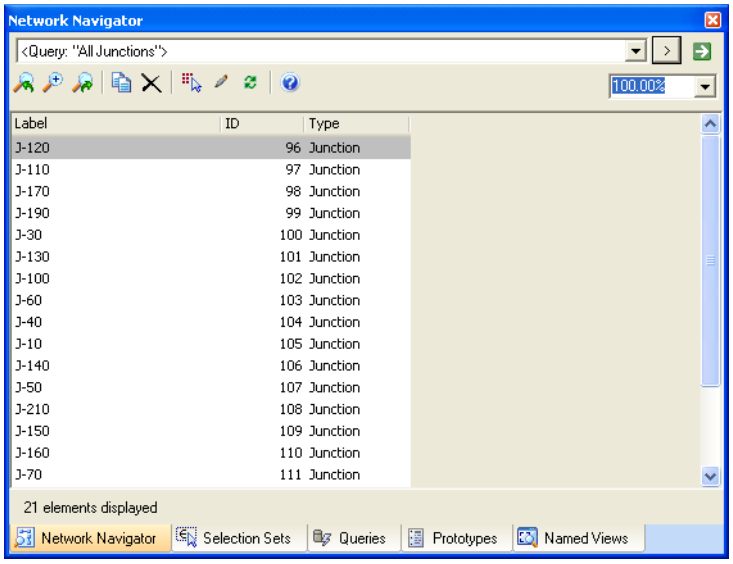
To create a report on a group of elements in a selection set

1. Open the Selection Sets Manager by selecting **View > Selection Sets** or clicking the **Selection Sets** button on the View toolbar.
2. In the Selection Sets Manager, highlight the selection set that contains elements you want to report on.
3. Click the **Select In Drawing** button in the Selection Sets Manager to highlight all of the selection set's elements in the drawing pane.
   - If there is only one selection set listed in the Selection Sets manager, you don't have to highlight it before clicking the Select In Drawing button.
4. Shift-click (hold down the **Shift** key and click the left mouse button) any selected elements that you do not want to include in the report.
5. Right-click and select **Report**. A report window displays the report.

Using the Network Navigator

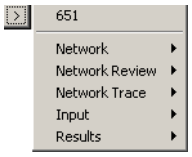
The Network Navigator consists of a toolbar and a table that lists the Label and ID of each of the elements contained within the current selection. The selection can include elements highlighted manually in the drawing pane, elements contained within a selection set, or elements returned by a query.

To open the Network Navigator, click the View menu and select the Network Navigator command, press <Ctrl+3>, or click the Network Navigator button  on the View toolbar.



The following controls are included in Network Navigator:

Query Selection List



Choose the element sets to use in the query. Once a query is selected, it can be executed when you click the > icon.



If there is already a Query listed in the list box, it can be run when the Execute icon is clicked.

Execute



Click to run the selected query.

Previous



Zooms the drawing pane view to the selected element at the magnification level specified in the Zoom Level menu.

Zoom To



Chooses the element below the currently selected one in the list.

Next



Specifies the magnification level at which elements are displayed in the drawing pane when the Zoom To command is initiated.

Copy



Copies the elements to the Windows clipboard.

Remove



Removes the selected element from the list.

Select In Drawing



Selects the listed elements in the drawing pane and performs a zoom extent based on the selection.

Highlight



When this toggle button is on, elements returned by a query will be highlighted in the drawing pane to increase their visibility.

Refresh Drawing



Refreshes the current selection.

Help



Opens WaterCAD V8*i* Help.

Predefined Queries

The Network Navigator provides access to a number of predefined queries grouped categorically, accessed by clicking the [>] button. Categories and the queries contained therein include:

Network

Network queries include “All Elements” queries for each element type, allowing you to display all elements of any type in the Network Navigator.

Network Review

Network Review Queries include the following:

- **Nodes In Close Proximity** - Identifies nodes within a specific tolerance.
- **Crossing Pipes** - Identifies pipes that intersect one another with no junction at the intersection.
- **Orphaned Nodes** - Identifies nodes that are not connected to a pipe in the model.
- **Orphaned Isolation Valves** - Identifies isolation valves that are not connected to a pipe in the model.
- **Dead End Nodes** - Identifies nodes that are only connected to one pipe.
- **Dead End Junctions** - Identifies junctions that are only connected to one pipe.
- **Pipe Split Candidates**- Identifies nodes near a pipe that may be intended to be nodes along the pipe. The tolerance value can be set for the maximum distance from the pipe where the node should be considered as a pipe split candidate.
- **Pipes Missing Nodes** - Identifies which pipes are missing either one or both end nodes.
- **Duplicate Pipes** - Identifies instances in the model where a pipe shares both end nodes with another pipe.

Network Trace

Network Trace Queries include the following:

- **Find Connected** - Locates all the connected elements to the selected element in the network.
- **Find Adjacent Nodes** - Locates all node elements connected upstream or downstream of the selected element or elements.
- **Find Adjacent Links** - Locates all link elements connected upstream or downstream of the selected element or elements.
- **Find Disconnected** - Locates all the disconnected elements in the network by reporting all the elements not connected to the selected element.
- **Find Shortest Path** - Select a Start Node and a Stop Node. The query reports the shortest path between the two nodes based upon the shortest number of edges.
- **Trace Upstream** - Locates all the elements connected upstream of the selected downstream element.
- **Trace Downstream** - Locates all the elements connected downstream of the selected upstream element.
- **Isolate** - Select an element that needs to be serviced. Run the query to locate the nearest isolation valves. In order to service the element, this will identify where shut off points and isolation valves are located.

- **Find Initially Isolated Elements** - Locates elements that are not connected or cannot be reached from any boundary condition.

Input

Input Queries include a number of queries that allow you to find elements that satisfy various conditions based on input data specified for them. Input queries include:

- **Duplicate Labels -** Locates duplicate labels according to parameters set by the user. See Using the Duplicate Labels Query for more information.
- **Elements With SCADA Data -** Locates elements that are have SCADA data associated with them.
- **Inactive Elements** - Locates elements that have been set to Inactive.
- **Pipes with Check Valves** - Locates pipes that have the Has Check Valve? input attribute set to True.
- **Controlled Elements** - Locates all elements that are referenced in a control Action.
- **Controlled Pumps** - Locates all pumps that are referenced in a control Action.
- **Controlled Valves** - Locates all valves that are referenced in a control Action.
- **Controlled Pipes** - Locates all pipes that are referenced in a control Action.
- **Controlling Elements** - Locates all elements that are referenced in a control Condition.
- **Initially Off Pumps** - Locates all pumps whose Status (Initial) input attribute is set to Off.
- **Initially Closed Control Valves** - Locates all control valves whose Status (Initial) input attribute is set to Closed.
- **Initially Inactive Control Valves** - Locates all control valves whose Status (Initial) input attribute is set to Inactive.
- **Initially Closed Pipes** - Locates all pipes whose Status (Initial) input attribute is set to Closed.
- **Fire Flow Nodes** - Locates nodes included in the group of elements specified in the Fire Flow Alternative's Fire Flow Nodes field.
- **Constituent Source Nodes -** Locates all nodes whose Is Constituent Source? input attribute is set to True.
- **Nodes with Non-Zero Initial Constituent Concentration -** Locates all nodes whose Concentration (Initial) input attribute value is something other than zero.
- **Tanks with Local Bulk Reaction Rate Coefficient -** Locates all tanks whose Specify Local Bulk Rate? input attribute is set to True.
- **Pipes with Local Reaction Rate Coefficients** - Locates all pipes whose Specify Local Bulk Reaction Rate? input attribute is set to True.

- **Pipes with Hyperlinks** - Locates all pipes that have one or more associated hyperlinks.
- **Nodes with Hyperlinks** - Locates all nodes that have one or more associated hyperlinks.

Results

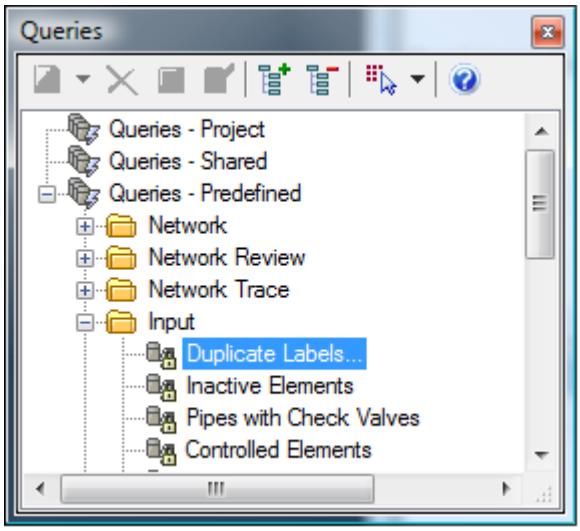
Results Queries include a number of queries that allow you to find elements that satisfy various conditions based on output results calculated for them. Results queries include:

- **Negative Pressures** - Locates all nodes that have negative calculated pressure results.
- **Pumps Operating Out of Range** - Locates all pumps whose Pump Exceeds Operating Range? result attribute displays True.
- **Pumps Cannot Deliver Flow or Head** - Locates all pumps whose Cannot Deliver Flow or Head? result attribute displays True.
- **Valves Cannot Deliver Flow or Head** - Locates all valves whose Cannot Deliver Flow or Head? result attribute displays True.
- **Empty Tanks** - Locates all tanks whose Status (Calculated) result attribute displays Empty.
- **Full Tanks** - Locates all tanks whose Status (Calculated) result attribute displays Full.
- **Off Pumps** - Locates all pumps whose Status (Calculated) result attribute displays Off.
- **Closed Control Valves** - Locates all control valves whose Status (Calculated) result attribute displays Closed.
- **Inactive Control Valves** - Locates all control valves whose Status (Calculated) result attribute displays Inactive.
- **Closed Pipes** - Locates all pipes whose Status (Calculated) result attribute displays Closed.
- **Failed Fire Flow Constraints** - Locates all elements whose Satisfies Fire Flow Constraints? result attribute displays False.
- **Self-Cleansing Pipes** - Locates all pipes that satisfy the user-defined criteria for self-cleansing pipes (Shear Stress, Velocity, or Shear Stress and Velocity).

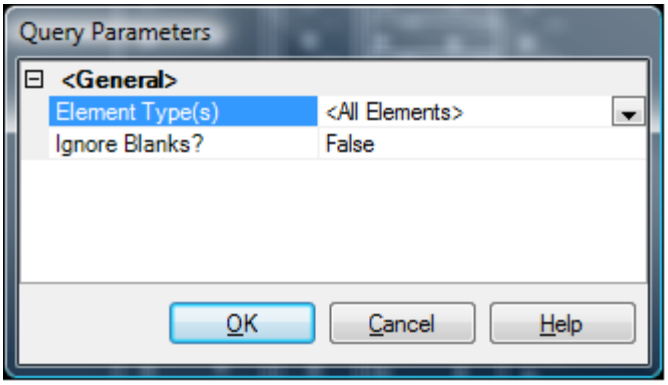
Using the Duplicate Labels Query

WaterCAD V8*i* internally keeps track of elements using a read-only ID property. In addition to this, users can and should identify elements using labels. The labels are purely for display and not used for data base management or hydraulic calculations. For the past several versions of the program, the models ran even if they contained duplicate or blank labels. On some occasions, however, duplicate labels could cause confusion (e.g. picking the wrong instance of an element in setting up a control). The Duplicate Labels query is a tool to find duplicate or blank labels.

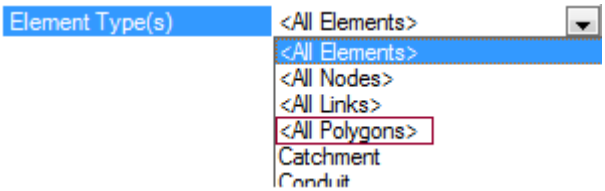
The Duplicate Labels query is accessed through View > Network Navigator > Queries - Predefined > Input > Duplicate Labels.



This opens the following dialog where the user can control the behavior of the query:

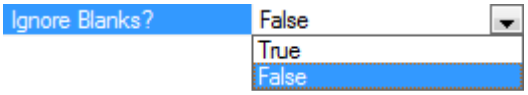


The element type parameter enables the user to search for duplicate queries across all elements or within a specific type of element.



Spot elevations are not included as a choice because duplicate spot elevations are not usually problematic.

The second choice in the dialog enables the user to control whether blank labels should be considered as duplicates.



The defaults for these parameters are to consider all elements and blank labels should be considered.

The query returns a list of elements with duplicate labels with their ID and Type. The user can highlight those elements in the drawing, zoom to individual elements and modify them as desired.

Using the Pressure Zone Manager

The Pressure Zone Manager is a tool for identifying elements that are located in a pressure zone based on the boundaries of the zone. It also provides the ability to conduct flow balance calculations for any pressure zone, color code by pressure zone and export information on elements in a zone to the Zone Manager.

It is important to distinguish between the Pressure Zone Manager and the Zone Manager. The pressure zone manager identifies which elements are included within a pressure zone. It is specific to the current scenario and is not a permanent property of the elements. A Zone is a property that can be assigned to any element. It can be based on any criteria you desire. Assignment of an element to a Zone based on what Pressure Zone it is in can be performed by identifying a representative element within a pressure zone and assigning that zone to every node element in the pressure zone. Zones are further described here: Zones)

The Pressure Zone Manager identifies elements in a pressure zone, by starting at one element and tracing through the network until it reaches a boundary element which can include closed pipes, closed isolation valves, pumps or any control valve. You can determine which types of elements can serve as pressure zone boundaries. Once all

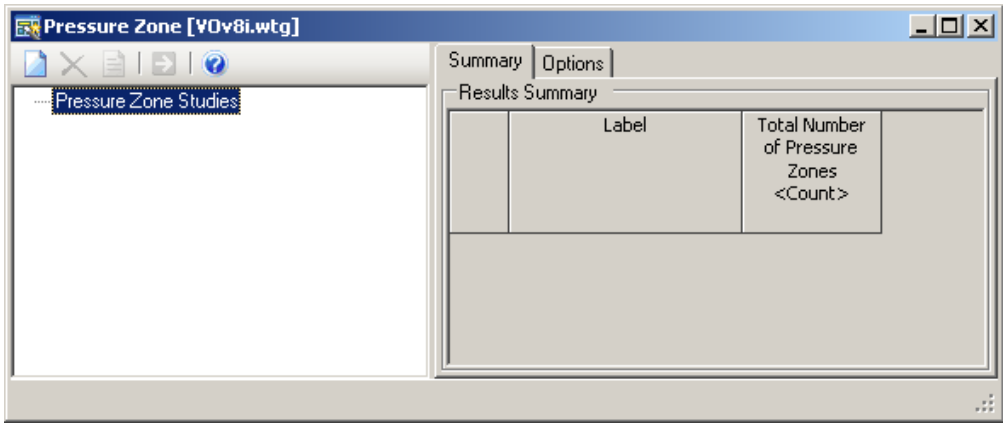
elements within a pressure zone have been identified, the pressure zone manager moves to an element outside of the pressure zone and searches for elements within that pressure zone. This continues until all elements have been assigned to a zone or are serving as zone boundaries.

You may find that the pressure zone manager has identified more pressure zones than are in the system. This is due to the fact that the manager assigns all elements to a pressure zone so that there are pressure zones for example, between the plant clearwell and the high service pumps or between the reservoir node representing the ground-water aquifer and the well pump. These "pressure zones" only contain a small number of elements.

Starting pressure zone manager

Start the pressure zone manager by selecting Analysis > Pressure Zone or clicking the Pressure Zone Manager button  .

When the pressure zone manager opens, you will see a left pane which lists the scenarios for which pressure zone studies have been set up. The first time, it will be blank. In the right pane, You see the Summary tab which lists the scenarios for which the pressure zone manager has been run and the number of pressure zones which were identified in the run.



To begin a pressure zone study, select New from the top of the left pane, and then pick which scenario will be used for the study. You can perform pressure zone studies for any scenario.

Specifying Boundary Elements

Once the scenario has been selected, you can define which elements are to be used as pressure zone boundary elements using the Options tab in the right pane. The user choose from the following settings:

1. Always use

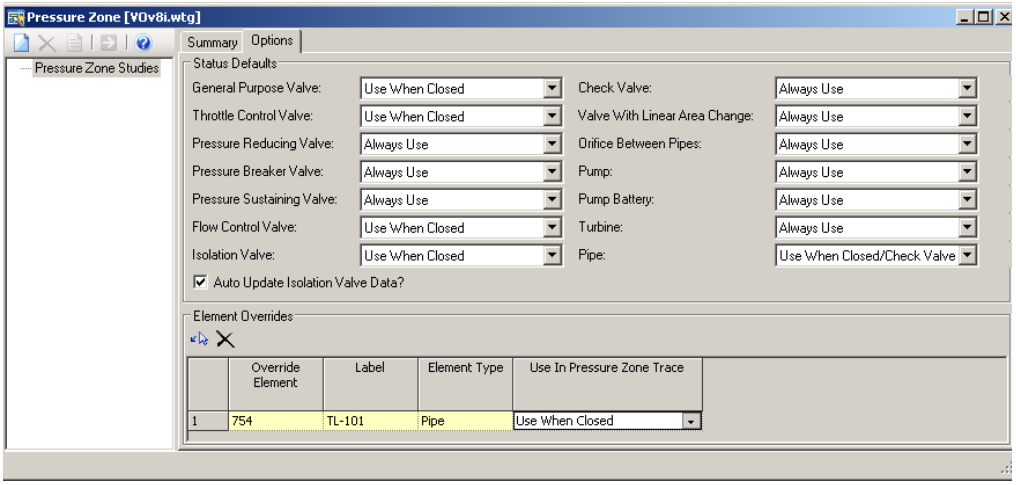
2. Use when closed

3. Do not use

4. (Pipes Only) Use when closed/Check valve

5. (Control Valves Only) Use When Active - When this is selected as the default status for a valve-type, elements of that valve-type will only be included as boundary nodes in the Pressure Zone tracing if their Status (Initial) field is set to "Active", and will be ignored otherwise.

6. (Control Valves Only) Use when Closed or Active - When this is selected as the default status for a valve-type, elements of that valve-type will only be included as boundary nodes in the Pressure Zone tracing if their Status (Initial) field is set to "Active" or "Closed", and will be ignored otherwise.

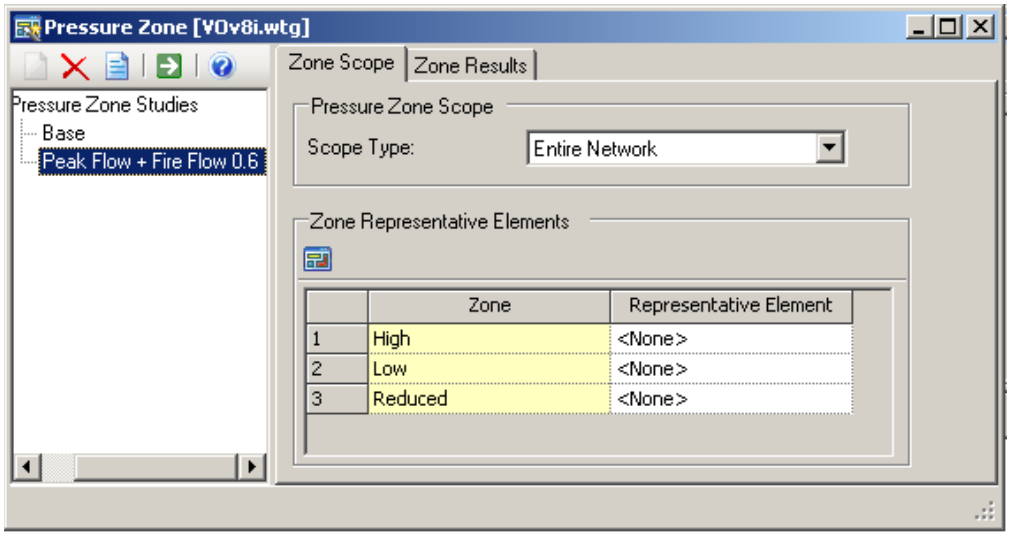


It is also possible to specify that an individual element behave differently from the default behaviors in the bottom right pane by clicking the Select from Drawing button at the top of the table and picking the element from the drawing.

Zone Scope

Once the settings have been established, select the scenario to be run in the left pane. Click the Zone Scope tab in the right pane.

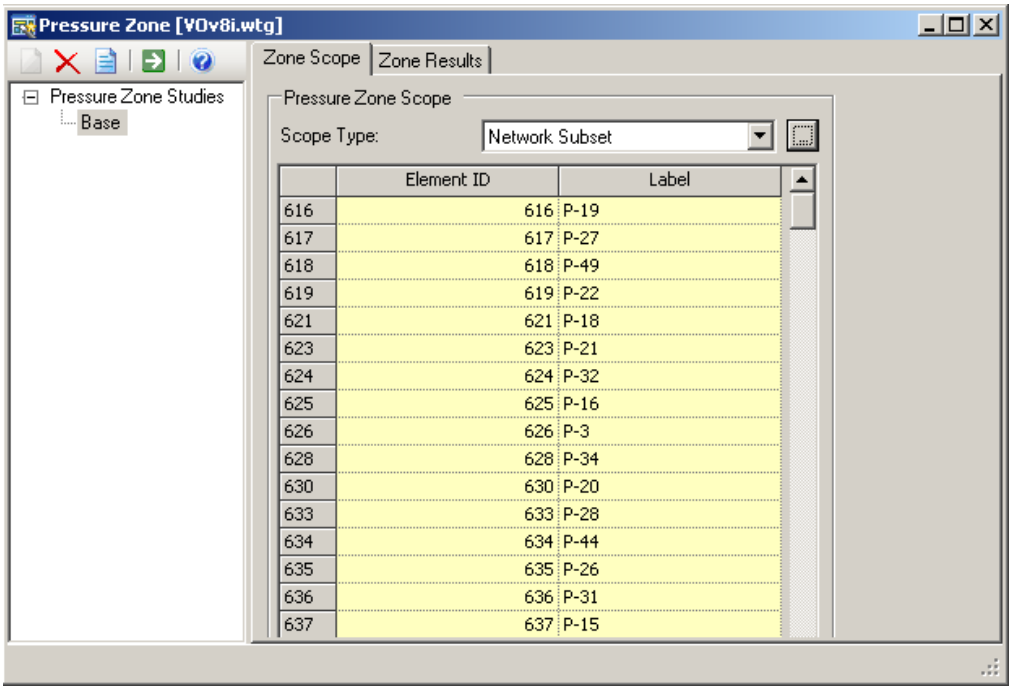
The first choice in the Zone Scope tab is whether to identify pressure zones for the entire network of a subset of the network. The default value is "Entire network".



If you want to run the pressure zone manager for a portion of the system, you should select Network Subset from the drop down menu and then click on the box to the right of the drop down arrow. This opens the drawing where you can make a selection using the standard selection tools as shown below. The fourth button enables you to select by drawing a polygon around the elements while the fifth button enables you to choose a previously created selection set. Remember to Right click "Done" when finished drawing the polygon.



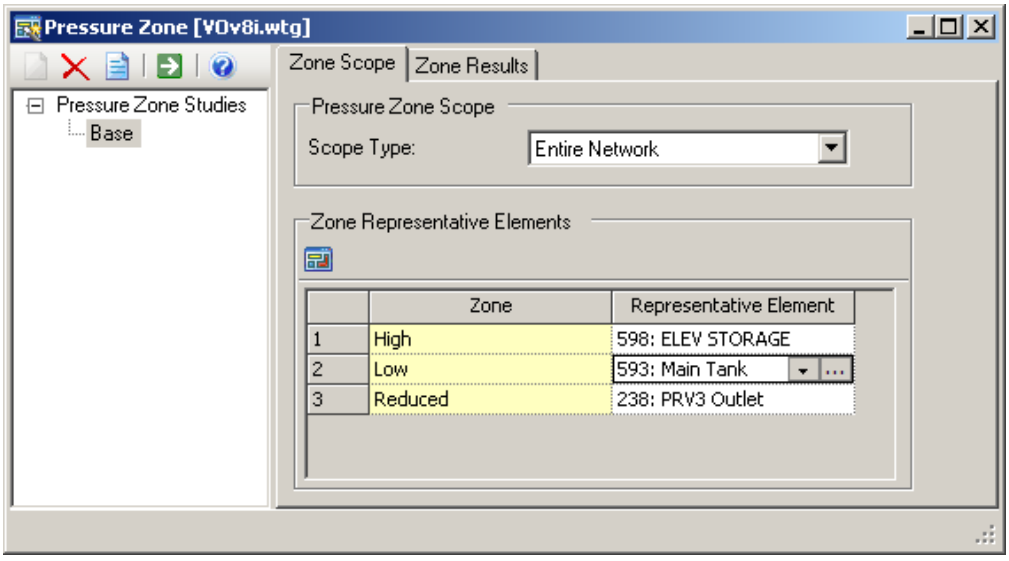
Upon picking the green check mark, the Zone Scope dialog opens again, displaying the elements selected.



Associating Pressure Zones with the "Zone" property

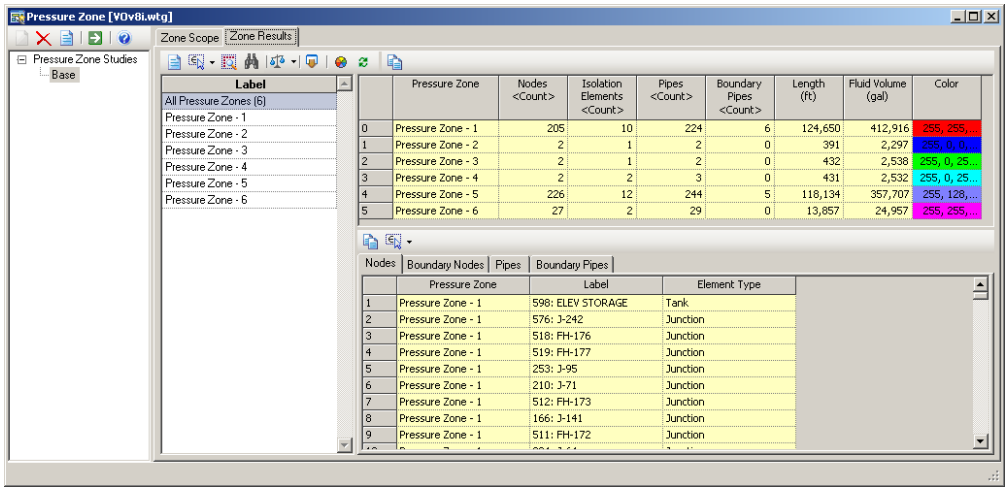
You can now run the pressure zone identification part of the pressure zone manager. However, if you want to associate pressure zones identified with Zones in the Zone Manager, the bottom of the right pane is the place to make that association. Each Zone is associated with a Representative Element - that is, an element that you are certain will be in the pressure zone associated with the Zone. For example, if Tank A is in the "Tank A Zone", then Tank A is a logical choice for the representative element. If a zone is to be named after the PRV feeding the zone, it is best to relabel the node on the downstream side of the PRV as something like "PRV Z Outlet" and choose that as the representative element. You can access the Zone Manager by selecting the button at the top of the lower right pane. All of the Zones in the Zone Manager are listed in the

column labeled Zone but you do not need to identify a representative element in each. It is best to set up Zones before starting the pressure zone manager. In that way, the drop down list under Representative Element on the Zone Scope tab (see below) will be populated.



Running Pressure Zone Manager

To identify pressure zones, select the Compute button (4th button on top of the left pane). The pressure zone manager runs and prepares statistics on each pressure zone as shown below.



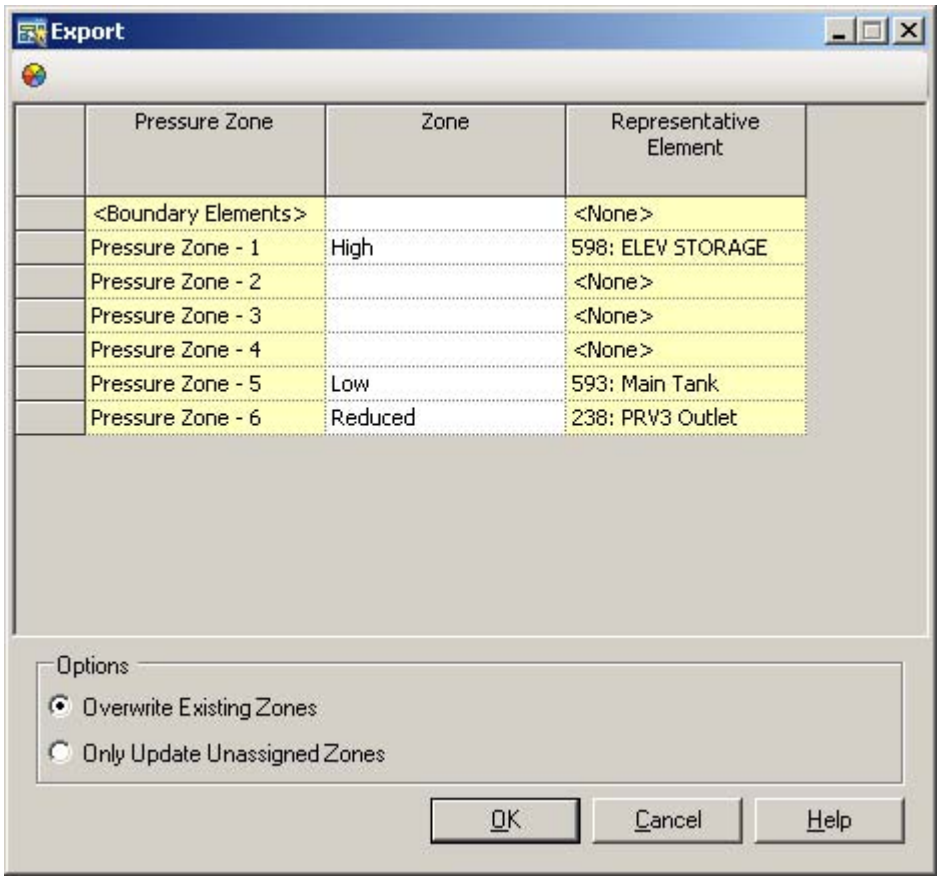
Overall Results

For each pressure zone, the number of nodes, the number of boundary (isolation) elements, the number of pipes, the length of pipe in the zone, the volume of water in the zone and the color associated with the zone in the drawing are displayed in the top right pane.

The lower portion of the right pane provides information on the individual elements in each pressure zone indicating the pipes and nodes in each zone and the pipes and nodes that serve as boundaries each in their own tab. You can also create selection sets corresponding to elements in each pressure zone by picking a pressure zone in the center pane (called Label), and then clicking the Create a Selection Set button on top of the lower right pane.

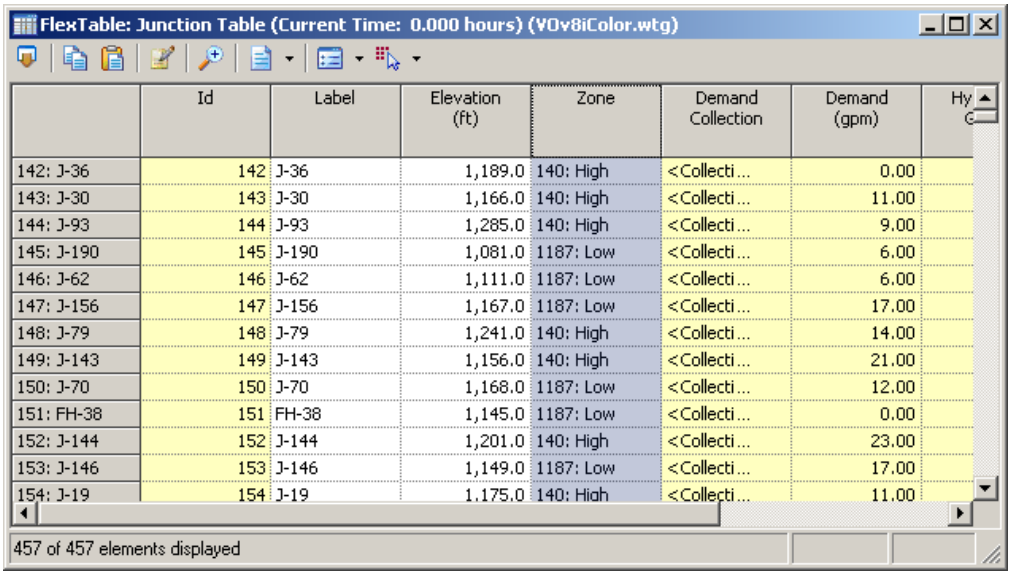
Exporting Pressure Zones to Zones

At this point, the pressure zones are labeled Pressure Zone - x, where x is a number indicating the order in which the pressure zone was identified. These pressure zones can be associated with the Zones using the fifth button, Export Pressure Zone. This opens up the Export dialog which lists the Zones that will be associated with the pressure zones based on representative elements.



The options at the bottom of the dialog control whether the Zone assignments that will be made will overwrite existing Zone assignments.

After selecting OK, each element in a pressure zone that has a representative element is assigned the Zone name associated with that representative element.



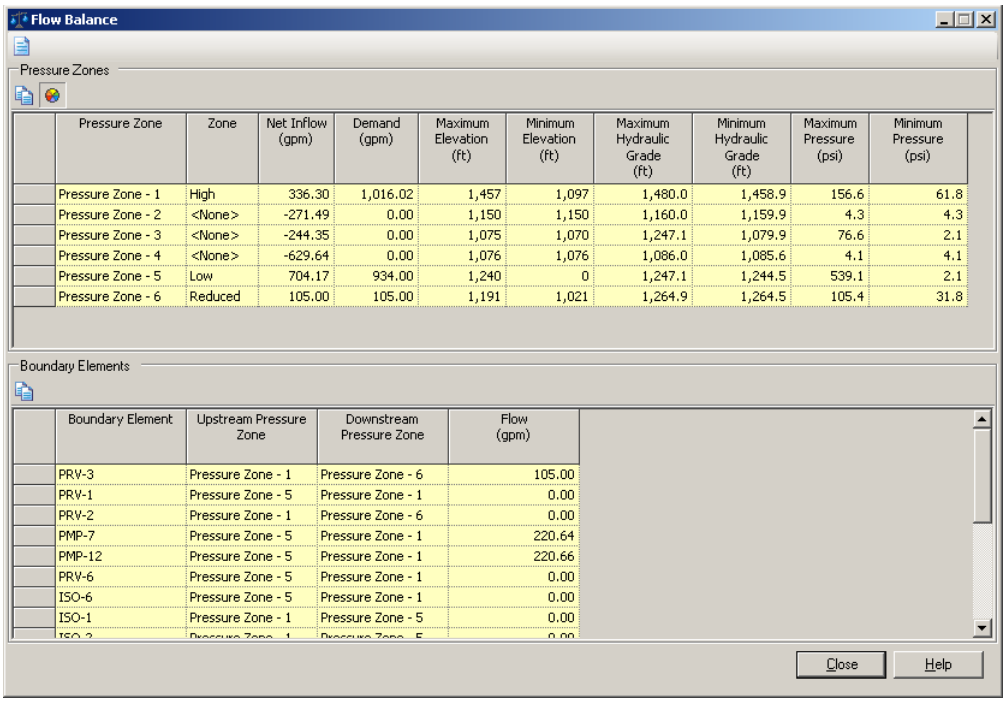
	Id	Label	Elevation (ft)	Zone	Demand Collection	Demand (gpm)	Hy
142: J-36	142	J-36	1,189.0	140: High	<Collecti...	0.00	
143: J-30	143	J-30	1,166.0	140: High	<Collecti...	11.00	
144: J-93	144	J-93	1,285.0	140: High	<Collecti...	9.00	
145: J-190	145	J-190	1,081.0	1187: Low	<Collecti...	6.00	
146: J-62	146	J-62	1,111.0	1187: Low	<Collecti...	6.00	
147: J-156	147	J-156	1,167.0	1187: Low	<Collecti...	17.00	
148: J-79	148	J-79	1,241.0	140: High	<Collecti...	14.00	
149: J-143	149	J-143	1,156.0	140: High	<Collecti...	21.00	
150: J-70	150	J-70	1,168.0	1187: Low	<Collecti...	12.00	
151: FH-38	151	FH-38	1,145.0	1187: Low	<Collecti...	0.00	
152: J-144	152	J-144	1,201.0	140: High	<Collecti...	23.00	
153: J-146	153	J-146	1,149.0	1187: Low	<Collecti...	17.00	
154: J-19	154	J-19	1,175.0	140: High	<Collecti...	11.00	

For more information, see Pressure Zone Export Dialog Box

Pressure Zone Flow Balance

The fourth button performs a flow balance on each pressure zone. For each Pressure Zone, it displays the Zone (if one is associated with the pressure zone), net inflow (flow across the boundaries but not including flow originating from tanks and reservoirs in the pressure zone), the demand in that zone, the minimum and maximum elevations in the pressure zone, the minimum and maximum hydraulic grade lines in the pressure zone, and the minimum and maximum pressure in the pressure zone. If

the scenario is not steady state, then the results correspond to the current time step. The lower pane displays the flow through each boundary element. If the hydraulics have not been calculated for this system, a message is given that the model needs to be calculated.



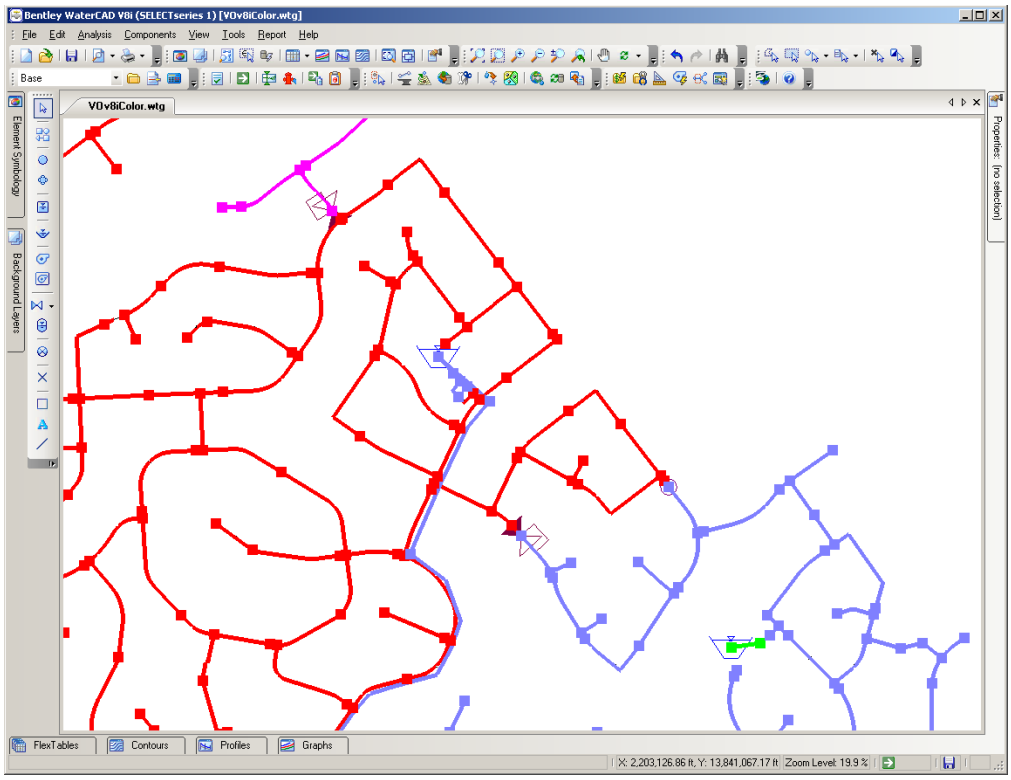
Pressure Zone	Zone	Net Inflow (gpm)	Demand (gpm)	Maximum Elevation (ft)	Minimum Elevation (ft)	Maximum Hydraulic Grade (ft)	Minimum Hydraulic Grade (ft)	Maximum Pressure (psi)	Minimum Pressure (psi)
Pressure Zone - 1	High	336.30	1,016.02	1,457	1,097	1,480.0	1,458.9	156.6	61.8
Pressure Zone - 2	<None>	-271.49	0.00	1,150	1,150	1,160.0	1,159.9	4.3	4.3
Pressure Zone - 3	<None>	-244.35	0.00	1,075	1,070	1,247.1	1,079.9	76.6	2.1
Pressure Zone - 4	<None>	-629.64	0.00	1,076	1,076	1,086.0	1,085.6	4.1	4.1
Pressure Zone - 5	Low	704.17	934.00	1,240	0	1,247.1	1,244.5	539.1	2.1
Pressure Zone - 6	Reduced	105.00	105.00	1,191	1,021	1,264.9	1,264.5	105.4	31.8

Boundary Element	Upstream Pressure Zone	Downstream Pressure Zone	Flow (gpm)
PRV-3	Pressure Zone - 1	Pressure Zone - 6	105.00
PRV-1	Pressure Zone - 5	Pressure Zone - 1	0.00
PRV-2	Pressure Zone - 1	Pressure Zone - 6	0.00
PMP-7	Pressure Zone - 5	Pressure Zone - 1	220.64
PMP-12	Pressure Zone - 5	Pressure Zone - 1	220.66
PRV-6	Pressure Zone - 5	Pressure Zone - 1	0.00
ISO-6	Pressure Zone - 5	Pressure Zone - 1	0.00
ISO-1	Pressure Zone - 1	Pressure Zone - 5	0.00

For more information, see Pressure Zone Flow Balance Tool Dialog Box.

Color Coding by Pressure Zone

The sixth button color codes the drawing by pressure zone. Each zone is colored according to the color displayed in the rightmost column of the table. In the image below, the main zone is blue, the red zone is boosted through a pump, the magenta zone is a reduced zone fed through a PRV and the green zone is a well.



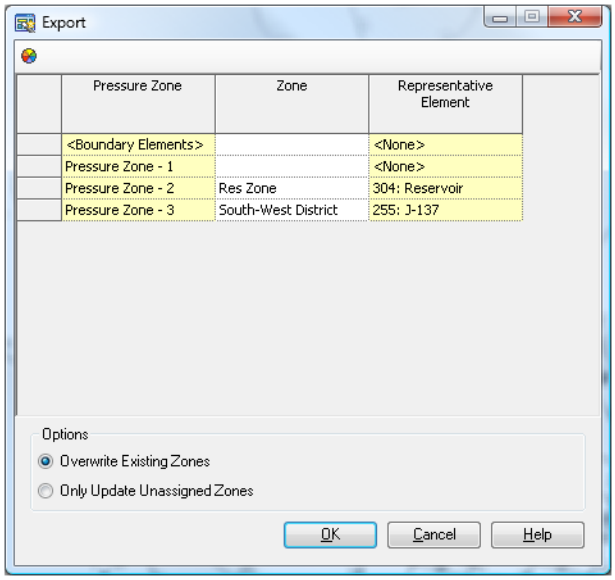
Other Pressure Zone Results

Other buttons such as Report, Refresh, Export to Selection Set, Zoom to and Copy behave as they do for other WaterCAD V8*i* features.

The results of a pressure zone analysis as stored in a .pzs file.

Pressure Zone Export Dialog Box

This dialog allows you to associate pressure zones with zones using representative elements.



The table of export data contains a row for each pressure zone, as well as a row for the boundary elements. The first column specifies the pressure zone. The second column specifies the zone, specified by you, to assign the elements of the pressure zone to. This comun consists of pull-down menus containing all of the model's zones. Additionally, there is an ellipsis (...) button that will bring up the Zone Manager if you need to add/remove/modify the model's zones (see Zones for more information). The third column is informational. It lists the representative element for the selected zone, which is specified in the Pressure Zone Manager (see Using the Pressure Zone Manager).

The special <Boundary Elements> pressure zone contains all of the boundary elements for every pressure zone. The other pressure zones each contain all of the elements in that pressure zone, excluding the boundary elements that seal off that pressure zone.

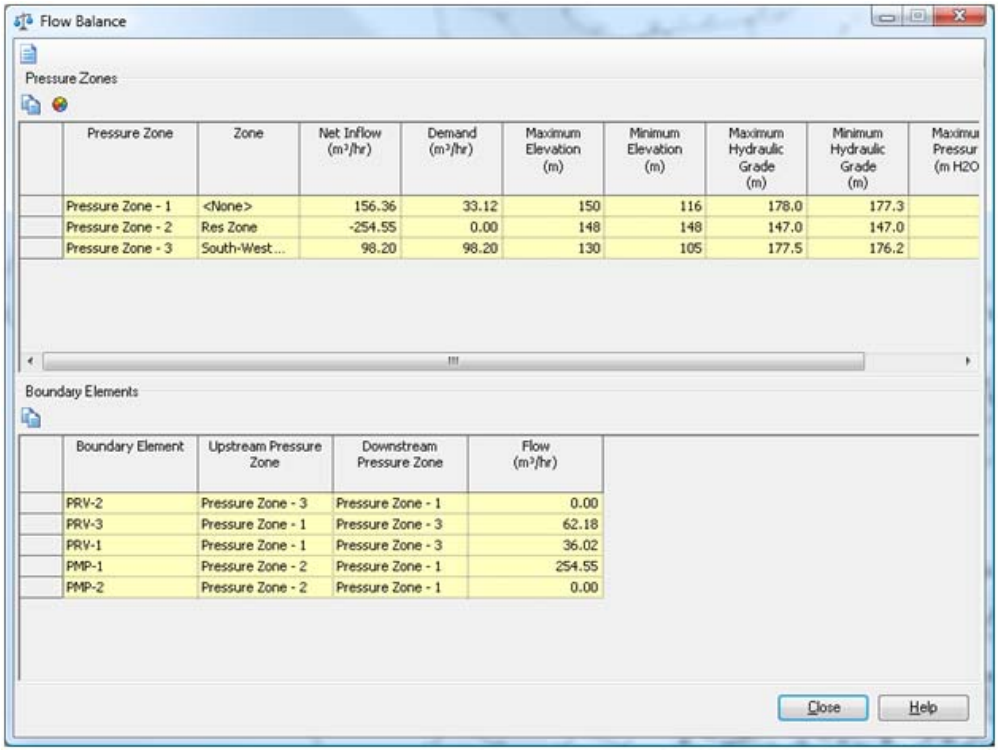
If you do not assign a zone to each pressure zone in the table before clicking the OK button, a warning will appear prompting you to do so.

The two Options radio buttons are mutually exclusive. "Overwrite Existing Zones" specifies that all elements in the pressure zones will be assigned to the corresponding zone chosen in the table. "Only Update Unassigned Zones" specifies that only those elements in the pressure zone that are not currently assigned to any zone will be assigned to the corresponding zone in the table. The exception is the <Boundary Elements> pressure zone, which will always be exported as if the "Overwrite Existing Zones" option is selected.

The "Highlight Pressure Zone In Drawing" toolbar button causes the elements of the pressure zone in the current row of the table to be highlighted in the drawing. This option gives allows you to see what elements are going to be affected by the export operation.

Pressure Zone Flow Balance Tool Dialog Box

The Flow Balance Tool dialog box allows you to perform a flow balance and/or a volume balance on each pressure zone.



For each Pressure Zone, it displays the Zone (if one is associated with the pressure zone), net inflow (flow across the boundaries but not including flow originating from tanks and reservoirs in the pressure zone) or net volume, the demand in that zone, the minimum and maximum elevations in the pressure zone, the minimum and maximum hydraulic grade lines in the pressure zone, and the minimum and maximum pressure in the pressure zone.

The Report button allows you to generate a preformatted report containg all of the data displayed in the tabels.

The Copy buttons (above the Pressure Zones and Boundary Elements tables) will copy the contents of the table to the clipboard in a format that is compatible with spreadsheet programs like Excel.

The Highlight Pressure Zone In Drawing button will toggle on/off highlighting of the the pressure zone for the currently active row in the Pressure Zone table.

For Volume balance, the sum of the flows over the run is found using the following formula:

$$V = \sum_{i=1}^{N} Q_i \, \Delta t_i$$

Where:

N = number of time steps

Q_i = flow in i-th time step (cfs)

Δt_i= time step duration for i-th time step

The value of Q_i is the net flow into the pressure zone at the start of the i-th time step. Δt_i is the difference in time between the start and end of that time step (because of pump cycling, the time step size changes).

Using Prototypes

Prototypes allow you to enter default values for elements in your network. These values are used while laying out the network. Prototypes can reduce data entry requirements dramatically if a group of network elements share common data.

For example, if a section of the network contains all 12-inch pipes, use the Prototype manager to set the Pipe Diameter field to 12 inches. When you create a new pipe in your model, its diameter attribute will default to 12 inches.

You can create prototypes in either of the following ways:

- From the Prototypes manager: The Prototypes manager consists of a toolbar and a list pane, which displays all of the elements available in WaterCAD V8*i*.
- From the Drawing Pane: Right-click an element to use the settings and attributes of that element as the current prototype.

Note: **Changes to the prototypes are not retroactive and will not affect any elements created prior to the change.**

If a section of your system has distinctly different characteristics than the rest of the system, adjust your prototypes before laying out that section. This will save time when you edit the properties later.

To open the Prototypes manager

Choose View > Prototypes

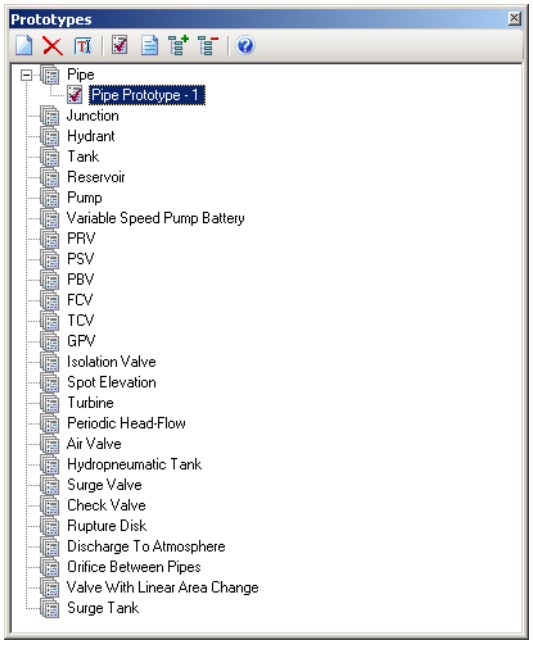
or

Press <Ctrl+6>

or

Click the Prototypes icon  from the View toolbar.

The Prototypes manager opens.



The list of elements in the Prototypes manager list pane is expandable and collapsible, once you've created additional prototypes. Click on the Plus sign to expand an element and see its associated prototypes. Click on the Minus sign to collapse the element.

Each element in the list pane contains a default prototype; you cannot edit this default prototype. The default prototypes contain common values for each element type; if you add elements to your model without creating new prototypes, the data values in the default prototypes appear in the Property Editor for that element type.

The toolbar contains the following icons:

New 	Creates a new prototype of the selected element.
Delete 	Deletes the prototype that is currently selected in the list pane.
Rename 	Renames the prototype that is currently selected in the list pane.
Make Current 	Makes the prototype that is currently highlighted in the list pane the default for that element type. When you make the current prototype the default, every new element of that type that you add to your model in the current project will contain the same common data as the prototype.

Report



Opens a report of the data associated with the prototype that is currently highlighted in the list pane.

Expand All



Opens all the Prototypes.

Collapse All



Closes all the Prototypes.

Help

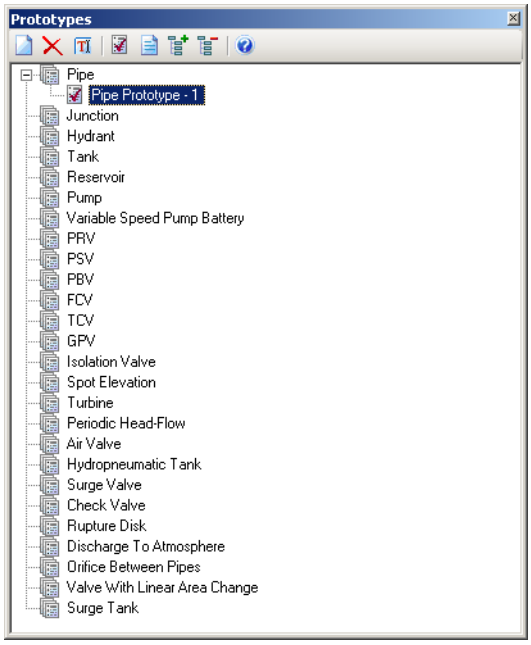


Displays online help for the Prototypes Manager.

To create Prototypes in the Prototypes Manager

1. Open your WaterCAD V8*i* project or start a new project.
2. Choose View > Prototypes or press <Ctrl+6>.

   The Prototypes Manager opens.



3. Select the element type for which you want to create a prototype, then click New.

The list expands to display all the prototypes that exist for that element type.

Each element type contains a default prototype, which is not editable, and any prototypes that you have created. The current set of default values for each element type is identified by the Make Current icon.

4. Double-click the prototype you just created. The Property Editor for the element type opens.
5. Edit the attribute values in the Property Editor as required.
6. To make the new prototype the default, click the **Make Current** button in the Prototypes Manager.

   The icon next to the prototype changes to indicate that the values in the prototype will be applied to all new elements of that type that you add to your current project.
7. Perform the following optional steps:
   - To rename a prototype, select the prototype in the list and click the **Rename** button.
   - To delete a prototype, select the prototype in the list and click the **Delete** button.
   - To view a report of the default values in the prototype, select the prototype in the list and click the **Report** button.

To create a Prototype from the Drawing View

1. Right-click the element you want to act as the current proptotype for newly created elements of that type.
2. Select **Create Prototype** from the context menu.
3. Enter a name for the new prototype in the **Create New Prototype** dialog that appears.
4. Click **OK**.

Zones

The Zones manager allows you to manipulate zones quickly and easily. Zones listed in the Zones manager can be associated with each nodal element using the Element Editors, Prototypes, or FlexTables. This manager includes a list of all of the available zones and a toolbar.

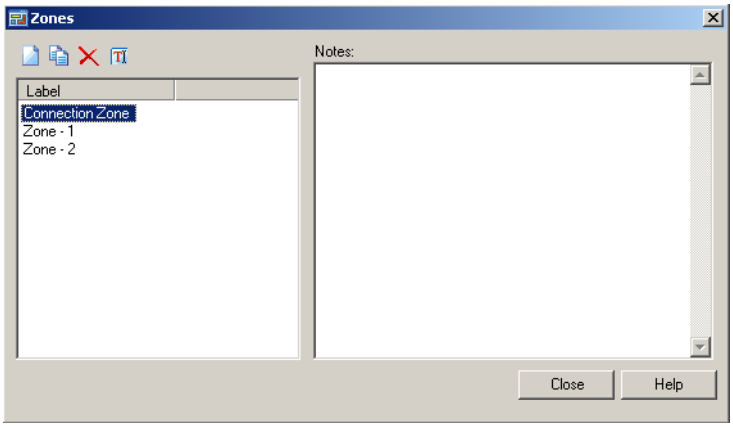
To open the Zones manager

Choose Components > Zones

or

Click the Zones icon  from the Components toolbar.

The Zones manager opens.



The toolbar contains the following icons:

New—Adds a new zone to the zone list.

Duplicate—Creates a copy of an existing zone.

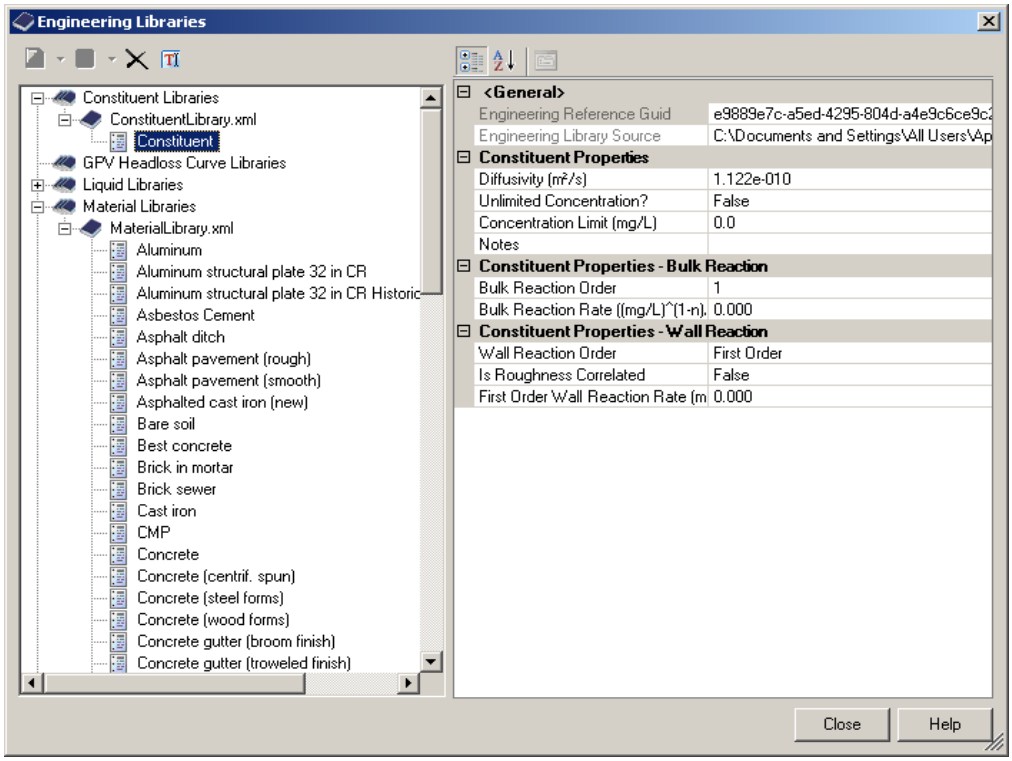
Delete—Deletes an existing zone. You can hold down the Ctrl key while clicking on items in the list to select multiple entries at once.

Rename - Renames the selected zone.

Notes - Enter information about the zone.

Engineering Libraries

Engineering Libraries are powerful and flexible tools that you use to manage specifications of common materials, objects, or components that are shared across projects. Some examples of objects that are specified through engineering libraries include constituents, pipe materials, patterns, and pump definitions.



You can modify engineering libraries and the items they contain by using the Engineering Libraries command in the Components menu.

You work with engineering libraries and the items they contain in the Engineering Libraries dialog box, which contains all of the project's engineering libraries. Individual libraries are compilations of library entries along with their attributes.

By default, each project you create in WaterCAD V8*i* uses the items in the default libraries. In special circumstances, you may wish to create custom libraries to use with one or more projects. You can do this by copying a standard library or creating a new library.

When you change the properties for an item in an engineering library, those changes affect all projects that use that library item. At the time a project is loaded, all of its engineering library items are synchronized to the current library. Items are synchronized based on their label. If the label is the same, then the item's values will be made the same.

The default libraries that are installed with Bentley WaterCAD V8*i* are editable. In addition, you can create a new library of any type and can then create new entries of your own definition.

- Library types are displayed in the Engineering Library manager in an expanding/collapsing tree view.
- Library types can contain categories and subcategories, represented as folders in the tree view.
- Individual library entries are contained within the categories, subcategories, and folders in the tree view.
- Libraries, categories, folders, and library entries are displayed in the tree view with their own unique icons. You can right-click these icons to display submenus with different commands.

Note: The data for each engineering library is stored in an XML file in your Bentley WaterCAD V8*i* program directory. We strongly recommend that you edit these files only using the built-in tools available by selecting Tools > Engineering Libraries.

Working with Engineering Libraries

When you select a library entry in the tree view, the attributes and attribute values associated with the entry are displayed in the editor pane on the right side of the dialog box.

Right-clicking a Library icon in the tree view opens a shortcut menu containing the following commands:

Create Library	Creates a new engineering library of the currently highlighted type.
Add Existing Library	Adds an existing engineering library that has been stored on your hard drive as an .xml file to the current project.
ProjectWise Add Existing Library	Adds an existing engineering library that is being managed by ProjectWise.

Working with Categories

Right-clicking a Category icon in the tree view opens a shortcut menu containing the following commands:

Add Item	Creates a new entry within the current library.
Add Folder	Creates a new folder under the currently highlighted library.
Save As	Saves the currently highlighted category as an .xml file that can then be used in future projects.
ProjectWise Save As	Saves the currently highlighted category to ProjectWise.
Remove	Deletes the currently highlighted category from the library.

Working with Folders

Right-clicking a Folder icon in the tree view opens a shortcut menu containing the following commands:

Add Item	Creates a new entry within the current folder.
Add Folder	Creates a new folder under the currently highlighted folder.
Rename	Renames the currently highlighted folder.
Delete	Deletes the currently highlighted folder and its contents.

Working with Library Entries

Right-clicking a Library Entry icon in the tree view opens a shortcut menu containing the following commands:

Rename	Renames the currently highlighted entry.
Delete	Deletes the currently highlighted entry from the library.

Engineering Libraries Dialog Box

The Engineering Libraries dialog box contains an explorer tree-view pane on the left, a library entry editor pane on the right, and the following icons above the explorer tree view pane:

New



Opens a submenu containing the following commands:

- **Create Library**—Creates a new engineering library.
- **Add Existing Library**—Adds an existing engineering library that has been stored on your hard drive as an .xml file to the current project.
- **ProjectWise Add Existing Library**—Adds an existing engineering library that is being managed by ProjectWise.

Delete



Removes the currently highlighted engineering library from the current project.

Rename



Renames the currently highlighted engineering library.

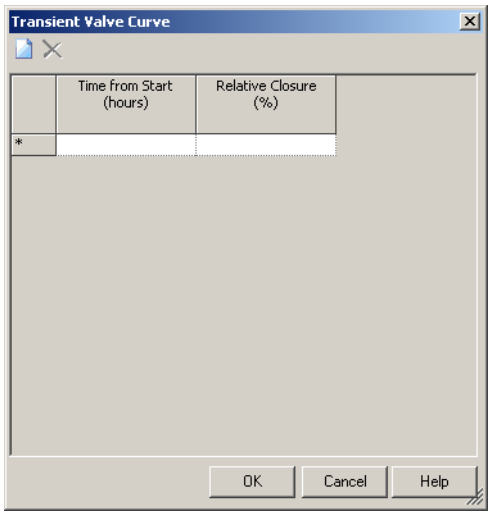
Sharing Engineering Libraries On a Network

You can share engineering libraries with other WaterCAD V8*i* users in your organization by storing the engineering libraries on a network drive. All users who will have access to the shared engineering library should have read-write access to the network folder in which the library is located.

To share an engineering library on a network, open the Engineering Libraries in WaterCAD V8*i* and create a new library in a network folder to which all users have read-write access.

Transient Valve Curve Editor

This dialog allows you to define pattern curves for the Air Flow Curve Engineering Library.



The following buttons are located above the curve points table on the left:

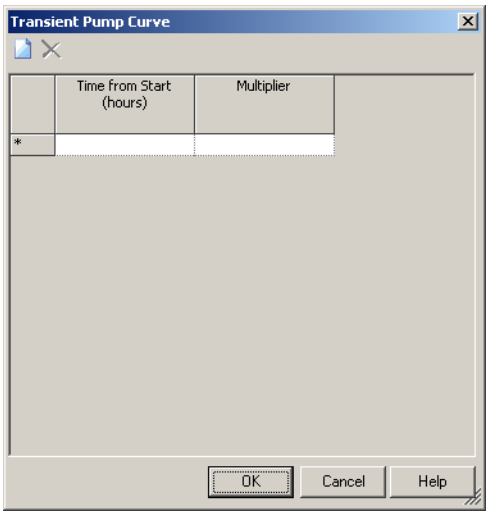
-  **New**—Creates a new row in the curve points table.
-  **Delete**—Deletes the currently highlighted row from the curve points table.

The curve points table contains the following columns:

- **Time From Start**—Lets you specify the amount of time from the Start Time of the pattern to the time step point being defined.
- **Relative Closure**—The percentage closed the valve is at the associated time.

Transient Pump Curve Editor

This dialog allows you to define pattern curves for the Air Flow Curve Engineering Library.



The following buttons are located above the curve points table on the left:

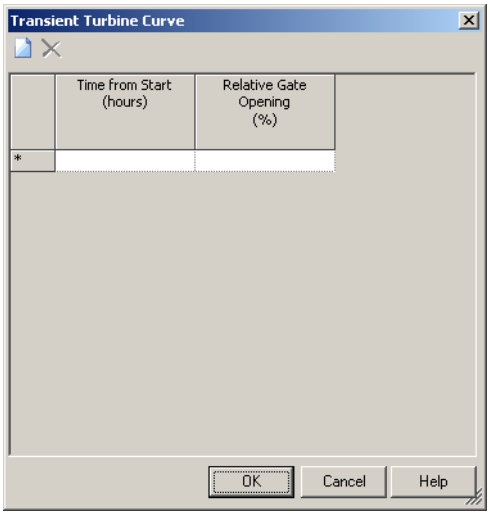
-  **New**—Creates a new row in the curve points table.
-  **Delete**—Deletes the currently highlighted row from the curve points table.

The curve points table contains the following columns:

- **Time From Start**—Lets you specify the amount of time from the Start Time of the pattern to the time step point being defined.
- **Multiplier**—Lets you specify the multiplier value associated with the time step point.

Transient Turbine Curve Editor

This dialog allows you to define pattern curves for the Air Flow Curve Engineering Library.



The following buttons are located above the curve points table on the left:

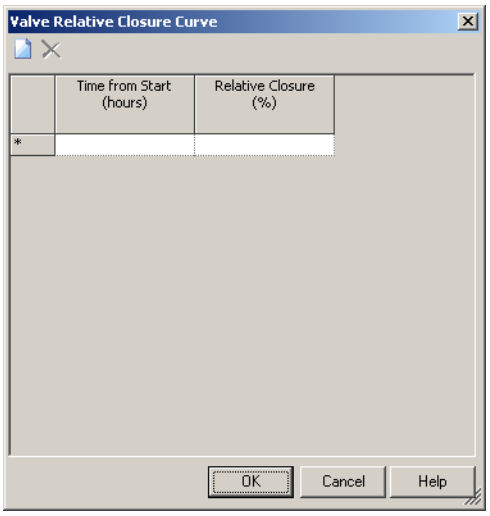
- **New**—Creates a new row in the curve points table.
- **Delete**—Deletes the currently highlighted row from the curve points table.

The curve points table contains the following columns:

- **Flow (Free Air)**—The volume of air flow at the associated pressure.
- **Relative Gate Opening**—The percentage compared to fully open for the turbine gate opening at the associated time step point.

Valve Relative Closure Curve Editor

This dialog allows you to define pattern curves for the Air Flow Curve Engineering Library.



The following buttons are located above the curve points table on the left:

-  **New**—Creates a new row in the curve points table.
-  **Delete**—Deletes the currently highlighted row from the curve points table.

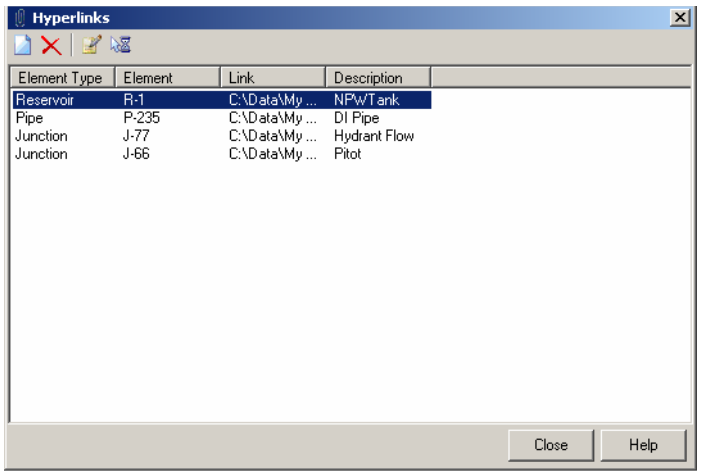
The curve points table contains the following columns:

- **Time From Start**—Lets you specify the amount of time from the Start Time of the pattern to the time step point being defined.
- **Relative Closure**—The percentage closed the valve is at the associated time.

Hyperlinks

The Hyperlinks feature is used to associate external files, such as pictures or movie files, with elements. You can Add, Edit, Delete, and Launch hyperlinks from the Hyperlinks manager.

To use hyperlinks, choose Tools > Hyperlinks. The Hyperlinks dialog box opens. The dialog box contains a toolbar and a tabular view of all your hyperlinks.



The toolbar contains the following icons:

New 
Creates a new hyperlink. Opens the Add Hyperlink dialog box.

Delete 
Deletes the currently selected hyperlink.

Edit 
Edits the currently selected hyperlink. Opens the Edit Hyperlink dialog box.

Launch 
Launches the external file associated with the currently selected hyperlink.

The table contains the following columns:

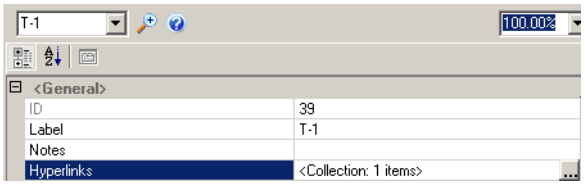
Element Type
Displays the element type of the element associated with the hyperlink.

Element
Displays the label of the element associated with the hyperlink.

Link
Displays the complete path of the hyperlink.

Description	Displays a description of the hyperlink, which you can optionally enter when you create or edit the hyperlink.

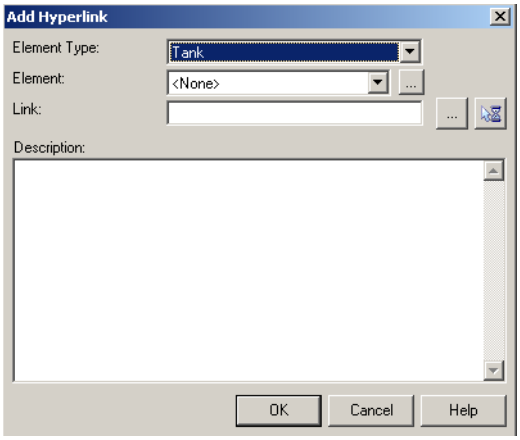
Once you have created Hyperlinks, you can open the Hyperlinks dialog box from within a Property dialog box associated with that Hyperlink.



Click the ellipsis (...) in the Hyperlinks field and the Hyperlinks dialog box opens.

Add Hyperlink Dialog Box

New hyperlinks are created in this dialog box.



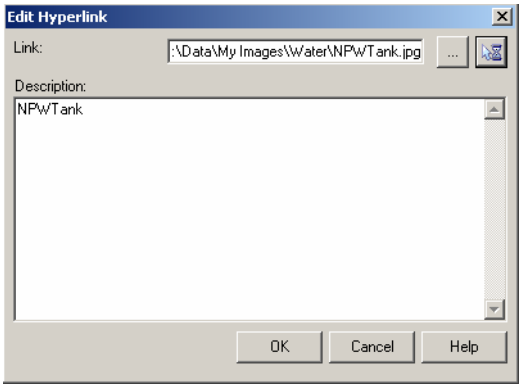
The Add Hyperlinks dialog box has the following controls:

Element Type	Select an element type from the drop-down list.
Element	Select an element from the drop-down list of specific elements from the model. Or click the ellipsis to select an element from the drawing.

Link	Click the ellipsis (...) to browse your computer and locate the file to be associated with the hyperlink. You can also enter the path of the external file by typing it in the Link field.
Description	Create a description of the hyperlink.

Edit Hyperlink Dialog Box

You edit existing hyperlinks in the Edit Hyperlink dialog box.

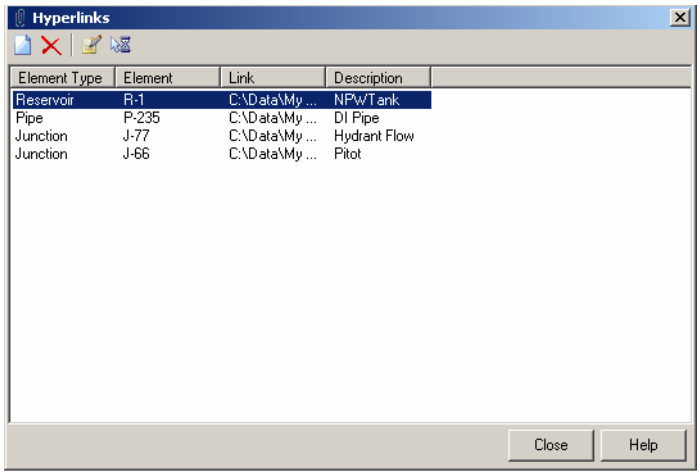


The Edit Hyperlinks dialog box contains the following controls:

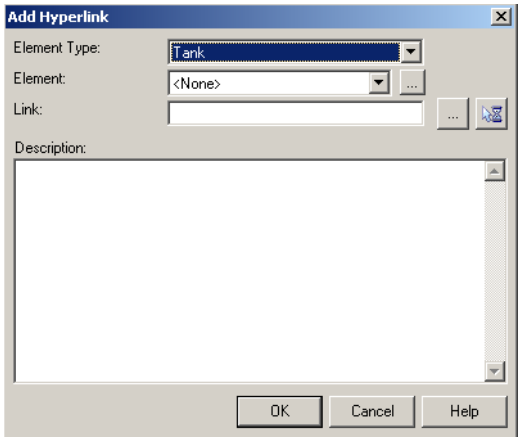
Link	Defines the complete path of the external file associated with the selected hyperlink. You can type the path yourself or click the ellipsis (...) to search your computer for the file. Once you have selected the file, you can test the hyperlink by clicking Launch 
Description	Accesses an existing description of the hyperlink or type a new description.

To Add a Hyperlink

1. Choose Tools > Hyperlink. The Hyperlinks dialog box opens.

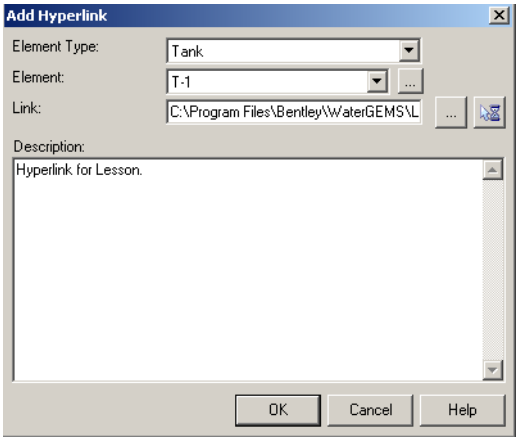


2. Click New to add a hyperlink. The Add Hyperlink dialog box opens.



3. Select the element type to associate an external file.
4. Click the ellipsis (...) to select the element in the drawing to associate with the hyperlink.
5. Click the ellipsis (...) to browse to the external file you want to use, select it and then click Open. This will add it to the Link field.

6. Add a description of your Hyperlink.

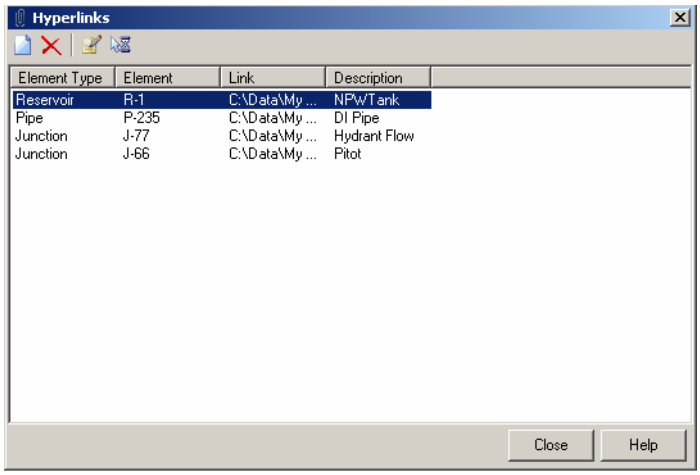


7. Click OK.

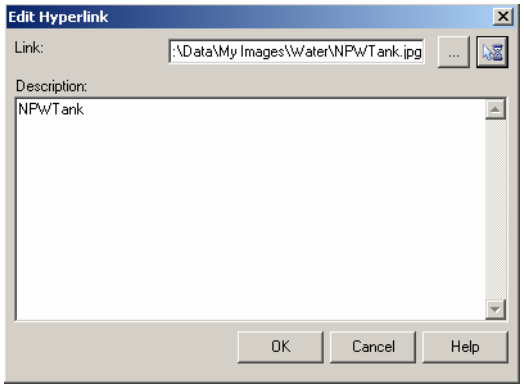
   You can add more than one associated file to an element using the hyperlink feature, but you must add the associations one at a time.

To Edit a Hyperlink

1. Choose Tools > Hyperlinks. The Hyperlinks dialog box opens.



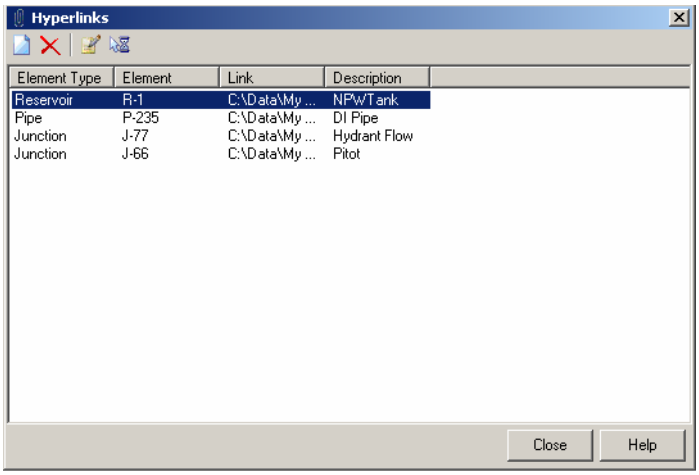
2. Select the element to edit and click Edit. The Edit Hyperlink dialog box opens.



3. Click the ellipsis (...) to browse to a new file to associate with the hyperlink.
4. Add a description.
5. Click OK

To Delete a Hyperlink

1. Choose Tools > Hyperlinks. The Hyperlinks dialog box opens.

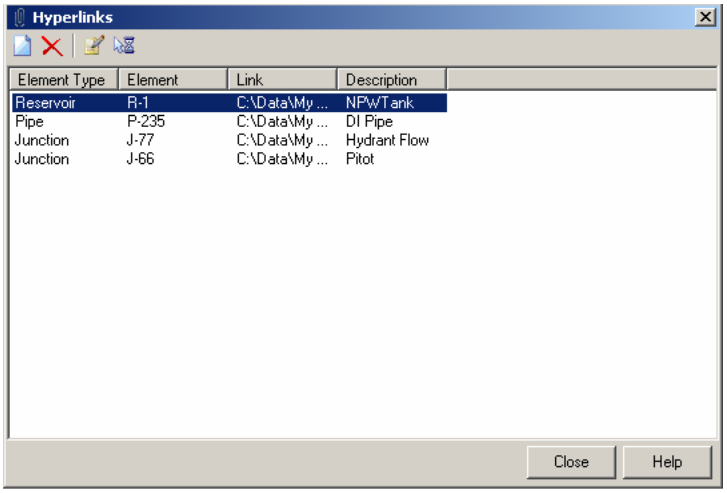


2. Select the element you want to delete.
3. Click Delete.

To Launch a Hyperlink

Hyperlinks can be launched from the Hyperlinks dialog box, the Add Hyperlink dialog box, and from the Edit Hyperlink dialog box. Launch in order to view the image or file associated with the element, or to run the program associated with the element.

1. Choose Tools > Hyperlinks. The Hyperlinks dialog box opens.



2. Select the element and click on the Hyperlinks icon. The hyperlink will launch.

Note: Click to open the Add or Edit dialog boxes and click Launch to open from there.

Using Queries

A query in Bentley WaterCAD V8*i* is a user-defined SQL expression that applies to a single element type. You use the Query Manager to create and store queries; you use the Query Builder dialog box to construct the actual SQL expression.

Queries can be one of the following three types:

- **Project queries**—Queries you define that are available only in the Bentley WaterCAD V8*i* project in which you define them.
- **Shared queries**—Queries you define that are available in all Bentley WaterCAD V8*i* projects you create. You can edit shared queries.
- **Predefined queries**—Factory-defined queries included with Bentley WaterCAD V8*i* that are available in all projects you create. You cannot edit predefined queries.

You can also use queries in the following ways:

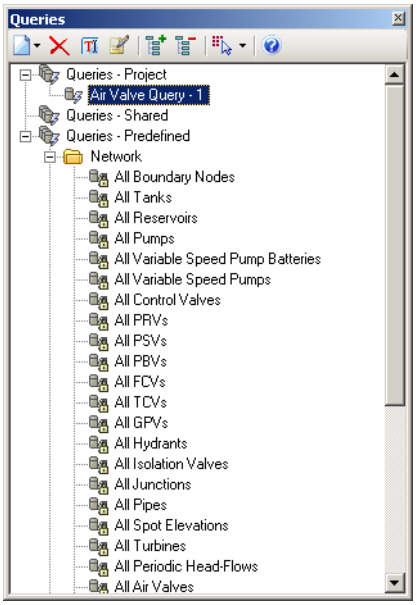
- Create dynamic selection sets based on one or more queries. For more information, see To create a Selection Set from a Query.
- Filter the data in a FlexTable using a query. For more information, see Sorting and Filtering FlexTable Data.
- You can use predefined queries in the Network Navigator. See Using the Network Navigator for more details.

For more information on how to construct queries, see Creating Queries.

Queries Manager

The Queries manager is a docking manager that displays all queries in the current project, including predefined, shared, and project queries. You can create, edit, or delete shared and project queries from within the Queries Manager, as well as use it to select all elements in your model that are part of the selected query.

To open the Queries manager, click the View menu and select the Queries command, press <Ctrl+5>, or click the Queries button  on the View toolbar.



The Queries manager consists of a toolbar and a tree view, which displays all of the queries that are associated with the current project.

The toolbar contains the following icons:

New



Contains the following commands:

- **Query**—Creates a new SQL expression as either a project or shared query, depending on which item is highlighted in the tree view.
- **Folder**—Creates a folder in the tree view, allowing you to group queries. You can right-click a folder and create queries or folders in that folder.

Delete



Deletes the currently-highlighted query or folder from the tree view. When you delete a folder, you also delete all of the queries it contains.

Rename



Renames the query or folder that is currently highlighted in the tree view.

Edit



Opens the Query Builder dialog box, allowing you to edit the SQL expression that makes up the currently-highlighted query.

Expand All

Opens all the Queries within all of the folders.



Collapse All

Closes all the Query folders.



Select in Drawing

Opens a submenu containing the following options:



- **Select in Drawing**—Selects the element or elements that satisfy the currently highlighted query.
- **Add to Current Selection**—Adds the element or elements that satisfy the currently highlighted query to the group of elements that are currently selected in the Drawing Pane.
- **Remove from Current Selection**—Removes the element or elements that satisfy the currently highlighted query from the group of elements that are currently selected in the Drawing Pane.
- **Select Within Current Selection**—Selects the element or elements that both satisfy the current query and are already selected in the Drawing Pane.

Help

Displays online help for the Query Manager.



Query Parameters Dialog Box

Some predefined queries require that a parameter be defined. When one of these queries is selected, the Query Parameters dialog box will open, allowing you to type the parameter value that will be used in the query. For example, when the Pipe Split Candidates query is used the Query Parameters dialog will open, allowing the Tolerance parameter to be defined.

Creating Queries

A query is a valid SQL expression that you construct in the Query Builder dialog box. You create and manage queries in the Query Manager. You also use queries to filter FlexTables and as the basis for a selection set.

To create a query from the Query manager

1. Choose View > Queries or click the Queries icon on the View toolbar, or press <CTRL+5>.
2. Perform one of the following steps:
   - To create a new project query, highlight **Queries - Project** in the list pane, then click the **New** button and select **Query**.
   - To create a new shared query, highlight **Queries - Shared** in the list pane, then click the **New** button and select **Query**.

   Note: You can also right-click an existing item or folder in the list pane and select New > Query from the shortcut menu.
3. In the Select Element Type dialog box, select the desired element type from the drop-down menu. The Query Builder dialog box opens.
4. All input and results fields for the selected element type appear in the Fields list pane, available SQL operators and keywords are represented by buttons, and available values for the selected field are listed in the Unique Values list pane. Perform the following steps to construct your query:
   a. Double-click the field you wish to include in your query. The database column name of the selected field appears in the preview pane.
   b. Click the desired operator or keyword button. The SQL operator or keyword is added to the SQL expression in the preview pane.
   c. Click the **Refresh** button above the Unique Values list pane to see a list of unique values available for the selected field. Note that the Refresh button is disabled after you use it for a particular field (because the unique values do not change in a single query-building session).
   d. Double-click the unique value you want to add to the query. The value is added to the SQL expression in the preview pane.

      Note: You can also manually edit the expression in the preview pane.

   e. Click the **Validate** button above the preview pane to validate your SQL expression. If the expression is valid, the word “VALIDATED” is displayed in the lower right corner of the dialog box.

f. Click the **Apply** button above the preview pane to execute the query. If you didn't validate the expression, the Apply button validates it before executing it.

g. Click **OK**.

5. Perform these optional steps in the Query Manager:
   - To create a new folder in the tree view, highlight the existing item or folder in which to place the new folder, then click the **New** button and select **Folder**. You can create queries and folders within folders.
   - To delete an existing query or folder, click the **Delete** button. When you delete a folder, you also delete all of its contents (the queries it contains).
   - To rename an existing query or folder, click the **Rename** button, then type a new name.
   - To edit the SQL expression in a query, select the query in the list pane, then click the **Edit** button. The Query Builder dialog box opens.
   - To quickly select all the elements in the drawing pane that are part of the currently highlighted query, click the **Select in Drawing** button.

Example Query

To create a query that finds all pipes with a diameter greater than 8 inches and less than or equal to 12 inches you would do the following:

1. In the **Queries** dialog, click the **New** button and select **Query**.
2. In the **Queries - Select Element Type** dialog, select **Pipe** and click **OK**.
3. In the **Query Builder** dialog, click the **()** (Parentheses) button.
4. Double-click **Diameter** in the **Fields** list.
5. Click the > (Greater Than) button.
6. Click the **Refresh** button above the **Unique Values** list. Double-click the value **8**.
7. In the Preview Pane, click to the right of the closing parenthesis.
8. Click the **And** button.
9. Click the **()** (Parentheses) button.
10. Double-click **Diameter** in the **Fields** list.
11. Click the <= (Less Than or Equal To) button.
12. Double-click the value **12** in the **Unique Values** list.

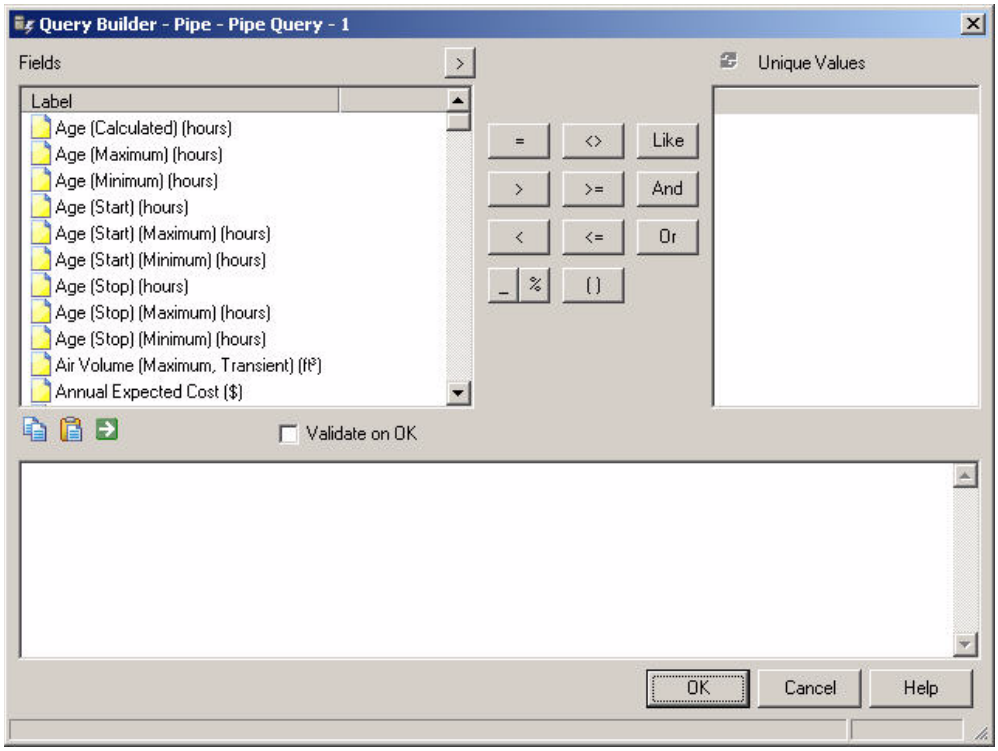
The final query will look like this:

```
(Physical_PipeDiameter > 8) AND (Physical_PipeDiameter <= 12)
```

See Using the Like Operator for more examples of query usage and syntax.

Query Builder Dialog Box

You construct the SQL expression that makes up your query in the Query Builder dialog box. The Query Builder dialog box is accessible from the Query manager and from within a FlexTable.



The top part of the dialog box contains all the controls you need to construct your query: a list pane displaying all available attributes for the selected element type, an SQL control panel containing available SQL keywords and operators, and list view that displays all the available values for the selected attribute. The bottom part of the dialog box contains a preview pane that displays your SQL expression as you construct it.

See Using the Like Operator for some examples of query usage and syntax.

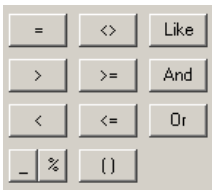
All the dialog box controls are described in the following table.

Fields



Lists all input and results fields applicable to the selected element type. This list displays the labels of the fields while the underlying database column names of the fields become visible in the preview pane when you add them to the expression. Double-click a field to add it to your SQL expression.

SQL Controls



These buttons represent all the SQL operators and controls that you can use in your query. They include =, >, <, **_**, **%**, <>, >=, <=, **[ ]**, **Like**, **And**, and **Or**. Click the appropriate button to add the operator or keyword to the end of your SQL expression, which is displayed in the preview pane.

Unique Values

When you click the **Refresh** button, this list displays all the available unique values for the selected field. Double-click a value in the list to add it to the end of your SQL expression, which is displayed in the preview pane. If you select a different field, you must click the **Refresh** button again to update the list of unique values for the selected field. When you first open the Query Builder dialog box, this list is empty.

Refresh



Updates the list of unique values for the selected field. This button is disabled after you use it for a particular field.

Copy



Copies the entire SQL expression displayed in the preview pane to the Windows clipboard.

Paste



Pastes the contents of the Windows clipboard into the preview pane at the location of the text cursor. For example, if your cursor is at the end of the SQL expression in the preview pane and you click the **Paste** button, the contents of your clipboard will be added to the end of the expression.

Validate on OK



Turn on to validate the SQL expression in the preview pane. If the expression is not valid, a message appears. When you turn on and your SQL expression passes validation, the word “VALIDATED” appears in the lower right corner of the dialog box.

Apply



Executes the query. The results of the query are displayed at the bottom of the Query Builder dialog box in the form “*x* of *x* elements returned.”

Preview Pane

Displays the SQL expression as you add fields, operators and/or keywords, and values to it.

Action

Allows you to select the operation to be performed on the elements returned by the query defined in the Preview pane. The following choices are available:

- **Create New Selection**—Creates a new selection containing the elements returned by the query.
- **Add to Current Selection**—Adds the elements returned by the query to the current selection.
- **Remove from Current Selection**—Removes the elements returned by the query from the current selection.
- **Select Within Current Selection**—Selects the element or elements that both satisfy the current query and are already selected in the Drawing Pane.

This control is only available when the Query Builder is accessed from the command Edit > Select By Attribute.

Note: If you receive a Query Syntax Error message notifying you that the query has too few parameters, check the field name you entered for typos. This message is triggered when the field name is not recognized.

Using the Like Operator

The LIKE operator does a pattern matching comparison. The operand to the right of the LIKE operator contains the pattern and the left hand operand contains the string to match against the pattern. A percent symbol ("%") in the LIKE pattern matches any sequence of zero or more characters in the string. An underscore ("_") in the LIKE pattern matches any single character in the string. Any other character matches itself or its lower/upper case equivalent (i.e. case-insensitive matching).

% and _ are wildcard characters. If the element(s) you are looking for contains one or more of those characters, you will need to prefix each one of those characters in the search term with \. E.g. J\%1

Query Examples

In order to get all elements of a given type whose label starts with a given letter(s) (e.g. J-1###), one could do a query such as:

```
Label LIKE 'J-1%'
```

In this case, the query would return elements with labels like J-1, J-100, J-101, but not J-01, J-001.

In order to get all elements of a given type whose label ends with a given letter(s) (e.g. ###100), one could do a query such as:

```
Label LIKE '%100'
```

In this case, the query would return elements with labels like J-100, J-10100, J-AA100, but not J-1000, J-100A.

In order to get all elements of a given type whose label contains a given letter(s) (e.g. #-1#), one could do a query such as:

```
Label LIKE '%-1%'
```

In this case, the query would return elements with labels like J-10, J-101, Node-10A, but not J10, J-20, J101.

In order to get all elements of a given type whose label ends with a single character, one could do a query such as:

```
Label LIKE 'J-1?'
```

In this case, the query would return elements with labels like J-1A, J-10, J-11, but not J-1, J-1AA, J1A.

When querying by referenced fields (i.e. zones for Junctions) where no referenced field exists (i.e. finding junctions that have no assigned zone) use the following query:

```
Physical_Zone IS NULL
```

User Data Extensions

User data extensions are a set of one or more attribute fields that you can define to hold data to be stored in the model. User data extensions allow you to add your own data fields to your project. For example, you can add a field for keeping track of the date of installation for an element or the type of area serviced by a particular element.

Note: The user data does not affect the hydraulic model calculations. However, their behavior concerning capabilities like editing, annotating, sorting and database connections is identical to any of the standard pre-defined attributes.

User data extensions exhibit the same characteristics as the predefined data used in and produced by the model calculations. This means that user data extensions can be imported or exported through database and shapefile connections, viewed and edited in the Property Editor or in FlexTables, included in tabular reports or element detailed reports, annotated in the drawing, color coded, and reported in the detailed element reports.

Note: The terms “user data extension” and “field” are used interchangeably here. In the context of the User Data Extension feature, these terms mean the same thing.

You define user data extensions in the User Data Extensions dialog box.

To define a user data extension

1. Select **Tools > User Data Extensions**.
2. In the list pane on the left, select the element type for which you want to define a new attribute field.
3. Click the **New** button to create a new user data extension. A user data extension with a default name appears under the element type. You can rename the new field if you wish.
4. In the properties pane on the right, enter the following:

- Type the name of the new field. This is the unique identifier for the field. The name field in the Property Editor is the name of the column in the data source.
- Type the label for the new field. This is the label that will appear next to the field for the user data extension in the Property Editor for the selected element type. This is also the column heading if the data extension is selected to appear in a FlexTable.
- Click the **Ellipses (...)** button in the Category field, then use the drop-down menu in the Select Category dialog box to select an existing category in which the new field will appear in the Property Editor. To create a new category, simply type the category name in the field.
- Type a number in the Field Order Index field. This is the display order of fields within a particular category in the Property Editor. This order also controls the order of columns in Alternative tables. An entry of 0 means the new field will be displayed first within the specified category.
- Type a description for the field. This description will appear at the bottom of the Property Editor when the field is selected for an element in your model. You can use this field as a reminder about the purpose of the field.
- Select an alternative from the drop-down menu in the Alternative field. This is the alternative that you want to extend with the new field.
- Select a data type from the drop-down menu in the Data Type field.
  - If you select **Enumerated**, an Ellipses (...) button appears in the Default Value field. Enumerated user data extensions are fields that present multiple choices.
- Enter the default value for the new field. If the data type is **Enumerated**, click the **Ellipses (...)** button to display the Enumeration Editor dialog box, where you define enumerated members.

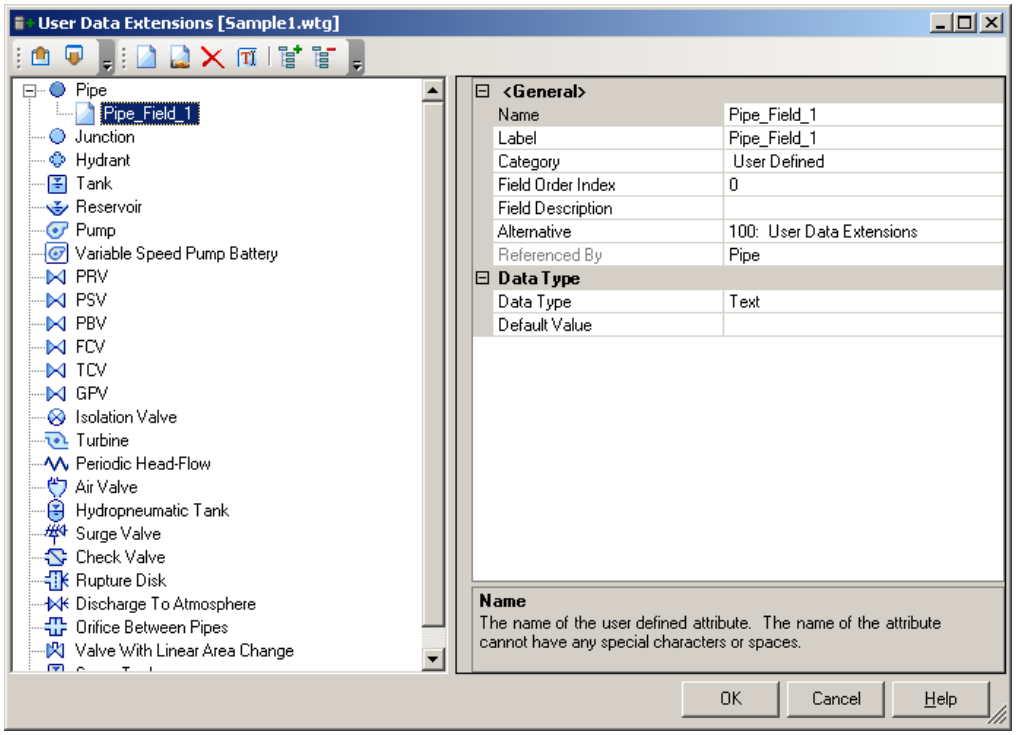
5. Perform the following optional steps:
   - To import an existing User Data Extension XML File, click the **Import** button, then select the file you want to import. User Data Extension XML Files contain the file name extension .xml or .udx.xml.
   - To export existing user data extensions, click the **Export to XML** button, then type the name of the udx.xml file. All user data extensions for all element types defined in the current project are exported.
   - To share the new field among two or more element types, select the user data extension in the list pane, then click the **Sharing** button or right-click and select **Sharing**. In the Shared Field Specification dialog box, select the check box next to the element or elements that will share the user data extension. The icon next to the user data extension changes to indicate that it is a shared field. For more information, see Sharing User Data Extensions Among Element Types on page 4-484.

- To delete an existing user data extension, select the user data extension you want to delete in the list pane, then click the **Delete** button, or right-click and select **Delete**.
- To rename the display label of an existing user data extension, select the user data extension in the list pane, click the **Rename** button or right-click and select **Rename**, then type the new display label.
- To expand the list of elements and view all user data extensions, click the **Expand All** button.
- To collapse the list of elements so that no user data extensions are displayed, click the **Collapse All** button.

6. Click **OK** to close the dialog box and save your user data extensions. The new field(s) you created will appear in the Property Editor for every instance of the specified element type in your model.

User Data Extensions Dialog Box

The User Data Extensions dialog box displays a summary of the user data extensions associated with the current project. The dialog box contains a toolbar, a list pane displaying all available WaterCAD V8*i* element types, and a property editor.



The toolbar contains the following controls:

	**Import**	Merges the user data extensions in a saved User Data Extension XML file (.udx.xml or .xml) into the current project. Importing a User Data Extension XML file will not remove any of the other data extensions defined in your project. User data extensions that have the same name as those already defined in your project will not be imported.
	**Export to XML**	Saves existing user data extensions for all element types in your model to a User Data Extension XML file (.udx.xml) for use in a different project.
	**Add Field**	Creates a new user data extension for the currently highlighted element type.
	**Share**	Shares the current user data extension with another element type. When you click this button, the Shared Field Specification dialog box opens. For more information, see Sharing User Data Extensions Among Element Types on page 4-484.
	**Delete Field**	Deletes the currently highlighted user data extension
	**Rename Field**	Renames the display label of the currently highlighted user data extension.
	**Expand All**	Expands all of the branches in the hierarchy displayed in the list pane.
	**Collapse All**	Collapses all of the branches in the hierarchy displayed in the list pane.

The property editor section of the dialog contains following fields, which define your new user data extension:

Attribute	Description
General	
Name	The unique identifier for the field. The name field in the Property Editor is the name of the column in the data source.
Label	The label that will appear next to the field for the user data extension in the Property Editor for the selected element type. This is also the column heading if the data extension is selected to appear in a FlexTable.
Category	The section in the Property Editor for the selected element type in which the new field will appear. You can create a new category or use an existing category. For example, you can create a new field for junctions and display it in the Physical section of that element's Property Editor.
Field Order Index	The display order of fields within a particular category in the Property Editor. This order also controls the order of columns in Alternative tables. An entry of 0 means the new field will be displayed first within the specified category.
Field Description	The description of the field. This description will appear at the bottom of the Property Editor when the field is selected for an element in your model. You can use this field as a reminder about the purpose of the field.
Alternative	Selects an existing alternative to extend with the new field.
Referenced By	Displays all the element types that are using the field. For example, if you create a field called "Installation Date" and you set it up to be shared, this field will show the element types that share this field. So for example, if you set up a field to be shared by junctions and catch basins, the Referenced By field would show "Manhole, Catch Basin".

Attribute	Description
Units	
Data Type	Specifies the data type for the user data extension. Click the down arrow in the field then select one of the following data types from the drop-down menu: • **Integer**—Any positive or negative whole number. • **Real**—Any fractional decimal number (for example, 3.14). It can also be unitized with the provided options. • **Text**—Any string (text) value up to 255 characters long. • **Long Text**—Any string (text) up to 65,526 characters long. • **Date/Time**—The current date. The current date appears by default in the format month/day/year. Click the down arrow to change the default date. • **Boolean**—True or False. • **Enumerated**—When you select this data type, an Ellipses button appears in the Default Value field. Click the **Ellipses (...)** button to display the Enumeration Editor dialog box, where you can add enumerated members and their associated values. For more information, see Enumeration Editor Dialog Box on page 4-486.
Default Value	The default value for the user data extension. The default value must be consistent with the selected data type. If you chose Enumerated as the data type, click the **Ellipses (...)** button to display the Enumeration Editor.
Dimension	Specifies the unit type. Click the drop-down arrow in the field to see a list of all available dimensions. This field is available only when you select **Real** as the Data Type.
Storage Unit	Specifies the storage units for the field. Click the drop-down arrow in the field to see a list of all available units; the units listed change depending on the Dimension you select. This field is available only when you select **Real** as the Data Type.
Numeric Formatter	Selects a number format for the field. Click the drop-down arrow in the field to see a list of all available number formats; the number formats listed change depending on the Dimension you select. For example, if you select **Flow** as the Dimension, you can select Flow, Flow - Pressurized Condition, Flow Tolerance, or Unit Load as the Numeric Formatter. This field is available only when you select **Real** as the Data Type.

Sharing User Data Extensions Among Element Types

You can share user data extensions across multiple element types in WaterCAD V8*i*. Shared user data extensions are displayed in the Property Editor for all elements types that share that field.

The icons displayed next to the user data extensions in the User Data Extensions dialog box change depending on the status of the field:

- Indicates a new unsaved user data extension. 
- Indicates a user data extension that has been saved to the data source. 
- Indicates a user data extension that is shared among multiple element types but has not been applied to the data source. 
- Indicates a user data extension that is shared among multiple element types and that has been applied to the data source.  Fields with this icon appear in the Property Editor for any elements of the associated element types that appear in your model.

Observe the following rules when sharing user data extensions:

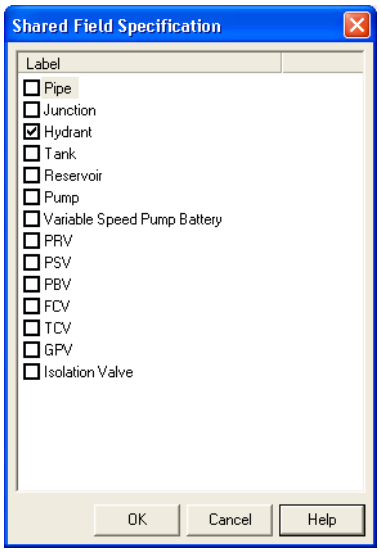
- You can select any number of element types with which to share the field. The list is limited to element types that support the Alternative defined for the Field. For example, the Physical Alternative may only apply to five of the element types. In this case, you will only see these five items listed in the Alternative drop-down menu.
- You cannot use the sharing feature to move a field from one element type to another. Validation is in place to ensure that only one item is selected and if it is the same as the original, default selection. If it is not, a message appears telling you that when sharing a field, you must select at least two element types, or select the original element type.
- To unshare a field that is shared among multiple element types, right-click the user data extension you want to keep in the list pane, then select **Sharing**. Clear all the element types that you do not want to share the field and click **OK**. If you leave only one element type checked in the Shared Field Specification dialog box, it must be the original element type for which you created the user data extension.
  - The fields that were located under the tank and pipe element type root nodes will be removed completely.
  - You can also unshare a field by using the **Delete** button or right-clicking and selecting **Delete**. This will unshare and delete the field.

To share a user data extension

1. Open the User Data Extensions dialog box by selecting **Tools > User Data Extensions**.
2. In the list pane, create a new user data extension to share or select an existing user data extension you want to share, then click the **Sharing** button.
3. In the Shared Field Specification dialog box, select the check box next to each element type that will share the user data extension.
4. Click **OK**.
5. The icon next to the user data extension in the list pane changes to indicate that it is a shared field.

Shared Field Specification Dialog Box

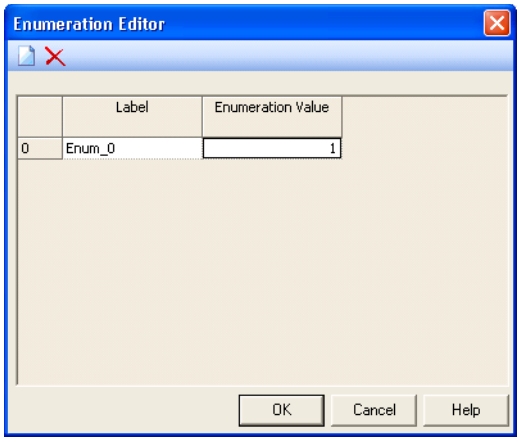
Select element types to share a user data extension in the Shared Field Specification dialog box. The dialog box contains a list of all possible element types with check boxes.



Select element types to share the current user data extension by selecting the check box next to the element type. Clear a selection if you no longer want that element type to share the current field.

Enumeration Editor Dialog Box

The Enumeration Editor dialog box opens when you select **Enumerated** as the Data Type for a user data extension, then click the **Ellipses (...)** button in the Default Value field. Enumerated fields are fields that contain multiple selections - you define these as members in the Enumeration Editor dialog box.



For example, suppose you want to identify pipes in a model of a new subdivision by one of the following states: Existing, Proposed, Abandoned, Removed, and Retired. You can define a new user data extension with the label “Pipe Status” for pipes, and select Enumerated as the data type. Click the **Ellipses (...)** button in the Default Value field in the Property Editor for the user data extension to display the Enumeration Editor dialog box. Then enter five members with unique labels (one member for each unique pipe status) and enumeration values in the table. After you close the User Data Extensions dialog box, the new field and its members will be available in the Property Editor for all pipes in your model. You will be able to select any of the statuses defined as members in the new Pipe Status field.

You can specify an unlimited number of members for each user data extension, but member labels and values must be unique. If they are not unique, an error message appears when you try to close the dialog box.

The dialog box contains a table and the following controls:

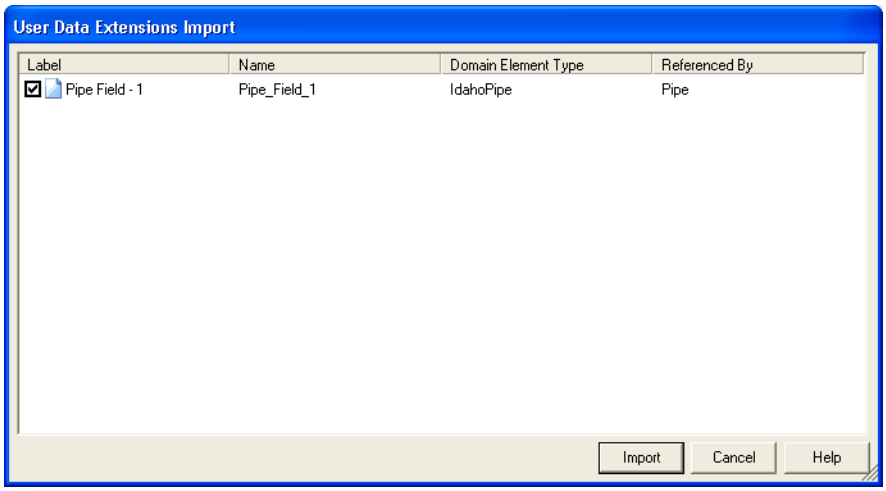
- **New—**Adds a new row to the table. Each row in the table represents a unique enumerated member of the current user data extension.
- **Delete—**Deletes the current row from the table. The enumerated member defined in that row is deleted from the user data extension.

Define enumerated members in the table, which contains the following columns:

- **Enumeration Member Display Label—**The label of the member. This is the label you will see in WaterCAD V8*i* wherever the user data extension appears (Property Editor, FlexTables, etc.).
- **Enumeration Value—**A unique integer index associated with the member label. WaterCAD V8*i* uses this number when it performs operations such as queries.

User Data Extensions Import Dialog Box

The Import dialog box opens after you initiate an Import command and choose the xml file to be imported. The Import dialog displays all of the elements contained within the selected xml file. Uncheck the boxes next to a domain element to ignore them during import.



Formula Dialog Box

This dialog allows you to define formulas for use with the Real (Formula) User Data Extension type.

You construct the formula using the available fields, operators, and functions. All the dialog box controls are described in the following table.

Fields



Lists all input and results fields applicable to the selected element type. This list displays the labels of the fields while the underlying database column names of the fields become visible in the preview pane when you add them to the formula. Double-click a field to add it to your formula.

Operators

These buttons represent all of the operators that can be used in the formula. Click the appropriate button to add the operator to the end of your formula , which is displayed in the preview pane. Besides the common options for options for adding, subtracting, multiplying and dividing values , there are also ( ) which allows for more complex formulas, and the caret (^) which is used for raising a value to the power of a value

Available Math Functions

Lists mathematical functions that can be used in the formula. If you hover over a function it will describe the number of required parameters and a brief description of what the function does.

Copy



Copies the entire formula displayed in the preview pane to the Windows clipboard.

Paste



Pastes the contents of the Windows clipboard into the preview pane at the location of the text cursor. For example, if your cursor is at the end of the formula in the preview pane and you click the **Paste** button, the contents of your clipboard will be added to the end of the formula.

Preview Pane

Displays the formula as you add fields, operators, and functions to it.

Property Grid Customizations Manager

The Property Grid Customizations Manager allows you to create customization profiles that define changes to the default user interface. Customization profiles allow you to turn off the visibility of properties in the Properties Editor.

Customization Profiles can be created for a single project or shared across projects. There are also a number of predefined profiles.

The Property Grid Customizations Manager consists of the following controls:

	**New**	This button opens a submenu containing the following commands: • Folder: This command creates a new folder under the currently highlighted node in the list pane. • Customization: This command creates a new customization profile under the currently highlighted node in the list pane.
	**Delete**	This button deletes the currently highlighted folder or customization profile.
	**Rename**	This button allows you to rename the currently highlighted folder or customization profile.
	**Duplicate**	This button allows you to make a copy of the highlighted customization profile.
	**Edit**	Opens the Customization Editor dialog allowing you to edit the currently highlighted customization profile.
	**Help**	Opens the online help.

Customization Editor Dialog Box

This dialog box allows you to edit the customization profiles that are created in the Customization Manager. In the Customization editor you can turn off the visibility of various properties in the Property Grid.

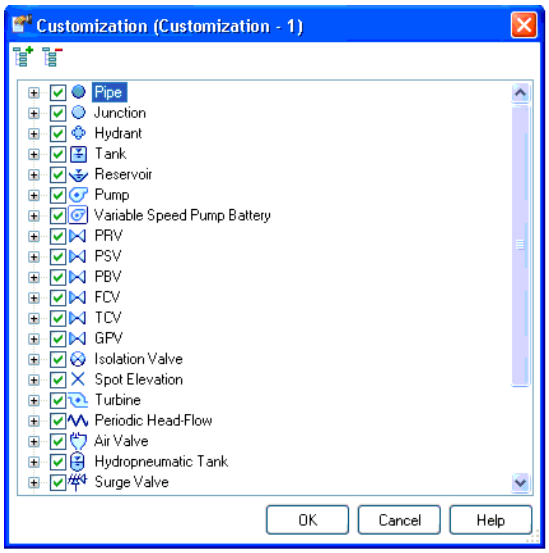
You can turn off any number of properties and/or entire categories of properties in a single customization profile.

To remove a property from the property grid:

1. Select the element type from the pulldown menu.
2. Find the property you want to turn off by expanding the node of the category the property is under.
3. Uncheck the box next to the property to be turned off.
4. Click OK.

To turn off all of the properties under a category:

1. Select the element type from the pulldown menu.
2. Uncheck the box next to the category to be turned off.
3. Click OK.



Tooltip Customization

Tooltip customization allows you to define what data is displayed in the tooltip that appears when you hover over an element in the drawing pane.

Tooltip Customization settings can be created for a single project or shared across projects. There are also a number of predefined profiles.

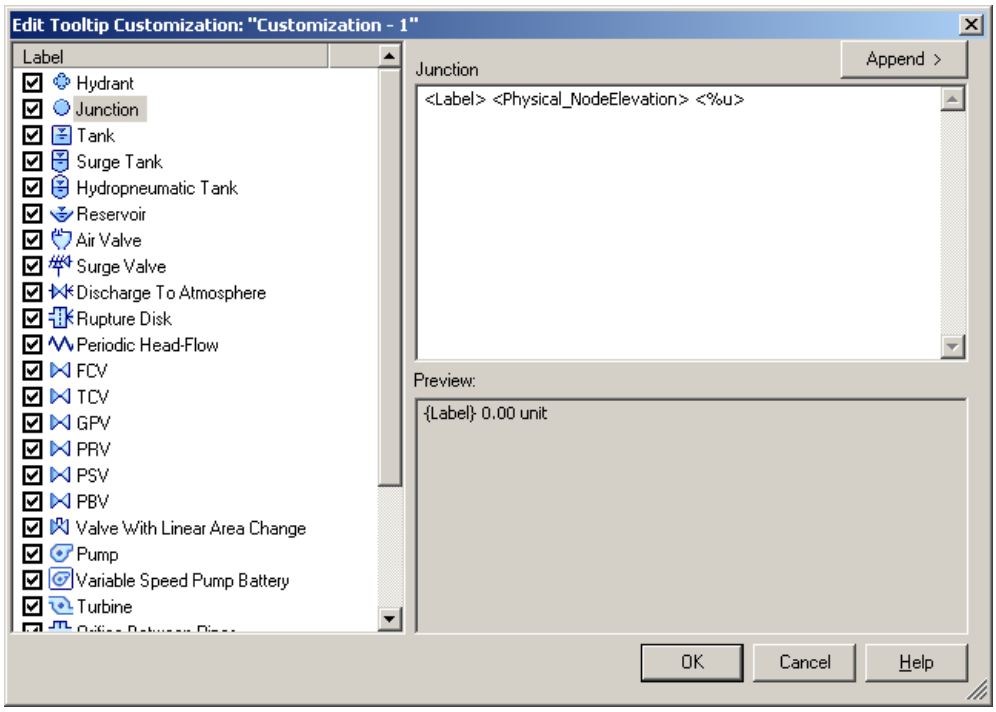
The Tooltip Customizations Manager consists of the following controls:

	**New**	This button opens a submenu containing the following commands: • Folder: This command creates a new folder under the currently highlighted node in the list pane. • Customization: This command creates a new customization profile under the currently highlighted node in the list pane.
	**Delete**	This button deletes the currently highlighted folder or customization profile.
	**Rename**	This button allows you to rename the currently highlighted folder or customization profile.
	**Duplicate**	This button allows you to make a copy of the highlighted customization profile.
	**Make Active**	This button allows you to make the currently highlighted customization profile the active one.
	**Edit**	Opens the Tooltip Customization Editor dialog allowing you to edit the currently highlighted customization profile.
	**Help**	Opens the online help.

See Tooltip Customization Editor for information on defining tooltip customizations.

Tooltip Customization Editor

This dialog allows you to define the tooltip customizations on a per-element basis.



On the left is a list of all of the element types. If the box for an element type is unchecked, no tooltip will be displayed for that element type.

Highlight an element type to define the tooltip in the pane on the upper right. You can type in the field or use the Append button to select from a number of predefined variables. After a tooltip using these variables has been defined, these variables will be populated with the associated values in the drawing pane after the model has been calculated.

The Preview pane displays an example of how the tooltip will look.

i-Models

The term “i-models” is used to describe a type of Bentley file (container) which can be used to share data between applications. The formal definition of an i-model is:

An immutable container for rich multi-discipline information published from known sources in a known state at a known time. It is a published rendition in a secure read-only container. It is a portable, self-describing and semantically rich data file.

i-models can be thought of as similar to shapefiles in that they provide ways to share data. They are immutable in that they cannot be modified (they are read-only). They reflect the state of the model file at the time the i-model was created.

i-model support is built on Bentley technology and is not automatically installed with WaterCAD V8*i* or other hydraulic products. The software to use i-models is installed with Microstation and other Microstation based products (versions 08.11.07 or later). If a user attempts to create an i-model and the support for i-model creation is not installed, an error message to download and install the necessary files is issued. The i-model files can be installed from the Bentley SELECTdownload site.

An i-model can contain all the elements and their properties for a model for a given scenario and time-step or the information can be filtered so that only a fraction of the elements and their properties are incorporated in the i-model.

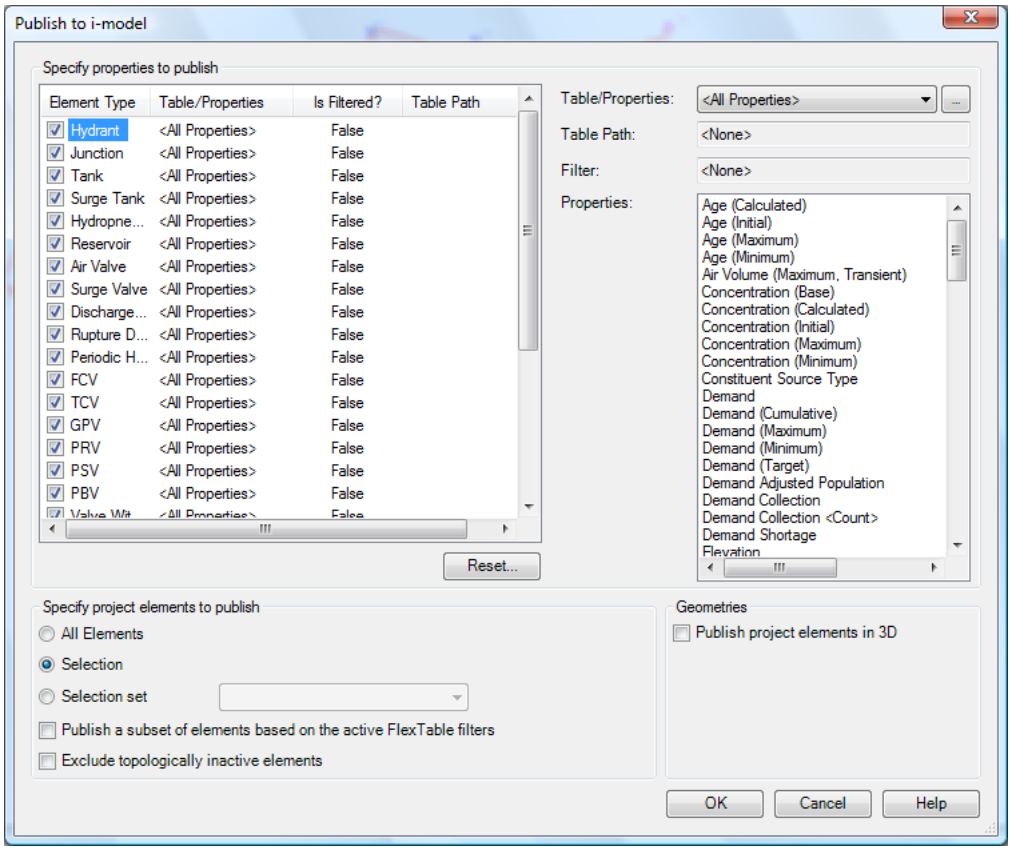
An i-model is generally much smaller than the .sqlite file for the hydraulic model even though it does contain results.

For details on publishing and viewing i-models, see Publishing an i-model and Viewing an i-model.

Publishing an i-model

To create an i-model, select File > Export > Publish i-model once the desired scenario and time-steps have been selected.

The following dialog opens with the defaults set so that all elements and properties are included in the i-model.



The top left pane is a summary of this element types are to be included in the i-model. If a box by the element type is checked, that element type is included. The Table/Properties column reflects the selections on the right side of the dialog in terms of which elements and properties are included.

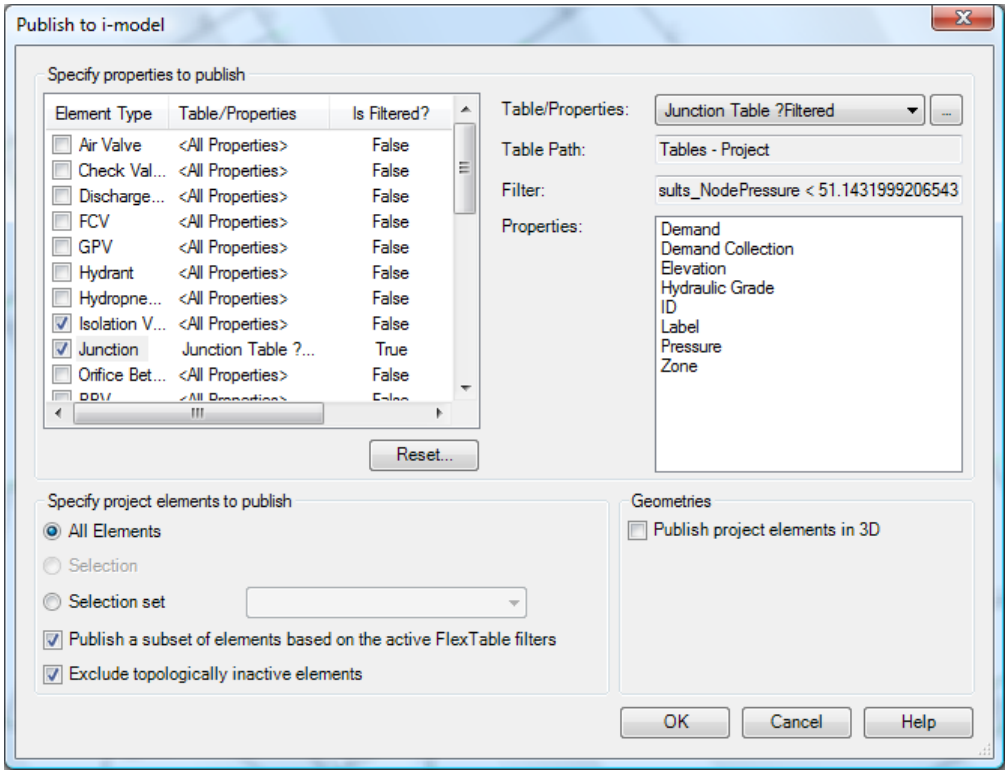
The bottom left portion of the dialog is used to identify which elements are to be included in the i-model. This can be specified individually for each element type.

If the "Publish a subset of elements based on the Flex Table filters" box is checked, only those elements that are in the filtered flex table will be included in the i-model.

If the "Exclude topologically inactive elements" box is checked, only active elements (Is active? = True) are included in the i-model.

The user will usually not need to include all element properties in the i-model. The right side of the dialog is to identify which properties of the elements are going to be included in the i-model. The default is "all properties". If the user wants to only include a subset of properties, the user should create a flex table with only those properties and select that flex table from the drop down list. Because it is possible to have

multiple flex tables with the same name (e.g. Pipe Table can be a predefined table or a Project table), the user can explicitly state the table path (e.g. Tables - Predefined or Tables - Project). If the flex table is filtered, the filter is displayed in the Filter box and in the left pane, the Is Filtered column is set to True for that element type.



The Properties box on the right side of the dialog shows the properties that are imported for that element type.

If the box for "Publish project elements in 3D" is selected, the elements will be published in 3D.

The main motivation behind allowing publishing geometries in 3D is to enable clash-detection. That feature is expected to be more important for gravity hydraulic products, but it is included with pressure-based applications as well. The basic functionality regarding this topic can be summarized as:

Node cells' z-coordinates are assigned according to their elevation values, at their cell's insertion point.

- 3D node cells in the cell-library are supported.
- Pipes are exported as cylinders, with partial toroidal shapes at their vertices.
- Pipe cylinder diameters match assigned diameter values.
- Pipe elevations in pressure applications are assumed to be at center of cylinders.

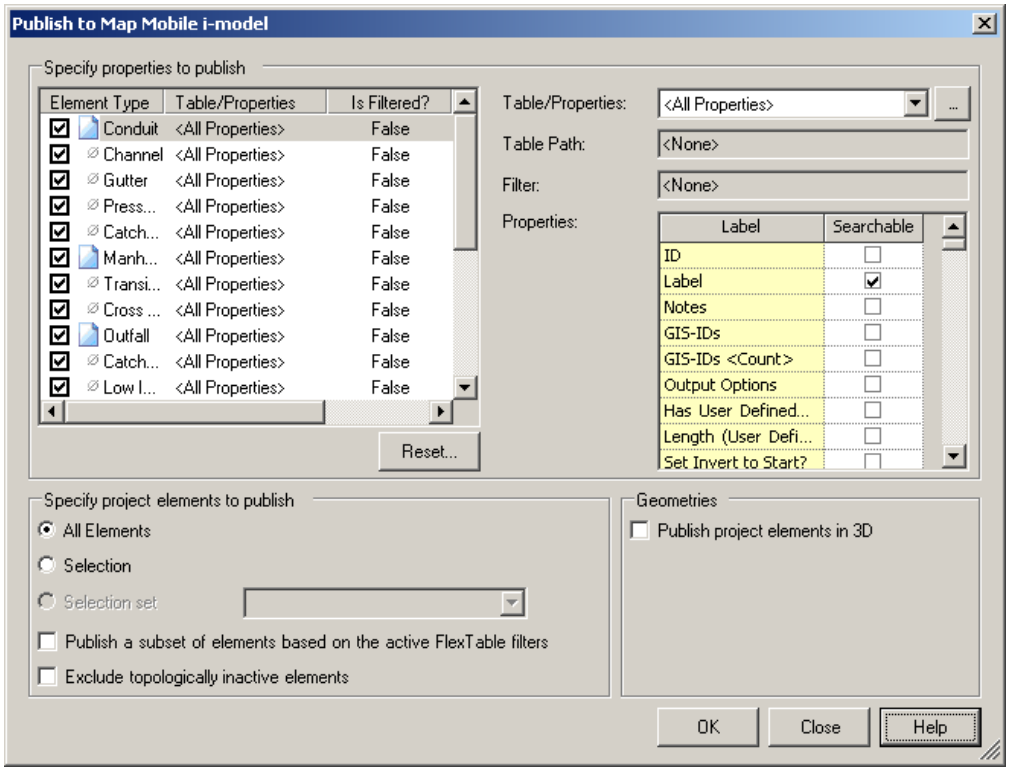
- Pipe elevations in gravity applications have more details to be aware of (e.g. rim, invert and crown elevations).
- References and any extra graphics published (e.g. annotations) are assigned a z-coordinate of 0.0.

When all settings are established for all element types, the user picks OK.

Upon starting the publishing, the user is asked for the file name for the .dgn file that will contain the i-model. The user names the file and path as with any other Windows application.

Publish to Map Mobile i-model

To publish to a Map Mobile i-model, select File > Export > Publish to Map Mobile i-model once the desired scenario and time-steps have been selected.



The Publish to Map Mobile i-model dialog box consists of the same controls as the Publish to i-model dialog. See Publishing an i-model on page 4-493 for more details on using this dialog.

You can use a geospatial reference (specified in the options dialog - see Options Dialog Box - Project Tab on page 4-129) when publishing from stand-alone. This spatial reference is applied to the i-model being published.

If publishing from MicroStation, a geospatial reference is used when publishing the i-model if one has been assigned.

Invalid geospatial references are ignored.

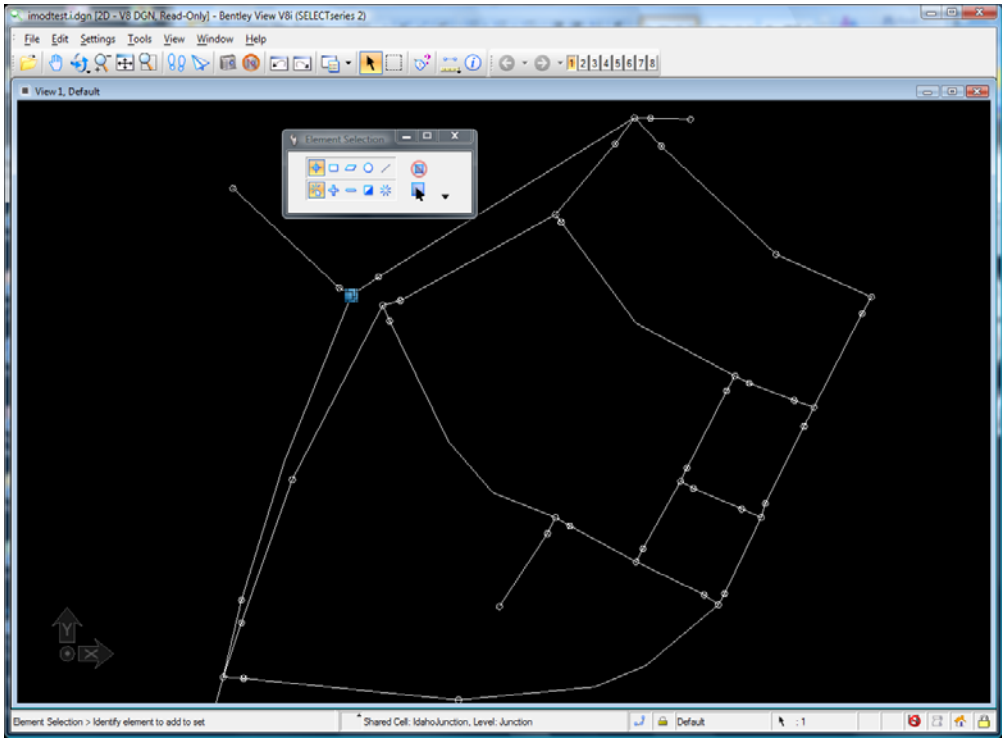
If specified correctly, GPS capabilities will be enabled in the Map Mobile app, including the ability to get directions to a selected element.

Viewing an i-model

It is anticipated that numerous applications will be able to view and use i-models. Initially, i-models can be view using

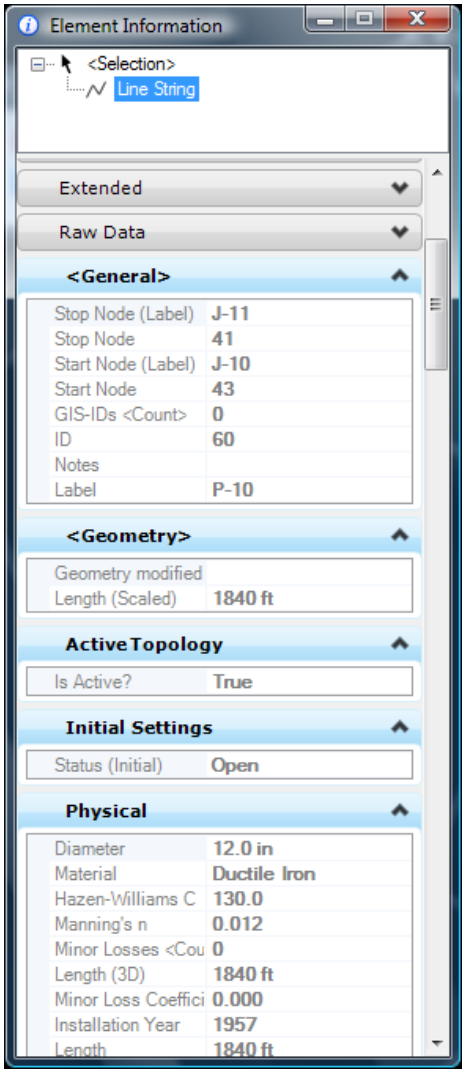
- Bentley View
- ProjectWise Navigator
- Microstation

In all of these applications, it is possible to open an i-model by browsing to the i-model when the application starts and opening the file.



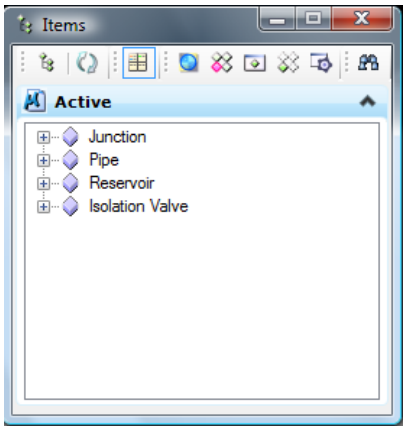
If the model is not visible, pick the "Fit View" button. This should make the model visible. From this view, it is possible to use other commands such as zooming and panning to navigate around the drawing.

To view the properties of individual elements, pick the Element Information button or pick Edit > Information in Bentley View or Review > Information in ProjectWise Navigator. The user can then select an element and its properties will be displayed.

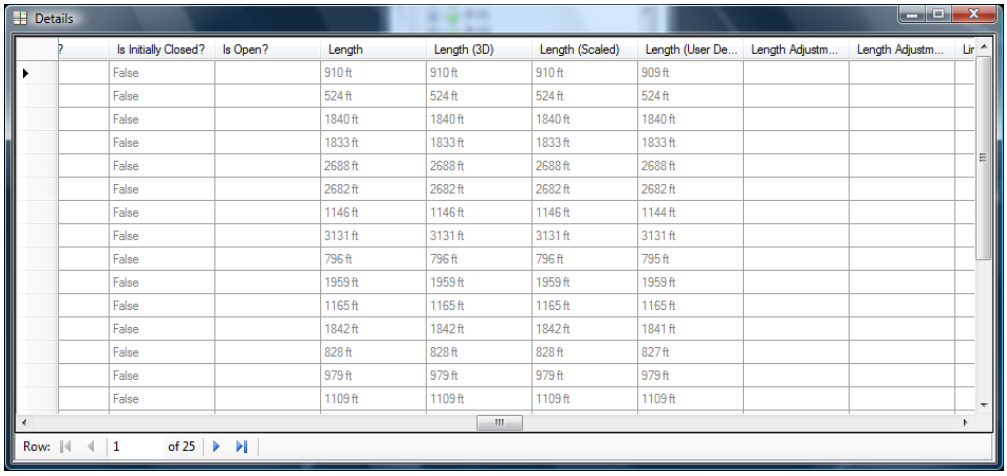


The user can collapse or expand any category in the window.

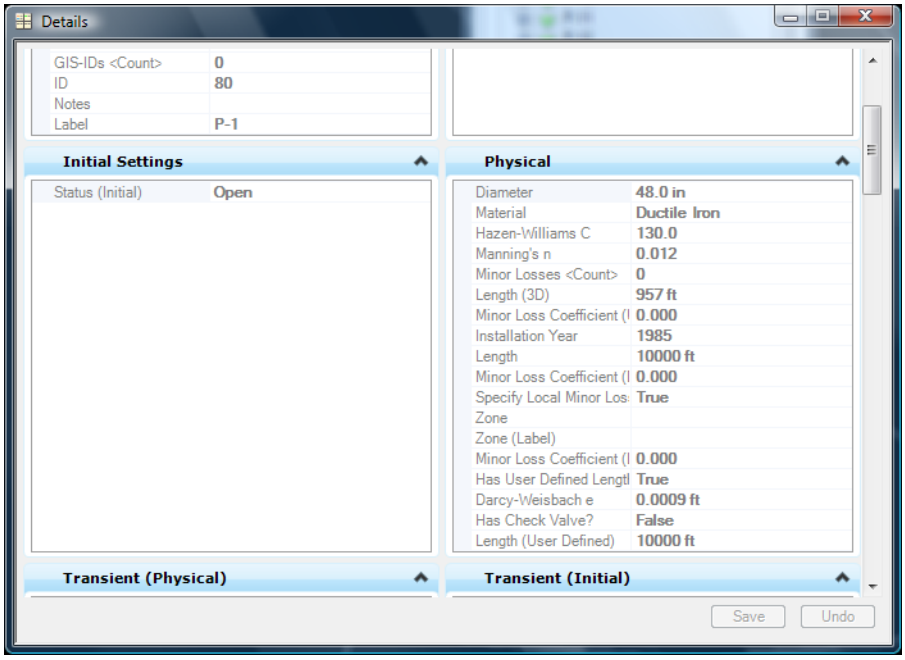
In Microstation and Navigator, it is also possible to view tabular element data for each element type by selecting File > Item browser. This opens the Items browser for element types as shown below:



Double clicking on one of the element types or picking the "Show Details" button from the top of the dialog, opens a table for that element type.

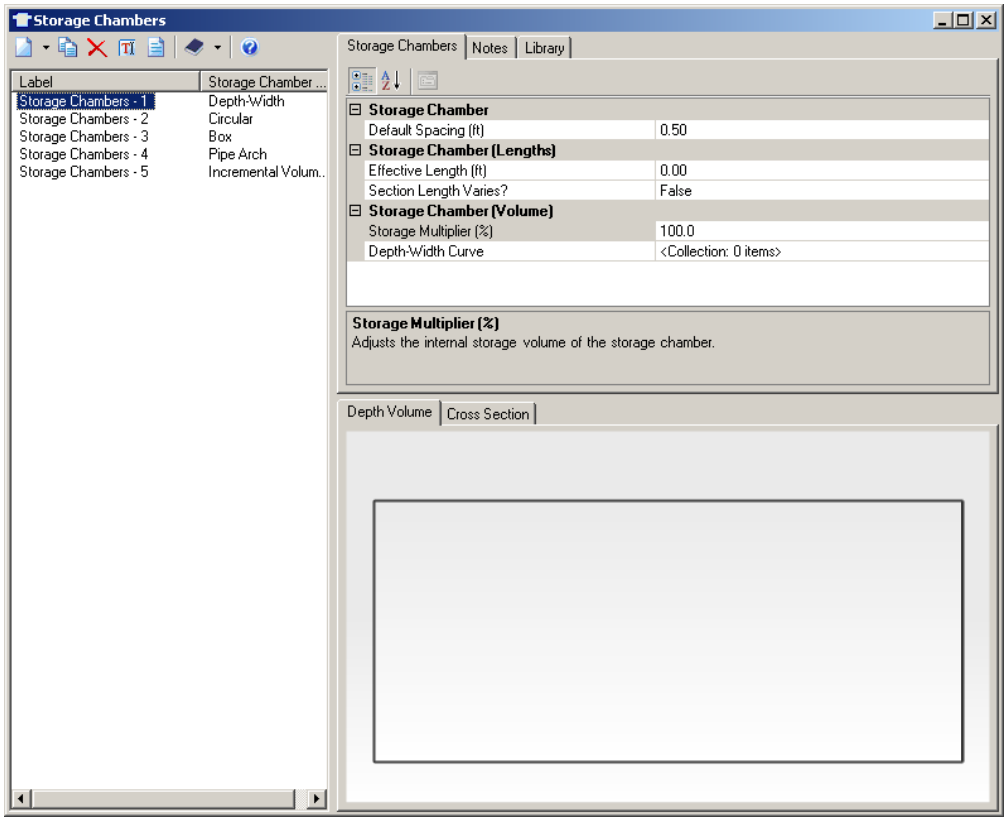


If the tree is expanded before selecting Show Details and an individual element is selected, the user will see properties for the selected element.



Storage Chambers Dialog Box

This dialog allows you to create and reuse prefabricated storage chambers. You can construct complex arrays of storage chambers which can be combined with other standard pond components to create a single storage entity to be used in the routing analysis.



The dialog consists of a list pane on the left and a tabbed section on the right. The list pane displays all of the storage chamber definitions associated with the project. Above the list pane are the following buttons:

-  **New**: Creates a new entry in the list pane.
-  **Duplicate**: Creates a copy of the entry that is currently highlighted in the list pane.
-  **Delete**: Removes the entry that is currently highlighted in the list pane

-  **Rename**: Allows you to enter a new label for the entry that is currently highlighted in the list pane.

-  **Report**: Generates a preformatted report that details the data associated with the entry that is currently highlighted in the list pane.

-  **Synchronization Options**: Clicking this button opens a submenu containing the following commands:
  - **Browse Engineering Library**: Opens the Engineering Library manager dialog, allowing you to browse the Storage Chamber Libraries.
  - **Synchronize From Library**: Lets you update a set of definitions previously imported from a Storage Chamber Library. The updates reflect changes that have been made to the library since it was imported.
  - **Synchronize To Library**: Lets you update an existing Storage Chamber Library using current definitions that were initially imported but have since been modified.
  - **Import From Library**: Lets you import a definition from an existing Storage Chamber Library.
  - **Export To Library**: Lets you export the current definition to an existing Storage Chamber Library.
  - **Connect to Library**—Opens the Engineering Library, allowing you to select a library to synchronize with the current entry.

-  **Help**: Opens the online help associated with this dialog.

The tabbed section on the right side of the dialog consists of the following tabs:

- **Storage Chambers**: The attributes displayed in the property editor section of the Storage Chambers tab will change depending on the type of storage chamber that is currently highlighted in the list pane. Depending on the node type, the property grid allows you to define the following attributes:
  - **Circular**: The following attributes are used to define Circular storage chambers:
    - **Default Spacing**: Set the default edge-to-edge spacing for the storage chamber when organized into rows on the Pond Element.
    - **Effective Length**: The length of the storage chamber when it is interlocked in a storage chamber row. If the Section Length varies, then the effective length represents the length of the middle sections.

- **Section Length Varies?**: If True, you can specify the effective lengths of the start and stop sections of a storage chamber row.
- **Start Section Length**: Specify the length of the storage chamber at the start of the storage chamber row.
- **End Section Length**: Specify the length of the storage chamber at the end of the storage chamber row.
- **Storage Multiplier**: Adjust the internal storage volume of the storage chamber.
- **Wall Thickness**: Set the thickness of the wall of the storage structure.
- **Nominal Diameter**: The internal diameter of the circular structure.

– **Box**: The following attributes are used to define Box storage chambers:

- **Default Spacing**: Set the default edge-to-edge spacing for the storage chamber when organized into rows on the Pond Element.
- **Effective Length**: The length of the storage chamber when it is interlocked in a storage chamber row. If the Section Length varies, then the effective length represents the length of the middle sections.
- **Section Length Varies?**: If True, you can specify the effective lengths of the start and stop sections of a storage chamber row.
- **Start Section Length**: Specify the length of the storage chamber at the start of the storage chamber row.
- **End Section Length**: Specify the length of the storage chamber at the end of the storage chamber row.
- **Storage Multiplier**: Adjust the internal storage volume of the storage chamber.
- **Wall Thickness**: Set the thickness of the wall of the storage structure.
- **Rise**: The internal rise of the storage structure.
- **Span**: The internal span of the storage structure.

– **Depth-Width Curve**: The following attributes are used to define Depth-Width Curve storage chambers:

- **Default Spacing**: Set the default edge-to-edge spacing for the storage chamber when organized into rows on the Pond Element.
- **Effective Length**: The length of the storage chamber when it is interlocked in a storage chamber row. If the Section Length varies, then the effective length represents the length of the middle sections.
- **Section Length Varies?**: If True, you can specify the effective lengths of the start and stop sections of a storage chamber row.
- **Start Section Length**: Specify the length of the storage chamber at the start of the storage chamber row.

- **End Section Length**: Specify the length of the storage chamber at the end of the storage chamber row.
- **Storage Multiplier**: Adjust the internal storage volume of the storage chamber.
- **Depth-Width Curve**: Opens the Depth-Width Curve dialog, allowing you to describe how the width of the storage chamber varies with depth.

– **Pipe Arch**: The following attributes are used to define Pipe Arch storage chambers:

- **Default Spacing**: Set the default edge-to-edge spacing for the storage chamber when organized into rows on the Pond Element.
- **Effective Length**: The length of the storage chamber when it is interlocked in a storage chamber row. If the Section Length varies, then the effective length represents the length of the middle sections.
- **Section Length Varies?**: If True, you can specify the effective lengths of the start and stop sections of a storage chamber row.
- **Start Section Length**: Specify the length of the storage chamber at the start of the storage chamber row.
- **End Section Length**: Specify the length of the storage chamber at the end of the storage chamber row.
- **Storage Multiplier**: Adjust the internal storage volume of the storage chamber.
- **Rise**: The internal rise of the storage structure.
- **Span**: The internal span of the storage structure.
- **Full Area**: Set the internal full area of the storage structure.
- **Bottom Radius**: Set the internal bottom radius of the pipe arch structure.
- **Corner Radius**: Set the internal corner radius of the pipe arch structure.
- **Bottom Distance**: Set the internal bottom distance of the pipe arch structure.
- **Top Radius**: Set the internal top radius of the pipe arch structure.

– **Volume Per Unit Length**: The following attributes are used to define Volume Per Unit Length storage chambers:

- **Default Spacing**: Set the default edge-to-edge spacing for the storage chamber when organized into rows on the Pond Element.
- **Effective Length**: The length of the storage chamber when it is interlocked in a storage chamber row. If the Section Length varies, then the effective length represents the length of the middle sections.
- **Section Length Varies?**: If True, you can specify the effective lengths of the start and stop sections of a storage chamber row.

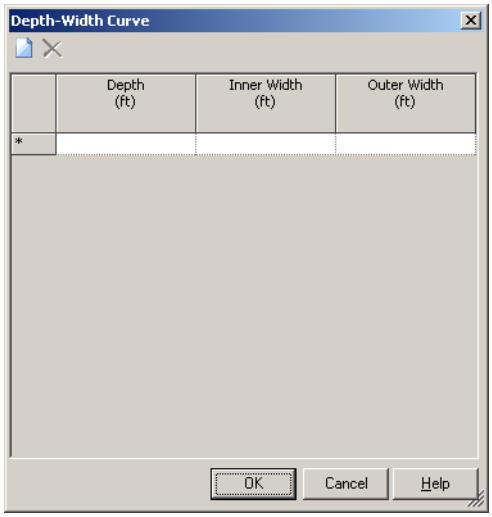
- **Start Section Length**: Specify the length of the storage chamber at the start of the storage chamber row.
- **End Section Length**: Specify the length of the storage chamber at the end of the storage chamber row.
- **Maximum Width**: Set the maximum span that occurs within the structure. This value is used for setting the spacing in the storage chamber system or pond element. It is not used for computing volumes.
- **Depth-Volume Per Unit Length Curve**: Opens the Depth-Volume Per Unit Curve dialog, allowing you to describe how the volume per unit length varies with depth.

- **Notes Tab**: This text field that allows you to enter descriptive notes that will be associated with the currently highlighted list pane entry.
- **Library Tab**: This tab displays information about the template that is currently highlighted in the list pane. If the template is derived from an engineering library, the synchronization details can be found here. If the template was created manually for this project, the synchronization details will display the message Orphan (local), indicating that the template was not derived from a library entry.

For more information about Storage Chambers, see Storage Chambers.

Depth-Width Curve Dialog Box

This dialog allows you to describe how the inner and outer width of the storage chamber vary with depth.

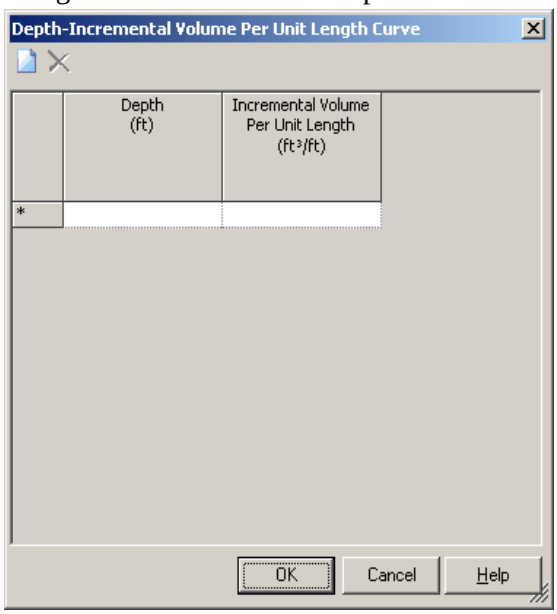


Click the New button to add a row. Click the Delete button to remove the currently highlighted row. Enter Depth vs Inner/Outer Width points to describe how the width of the storage chamber varies with depth.

For more information about Depth-Width Curves, see Depth-Width Curve Type.

Depth-Incremental Volume Per Unit Length Curve Dialog Box

This dialog allows you to describe how the incremental volume per unit length of the storage chamber varies with depth.



Click the New button to add a row. Click the Delete button to remove the currently highlighted row. Enter Depth vs Incremental Volume points to describe how the incremental volume per unit length of the storage chamber varies with depth.

Using ModelBuilder to Transfer Existing Data

5

ModelBuilder lets you use your existing GIS asset to construct a new WaterCAD V8*i* model or update an existing WaterCAD V8*i* model. ModelBuilder supports a wide variety of data formats, from simple databases (such as Access and DBase), spread-sheets (such as Excel), GIS data (such as shape files), to high end data stores (such as Oracle, and SQL Server), and more.

Using ModelBuilder, you map the tables and fields contained within your data source to element types and attributes in your WaterCAD V8*i* model. The result is that a WaterCAD V8*i* model is created. ModelBuilder can be used in any of the Bentley WaterCAD V8*i* platforms - Stand-Alone, MicroStation mode, AutoCAD mode, or ArcGIS mode.

Note: ModelBuilder lets you bring a wide range of data into your model. However, some data is better suited to the use of the more specialized WaterCAD V8*i* modules. For instance, LoadBuilder offers many powerful options for incorporating loading data into your model.

ModelBuilder is the first tool you will use when constructing a model from GIS data. The steps that you take at the outset will impact how the rest of the process goes. Take the time now to ensure that this process goes as smoothly and efficiently as possible:

- Preparing to Use ModelBuilder
- Reviewing Your Results

Preparing to Use ModelBuilder

- **Determine the purpose of your model**—Once you establish the purpose of your model, you can start to make decisions about how detailed the model should be.

- **Get familiar with your data**—ModelBuilder supports several data source types, including tabular and geometric. Tabular data sources include spreadsheets, databases, and other data sources without geometric information. Some supported tabular data source types include Microsoft Excel, and Microsoft Access files. Geometric data sources, while also internally organized by tables, include geometric characteristics such as shape type, size, and location. Some supported geometric data source types include the major CAD and GIS file types

  If you obtained your model data from an outside source, you should take the time to get acquainted with it in its native platform. For example, review spatial and attribute data directly in your GIS environment. Do the nodes have coordinate information, and do the pipes have start and stop nodes specified? If not, the best method of specifying network connectivity must be determined.

  Contact those involved in the development of the GIS to learn more about the GIS tables and associated attributes. Find out the purpose of any fields that may be of interest, ensure that data is of an acceptable accuracy, and determine units associated with fields containing numeric data.

  Ideally, there will be one source data table for each WaterCAD V8*i* element type. This isn't always the case, and there are two other possible scenarios:

  Many tables for one element type—In this case, there may be several tables in the datasource corresponding to a single GEMS modeling element, component, or collection. In this case each data source table must be individually mapped to the WaterCAD V8*i* table type, or the tables must be combined into a single table from within its native platform before running ModelBuilder.

  One table containing many element types—In this case, there may be entries that correspond to several WaterCAD V8*i* table types in one datasource table. You should separate these into individual tables before running ModelBuilder. The one case where a single table can work is when the features in the table are ArcGIS subtypes. ModelBuilder handles these subtypes by treating them as separate tables when setting up mappings. See Subtypes for more information.

  Note: If you are working with an ArcGIS data source, note that ModelBuilder can only use geodatabases, geometric networks, and coverages in ArcGIS mode. See ESRI ArcGIS Geodatabase Support for additional information.

- **All mappings should be contained in a single ModelBuilder connection**—ModelBuilder will ensure that data is synchronized into the model in the correct order using this technique. If multiple connections are to be used instead, then the user should run the individual ModelBuilder connections to get the following data synchronization order: Components, Nodes, Pipes, polygon data (if any), Directed Nodes (i.e. node types with a Downstream Pipe field), and finally collection data. If pipes are brought in first it could create node elements which may not be desired and could result in model run errors.

- **Preparing your data**—When using ModelBuilder to get data from your data source into your model, you will be associating rows in your data source to elements in WaterCAD V8*i*. Your data source needs to contain a Key/Label field that can be used to uniquely identify every element in your model. The data source tables should have identifying column labels, or ModelBuilder will interpret the first row of data in the table as the column labels. Be sure data is in a format suited for use in ModelBuilder. Where applicable, use powerful GIS and Database tools to perform Database Joins, Spatial Joins, and Update Joins to get data into the appropriate table, and in the desired format.

  Note: When working with ID fields, the expected model input is the WaterCAD V8*i* ID. After creating these items in your WaterCAD V8*i* model, you can obtain the assigned ID values directly from your WaterCAD V8*i* modeling file. Before synchronizing your model, get these WaterCAD V8*i* IDs into your data source table (e.g., by performing a database join).

- **Preparing your CAD Data**—In previous versions of WaterCAD V8*i*, the Polyline-to-Pipe feature was used to import CAD data into a WaterCAD V8*i* model. In v8, CAD data is imported using ModelBuilder. When using ModelBuilder to import data from your CAD file into your model, you will be associating cells in your CAD drawing with elements in WaterCAD V8*i*.

  Different CAD cells will be recognized as different element types and presented as tables existing in your CAD data source. It is recommended that you natively export your AutoCAD .dwg or MicroStation .dgn files first as a .dxf file, then select this .dxf as the data source in ModelBuilder. Your data source will most likely not contain a Key/Label field that can be used to uniquely identify every element in your model, so ModelBuilder will automatically generate one for you using the default "<label>". This "<label>" field is a combination of an element's cell type label, its shape type, and a numeric ID that represents the order in which it was created.

- **Build first, Synchronize later**—ModelBuilder allows you to construct a new model or synchronize to an existing model. This gives you the ability to develop your model in multiple passes. On the first pass, use a simple connection to build your model. Then, on a subsequent pass, use a connection to load additional data into your model, such as supporting pattern or collection data.

Note: Upon completion of your ModelBuilder run, it is suggested you use the Network Navigator to identify any connectivity or topological problems in your new model. For instance, Pipe Split Candidates can be identified and then automatically modified with the Batch Split Pipe Tool (see Batch Pipe Split Dialog Box). See Using the Network Navigator for more information.

- **Going Beyond ModelBuilder**—Keep in mind that there are additional ways to get data into your model. ModelBuilder can import loads if you have already assigned a load to each node. If, however, this information is not available from the GIS data, or if your loading data is in a format unrecognized by ModelBuilder (meter data, etc.), use LoadBuilder; this module is a specialized tool for getting this data into your model. In addition, with its open database format, WaterCAD V8*i* gives you unprecedented access to your modeling data.

  One area of difficulty in building a model from external data sources is the fact that unless the source was created solely to support modeling, it most likely contains much more detailed information than is needed for modeling. This is especially true with regard to the number of piping elements. It is not uncommon for the data sources to include every service line and hydrant lateral. Such information is not needed for most modeling applications and should be removed to improve model run time, reduce file size, and save costs.

- **Importing Collections**—When you are importing a collection, values will always override existing collection items in the model. In order to preserve existing items, they need to be combined with the new values and import them together.

  For example importing "Junction, Demand Collection", incoming demand rows will override the existing demand collection, not append to it.

  If you want to keep the existing demands, you should first export those values (copy-paste is usually easiest) to your data source (e.g. spreadsheet, shapefile) and make those demands part of the data you are importing. In this way ModelBuilder will import both the original and new demands.

ModelBuilder Connections Manager

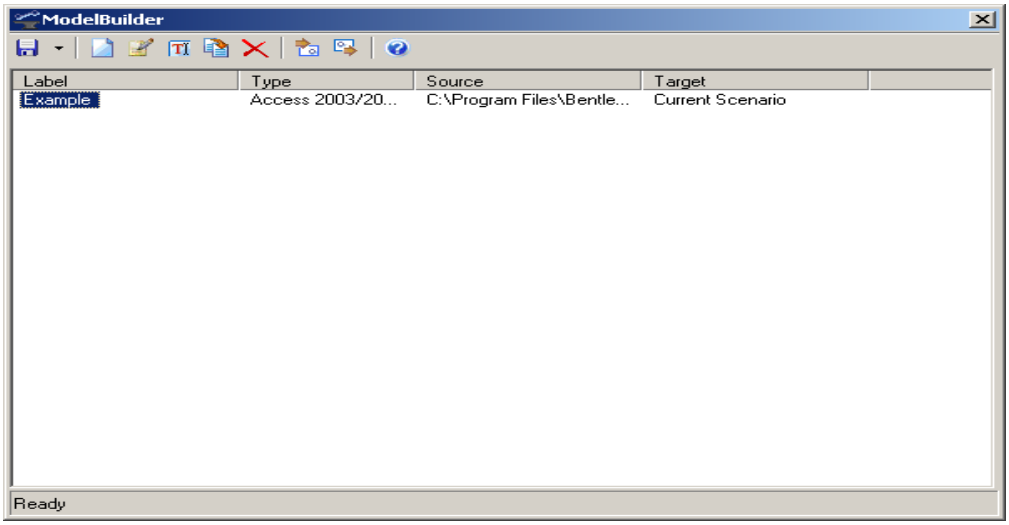
ModelBuilder can be used in any of the Bentley WaterCAD V8*i* platforms - Stand-Alone, MicroStation mode, AutoCAD mode, or ArcGIS mode.

To access ModelBuilder: Click the Tools menu and select the ModelBuilder command, or click the ModelBuilder button  .

The ModelBuilder Connections manager allows you to create, edit, and manage ModelBuilder connections to be used in the model-building/model-synchronizing process. Each item in this manager represents a "connection" which contains the set of directions for moving data between a source to a target. ModelBuilder connections are not stored in a particular project, but are stored in an external xml file, with the following path:

Windows XP: C:\Documents and Settings\<username>\Application Data\Bentley\<productname>\<productversion>\ModelBuilder.xml

Windows Vista: C:\Users\<username>\AppData\Roaming\Bentley\<product-name>\<productversion>\ModelBuilder.xml



At the center of this window is the **Connections List** which displays the list of connections that you have defined.

There is a **toolbar** located along the top of the Connections list.

The set of buttons on the left of the toolbar allow you to manage your connections:

	**Import/Export**	Click this button to import or export a ModelBuilder Connection file (.mbc).
	**New**	Create a new connection using the ModelBuilder Wizard.
	**Edit**	Edit the selected connection using the ModelBuilder Wizard.
	**Rename**	Rename the selected connection.
	**Duplicate**	Create a copy of the selected connection.

	**Delete**	Permanently Remove the selected connection.
	**Build Model**	Starts the ModelBuilder build process using the selected connection. This is also referred to as "synching in" from an external data source to a model. Excluding some spatial option overrides, a build operation will update your model with new elements, components, and collections that already exist in the model. Only table types and fields that are mapped will be updated. Depending upon the configuration of synchronization options in the selected connection, if an element in your data source does not already exist in your model, it may be created. If the element exists, only the fields mapped for that table type may be updated. ModelBuilder will not override element properties not specifically associated with the defined field mappings. A Build Model operation will update existing or newly created element values for the current scenario/ alternative, or you can optionally create new child scenario/alternatives to capture any data difference.
	**Sync Out**	Starts the ModelBuilder synchronize process using the selected connection. Unless specifically overridden, a Sync Out operation will only work for existing and new elements. On a Sync Out every element in your target data source that also exists in your model will be refreshed with the current model values. If your model contains elements that aren't contained in your data source, those data rows can optionally be added to your target data file. Only those properties specified with field mappings will be synchronized out to the data source. A Sync Out operation will refresh element properties in the data source with the current model values for the current scenario/alternative.
	**Help**	Displays online help.

After initiating a Build or Sync command, ModelBuilder will perform the selected operation. During the process, a progress-bar will be displayed indicating the step that ModelBuilder is currently working on.

When ModelBuilder completes, you will be presented with a summary window that outlines important information about the build process. We recommend that you save this summary so that you can refer to it later.

Note: Because the connections are stored in a separate xml file rather than with the project file, ModelBuilder connections are preserved even after Bentley WaterCAD V8*i* is closed.

Specify Datasource Location

This dialog allows you to specify the datasource associated with the ModelBuilder connection that is currently highlighted in the ModelBuilder connections manager. Click the Browse button and select the datasource file.

Microsoft Access Database Engine Version

The 64 bit version of this Bentley software requires the "64-bit Access Database Engine" (not included with this Bentley software) to be able to support newer MSOffice file formats which can be used in ModelBuilder and SCADAConnect. If you do not have a compatible version of the Access Database Engine installed and wish to connect to these data sources, either download and install the 64-bit Access Database Engine from Microsoft using the following link: **http://www.microsoft.com/en-us/download/details.aspx?id=13255** or alternatively, use the 32 bit version of the software, which can be accessed from **C:\Program Files (x86)\Bentley\.exe**, which supports these formats without requiring additional components.

ModelBuilder Wizard

The ModelBuilder Wizard assists in the creation of ModelBuilder connections. The Wizard will guide you through the process of selecting your data source and mapping that data to the desired input of your model.

Tip: The ModelBuilder Wizard can be resized, making it easier to preview tables in your data source. In addition, Step 1 and Step 3 of the wizard offer a vertical split bar, letting you adjust the size of the list located on the left side of these pages.

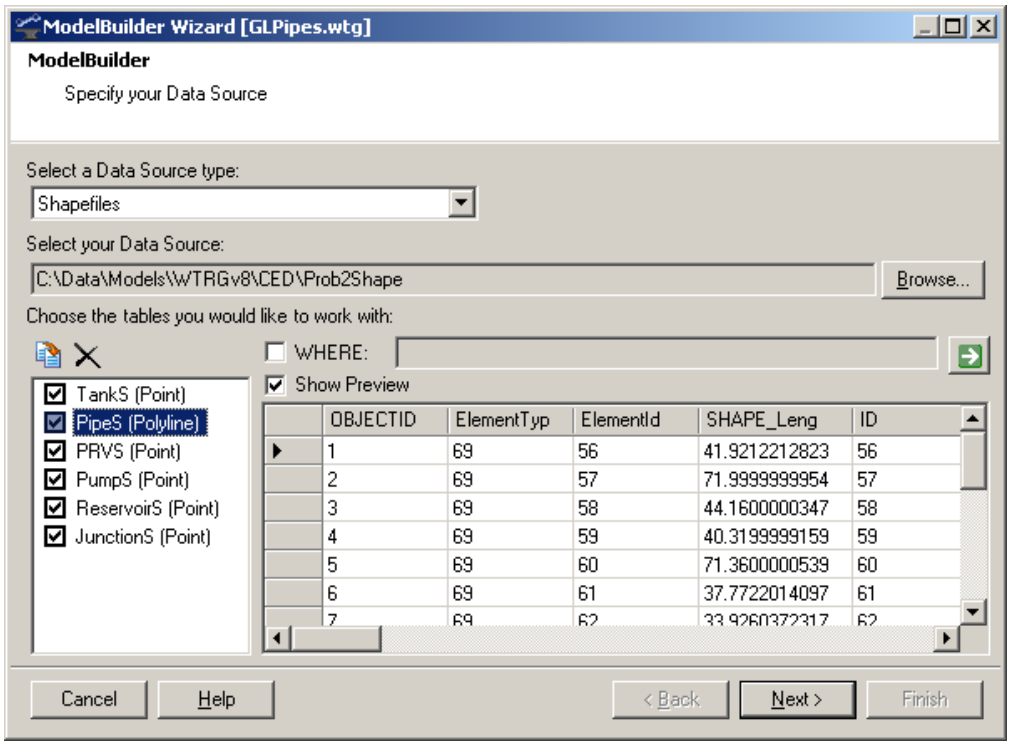
There are 6 steps involved:

- Step 1—Specify Data Source

- Step 2—Specify Spatial Options
- Step 3 - Specify Element Create/Remove/Update Options
- Step 4—Additional Options
- Step 5—Specify Field mappings for each Table/Feature Class
- Step 6—Build operation Confirmation

Step 1—Specify Data Source

In this step, the data source type and location are specified. After selecting your data source, the desired database tables can be chosen and previewed.



The following fields are available:

- **Data Source** type (drop-down list)—This field allows you to specify the type of data you would like to work with.

  Note: If your specific data source type is not listed in the Data Source type field, try using the OLE DB data source type. OLE DB can be used to access many database systems (including ORACLE, and SQL Server, to name a few).

- **Data Source** (text field)—This read-only field displays the path to your data source.

- **Browse** (button)—This button opens a browse dialog box that allows you to interactively select your data source.

  Note: Some Data Source types expect you to choose more than one item in the Browse dialog box. For more information, see Multi-select Data Source Types**.**

- **Table/Feature Class** (list)—This pane is located along the left side of the form and lists the tables/feature classes that are contained within the data source. Use the check boxes (along the left side of the list) to specify the tables you would like to include.

  Tip: The list can be resized using the split bar (located on the right side of the list).

  Right-click to Select All or Clear the current selection in the list.

  ModelBuilder has built in support for ArcGIS Subtypes. For more information, see ESRI ArcGIS Geodatabase Support**.**

- **Duplicate Table** (button)  —The duplicate table button is located along the top of the Table/Feature Class list. This button allows you to make copies of a table, which can each be mapped to a different element type in your model. Use this in conjunction with the WHERE clause.

- **Remove Table** (button)  —The remove table button can be used to remove a table from the list.

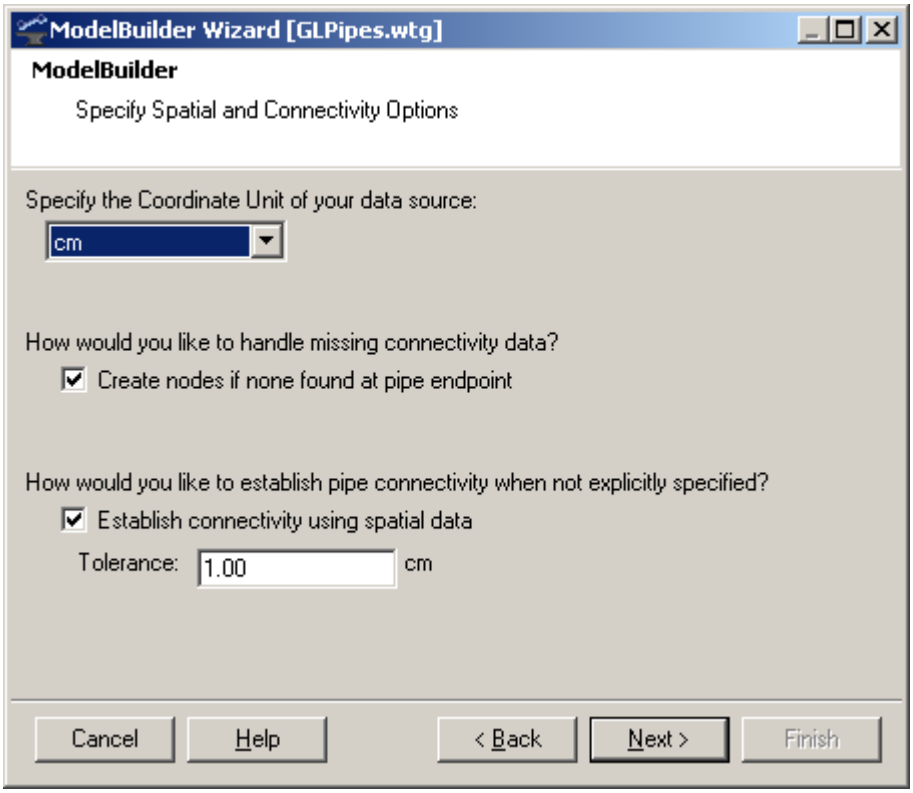
- **WHERE Clause** (field)—Allows you to create a SQL query to filter the tables. When the box is checked, only tables that meet the criteria specified by the WHERE clause will be displayed. Click the  button to validate the query and to refresh the preview table.

- **Preview Pane**—A tabular preview of the highlighted table is displayed in this pane when the **Show Preview** check box is enabled.

Note: If both nodes and pipes are imported in the same ModelBuilder connection, nodes will be imported first regardless of the order they are listed here.

Step 2—Specify Spatial Options

In this step you will specify the spatial options to be used during the ModelBuilder process. The spatial options will determine the placement and connectivity of the model elements. The fields available in this step will vary depending on the data source type.



- **Specify the Coordinate Unit of your data source** (drop-down list)—This field allows you to specify the coordinate unit of the spatial data in your data source. The default unit is the unit used for coordinates.

- **Create nodes if none found at pipe endpoint** (check box)—When this box is checked, ModelBuilder will create a pressure junction at any pipe endpoint that: a) doesn't have a connected node, and b) is not within the specified tolerance of an existing node. This field is only active when the Establish connectivity using spatial data box is checked. (This option is not available if the connection is bringing in only point type geometric data.)

  ModelBuilder will not create pipes unless a valid start/stop node exists. Choose this option if you know that there are nodes missing from your source data. If you expect your data to be complete, then leave this option off and if this situation is detected ModelBuilder will report errors for your review. For more information see Specifying Network Connectivity in ModelBuilder.

- **Establish connectivity using spatial data** (check box)—When this box is checked, ModelBuilder will connect pipes to nodes that fall within a specified tolerance of a pipe endpoint. (This option is available if the connection is bringing in only polyline type geometric data.) Use this option, when the data source does not explicitly name the nodes at the end of each pipe. For more information, see Specifying Network Connectivity in ModelBuilder.

- **Tolerance** (numeric field)—This field dictates how close a node must be to a pipe endpoint in order for connectivity to be established. The Tolerance field is only available when the Establish connectivity using spatial data box is checked. (This option is available if the connection is bringing in only polyline type geometric data.) Tolerances should be set as low as possible so that unintended connections are not made. If you are not sure what tolerance to use, try doing some test runs. Use the Network Review queries to evaluate the success of each trial import.

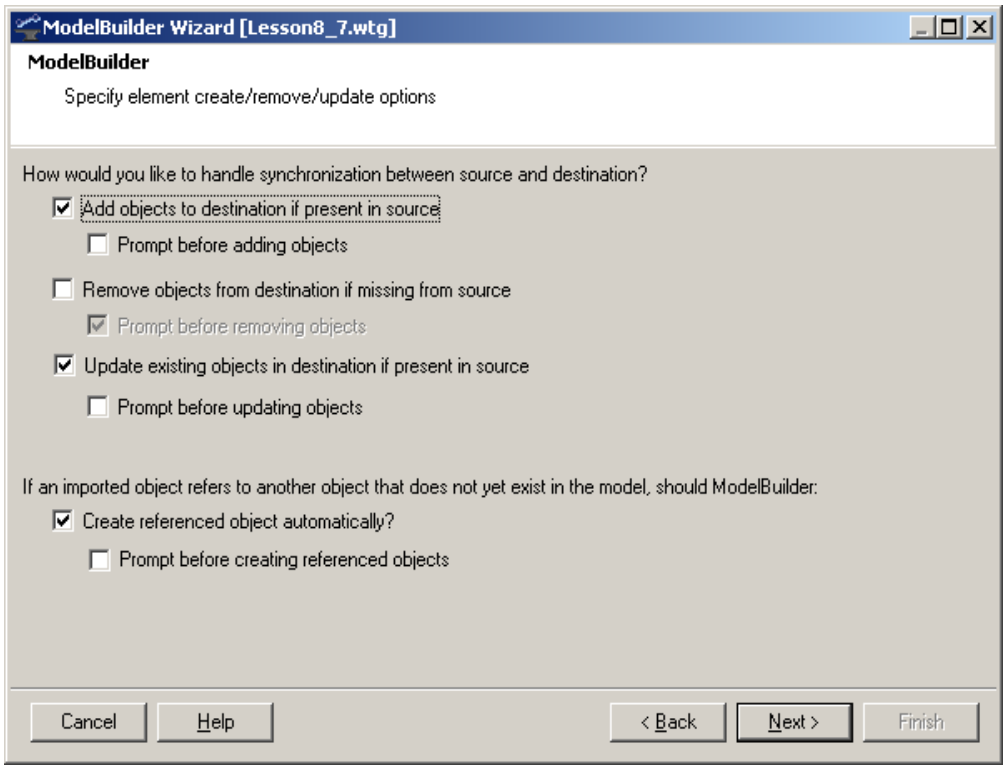
  Note: Pipes will be connected to the closest node within the specified tolerance.

  The unit associated with the tolerance is dictated by the Specify the Coordinate Unit of your data source field.

  For more information, see Specifying Network Connectivity in ModelBuilder.

Step 3 - Specify Element Create/Remove/Update Options

Because of the variety of different data sources and they way those sources were created, the user has a wide variety of options to control the behavior of ModelBuilder.



How would you like to handle synchronization between source and destination?:

- **Add objects to destination if present in source** (check box)-When this box is checked, ModelBuilder will automatically add new elements to the model for "new" records in the data source when synching in (or vice-versa when synching out).

  This is checked by default since a user generally wants to add elements to the model (especially if this is the initial run of ModelBuilder). This should be unchecked if new elements have been added to the source file since the model was created but the user does not want them in the model (e.g. proposed piping).

  – **Prompt before adding objects** (check box)-When this box is checked, ModelBuilder will pause during the synchronization process to present a confirmation message box to the user each time an element is about to be created in the model or data-source.

- **Remove objects from destination if missing from source** (check box)-When this box is checked, ModelBuilder will delete elements from the model if they do not exist in the data source when synching in (or vice-versa when synching out). This option can be useful if you are importing a subset of elements.

  This is used if abandoned pipes have been deleted from the source file and the user wants them to automatically be removed from the model by ModelBuilder.

  – **Prompt before removing objects** (check box)-When this box is checked, ModelBuilder will pause during the synchronization process to present a confirmation message box to the user each time an element is about to be deleted from the model.

- **Update existing objects in destination if present in source** (check box) - If checked, this option allows you to control whether or not properties and geometry of existing model elements will be updated when synching in (or vice-versa when synching out). Turning this option off can be useful if you want to synchronize newly added or removed elements, while leaving existing elements untouched.

  – **Prompt before updating objects** (check box)-When this box is checked, ModelBuilder will pause during the synchronization process to present a confirmation message box to the user each time an element is about to be updated.

If an imported object refers to another object that does not yet exist in the model, should ModelBuilder:

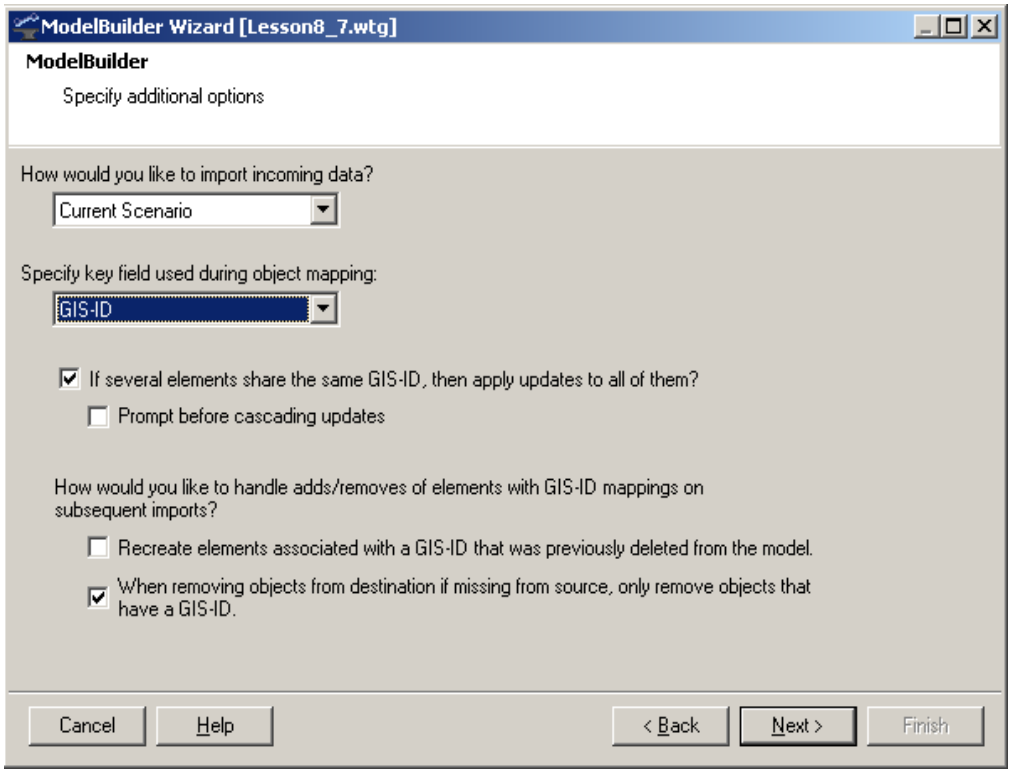
- **Create referenced element automatically?** (check box)-When this box is checked, ModelBuilder will create any domain and/or support elements that are referenced during the import process.

  – **Prompt before creating referenced elements** (check box)-When this box is checked, ModelBuilder will pause during model generation to present a confirmation message box to the user each time a specified referenced element could not be found, and is about to be created for the model.

  "Referenced elements" refers to any support or domain element that is referenced by another element. For example, Pumps can refer to Pump Definition support-elements, Junctions can refer to Zone support-elements, and Pumps can refer to a downstream Pipe domain-element. Node domain-elements that get created as a result of being referenced during the ModelBuilder process will use a default coordinate of 0, 0.

Note: These options listed above apply to elements (pipes and nodes) as well as support elements (such as Zones or Controls).

Step 4—Additional Options



- **How would you like to import incoming data?** (drop-down list) - This refers to the scenario (and associated alternatives) into which the data will be imported. The user can import the data into the Current Scenario or a new child scenario. If the latter is selected, a new child scenario (and child alternatives) will be created for any data difference between the source and the active scenario.

Note: **If there is no data change for a particular alternative, no child alternative will be created in that case.**

New scenario and alternatives will be automatically labeled "Created by ModelBuilder" followed by the date and time when they were created.

- **Specify key field used during object mapping** (drop-down list) - The key field represents the field in the model and data source that contains the unique identifier for associating elements in your model to records in your data source. Refer to the "Key Field (Model)" topic in the next section for additional guidance on how this setting applies to ModelBuilder. ModelBuilder provides three choices for Key Field:
  - **Label** - The element "Label" will be used as the key for associating model elements with data source records. Label is a good choice if the identifier field in your data-source is unique and represents the identifier you commonly use to refer to the record in your GIS.
  - **<custom>** - Any editable text field in your model can be used as the key for associating model elements with data source records. This is a good choice if you perhaps don't use labels on every element, or if perhaps there are duplicate labels in your data source.
  - **GIS-ID** - The element "GIS-ID" field will be used as the key for associating model elements with data source elements. The GIS-ID field offers a number of advanced capabilities, and is the preferred choice for models that you plan to keep in sync with your GIS over a period of time.

    Refer to the section The GIS-ID Property for more information.

The following options only apply when using the advanced GIS-ID key field option.

- **If several elements share the same GIS-ID, then apply updates to all of them?** (check box) - When using the GIS-ID option, ModelBuilder allows you to maintain one-to-many, and many-to-one relationships between records in your GIS and elements in your Model.

  For example, you may have a single pipe in your GIS that you want to maintain as multiple elements in your Model because you have split that pipe into two pipes elements in the model. You may accomplish this using the native WaterCAD V8*i* layout tools to split the pipe with a node; the newly created pipe segment will be assigned the same GIS-ID as the original pipe (establishing a one-to-many relationship). By using this option, when you later synchronize from the GIS into your model, any data changes to the single pipe record in your GIS can be cascaded to both pipes elements in your model (e.g. so a diameter change to a single record in the GIS would be reflected in both elements in the model).
  - **Prompt before cascading updates** (check box) - When this box is checked, ModelBuilder will pause during model generation to present a confirmation message box to the user each time a cascading update is about to be applied.

- **How would you like to handle add/removes of elements with GIS-ID mappings on subsequent imports**? - These options are useful for keeping your GIS and Model synchronized, while maintaining established differences.

  – **Recreate elements associated with a GIS-ID that was previously deleted from the model** (check box) - By default, ModelBuilder will not recreate elements you remove from your model that are associated with a records (with GIS-ID mappings) that are still in your GIS. This behavior is useful when you want to perform GIS to model synchronizations, but have elements that exist in your GIS that you do not want in your model.

    For example, after creating your model from GIS, you may find redundant nodes when performing a Network Navigator, "Nodes in Close Proximity" network review query. You may choose to use the "Merge Nodes in Close Proximity" feature to make the correction in your model (deleting the redundant nodes from your model). Normally, when you later synchronize from your GIS to your model, missing elements would be recreated and your correction would be lost. However, WaterCAD V8*i* now maintains the history of elements (with GIS-ID's) that were removed from your model; this option allows you to control whether or not those elements get recreated.

  – **When removing objects from destination if missing from source, only remove objects that have a GIS-ID.** (check box) - This option is useful when you have elements that are missing from your GIS that you want to keep in your model (or vice-versa).

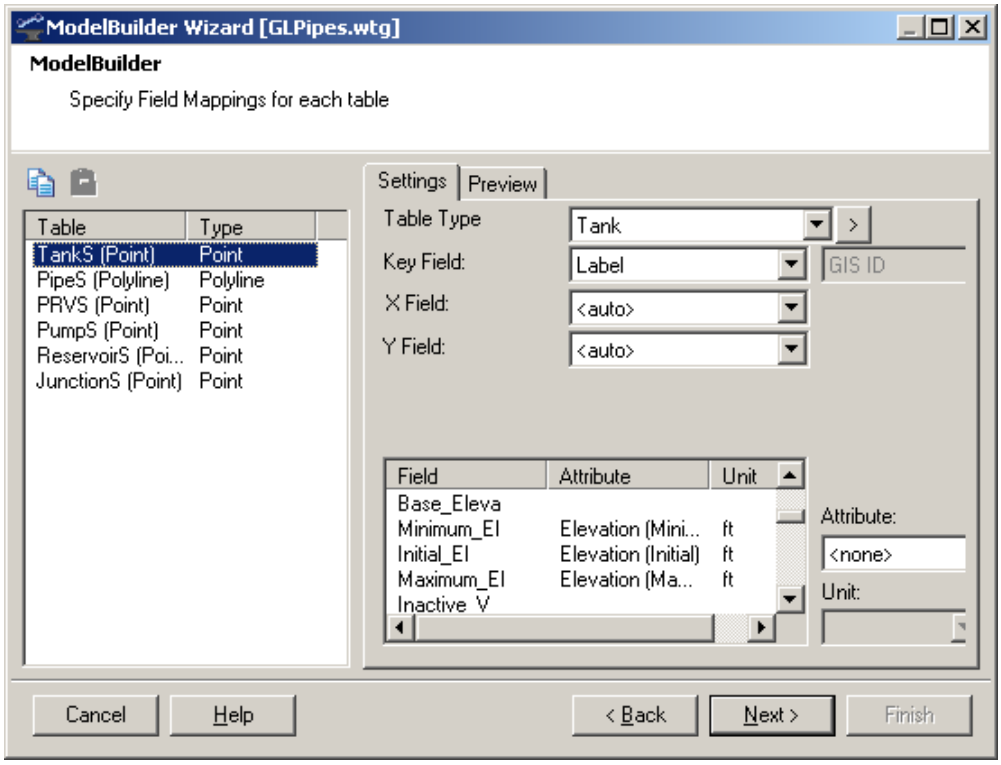
    For example, if you build your model from your GIS (using the GIS-ID option, a GIS-ID will be assigned to newly created elements in your model. If you later add elements to your model (they will not be assigned a GIS-ID); on subsequent synchronizations, this option (if checked) will allow you to you retain those model specific elements that do not exist in your GIS. For example, you may have a proposed land development project in your model that does not exist in the GIS. These elements will not have a GIS-ID because they were not imported from the GIS. If this box is checked, the new elements will not be removed on subsequent runs of ModelBuilder.

Note: This setting only applies if the "Remove objects from destination if missing from source" option is checked.

When you do make connectivity changes to your model, it is often beneficial to make those same changes to the GIS. However, this is not always possible; and in some cases is not desirable -- given the fact that Modeling often has highly specialized needs that may not be met by a general purpose GIS.

Step 5—Specify Field mappings for each Table/Feature Class

In this step, data source tables are mapped to the desired modeling element types, and data source fields are mapped to the desired model input properties. You will assign mappings for each Table/Feature Class that appears in the list; Step 1 of the wizard can be used to exclude tables, if you wish.



- **Tables** (list)-This pane, located along the left side of the dialog box, lists the data source Tables/Feature Classes to be used in the ModelBuilder process. Select an item in the list to specify the settings for that item.

  Note: The tables list can be resized using the splitter bar.

  There are two toolbar buttons located directly above Tables list (these buttons can be a great time saver when setting up multiple mappings with similar settings).

– **Copy Mappings** (button)-This button copies the mappings (associated with the currently selected table) to the clipboard.

– **Paste Mappings** (button)-This button applies the copied mappings to the currently selected table.

• **Settings Tab**-The Settings tab allows you to specify mappings for the selected item in the Tables list.

The top section of the Settings tab allows you to specify the common data mappings:

– **Table Type** (drop-down list)-This field, which contains a list of all of the WaterCAD V8*i*/Hammer element types, allows you to specify the target modeling element type that the source table/feature class represents. For example, a source table that contains pipe data should be associated with the Pressure Pipe element type.

There are three categories of Table Types: Element Types, Components, and Collections. For geometric data sources, only Element Types are available. However with tabular data sources all table types can be used. The categorized menu accessed by the [>] button assists in quicker selection of the desired table type.

- **Element Types**-This category of Table Type includes geometric elements represented in the drawing view such as pipes, junctions, tanks, etc.

- **Components**-This category of Table Type includes the supporting data items in your model that are potentially shared among elements such as patterns, pump definitions, and controls.

- **Collections**-This category of Table Type includes table types that are typically lists of 2-columned data. For instance, if one table in your connection consists of a list of (Time From Start, Multiplier) pairs, use a Pattern collection table type selection.

– **Key Fields** - This pair of key fields allows you to control how records in your data source are associated with elements in the model. The Key Fields element mapping consists of two parts, a data-source part and a model part:

- **Key Field (Data Source)** (drop-down list)-Choose the field in your data source that contains the unique identifier for each record.

Note: If you plan to maintain synchronizations between your model and GIS, it is best to define a unique identifier in your data source for this purpose. Using an identifier that is unique across all tables is critical if you wish to maintain explicit pipe start/stop connectivity identifiers in your GIS.

When working with ArcGIS data sources, OBJECTID is not a good choice for Key field (because OBJECTID is only unique for a particular Feature Class).

For one-time model builds -- if you do not have a field that can be used to uniquely identify each element -- you may use the <label> field (which is automatically generated by ModelBuilder for this purpose).

- **Key Field (Model)** (drop-down-list) - This field is only enabled if you specified <custom> in the "Specify key field to be used in object mapping?" option in the previous step. If you specified "GIS-ID' or "Label" the field will be disabled.

  If you specified <custom>, then you will be presented with a list of the available text fields for that element type. Choose a field that represents the unique alphanumeric identifier for each element in your model.

Note: You can define a text User Data Extensions property for use as your <custom> model key field.

The <custom> key field list is limited to read-write text fields. This is because during import, the value of this field will be assigned as new elements in your model are created. Therefore, the models internal (read-only) element ID field cannot be used for this purpose.

The following optional fields are available for Pipe element types:

- **Start/Stop** - Select the fields in a pipe table that contain the identifier of the start and stop nodes. Specify <none> if you are using the spatial connectivity support in ModelBuilder (or if you want to keep connectivity unchanged on update). For more information, see Specifying Network Connectivity in ModelBuilder.

Note: When working with an ArcGIS Geometric Network data source, these fields will be set to <auto> (indicating that ModelBuilder will automatically determine connectivity from the geometric network).

These fields are available for Node element types:

- **X/Y Field** - These fields are used to specify the node X and Y coordinate data. This field only applies to point table types.

Note: The Coordinate Unit setting in Step 2 of the wizard allows you to specify the units associated with these fields.

When working with ArcGIS Geodatabase, shape file and CAD data sources, these fields will be set to <auto> (indicating that ModelBuilder will automatically determine node geometry from the data source).

These optional fields are available for Pump element types:

- **Suction Element** (drop-down list)-For tables that define pump data, select a pipe label or other unique identifier to set the suction element of the Pump.
- **Downstream Edge** (drop-down list)-For tables that define pump or valve data, select a pipe label or other unique identifier to set the direction of the pump or valve.

The bottom section of the Settings tab allows you to specify additional data mappings for each field in the source.

- **Field** - Field refers to a field in the selected data source. The Field list displays the associations between fields in the database to properties in the model.
- **Property** (drop-down list)-Property refers to a Bentley WaterCAD V8*i* property. Use the Property drop-down list to map the highlighted field to the desired property.
- **Unit** (drop-down list)-This field allows you to specify the units of the values in the database (no conversion on your part is required). This field only applies if the selected model property is unitized.

- **Preview Tab**-The Preview tab displays a tabular preview of the currently highlighted source data table when the Show Preview check box is checked.

To map a field in your table to a particular Bentley WaterCAD V8*i* property:

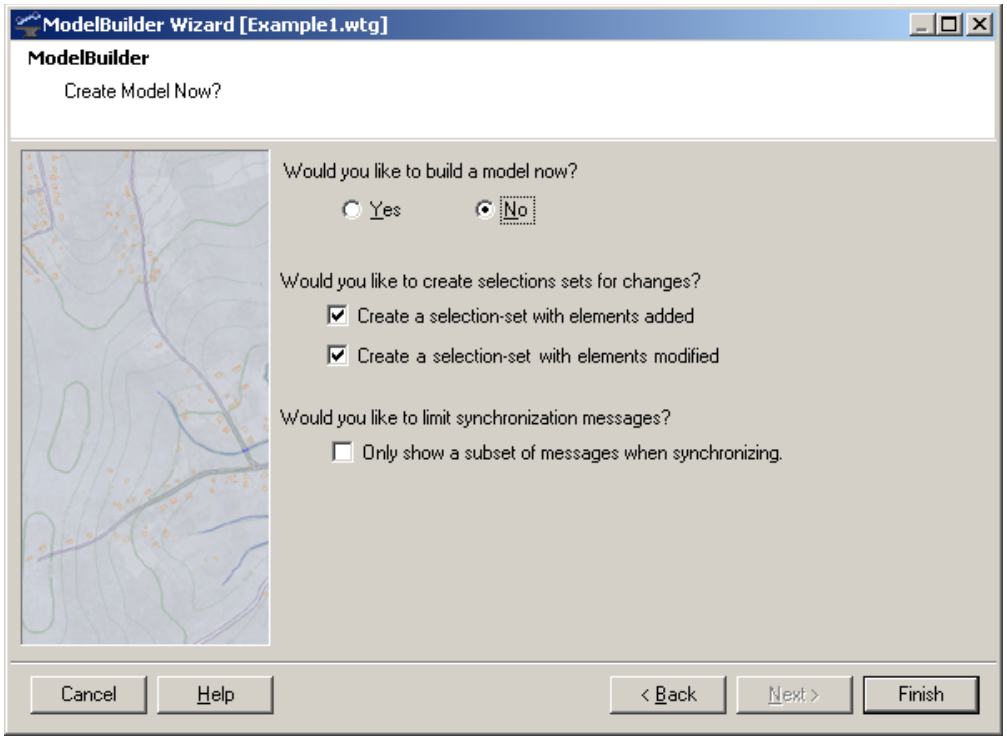
1. In the **Field** list, select the data source field you would like to define a mapping for.
2. In the **Property** drop-down list, select the desired Bentley WaterCAD V8*i* target model property.
3. If the property is unitized, specify the unit of this field in your data source in the **Unit** drop-down list.

To remove the mapping for a particular field:

1. Select the field you would like to update.
2. In the **Property** drop-down list, select **<none>**.

Step 6—Build operation Confirmation

In this step, you are prompted to build a new model or update an existing model.



To build a new model, click the **Yes** radio button under **Would you like to build the model now?**.

If you choose **No**, you will be returned to the **ModelBuilder Manager** dialog. The connection you defined will appear in the list pane. To build the model from the ModelBuilder Manager, highlight the connection and click the **Build Model** button.

Create Selection Set options: Often a user wants to view the elements that have been affected by a ModelBuilder operation. To do this, ModelBuilder can create selection sets which the user can view and use within the application.

- To create a selection set containing the elements added during the ModelBuilder, check the box next to "**Create selection set with elements added**."
- To create a selection set containing the elements for which the properties or geometry were modified during the ModelBuilder, check the box next to "**Create selection set with elements modified**."

Only show a subset of messages when synchronizing: Depending on the Model-Builder configuration and the external data, there are situations when a very large number of messages may be generated during the ModelBuilder synchronization. Generating these messages adds some overhead and can use up a large amount of memory. Checking this box will limit the number of messages that are generated for each specific message type.

Note: Selection sets created as a result of these options will include the word "ModelBuilder" in their name, along with the date and time (e.g. "Elements added via ModelBuilder - mm/dd/yyyy hh:mm:ss am/pm")

Reviewing Your Results

At the end of the model building process, you will be presented with statistics, and a list of any warning/error messages reported during the process. You should closely review this information, and be sure to save this data to disk where you can refer to it later.

Note: Refer to the section titled ModelBuilder Warnings and Error Messages **to determine the nature of any messages that were reported.**

Refer to the Using the Network Navigator and Manipulating Elements topics for information about reviewing and correcting model connectivity issues.

Multi-select Data Source Types

When certain Data Source types are chosen in Step 1 of the ModelBuilder Wizard (see Step 1—Specify Data Source), multiple items can be selected for inclusion in your ModelBuilder connection.

After clicking the **Browse** button to interactively specify your data source, use standard Windows selection techniques to select all items you would like to include in the connection (e.g., Ctrl+click each item you would like to include).

The following are multi-select Data Source types:

- ArcGIS Geodatabase Features
- Shape files
- DBase and HTML Export.

ModelBuilder Warnings and Error Messages

Errors and warnings that are encountered during the ModelBuilder process will be reported in the ModelBuilder Summary.

For more information, see:

- ModelBuilder Warnings
- ModelBuilder Error Messages

ModelBuilder Warnings

Warning messages include:

1. *Some rows were ignored due to missing key-field values.*

   ModelBuilder encountered missing data (e.g., null or blank) in the specified Key/Label field for rows in your data source table. Without a key, ModelBuilder is unable to associate this source row with a target element, and must skip these items. This can commonly occur when using a spreadsheet data source. To determine where and how often this error occurred, check the Statistics page for the message **<x> row(s) ignored due to missing key-field values**.

2. *Unable to create pipe <element>; start and/or stop node could not be found.*

   Pipes can only be created if its start and stop nodes can be established. If you are using Explicit connectivity, a node element with the referenced start or stop label could not be found. If you are using implicit connectivity, a node element could not be located within the specified tolerance. For more information, see Specifying Network Connectivity in ModelBuilder.

3. *Unable to update pipe <element> topology; (start or stop) node could not be found.*

   This error occurs when synchronizing an existing model, and indicates that the pipe connectivity could not be updated. For more information, see warning message #2 (above).

4. *The downstream edge for <element> could not be found.*

   ModelBuilder was unable to set a Pump direction because a pipe with the referenced label could not be found.

5. *Directed Node <element> direction is ambiguous.*

   ModelBuilder was unable to set the direction of the referenced pump or valve because direction could not be implied based on the adjacent pipes (e.g. there should be one incoming and one outgoing pipe).

ModelBuilder Error Messages

Note: If you encounter these errors or warnings, we recommend that you correct the problems in your original data source and re-run ModelBuilder (when applicable).

Error messages include:

1. *Unable to assign <attribute> for element <element>.*

   Be sure that the data in your source table is compatible with the expected WaterCAD V8*i* format. For more information, see Preparing to Use ModelBuilder.

2. *Unable to create <element type> <element>.*

   This message indicates that an unexpected error occurred when attempting to create a node element.

3. *Unable to create pipe <element> possibly due to start or stop connectivity constraints.*

   This message indicates that this pipe could not be created, because the pump or valve already has an incoming and outgoing pipe. Adding a third pipe to a pump or valve is not allowed.

4. *Unable to update pipe <element> topology; possibly due to start element connectivity constraints.*

   This error occurs when synchronizing. For more information, see error message #3 (above).

5. *Operation terminated by user.*

   You pressed the Cancel button during the ModelBuilder process.

6. *Unable to create < element>; pipe start and stop must be different.*

   This message indicates that the start and stop specified for this pipe refer to the same node element.

7. *Unable to update <element> topology; pipe start and stop must be different.*

   This message indicates that the start and stop specified for this pipe refer to the same node element.

8. *Unable to update the downstream edge for <element>.*

   An unexpected error occurred attempting to set the downstream edge for this pump or valve.

9. *Nothing to do. Some previously referenced tables may be missing from your data source.*

This data source has changed since this connection was created. Verify that tables/feature-classes in your data source have not been renamed or deleted.

10. *One or more input features fall outside of the XYDomain.*

    This error occurs when model elements have been imported into a new geodatabase that has a different spatial reference from the elements being created. Elements cannot be created in ArcMAP if they are outside the spatial bounds of the geodatabase.

    The solution is to assign the correct X/Y Domain to the new geodatabase when it is being created:

1. In the Attach Geodatabase dialog that appears after you initialize the Create New Project command, click the Change button.
2. In the Spatial Reference Properties dialog that appears, click the Import button.
3. Browse to the datasource you will be using in ModelBuilder and click Add.
4. Back in the Spatial Reference Properties dialog, click the X/Y Domain tab. The settings should match those of the datasource.
5. Use ModelBuilder to create the model from the datasource.

ESRI ArcGIS Geodatabase Support

ModelBuilder was built using ArcObjects, and supports the following ESRI ArcGIS Geodatabase functionality. See your ArcGIS documentation for more information about ArcObjects. For more information, see:

- Geodatabase Features
- Geometric Networks
- ArcGIS Geodatabase Features versus ArcGIS Geometric Network
- Subtypes
- SDE (Spatial Database Engine)

Geodatabase Features

ModelBuilder provides direct support for working with Geodatabase features. A feature class is much like a shapefile, but with added functionality (such as subtypes).

The geodatabase stores objects. These objects may represent nonspatial real-world entities, such as manufacturers, or they may represent spatial objects, such as pipes in a network. Objects in the geodatabase are stored in feature classes (spatial) and tables (nonspatial).

The objects stored in a feature class or table can be organized into subtypes and may have a set of validation rules associated with them. The ArcInfo™ system uses these validation rules to help you maintain a geodatabase that contains valid objects.

Tables and feature classes store objects of the same type—that is, objects that have the same behavior and attributes. For example, a feature class called WaterMains may store pressurized water mains. All water mains have the same behavior and have the attributes ReferenceID, Depth, Material, GroundSurfaceType, Size, and PressureRating.

Geometric Networks

ModelBuilder has support for Geometric Networks, and a new network element type known as Complex Edge. When you specify a Geometric Network data source, ModelBuilder automatically determines the feature classes that make up the network. In addition, ModelBuilder can automatically establish model connectivity based on information in the Geometric Network.

ArcGIS Geodatabase Features versus ArcGIS Geometric Network

Note: See your ArcGIS documentation for more information about Geometric Networks and Complex Edges.

When working with a Geometric Network, you have two options for constructing your model—if your model contains Complex Edges, then there is a distinct difference. A Complex Edge can represent a single feature in the Geodatabase, but multiple elements in the Geometric Network.

For example, when defining your Geometric Network, you can connect a lateral to a main without splitting the main line. In this case, the main line will be represented as a single feature in the Geodatabase but as multiple edges in the Geometric Network.

Depending on the data source type that you choose, ModelBuilder can see either representation. If you want to include every element in your system, choose ArcGIS Geometric Network as your data source type. If you want to leave out laterals and you want your main lines to be represented by single pipes in the model, choose **ArcGIS Geodatabase Features** as your data source type.

Subtypes

Tip: Shapefiles can be converted into Geodatabase Feature Classes if you would like to make use of Subtypes. See your ArcGIS documentation for more information.

If multiple types of WaterCAD V8*i* elements have their data stored in a single geodatabase table, then each element must be a separate ArcGIS subtype. For example, in a valve table PRVs may be subtype 1, PSVs may be subtype 2, FCVs may be subtype 3, and so on. With subtypes, it is not necessary to follow the rule that each GIS/database feature type must be associated with a single type of GEMS model element. Note that the subtype field must be of the integer type (e.g., 1, 2) and not an alphanumeric field (e.g., PRV). For more information about subtypes, see ArcGIS Help.

ModelBuilder has built in support for subtypes. After selecting your data source, feature classes will automatically be categorized by subtype. This gives you the ability to assign mappings at the subtype level. For example, ModelBuilder allows you to exclude a particular subtype within a feature class, or associate each subtype with a different element type.

SDE (Spatial Database Engine)

ModelBuilder lets you specify an SDE Geodatabase as your data source. See your ESRI documentation for more information about SDE.

Specifying Network Connectivity in ModelBuilder

When importing spatial data (ArcGIS Geodatabases or shapefile data contain spatial geometry data that ModelBuilder can use to establish network connectivity by connecting pipe ends to nodes, creating nodes at pipe endpoints if none are found.), ModelBuilder provides two ways to specify network connectivity:

- **Explicit connectivity**—based on pipe **Start** node and **Stop** node (see Step 3 - Specify Element Create/Remove/Update Options).
- **Implicit connectivity**—based on spatial data. When using implicit connectivity, ModelBuilder allows you to specify a **Tolerance**, and provides a second option allowing you to **Create nodes if none found** (see Step 2—Specify Spatial Options).

The method that you use will vary depending on the quality of your data. The possible situations include (in order from best case to worst case):

- You have pipe start and stop information—Explicit connectivity is definitely the preferred option.

- You have some start and stop information—Use a combination of explicit and implicit connectivity (use the **Spatial Data** option, and specify pipe Start/Stop fields). If the start or stop data is missing (blank) for a particular pipe, Model-Builder will then attempt to use spatial data to establish connectivity.
- You do not have start and stop information—Implicit connectivity is your only option. If your spatial data is good, then you should reduce your connectivity **Tolerance** accordingly.
- You do not have start and stop information, and you do not have any node data (e.g., you have GIS data that defines your pipes, but you do not have data for nodes)—Use implicit connectivity and specify the **Create nodes if none found** option; otherwise, the pipes cannot be created.

Note: If pipes do not have explicit Start/Stop nodes and "Establish connectivity using spatial data" is not checked, the pipes will not be connected to the nodes and a valid model will not be produced.

Other considerations include what happens when the coordinates of the pipe ends do not match up with the node coordinates. This problem can be one of a few different varieties:

1. **Both nodes and pipe ends have coordinates, and pipes have explicit Start/Stop nodes**—In this case, the node coordinates are used, and the pipe ends are moved to connect with the nodes.
2. **Nodes have coordinates but pipes do not have explicit Start/Stop nodes**—The nodes will be created, and the specified tolerance will be used to connect pipe ends within this tolerance to the appropriate nodes. If a pipe end does not fall within any node's specified tolerance, a new node can be created using the **Create nodes if none found** option.
3. **Pipe ends have coordinates but there are no junctions**—New nodes must be created using the **Create nodes if none found** option. Pipe ends are then connected using the tolerance that is specified. . Subsequent pipe ends could then connect to any newly added nodes if they fall within the specified tolerance.

Another situation of interest occurs when two pipes cross but aren't connected. If, at the point where the pipes cross, there are no pipe ends or nodes within the specified tolerance, then the pipes will not be connected in the model. If you intend for the pipes to connect, then pipe ends or junctions must exist within the specified tolerance.

Refer to the Using the Network Navigator and Manipulating Elements topics for information about reviewing and correcting model connectivity issues.

Sample Spreadsheet Data Source

Note: Database formats (such as MS Access) are preferable to simple spreadsheet data sources. The sample below is intended only to illustrate the importance of using expected data formats.

Here are two examples of possible data source tables. The first represents data that is in the correct format for an easy transition into ModelBuilder, with no modification. The second table will require adjustments before all of the data can be used by ModelBuilder.

Table 5-1: Correct Data Format for ModelBuilder

Label	Roughness_C	Diam_in	Length_ft	Material_ID	Subtype
P-1	120	6	120	3	2
P-2	110	8	75	2	1
P-3	130	6	356	2	3
P-4	100	10	729	1	1

Table 5-2: Data Format Needs Editing for ModelBuilder

P-1	120	.5	120	PVC	Phase2
P-2	110	.66	75	DuctIron	Lateral
P-3	130	.5	356	PVC	Phase1
P-4	100	.83	729	DuctIron	Main
P-5	100	1	1029	DuctIron	Main

In Data Format Needs Editing for ModelBuilder, no column labels have been specified. ModelBuilder will interpret the first row of data in the table as the column labels, which can make the attribute mapping step of the ModelBuilder Wizard more difficult unless you are very familiar with your data source setup.

Correct Data Format for ModelBuilder is also superior to Data Format Needs Editing for ModelBuilder in that it clearly identifies the units that are used for unitized attribute values, such as length and diameter. Again, unless you are very familiar with your data source, unspecified units can lead to errors and confusion.

Finally, Data Format Needs Editing for ModelBuilder is storing the Material and Subtype attributes as alphanumeric values, while ModelBuilder uses integer ID values to access this input. This data is unusable by ModelBuilder in alphanumeric format, and must be translated to an integer ID system in order to read this data.

The GIS-ID Property

All elements in WaterCAD V8*i* have an editable GIS-ID property which can be used for maintaining associations between records in your source file and elements in your model. These associations can be one-to-one, one-to-many, or many-to-one.

ModelBuilder can take advantage of this GIS-ID property, and has advanced logic for keeping your model and GIS source file synchronized across the various model to GIS associations.

The GIS-ID is a unique field in the source file which the user selects when Model-Builder is being set up. In contrast to using Label (which is adequate if model building is a one time operation) as the key field between the model and the source file, a GIS-ID has some special properties which are very helpful in maintaining long term updating of the model as the data source evolves over time.

In addition, WaterCAD V8*i* will intelligently maintain GIS-ID as you use the various tools to manipulate elements (Delete, Morph, Split, Merge Nodes in Close Proximity).

- When an element with one or more GIS-IDs is deleted, ModelBuilder will not recreate it the next time a synchronization from your GIS occurs if the "Recreate elements associated with a GIS-ID that was previously deleted from the model" option is left unchecked.
- When an element with one or more GIS-IDs is morphed, the new element will preserve those GIS-IDs. The original element will be considered as "deleted with GIS-IDs", which means that it will not be recreated by default (see above).
- When a link is split, the two links will preserve the same GIS-IDs the original pipe had. On subsequent ModelBuilder synchronizations, any data-change occurring for the associated record in the GIS can be cascaded into all the split link segments (see ModelBuilder - additional options).
- When nodes in close proximity are merged, the resulting node will preserve the GIS-IDs of all the nodes that were removed. On subsequent ModelBuilder synchronizations into the model, if there are data-update conflicts between the records in the GIS associated with the merged node in the model, updates from the first GIS-ID listed for the merged node will be preserved in the model. Note that in this case, the geometry of the merged node can't be updated in the model. For synchronizations going from the model to the GIS, data-updates affecting merged-nodes can be cascaded into all the associated records in the GIS (see ModelBuilder - additional options).

To support these relationship (specifically one to many), GIS-ID are managed as a collection property (capable of holding any number of GIS identifiers).

A variety of model element(s) to GIS record(s) associations can be specified:

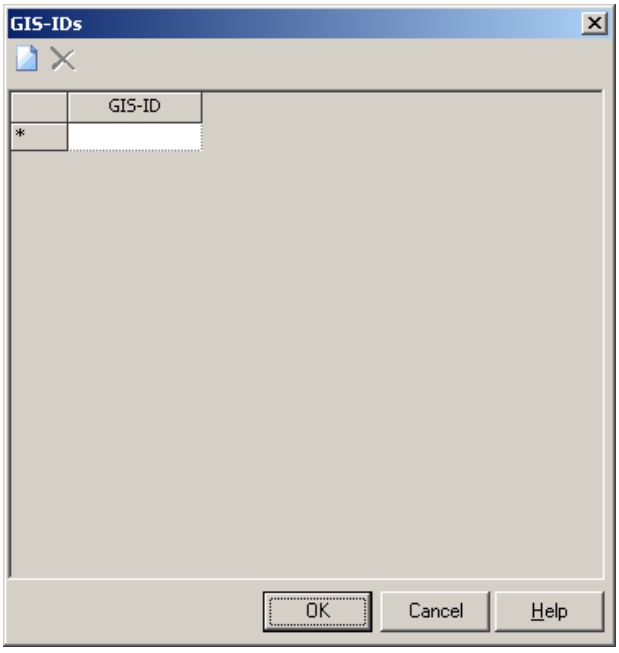
- If the GIS-ID collection is empty, there is no association between the GIS and this element.
- If there is a single entry, this element is associated with one record in the GIS.
- If there are multiple entries, this element is associated with multiple records in the GIS.
- More than one element in the model can have the same GIS-ID, meaning multiple records on the model are associated with a single record in the GIS.

Note: You can also manually edit the GIS-ID property to review or modify the element to

GIS association(s).

GIS-ID Collection Dialog Box

This dialog box allows you to assign one or more GIS-IDs to the currently selected element.



See The GIS-ID Property for more information on using GIS-IDs.

Specifying a SQL WHERE clause in ModelBuilder

The simplest form of a WHERE clause consists of "Column name - comparison operator - value". For example, if you want to process only pipes in your data source that are ductile iron, you would enter something like this:

```
Material = 'Ductile Iron'
```

String values must be enclosed in single quotes.

Column names are not case sensitive. Column names that contain a space must be enclosed in brackets:

```
[Pipe Material] = 'Ductile Iron'
```

Brackets are optional for columns names that do not contain a space.

Supported comparison operators are: <, >, <=, >=, <>, =, IN and LIKE.

Multiple logical statements can be combined by using AND, OR and NOT operators. Parentheses can be used to group statements and enforce precedence.

The * and % wildcard can be used interchangeably in a LIKE statement. A wildcard is allowed at the beginning and/or end of a pattern. Wildcards are not allowed in the middle of a pattern. For example:

```
PipeKey LIKE 'P-1*'
```

is valid, while:

```
PipeKey LIKE 'P*1'
```

is not.

Modelbuilder Import Procedures

You can use ModelBuilder to import pump definitions, pump curves, and patterns.

- Importing Pump Definitions Using ModelBuilder
- Using ModelBuilder to Import Pump Curves
- Using ModelBuilder to Import Patterns
- Using ModelBuilder to Import Time Series Data

Importing Pump Definitions Using ModelBuilder

Pump definition information can be extracted from an external data source using ModelBuilder.

Most of this importing is accomplished by setting up mappings under the **Pump Definition** Table Type. However, to import multipoint head, efficiency or speed vs. efficiency curves, the tabular values must be imported under Table Types: **Pump Definition - Pump Curves**, **Pump Definition - Flow-Efficiency Curve**, and **Pump Definition - Speed-Efficiency Curve** respectively.

The list of properties that can be imported under **Pump Definition** is given below. The only property in the list that is required is a **Key** or **Label**. Most of the properties are numerical values.

- BEP Efficiency
- BEP Flow
- Define BEP Max Flow?
- Design Flow
- Design Head
- GemsID (imported)
- Is Variable Speed Drive?
- Max Extended Flow
- Max Operating Flow
- Max Operating Head
- Motor Efficiency
- Notes
- Pump Definition Type (ID)
- Pump Definition Type (Label)
- Pump Efficiency
- Pump Efficiency (ID)
- Pump Efficiency (Label)
- Pump Power
- Shutoff Head
- User Defined BEP Max Flow

Those properties that are text such as Pump Efficiency and Pump Definition Type are alphanumeric and must be spelled correctly. For example **Standard (3 Point)** must be spelled exactly as shown in the **Pump Definition** drop down. Properties with a question mark above, require a **TRUE** or **FALSE** value. Those with ID next to the name are internal IDs and are usually only useful when syncing out from a model.

To import data, create a table in a data source (e.g. spreadsheet, data base), and then create columns/fields for each of the properties to be imported. In Excel for example, the columns are created by entering column headings in the first row of a sheet for each of the properties. Starting with the second row in the table, there will be one row for each pump definition to be imported.

Once the table is created in the source file, the file must be saved before it can be imported.

In the **Specify you data source** step in the wizard, the user indicates the source file name and the sheet or table corresponding to the pump definition data. In the **Specify field mappings for each table** step, the user selects **Pump Definition** as the table type, indicates the name of the pump definition in the **Key>Label** field and then maps each of the fields to be imported with the appropriate property in the Attribute drop down.

When syncing out from the model to a data table, the table must contain column headings for each of the properties to be exported. The names of the columns in the source table do not need to be identical to the property names in the model.

Importing can best be illustrated with an example. Given the data and graphs for three pump definitions shown in the graph below, the table below the graph shows the format for the pump curve definition import assuming that a standard 3 point curve is to be used for the head curve and a best efficiency curve is to be used for the efficiency curve. All three pumps are rated at 120 ft of TDH at 200 gpm.

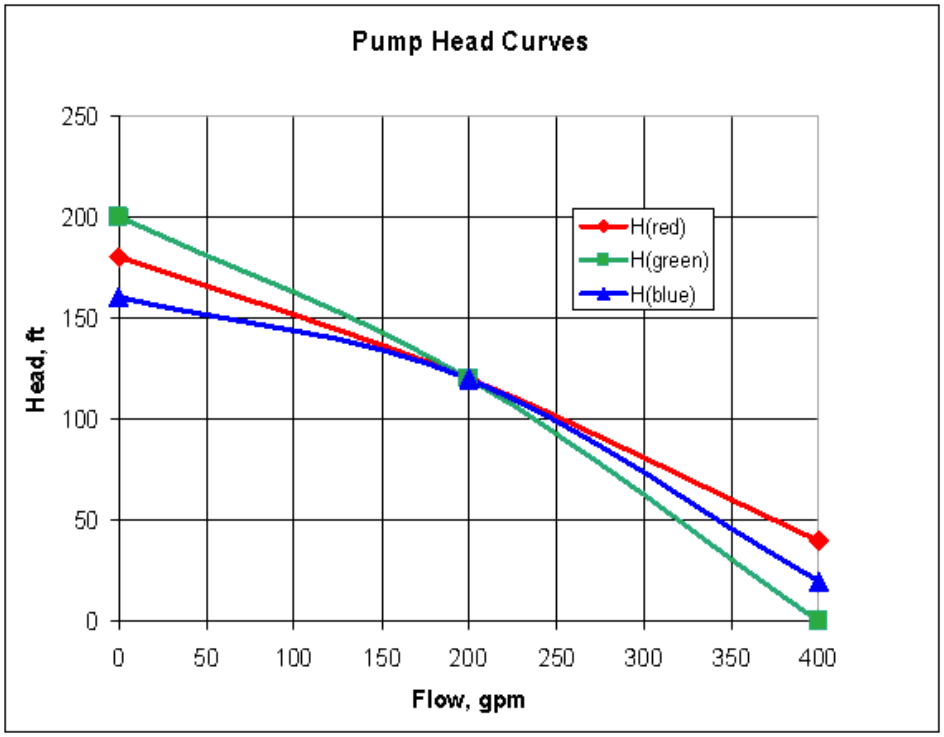


Table 5-3: Format of Pump Definition Import Data

Q, gpm	H (red)	H (green)	H (blue)
0	180	200	160
200	120	120	120
400	40	0	20
BEPe	70	69	65

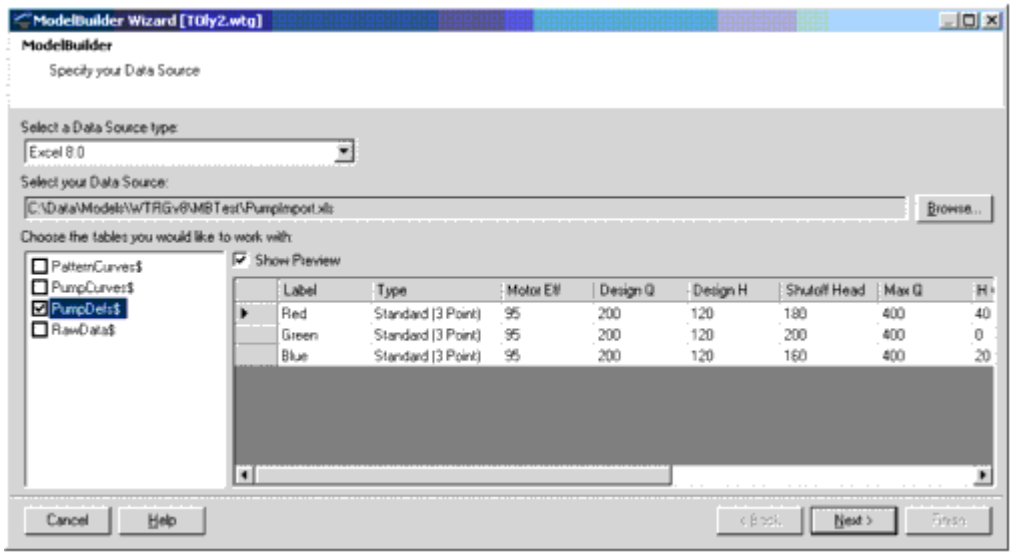
All three pumps have 95% motor efficiency and a BEP flow of 200.

The data source is created in an Excel spreadsheet.

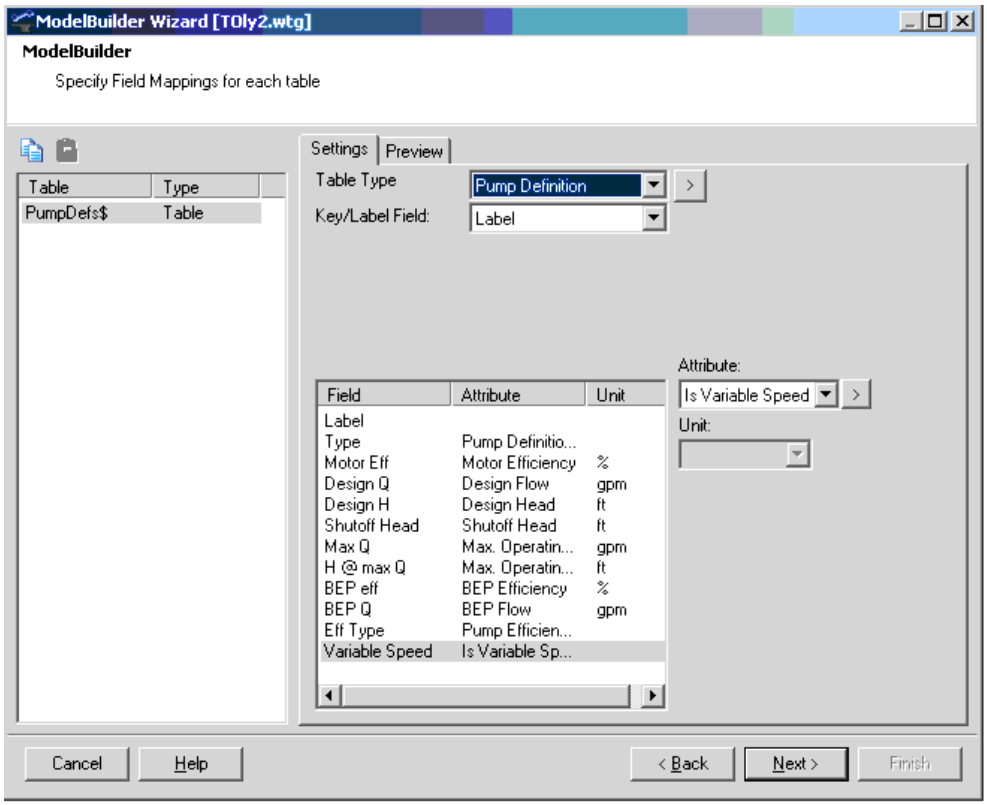
Table 5-4: Excel Data Source Format

Label	Type	Motor Eff	Design Q	Design H	Shutoff Head	Max Q	H @ Max Q	BEP Eff	BEP Q	Eff Type	Variable Speed
Red	Standard (3 Point)	95	200	120	180	400	40	70	200	Best Efficiency Point	FALSE
Green	Standard (3 Point)	95	200	120	200	400	0	69	200	Best Efficiency Point	FALSE
Blue	Standard (3 Point)	95	200	120	160	400	20	65	200	Best Efficiency Point	FALSE

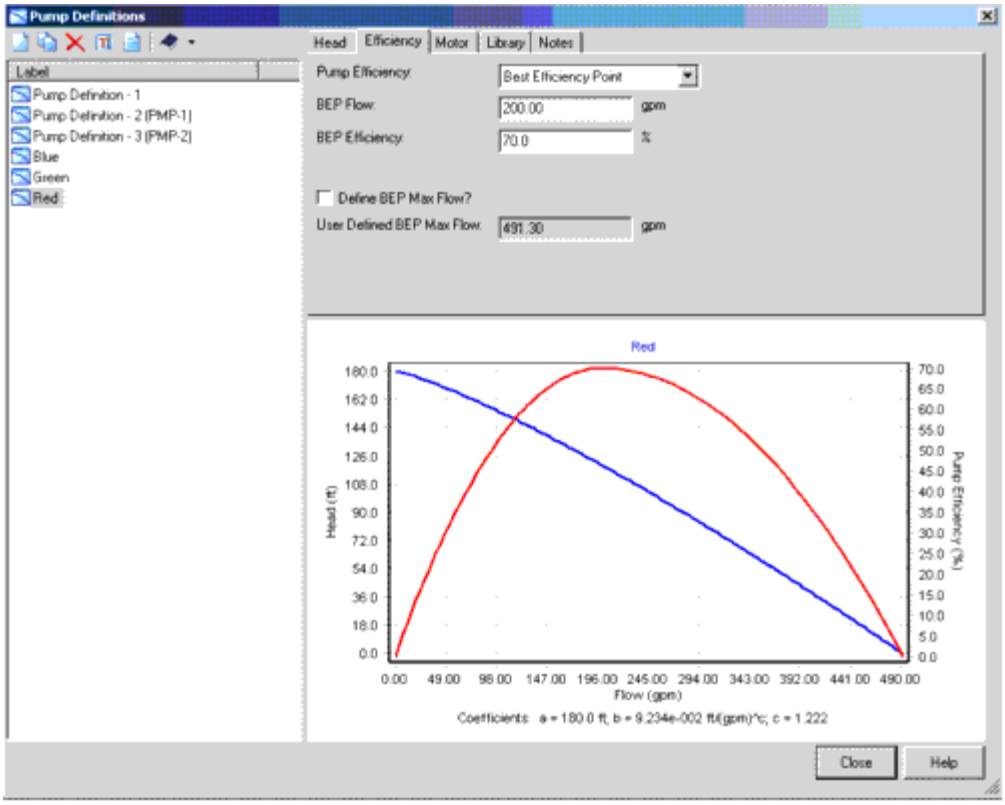
The data source step in ModelBuilder wizard looks like this:



The field mappings should look like the screen below:



After the import, the three pumps are listed in the Pump Definitions. The curve for the "Red" pump is shown below:



Using ModelBuilder to Import Pump Curves

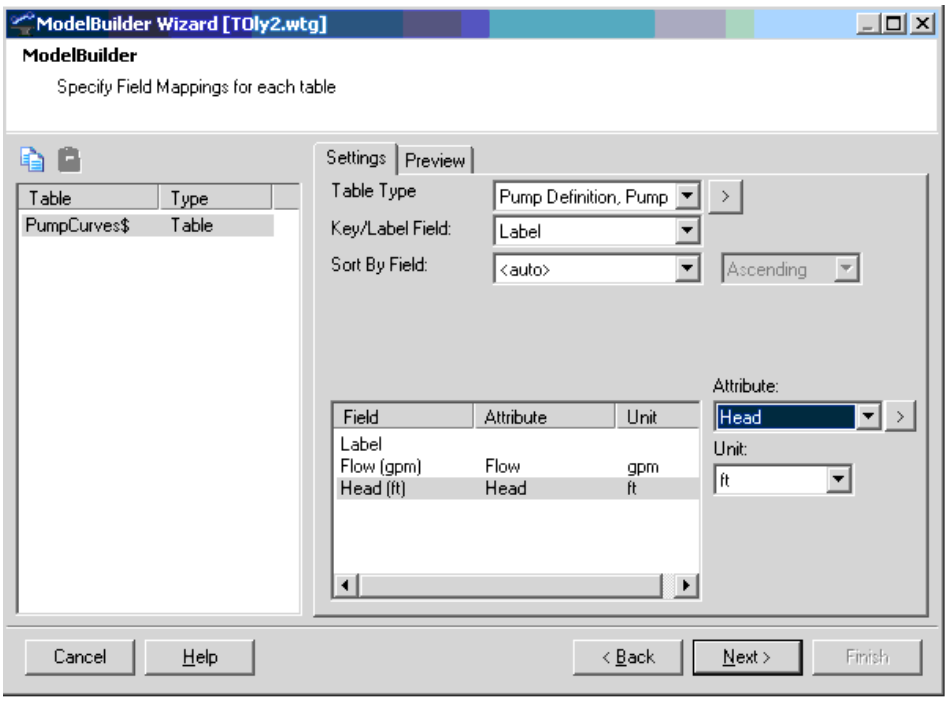
While most pump definition information can be imported using the Pump Definition Table Type, tabular data including

1. Multipoint pump-head curves,
2. Multipoint pump-efficiency curves and
3. Multipoint speed-efficiency curves

must be imported in their own table types.

To import these curves, first set up the pump definition type either manually in the Pump Definition dialog or by importing the pump definition through ModelBuilder. The Pump definition type would be **Multiple Point**, the efficiency type would be **Multiple Efficiency Points** or the **Is variable speed drive?** box would be checked.

In the field mapping step of the ModelBuilder wizard, the user the Table Type, **Pump Definition - Pump Curve** and would use the mappings shown below:



The example below shows an example of importing a Pump Head Curve. The process and format are analogous for flow-efficiency and speed-efficiency curves.

For the pump curves shown in the figure below, the data table needed is given. Several pump definitions can be included in the single table as long as they have different labels.

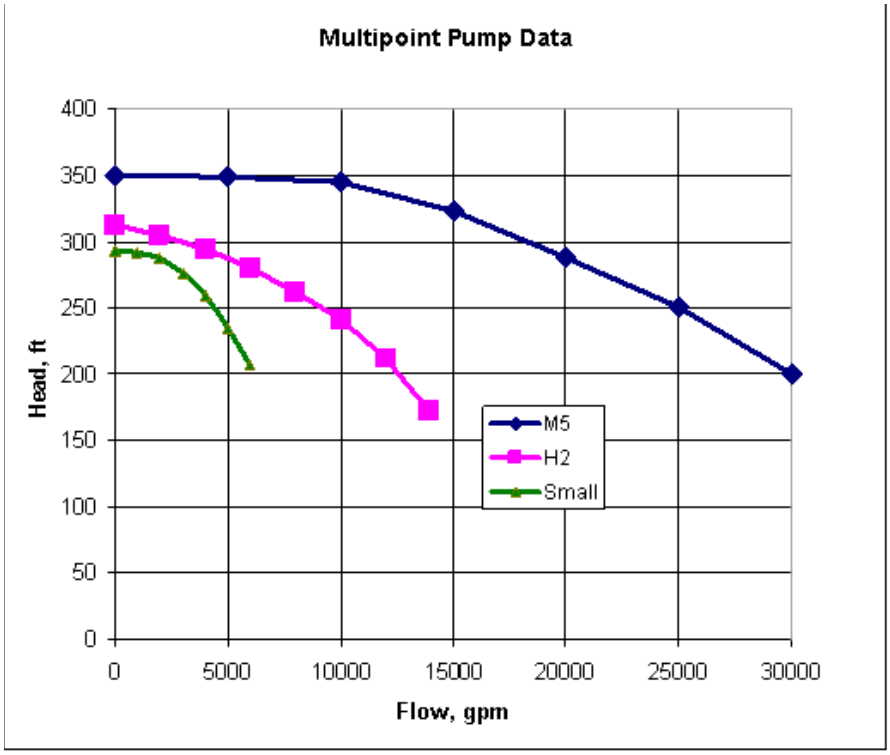


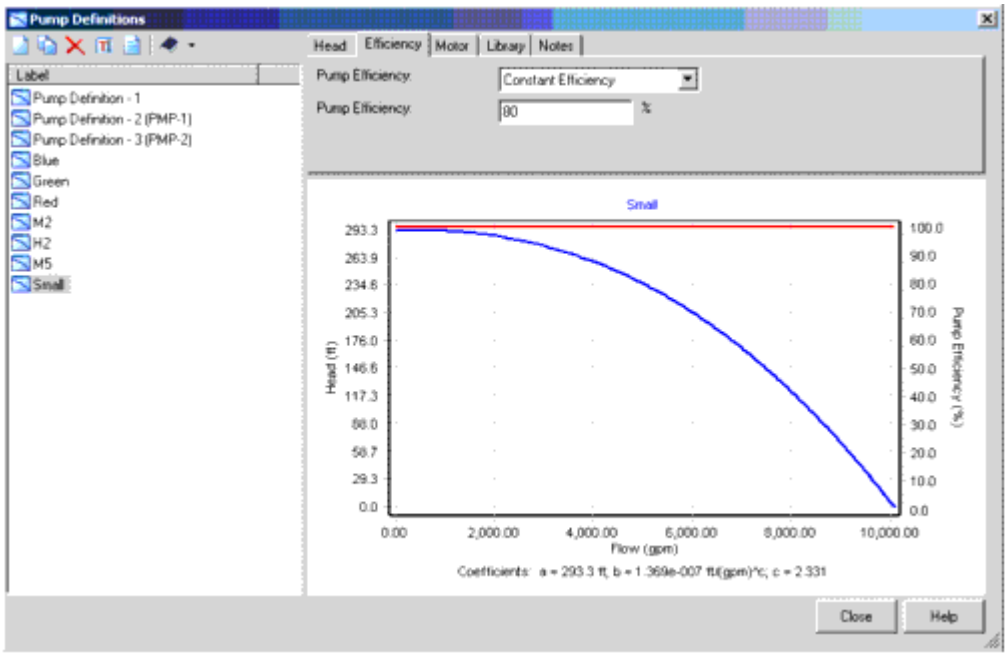
Table 5-5: Pump Curve Import Data Format

Label	Flow (gpm)	Head (ft)
M5	0	350
M5	5000	348
M5	10000	344
M5	15000	323
M5	20000	288
M5	25000	250
M5	30000	200
H2	0	312
H2	2000	304

Table 5-5: Pump Curve Import Data Format

H2	4000	294
H2	6000	280
H2	8000	262
H2	10000	241
H2	12000	211
H2	14000	172
Small	0	293
Small	1000	291
Small	2000	288
Small	3000	276
Small	4000	259
Small	5000	235
Small	6000	206

Upon running ModelBuilder to import the table above, three pump definitions would be created. The one called "Small" is shown below.



Using ModelBuilder to Import Patterns

Patterns can be imported into the model from external tables using ModelBuilder. This is a two step process.

1. Description of the pattern
2. Import tabular data

In general, the steps of the import are the same as described in the ModelBuilder documentation. The only steps unique to patterns are described below. All the fields except the **Key/Label** fields are optional

The source data files can be any type of tabular data including spreadsheets and data base tables.

Alphanumeric fields such as those which describe the month or day of the week must be spelled exactly as used in the model (e.g. January not Jan, Saturday not Sat).

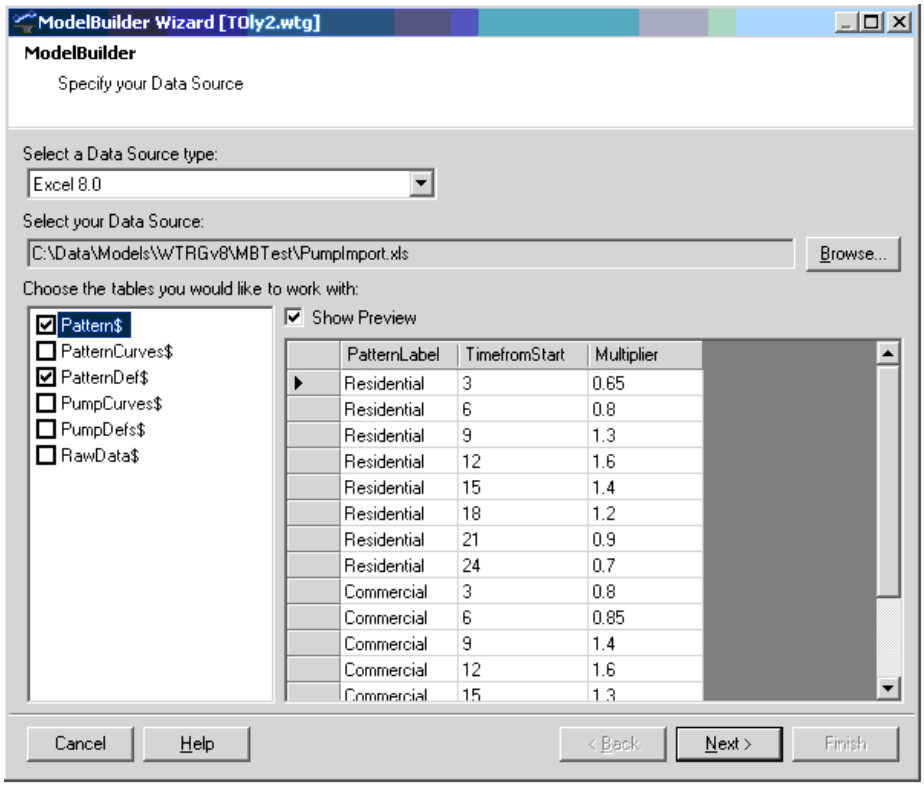
The list of model attributes which can be imported are given below.

- Label
- MONTH [January, February,…]

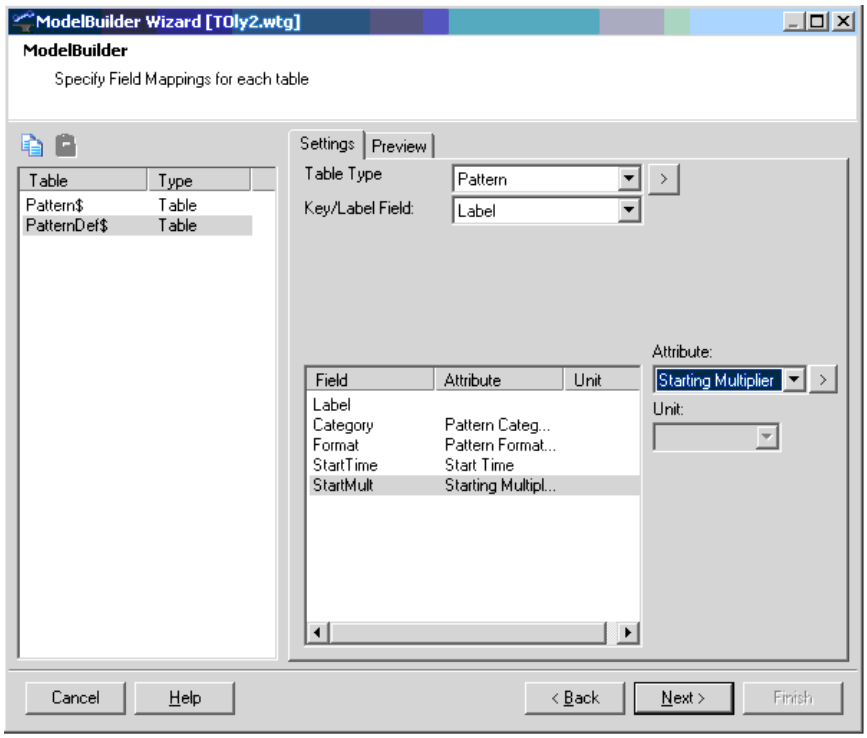
- DAY [Sunday, Monday,…]
- Pattern category type (Label) [Hydraulic, Reservoir…]
- Pattern format (Label) [Stepwise , Continuous]
- Start Time
- Starting Multiplier

The month and day are the actual month or day of week, not the word "MONTH". Labels must be spelled correctly.

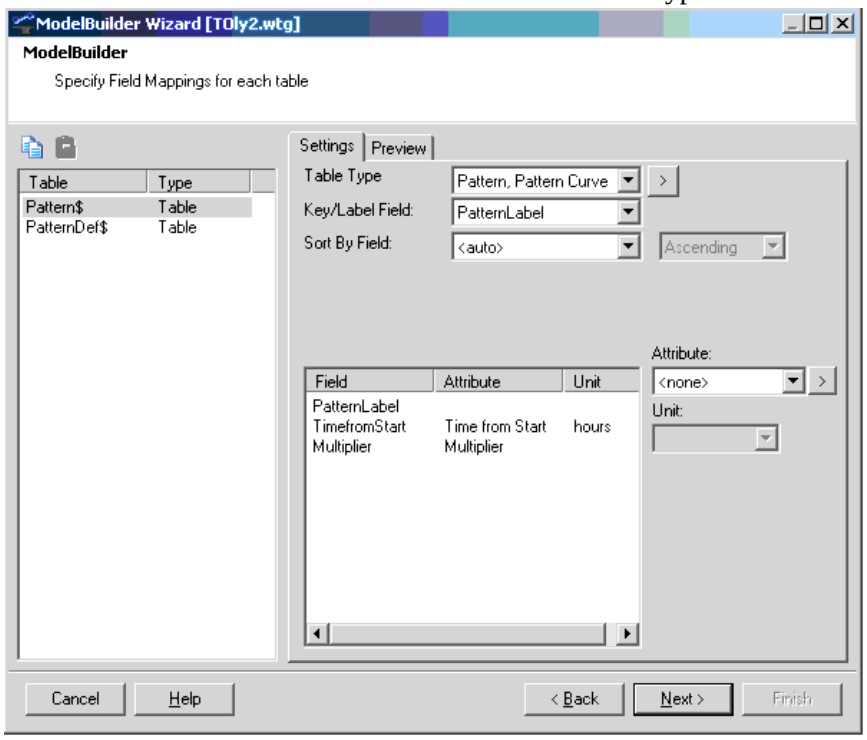
To import patterns, start ModelBuilder, create a new set of instructions, pick the file type, browse to the data file and pick the tables in that file to be imported. Checking the **Show Preview** button enables you to view the data before importing.



Then proceed to the **Field Mapping** step of ModelBuilder to set up the mappings for the Pattern in the Pattern Table Type. **Fields** refers to the name in the source table, **Attributes** refers to the name in the model.



And the actual Pattern Curve in the Pattern Curve table type.



The tables below show the pattern definition data and the pattern curve for two step-wise curves labeled Commercial and Residential. These data must be stored in two different tables although they may be and ideally should be in the same file.)

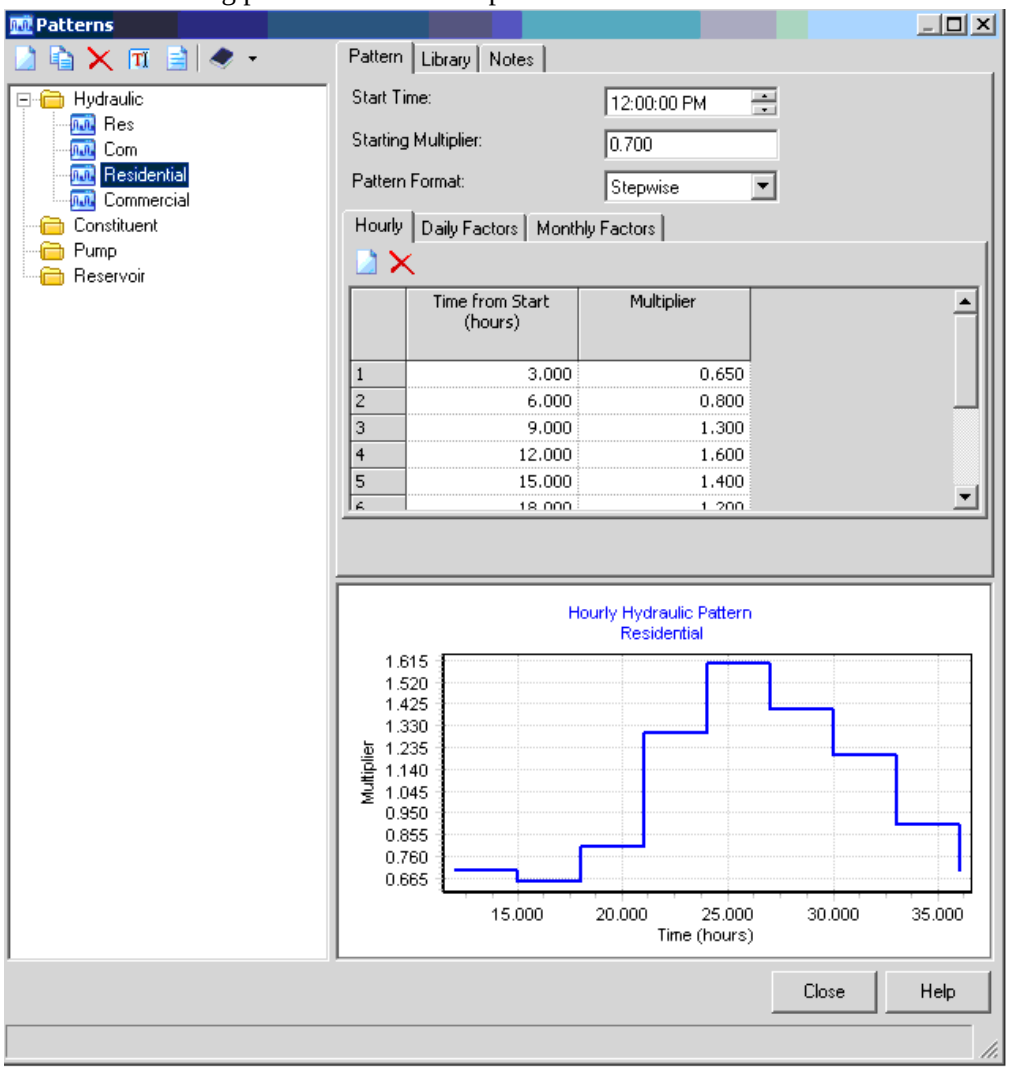
Table 5-6: Pattern Definition Import Data Format

Label	Category	Format	StartTime	StartMult
Residential	Hydraulic	Stepwise	12:00 PM	0.7
Commercial	Hydraulic	Stepwise	12:00 PM	0.8

Table 5-7: Pattern Curve Import Data Format

PatternLabel	TimeFromStart	Multiplier
Residential	3	0.65
Residential	6	0.8
Residential	9	1.3
Residential	12	1.6
Residential	15	1.4
Residential	18	1.2
Residential	21	0.9
Residential	24	0.7
Commercial	3	0.8
Commercial	6	0.85
Commercial	9	1.4
Commercial	12	1.6
Commercial	15	1.3
Commercial	18	0.9
Commercial	21	0.8
Commercial	24	0.8

One of the resulting patterns from this import is shown below:



Using ModelBuilder to Import Time Series Data

Time Series data maps onto the following *two* table types in ModelBuilder: **Time Series**, and **Time Series Collection**. The “Time Series" mapping represents entries in the TreeView along the left of the form (including the simple "Start Date Time", "Element", and "Notes" values shown on the right). The "Time Series Collection" mapping represents the tabular data shown in the table at the bottom right of the form.

Export Sample Time Series Data

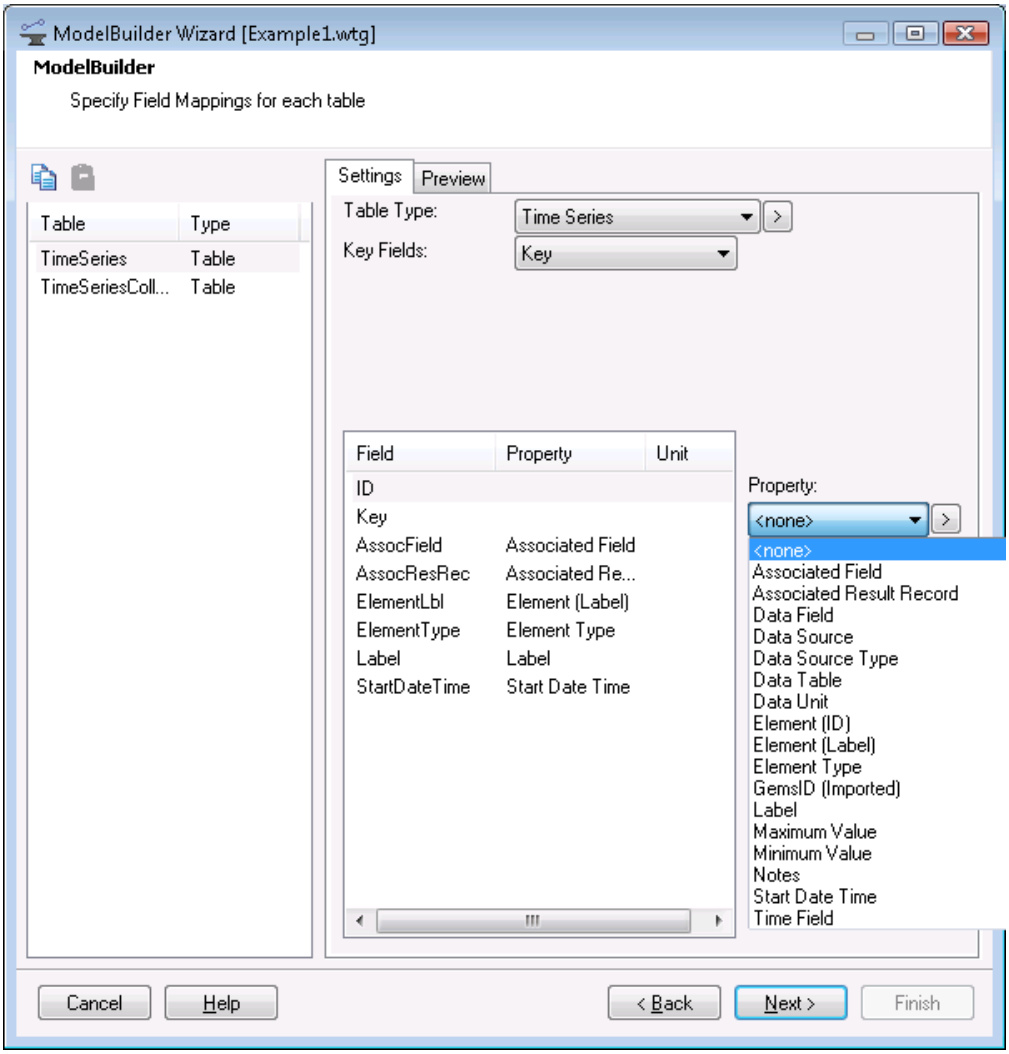
To automatically determine the appropriate values for handling Pipe Flow time series data, we're going to first export a sample from WaterCAD V8*i* to Excel.

First, create a sample Pipe Flow time series in WaterCAD V8*i* as shown above.

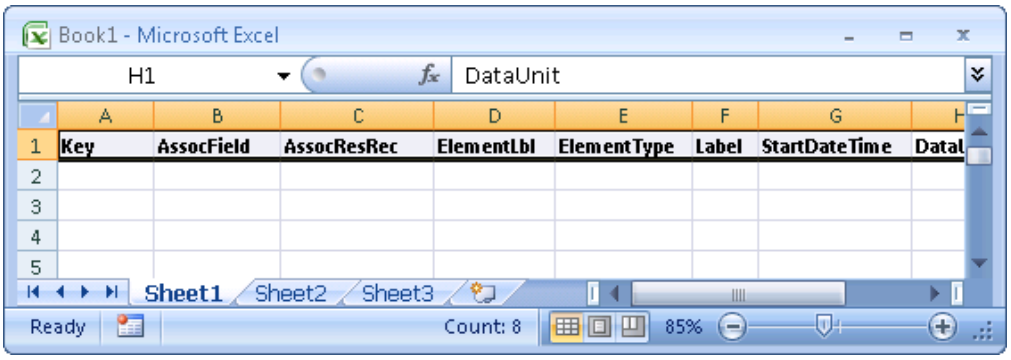
Next, create a new Excel .xls file. We'll need two "sheets" to receive the data (the default "Sheet1" and "Sheet2" will do).

Note: We recommend that you choose MSAccess over MSExcel if possible; there is no explicit way to specify the data-type of a column in Excel, which can result in some problems. You mentioned Excel in your post (and I didn't encounter any data-type problems), so I'll go with that here.

Time Series: This is the more difficult of the two Excel sheets we need to set up. To determine the columns to define in Excel, create a temporary ModelBuilder connection and get to the "Specify Field Mappings" step (you won't be saving this connection, so to get past Step 1 of the Wizard, just pick any data source). Navigate to this step, choose the Time Series table type, and click on the "Property" drop-down field:



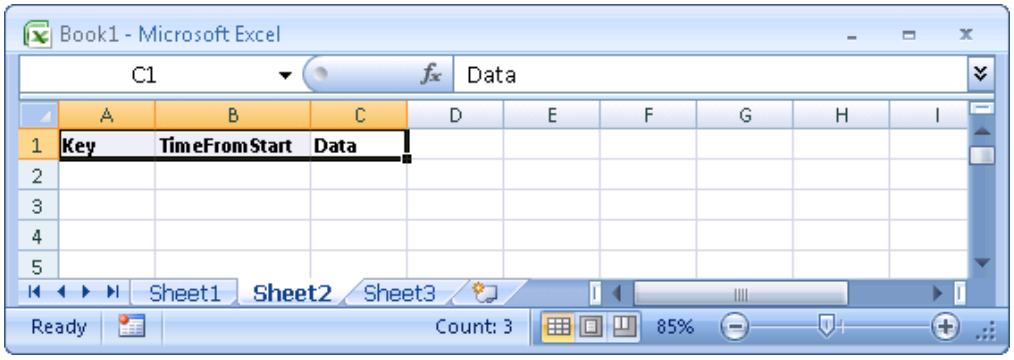
Click on the **Sheet1** tab in Excel to define the necessary columns for the "Time Series" table (You don't need all of these columns for Flow Data, but go ahead and define them all to be sure we don't miss any that are required for your use-case). It should look something like this:



Time Series Collection

Again, get to the "Specify Field Mappings" step in ModelBuilder, choose the "Time Series Collection" table type, and click on the "Property" drop-down field to determine the columns to define.

Click on the **Sheet2** tab in Excel and define the necessary columns for the "Time Series Collection" table. It should look something like this:



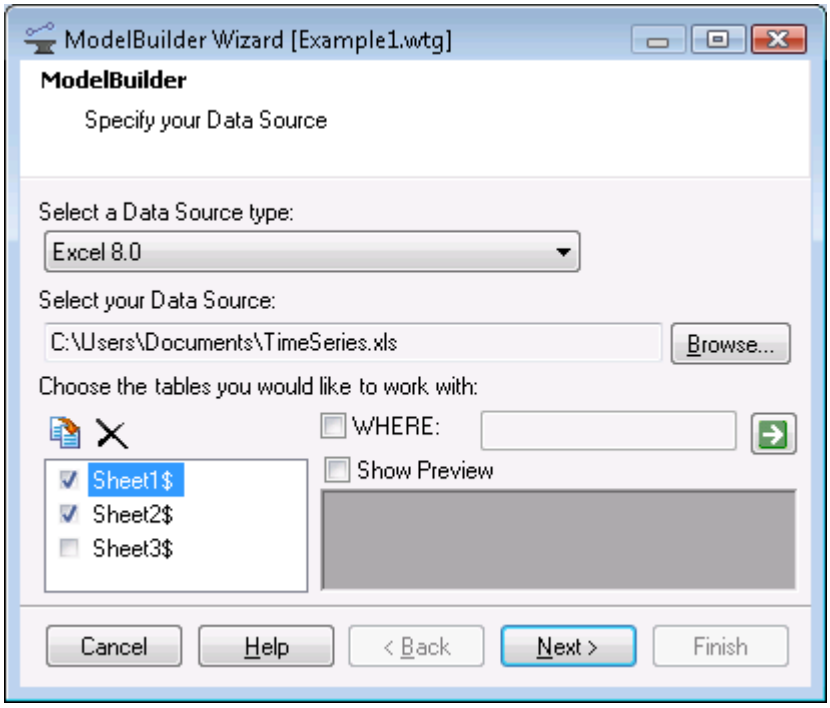
Save and close your spreadsheet.

Define the ModelBuilder Connection

Now we're ready to create the ModelBuilder connection to this spreadsheet.

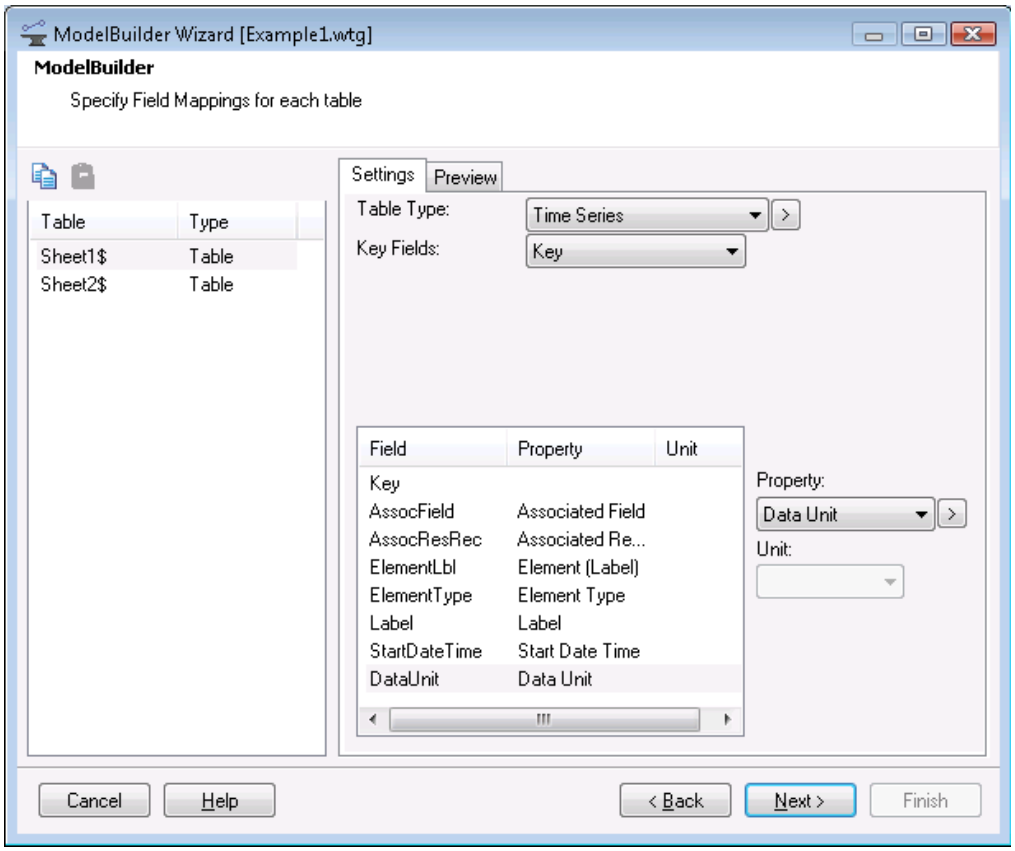
Open ModelBuilder and create a new Connection.

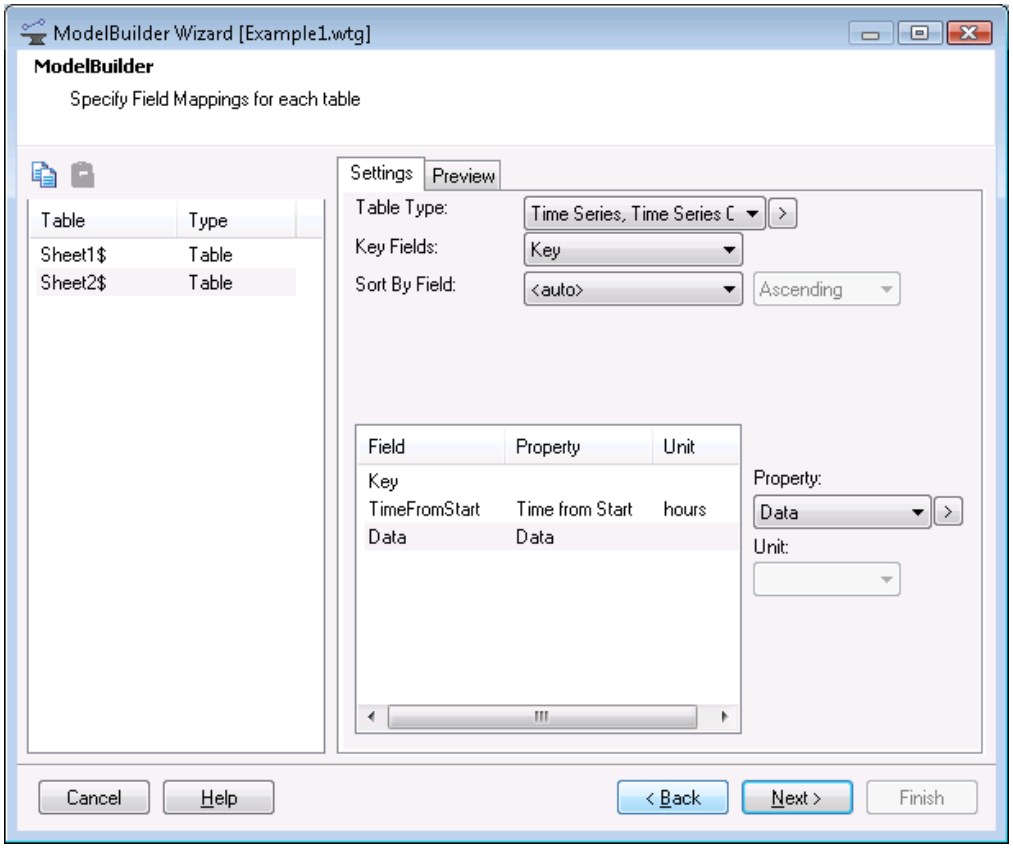
In step 1 of the Wizard, choose "Excel" as the data source type, browse to the Excel spreadsheet that you created to select it. You should see Sheet1 and Sheet2 in the list of available tables, select those (and unselect any others that appear).



Navigate through the next few steps, just use the defaults there.

When you reach the Mapping Step, set things up for Sheet1 and Sheet2 as shown below:





Navigate to the end of the Wizard.

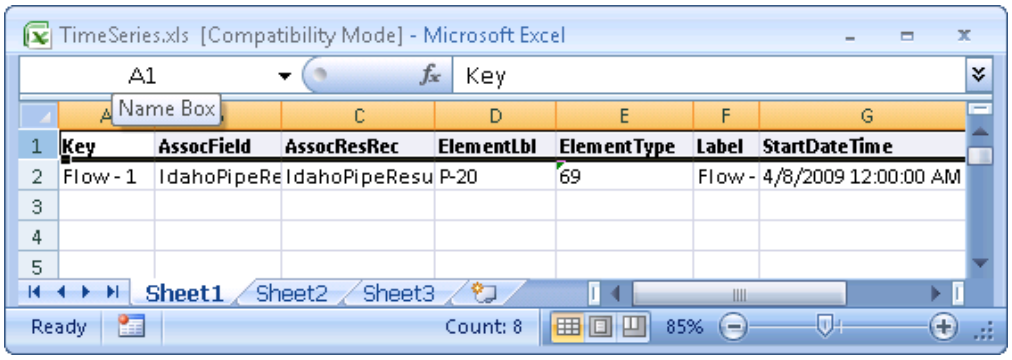
On the last step, click "No" for the "Would you like to build a model now?" prompt and click [Finish].

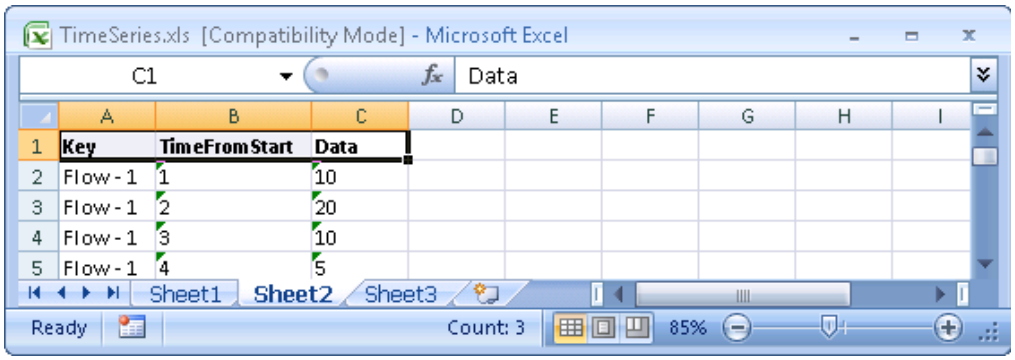
Synchronize Out from ModelBuilder

Choose the connection you just defined (be sure to close the Excel spreadsheet you just defined), and click the Sync Out toolbar button.

The sample time series data from WaterCAD V8*i* will now be available in the Excel spreadsheet you created.

Using that as a go-by, you should be able to enter the data in the appropriate format to import in to WaterCAD V8*i*.





Oracle as a Data Source for ModelBuilder

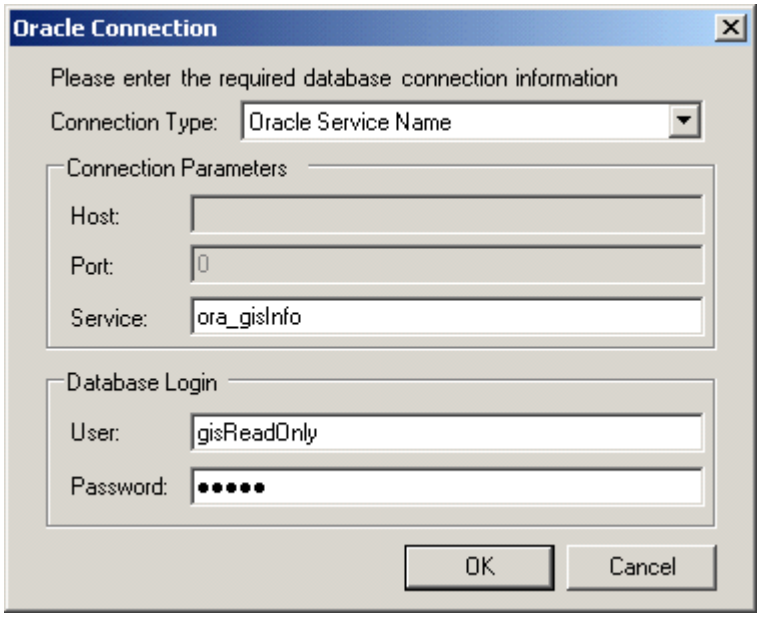
WaterCAD V8*i* makes it possible to import data to create a model from an Oracle database. To use this database, the user must have Oracle 11g Client software installed on the same computer in which WaterCAD V8*i* is running and it must be connected t the Oracle Server.

The user needs to understand the nature of the data stored in Oracle and the way it is stored. For example, the user must know if the data are stored as simple tabular data or whether the data are spatial data associated with polygons, lines, and points. The user needs to decide which fields in the database are to be imported into WaterCAD V8*i*.

It is possible to connect to an Oracle database from WaterCAD V8*i* using any supported CAD/GIS platform. Start ModelBuilder the same as with any other data source (see ModelBuilder Connections Manager). However, when the user browses for a data source some additional information is required.

When the user Browses for an Oracle datasource, ModelBuilder opens an Oracle login form. The user can enter just a service name if they have setup an alias on their system for the Oracle datasource. The user should contact their administrator for details on how to setup this alias. Otherwise, the user must enter all of the connection informa-

tion, which includes the computer/host that Oracle is running on, the network port number that Oracle is using, and the raw Oracle service name. Again, the user should contact their administrator for those details. The user must also supply a valid Oracle username and password to log into the data source.



On the mapping form in ModelBuilder, there is a Generator (Sync out) combo-box. The user only needs to select a sequence generator in this box if they plan to sync out to Oracle and have ModelBuilder create new records in Oracle. The Oracle sequence generator is an object that is created in Oracle by the administrator. It allows Oracle to create records with unique Oracle identifiers, which is may be required when creating new records. ModelBuilder will display the available sequence generators that are available for use.

Oracle/ArcSDE Behavior

If creating a ModelBuilder connection to an ArcSDE data source, you can always use the Geodatabase and/or Geometric Network connection types when running in the ArcGIS platform. If the ArcSDE has an Oracle database as the back end data store, and ArcSDE has been configured to use Oracle's native geometry type (i.e. SDO_GEOMETRY), you can also use the Oracle connection in ModelBuilder to interact directly with the Oracle data, which has the benefit of being an option in any platform, such as Microstation. However you should not synchronize data from the model out to the Oracle connection if it's the back end of an ArcSDE data source, as that may cause problems for the ArcSDE.

Applying Elevation Data with TRex

6

The Importance of Accurate Elevation Data

Numerical Value of Elevation

Record Types

Calibration Nodes

TRex Terrain Extractor

The Importance of Accurate Elevation Data

Obtaining node elevation data for input into a water distribution model can be an expensive, time-consuming process. In some cases, very accurate elevation data may be critical to the model's utility; in other cases it can represent a significant resource expenditure. In order to decide on the appropriate level of quality of elevation data to be gathered, it is important to understand how a model uses this data.

Elevation data for nodes is not directly used in solving the network equations in hydraulic models. Instead, the models solve for hydraulic grade line (HGL). Once the HGL is calculated and the numerical solution process is essentially completed, the elevations are then used to determine pressure using the following relationship:

$$p = (\mathrm{HGL} - z)\rho g$$

Where: p = pressure (lb./ft.^{2}, N/m^2)

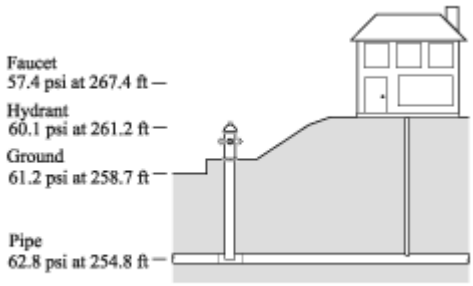
HGL	=	hydraulic grade line (ft., m)
z	=	node elevation (ft., m)
ρ	=	density of water (slugs/ft.^{3}, kg/m^3)
g	=	gravitational acceleration (ft./sec.^{2}, m/sec.^{2})

If the modeler is only interested in calculating flows, velocities, and HGL values, then elevation need not be specified. In this case, the pressures at the nodes will be computed assuming an elevation of zero, thus resulting in pressures relative to a zero elevation.

If the modeler specifies pump controls or pressure valve settings in pressure units, then the model needs to compute pressures relative to the elevation of the nodes being tested. In this case, the elevation at the control node or valve would need to be specified (or else the model will assume zero elevation). Therefore, an accurate elevation value is required at each key node where pressure is of importance.

Numerical Value of Elevation

The correct elevation of a node is the elevation at which the modeler wants to know the pressure. The relationship between pressure and elevation is illustrated as follows:



Notice that an HGL of 400 ft. calculated at the hydrant is independent of elevation. However, depending on which elevation the modeler entered for that node, the pressure can vary as shown. Usually modelers use ground elevation as the elevation for the node.

Accuracy and Precision

How accurate must the elevation data be? The answer depends on the accuracy desired in pressure calculations vs. the amount of labor and cost allotted for data collection. For example, the HGL calculated by the model is significantly more precise than any of the elevation data. Since 2.31 ft.of elevation translates into 1 psi of pressure (for water), calculating pressure to 1 psi precision requires elevation data that is accurate to roughly 2 ft. Elevation data that is accurate to the nearest 10 ft. will result in pressure that is accurate to roughly 4 psi.

The lack of precision in elevation data (and pressure results) also leads to questions regarding water distribution design. If design criteria state that pressure must exceed 20 psi and the model gives a pressure of 21 (+/- 4) psi or 19 (+/-4) psi, the engineer relying on the model will have to decide if this design is acceptable.

Obtaining Elevation Data

In building the large models that are used today, collecting elevation data is often a time-consuming process. A good modeler wants to devote the appropriate level of effort to data collection that will yield the desired accuracy at a minimum cost. Some of the data collection options are:

- USGS Topographic Maps
- Surveying from known benchmarks
- Digital Elevation Models (DEMs)
- SDTS Digital Elevation Models
- Digital Ortho-Rectified Photogrammetry
- Contour Maps (contour shapefiles)
- As-built Plans
- Global Positioning Systems (GPS).

The data type used by the Elevation Extractor is Digital Elevation Models (DEMs). Digital Elevation Models, available from the USGS, are computer files that contain elevation data and routines for interpolating that data to arrive at elevations at nearby points. DEM data are recorded in a raster format, which means that they are represented by a uniform grid of cells of a specified resolution (typically 100 ft.). The accuracy of points interpolated from the grid depends on the distance from known

benchmarks and is highly site-specific. However, it is usually on the order of 5 to 10 ft. when the ground slopes continuously. If there are abrupt breaks in elevation corresponding to road cuts, levees, and cliffs, the elevations taken from the DEMs can be inaccurate.

DEMs are raster files containing evenly spaced elevation data referenced to a horizontal coordinate system. In the United States, the most commonly used DEMs are prepared by the U.S. Geological Survey (USGS). Horizontal position is determined based on the Universal Transverse Mercator coordinate system referenced to the North American Datum of 1927 (NAD 27) or 1983 (NAD 83), with distances given in meters. In the continental U.S., elevation values are given in meters (or in some cases feet) relative to the National Geodetic Vertical Datum (NGVD) of 1929.

DEMs are available at several scales. For water distribution, it is best to use the 30-meter DEMs with the same spatial extents as the 7.5-minute USGS topographic map series. These files are referred to as large-scale DEMs. The raster grids for the 7.5-minute quads are 30 by 30 meters. There is a single elevation value for each 900 square meters. (Some maps are now available with grid spacing as small as 10 by 10 meters, and more are being developed.) Ideally, some interpolation is performed to determine the elevation value at a given point. The DEMs produce the best accuracy in terms of point elevations in areas that are relatively flat with smooth slopes but have poorer accuracy in areas with large, abrupt changes in elevation, such as cliffs and road cuts.

The Spatial Data Transfer Standard, or SDTS, is a standard for the transfer of earth-referenced spatial data between dissimilar computer systems. The SDTS provides a solution to the problem of spatial data transfer from the conceptual level to the details of physical file encoding. Transfer of spatial data involves modeling spatial data concepts, data structures, and logical and physical file structures. In order to be useful, the data to be transferred must also be meaningful in terms of data content and data quality. SDTS addresses all of these aspects for both vector and raster data structures.

The SDTS spatial data model can be made up of more than one spatial object (referred to as aggregated spatial objects), which can be thought of as data layers in the Point or Topological Vector profiles. A Raster Profile can contain multiple raster object record numbers, which are part of the RSDF module of a Raster Profile data set. Multiple raster object record numbers must be converted into separate grids by converting each raster object record number one at a time into an Output grid.

LIDAR is relatively new technology which determines elevation using a light signal from an airplane. LIDAR elevation data is collected using an aerial transmitter and sensor and is significantly more accurate and expensive than traditional DEM data. LIDAR data can be produced in a DEM format and is becoming more widely available.

Record Types

USGS DEM files are organized into these record types:

- **Type A** records contain information about the DEM, including name, boundaries, and units of measure.
- **Type B** records contain elevation data arranged in “profiles” from south to north, with the profiles organized from west to east.
- **Type C** records contain statistical information on the accuracy of the DEM.

There is one Type A and one Type C record for each DEM. There is one Type B record for each south-north profile.

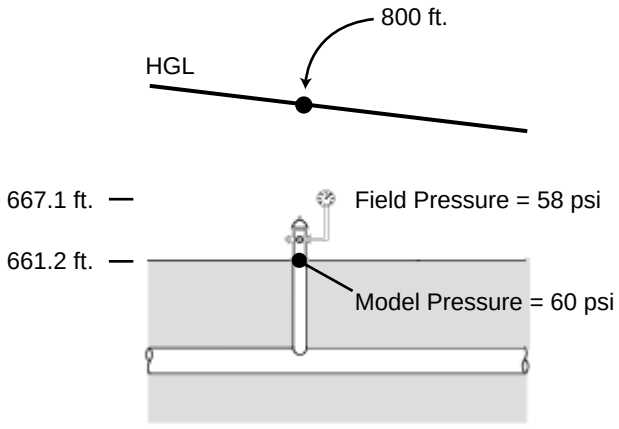
DEMs are classified by the method with which they were prepared and the corresponding accuracy standard. Accuracy is measured as the root mean square error (RMSE) of linearly interpolated elevations from the DEM compared to known elevations. The levels of accuracy, from least accurate to most accurate, are described as follows:

- **Level One DEMs** are based on high altitude photography and have a vertical RMSE of 7 meters and a maximum permitted RMSE of 15 meters.
- **Level Two DEMs** are based on hypsographic and hydrographic digitizing with editing to remove identifiable errors. The maximum permitted RMSE is one-half of the contour interval.
- **Level Three DEMs** are based on digital line graphs (DLG) and have a maximum RMSE of one-third of the contour interval.

DEMs will not replace elevation data obtained from field-run surveys, high-quality global positioning systems, or even well-calibrated altimeters. They can be used to avoid potential for error which can be involved in manually interpolating points.

Calibration Nodes

An elevation accuracy of 5 ft. is adequate for most nodes; therefore, a USGS topographic map is typically acceptable. However, for nodes to be used for model calibration, a higher level of accuracy is desirable. Consider a situation where both the model and the actual system have exactly the same HGL of 800 ft. at a node (see figure below). The elevation of the ground (and model node) is 661.2 ft. while the elevation of the pressure gage used in calibration is 667.1 ft. The model would predict a pressure of 60.1 psi while the gage would read 57.5 psi even though the model is correct.



A similar error could occur in the opposite direction with an incorrect pressure appearing accurate because an incorrect elevation is used. This is one reason why model calibration should be done by comparing modeled and observed HGL values and not pressures.

TRex Terrain Extractor

The TRex Terrain Extractor was designed to expedite the elevation assignment process by automatically assigning elevations to the model features according to the elevation data stored within Digital Elevation Models.

Digital Elevation Models were chosen because of their wide availability and since a reasonable level of accuracy can be obtained by using this data type depending on the accuracy of the DEM/DTM.

The TRex Terrain Extractor can quickly and easily assign elevations to any or all of the nodes in the water distribution model. All that is required is a valid Digital Elevation Model. Data input for TRex consists of:

1. Specify the GIS layer that contains the DEM from which elevation data will be extracted.
2. Specify the measurement unit associated with the DEM (feet, meters, etc.).
3. Select the model features to which elevations should be applied; all model features or a selection set of features can be chosen.

TRex then interpolates an elevation value for each specific point occupied by a model feature. The final step of the wizard displays a list of all of the features to which an elevation was applied, along with the elevation values for those features. These elevation values can then be applied to a new physical properties alternative, or an existing one. In some cases, you might have more accurate information for some nodes (e.g., survey elevation from a pump station). In those cases, you should create the elevation data using DEM data and manually overwrite the more accurate data for those nodes.

The TRex Terrain Extractor simplifies the process of applying accurate elevation data to water distribution models. As was shown previously, accurate elevation data is vital when accurate pressure calculations and/or pressure-based controls are required for the water distribution model in question. All elevation data for even large distribution networks can be applied by completing a few steps.

In the US, DEM data is usually available in files corresponding to a single USGS 7.5 minute quadrangle map. If the model covers an area involving several maps, it is best to mosaic the maps into a single map using the appropriate GIS functions as opposed to applying TRex separately for each map.

When using TRex, it is necessary that the model and the DEM be in the same coordinate system. Usually the USGS DEMs are in the UTM (Universal Transverse Mercator) with North American Datum 1983 (NAD83) in meters, although some may use NAD27. Models are often constructed using a state plane coordinate system in feet. Either the model or DEM must be converted so that the two are in the same coordinate system for TRex to work. Similarly, the vertical datum for USGS is based on national Vertical Geodetic Datum of 1929. If the utility has used some other datum for vertical control, then these differences need to be reconciled.

The TRex Terrain Extractor can read the USGS DEM raster data in SDTS format. Raster profiles provide a flexible way to encode raster data. The SDTS standard contains small limited subsets called profiles. In a raster transfer, there should be one RSDF module, one LDEF module and one or more cell modules. Each record in the RSDF module denotes one raster object. Each raster object can have multiple layers. Each layer is encoded as one record in the LDEF module. The actual grid data is stored in the cell module which is referenced by the layer record. A typical USGS DEM data set contains one RSDF record, one LDEF record and one cell file.

TRex Wizard

The TRex Wizard steps you through the process of automatically assigning elevations to specified nodes based on data from a Digital Elevation Model or a Digital Terrain Model.

TRex can load elevation data into model point features (nodes) from a variety of file types including both vector and raster files. To use raster files as the data source, the ArcGIS platform must be used. With a vector data source, it is possible to use any platform. Vector data must consist of either points with an elevation or contours with an elevation.

It is important to understand the resolution, projection, datum, units and accuracy of any source file that will be used to load elevation data for nodes.

In the United States, elevation data can be obtained at the USGS National Map Seamless Server. The vertical accuracy may only be +/- 7 to 15 m.

Step 1: File Selection

The elevation data source and features to which elevations will be assigned are specified in the File Selection dialog of the TRex wizard. Valid elevation data sources include:

- Vector files such as DXF and SHP files
- LandXML files
- InRoads .dtm (Microstation platform only)
- Geopack .tin (32-bit version only)
- Bentley MX .fil
- Bentley .dgn (Microstation platform only)

DXF files are able to contain both points and lines, therefore the user must indicate whether the node elevations should be built based on the points in the DXF, or based on the contour lines in the DXF.

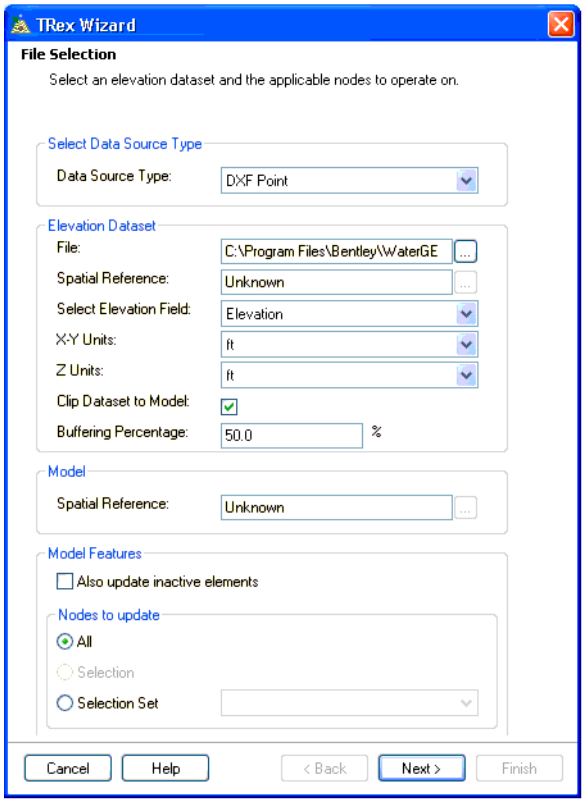
Shapefiles are not allowed to contain mixed geometric data, so TRex can safely determine whether to build the elevation map based on either elevation point data or elevation contour lines. The Model Spot Elevation data source type uses existing spot elevation nodes in the model, which must already have correct elevation values assigned. Using these as the data source, TRex can determine the elevations for the other nodes in the model.

Bentley MX (.fil) files can contain multiple terrain models; you must select a single model to use as the elevation data source.

When running under the ArcGIS platform, additional raster data sources are also available for direct use in TRex, including TIN, Rasters(grid), USGS(DEM), and SDTS(DDF) files.

These data sources are often created in a specific spatial reference, meaning that the coordinates in the data source will be transformed to a real geographic location using this spatial reference. Care must be taken when laying out the model to ensure that the model coordinates, when transformed by the model's spatial reference (if applicable), will overlay the elevation data source in this 'global' coordinate system. If the model and elevation data source's data don't overlay each other, TRex will be unable to interpolate elevation data. GIS products such as Bentley Map and ArcGIS can be used to transform raster source data into a spatial reference that matches that of the model.

If you are unable to run TRex under ArcGIS (i.e. you are using stand-alone or a CAD platform), ArcGIS can generally be used to convert the raster data to a point shapefile that approximates the raster data source. Shapefiles can be always be used in TRex, regardless of the platform that TRex is running.



- **Data Source Type**—This menu allows you to choose the type of file that contains the input data you will use.
- **File**—This field displays the path where the data file is located. Use the browse button to find and select the desired file.
- **Spatial Reference (ArcGIS Mode Only)**—Click the Ellipsis (...) next to this field to open the Spatial Reference Properties dialog box, allowing you to specify the spatial reference being used by the elevation data file.
- **Select Elevation Field**—Select the elevation unit.
- **X-Y Units**—This menu allows the selection of the measurement unit type associated with the X and Y coordinates of the elevation data file.
- **Z Units**—This menu allows the selection of the measurement unit type associated with the Z coordinates of the elevation data file.
- **Clip Dataset to Model—**In some cases, the data source contains elevation data for an area that exceeds the dimensions of the area being modeled. When this box is checked, TRex will calculate the model's bounding box, find the larger dimension (width or height), calculate the Buffering Percentage of that dimension, and increase both the width and height of the model bounding box by that amount.

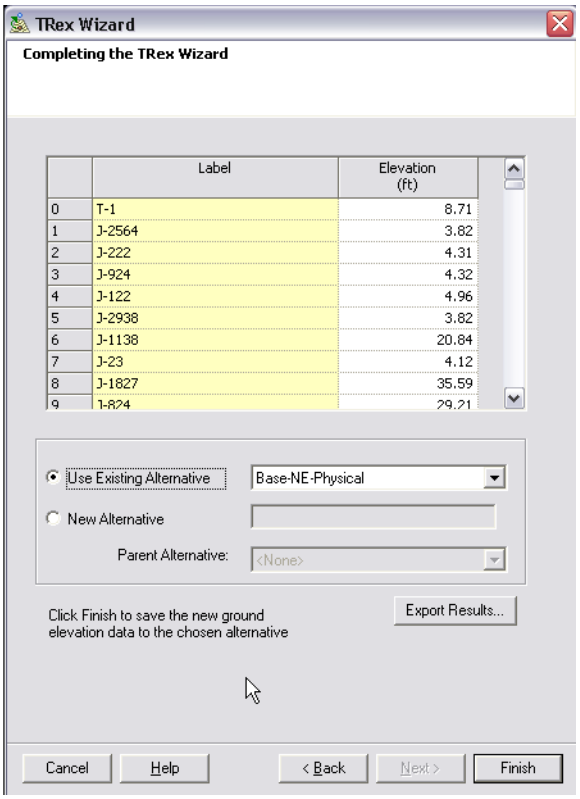
Then any data point that falls outside of the new bounding box will not be used to generate the elevation mesh. If this box isn't checked, all the source data points are used to generate the elevation mesh. Checking this box should result in faster calculation speed and use less memory.

- **Buffering Percentage**—This field is only active when the Clip Dataset to Model box is checked. The percentage entered here is the percentage of the larger dimension (width or height) of the model's bounding box that will be added to both the bounding box width and height to find the area within which the source data points will be used to build the elevation mesh.
- **Spatial Reference (ArcGIS Mode Only)**—Click the Ellipsis (...) next to this field to open the Spatial Reference Properties dialog box, allowing you to specify the spatial reference being used by the WaterCAD V8*i* model file.
- **Also update inactive elements**—Check this box to include inactive elements in the elevation assignment operation. When this box is unchecked, elements that are marked Inactive will be ignored by TRex.
- **All**—When this button is selected, TRex will attempt to assign elevations to all nodes within the WaterCAD V8*i* model.
- **Selection**—When this button is selected, TRex will attempt to assign elevations to all currently highlighted nodes.
- **Selection Set**—When this is selected, the Selection Set menu is activated. When the Selection Set button is selected, TRex will assign elevations to all nodes within the selection set that is specified in this menu.

Note: If the WaterCAD V8*i* model (which may or may not have a spatial reference explicitly associated with it) is in a different spatial reference than the DEM/DTM (which does have a spatial reference explicitly associated with it), then the features of the model will be projected from the model's spatial reference to the spatial reference used by the DEM/DTM.

Step 2: Completing the TRex Wizard

The results of the elevation extraction process are displayed and the results can be applied to a new or existing physical alternative.



- **Results Preview Pane**—This tabular pane displays the elevations that were calculated by TRex. The table can be sorted by label by clicking the Label column heading and by elevation by clicking the Elevation column heading. You can filter the table by right-clicking a column in the table and selecting the **Filter...Custom** command. You can also right-click any of the values in the elevation column to change the display options.
- **Use Existing Alternative**—When this is selected, the results will be applied to the physical alternative that is selected in the Use Existing Alternative menu. This menu allows the selection of the physical alternative to which the results will be applied.
- **New Alternative** —When this is selected, the results will be applied to a new physical alternative. First, the currently active physical alternative will be duplicated, then the results generated by TRex will be applied to the newly created alternative. The name of this new alternative must be supplied in the New Alternative text field.

- **Parent Alternative**—Select an alternative to duplicate from the menu, or select <None> to create a new Base alternative.
- **Export Results**—This exports the results generated by TRex to a tab or comma-delimited text file (.TXT). These files can then be re-used by WaterCAD V8*i* or imported into other programs.

- Click Finish when complete, or Cancel to close without making any changes.

TRex Supported Terrain Models

TRex can import terrain models created in InRoads, MXROAD or GEOPAK, however not all terrain model types are currently supported on all platforms. The following table shows which terrain models are supported in each WaterGEMS/ WaterCAD/HAMMER platform.:

Table 6-1: TRex Supported Terrain Models

Platform	InRoads	GEOPAK	Bentley MX
Stand Alone x86	No	Yes	Yes
Stand Alone x64	No	Partial	No
Microstation	Yes	Yes	Yes
AutoCAD x86	No	Yes	Yes
AutoCAD x64	No	Partial	No
ArcGIS	No	Yes	Yes

Allocating Demands using LoadBuilder

7

Using GIS for Demand Allocation

Using LoadBuilder to Assign Loading Data

Generating Thiessen Polygons

Demand Control Center

Unit Demand Control Center

Pressure Dependent Demands

Using GIS for Demand Allocation

The consumption of water is the driving force behind the hydraulic dynamics occurring in water distribution systems. When simulating these dynamics in your water distribution model, an accurate representation of system demands is as critical as precisely modeling the physical components of the model.

To realize the full potential of the model as a master planning and decision support tool, you must accurately allocate demands while anticipating future demands. Collecting the necessary data and translating it to model loading data must be performed regularly to account for changes to the network conditions. Due to the difficulties involved in manually loading the model, automated techniques have been developed to assist the modeler with this task.

Spatial allocation of demands is the most common approach to loading a water distribution model. The spatial analysis capabilities of GIS make these applications a logical tool for the automation of the demand allocation process.

LoadBuilder leverages the spatial analysis abilities of your GIS software to distribute demands according to geocoded meter data, demand density information, and coverage polygon intersections.

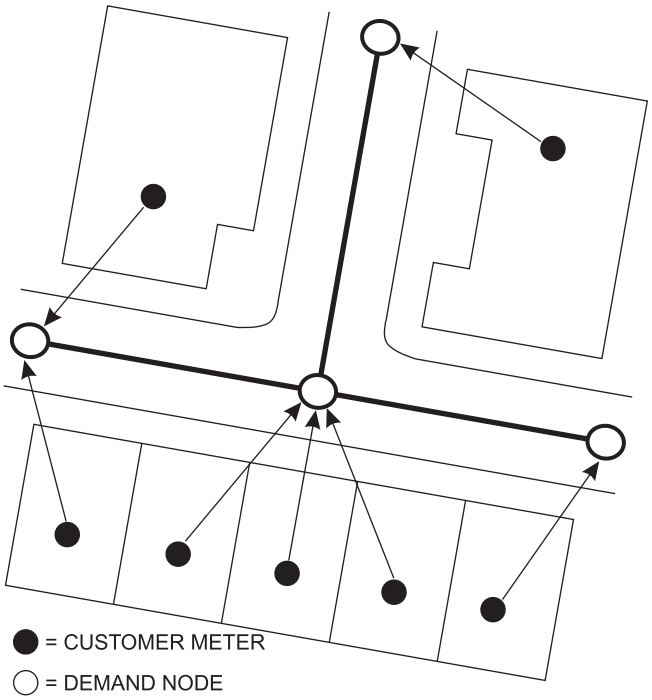
LoadBuilder greatly facilitates the tasks of demand allocation and projection. Every step of the loading process is enhanced, from the initial gathering and analysis of data from disparate sources and formats to the employment of various allocation strategies.

The following are descriptions of the types of allocation strategies that can be applied using LoadBuilder.

Allocation

This uses the spatial analysis capabilities of GIS to assign geocoded (possessing coordinate data based on physical location, such as an x-y coordinate) customer meters to the nearest demand node or pipe. Assigning metered demands to nodes is a point-to-point demand allocation technique, meaning that known point demands (customer meters) are assigned to network demand points (demand nodes). Assigning metered demands to pipes is also a point-to-point assignment technique, since demands must still be assigned to node elements, but there is an additional step involved. When using the Nearest Pipe meter assignment strategy, the demands at a meter are assigned to the

nearest pipe. From the pipe, the demand is then distributed to the nodes at the ends of the pipe by utilizing a distribution strategy. Meter assignment is the simplest technique in terms of required data, because there is no need for service polygons to be applied (see Figure below).



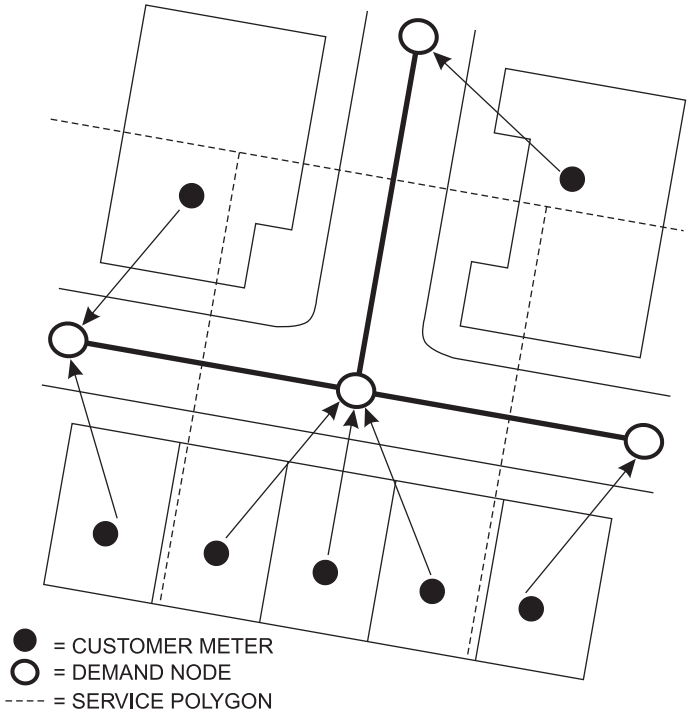
Meter assignment can prove less accurate than the more complex allocation strategies because the nearest node is determined by straight-line proximity between the demand node and the consumption meter. Piping routes are not considered, so the nearest demand node may not be the location from which the meter actually receives its flow. In addition, the actual location of the service meter may not be known.

The geographic location of the meter in the GIS is not necessarily the point from which water is taken from the system, but may be the centroid of the land parcel, the centroid of building footprint, or a point along the frontage of the building. Ideally, these meter points should be placed at the location of the tap, but the centroid of the building or land parcel may be all that is known about a customer account.

Note: In LoadBuilder, the Nearest Node and Nearest Pipe strategies are also in the Allocation loading method.

Billing Meter Aggregation

Billing Meter aggregation is the technique of assigning all meters within a service polygon to a specified demand node (see Figure below). Service polygons define the service area for each of the demand nodes.



Meter Aggregation is a polygon-to-point allocation technique, because the service areas are contained in a GIS polygon layer, while again, the demand nodes are contained in a point layer. The demands associated with the meters within each of the service area polygons is assigned to the respective demand node points.

Due to the need for service polygons, the initial setup for this approach is more involved than the meter assignment strategy, the trade-off being greater control over the assignment of meters to demand nodes. Automated construction of the service polygons may not produce the desired results, so it may be necessary to manually adjust the polygon boundaries, especially at the edges of the drawing.

Note: In LoadBuilder, the Billing Meter Aggregation strategy falls into the meter aggregation category of loading methods.

Distribution

This strategy involves distributing lump-sum area water use data among a number of service polygons (service areas) and, by extension, their associated demand nodes. The lump-sum area is a polygon for which the total (lump-sum) water use of all of the service areas (and their demand nodes) within it is known (metered), but the distribution of the total water use among the individual nodes is not. The water use data for these lump-sum areas can be based on system meter data from pump stations, treatment plants or flow control valves, meter routes, pressure zones, and traffic analysis zones (TAZ). The lump sum area for which a flow is known must be a GIS polygon. There is one flow rate per polygon, and there can be no overlap of or open space between the polygons.

The known flow within the lump-sum area is generally divided among the service polygons within the area using one of two techniques: equal distribution or proportional distribution:

- The **equal flow distribution option** simply divides the known flow evenly between the demand nodes. The equal flow distribution strategy is illustrated in the diagram below. The lump-sum area in this case is a polygon layer that represents meter route areas. For each of these meter route polygons, the total flow is known. The total flow is then equally divided among the demand nodes within each of the meter route polygons (See Figure).
- The **proportional distribution option (by area or by population)** divides the lump-sum flow among the service polygons based upon one of two attributes of the service polygons-the area or the population. The greater the percentage of the lump-sum area or population that a service polygon contains, the greater the percentage of total flow that will be assigned to that service polygon.

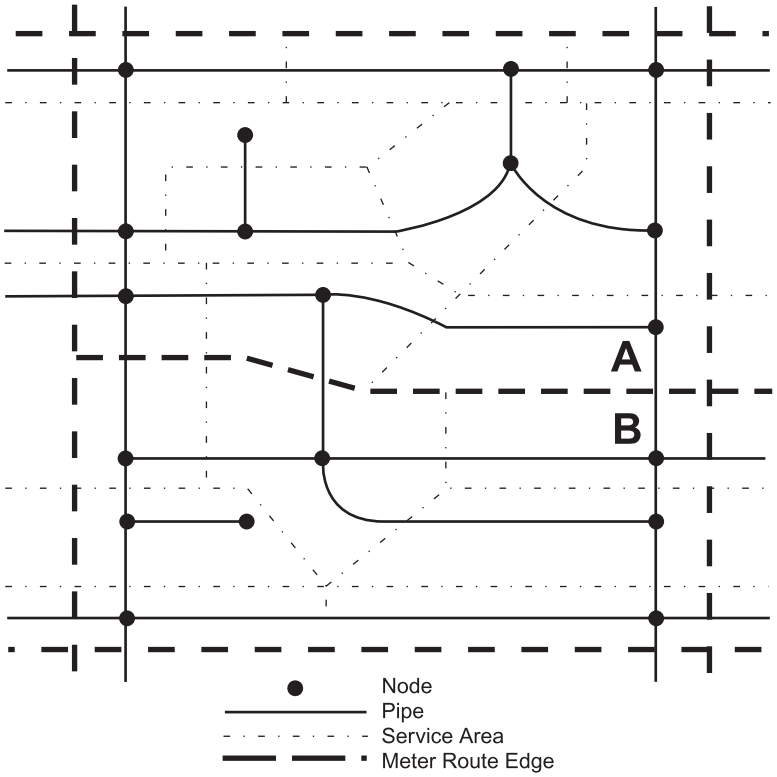
Note: In addition to the distribution options listed above, LoadBuilder allows Nearest node and Farthest node strategies as well.

Each service polygon has an associated demand node, and the flow that is calculated for each service polygon is assigned to this demand node. For example, if a service polygon consists of 50 percent of the lump-sum polygon's area, then 50 percent of the flow associated with the lump-sum polygon will be assigned to the demand node associated with that service polygon. This strategy requires the definition of lump-sum area or population polygons in the GIS, service polygons in the model, and their related demand nodes. Sometimes the flow distribution technique must be used to

assign unaccounted-for-water to nodes, and when any method that uses customer metering data as opposed to system metering data is implemented. For instance, when the flow is metered at the well, unaccounted-for-water is included; when the customer meters are added together, unaccounted-for-water is *not* included.

Note: In LoadBuilder, the Equal Flow Distribution, Proportional Distribution by Area, and Proportional Distribution by Population strategies fall within the flow distribution category of loading methods.

In the following figure, the total demand in meter route A may be 55 gpm (3.48 L/s) while in meter route B the demand is 72 gpm (4.55 L/s). Since there are 11 nodes in meter route A, if equal distribution is used, the demand at each node would be 5 gpm (0.32 L/s), while in meter route B, with 8 nodes, the demand at each node would be 9 gpm (0.57 L/s).



Point Demand Assignment

A point demand assignment technique is used to directly assign a demand to a demand node. This strategy is primarily a manual operation, and is used to assign large (generally industrial or commercial) water users to the demand node that serves the consumer in question. This technique is unnecessary if all demands are accounted for using one of the other allocation strategies.

Projection

Automated techniques have also been developed to assist in the estimation of demands using land use and population density data. These are similar to the Flow Distribution allocation methods except that the type of base layer that is used to intersect with the service layer may contain information other than flow, such as land use or population.

This type of demand estimation can be used in the projection of future demands; in this case, the demand allocation relies on a polygon layer that contains data regarding expected future conditions. A variety of data types can be used with this technique, including future land use, projected population, or demand density (in polygon form), with the polygons based upon traffic analysis zones, census tracts, planning districts, or another classification. Note that these data sources can also be used to assign current demands; the difference between the two being the data that is contained within the source. If the data relates to projected values, it can be used for demand projections.

Many of these data types do not include demand information, so further data conversion is required to translate the information contained in the future condition polygons into projected demand values. This entails translating the data contained within your data source to flow, which can then be applied using LoadBuilder.

After an appropriate conversion method is in place, the service layer containing the service areas and demand nodes is overlaid with the future condition polygon layer(s). A projected demand for each of the service areas can then be determined and assigned to the demand nodes associated with each service polygon. The conversion that is required will depend on the source data that is being used. It could be a matter of translating the data contained within the source, such as population, land area, etc. to flow, which can then be used by LoadBuilder to assign demands.

Depending on how the layers intersect, service areas may contain multiple demand types (land uses) that are added and applied to the demand node for that service polygon.

Using LoadBuilder to Assign Loading Data

LoadBuilder simplifies and expedites the process of assigning loading data to your model, using a variety of source data types.

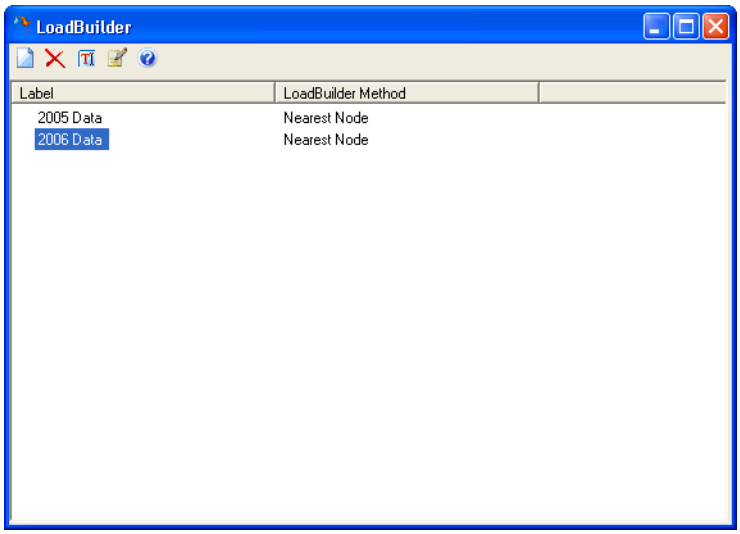
Note: The loading output data generated by LoadBuilder is a Base Flow, i.e., a single value that remains constant over time.

After running LoadBuilder and exporting the results, you may need to modify your data to reflect changes over time by applying patterns to the base flow values.

LoadBuilder Manager

The LoadBuilder manager provides a central location for the creation, storage, and management of Load Build templates.

Go to Tools > Loadbuilder or click  .



The following are available from this dialog box:

New 	Opens the LoadBuilder Wizard.
Delete 	Deletes an existing LoadBuilder template.
Rename 	Renames an existing LoadBuilder template.
Edit 	Opens the LoadBuilder Wizard with the settings associated with the currently highlighted definition loaded.
Help 	Opens the context-sensitive online help.

LoadBuilder Wizard

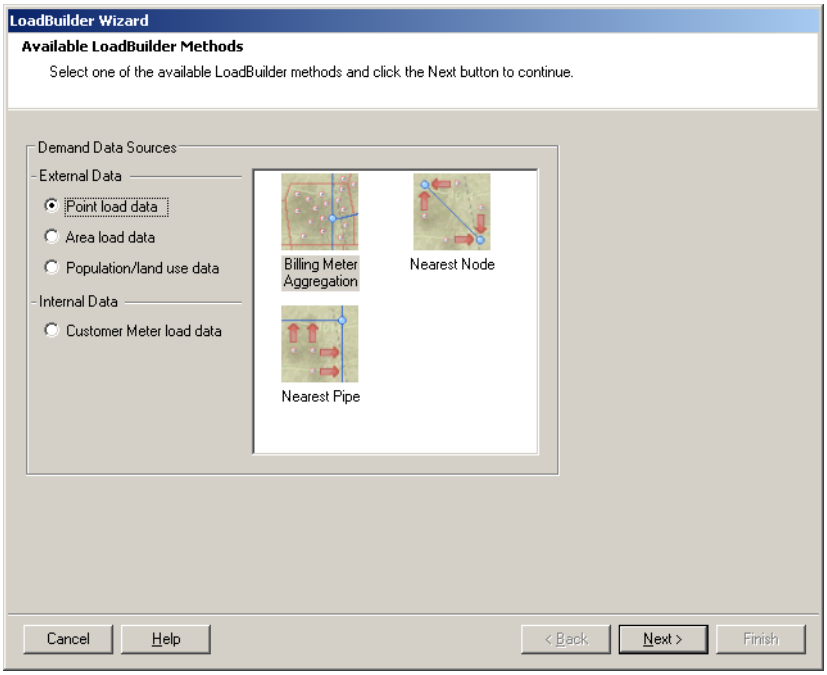
The LoadBuilder wizard assists you in the creation of a new load build template by stepping you through the procedure of creating a new load build template. Depending on the load build method you choose, the specific steps presented in the wizard will vary.

Note: The loading output data generated by LoadBuilder is a Base Flow, i.e., a single value that remains constant over time.

After running LoadBuilder and exporting the results, you may need to modify your data to reflect changes over time by applying patterns to the base flow values.

Step 1: Available LoadBuilder Methods

In this step, the Load Method to be used is specified. The next steps will vary according to the load method that is chosen. The load methods are divided into three categories; the desired category is selected by clicking the corresponding button. Then the method is chosen from the Load Demand types pane.



The available load methods are as follows:

Point Load Data

- **Billing Meter Aggregation**—This loading method assigns all meters within a service polygon to the specified demand node for that service polygon.



- **Nearest Node**—This loading method assigns customer meter demands to the closest demand junction.



Nearest Node

- **Nearest Pipe**—This loading method assigns customer meter demands to the closest pipe, then distributes demands using user-defined criteria.



Nearest Pipe

Area Load Data

- **Equal Flow Distribution**—This loading method equally divides the total flow contained in a flow boundary polygon and assigns it to the nodes that fall within the flow boundary polygon.



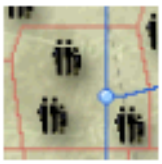
Equal Flow Distribution

- **Proportional Distribution by Area**—This load method proportionally distributes a lump-sum flow among a number of demand nodes based upon the ratio of total service area to the area of the node's corresponding service polygon.



Proportional Distribution By Area

- **Proportional Distribution by Population**—This load method proportionally distributes a lump-sum demand among a number of demand nodes based upon the ratio of total population contained within the node's corresponding service polygon.



Proportional Distribution ...

- **Unit Line**—This load method divides the total demand in the system (or in a section of the system) into 2 parts: known demand (metered) and unknown demand (leakage and unmeasured user demand).



Unit line

See Unit Line Method for more details.

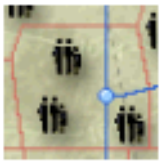
Population/Land Use Data

- **Projection by Land Use**—This method allocates demand based upon the density per land use type of each service polygon.



Projection By Land Use

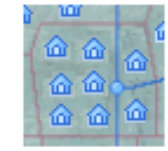
- **Load Estimation by Population**—This method allocates demand based upon user-defined relationships between demand per capita and population data.



Load Estimation By Population

Customer Meter Load Data

- **Customer Meter Aggregation**—This loading method assigns all customers within a service polygon to the specified demand node for that service polygon.



- **Nearest Node**—This loading method assigns customer meter demands to the closest demand junction.



- **Nearest Pipe**—This loading method assigns customer meter demands to the closest pipe, then distributes demands using user-defined criteria.



Step 2: Input Data

The available controls in this step will vary according to the load method type that was specified as follows:

- **Billing Meter Aggregation**—Input Data—The following fields require data to be specified:
  - **Service Area Layer**—Specify the polygon feature class or shapefile that defines the service area for each demand node.
  - **Node ID Field**—Specify the source database field that contains identifying label data.

    Note: ElementID is the preferred Junction ID value because it is always unique to any given element.

  - **Billing Meter Layer**—Specify the point feature class or shapefile that contains the geocoded billing meter data.
  - **Load Type Field**—Specify the source database field that contains load type data. Load Type is an optional classification that can be used to assign composite loads to nodes, which enables different behaviors, multipliers, and patterns to be applied in various situations. For example, possible load types may include Residential, Commercial, Industrial, etc. To make use of the Load Type classification, your source database must include a column that contains this data.
  - **Usage Field**—Specify the source database field that contains usage data. The usage field in the source database must contain flow data. Also, use to select the unit associated with the usage field value.
- **Nearest Node**—Input Data—The following fields require data to be specified:
  - **Node Layer**—Specify the feature class or shapefile that contains the nodes that the loads will be assigned to.
  - **Node ID Field**—Specify the feature class database field that contains the unique identifying label data.

    Note: ElementID is the preferred node ID value because it is always unique to any given element.

  - **Billing Meter Layer**—Specify the feature class or shapefile that contains the geocoded billing meter data.

- **Load Type Field**—Specify the source database field that contains load type data. Load Type is an optional classification that can be used to assign composite loads to nodes, which enables different behaviors, multipliers, and patterns to be applied in various situations. For example, possible load types may include Residential, Commercial, Industrial, etc. To make use of the Load Type classification, your source database must include a column that contains this data.
- **Usage Field**—Specify the source database field that contains usage data. The usage field in the source database must contain flow data. Also, use to select the unit associated with the usage field value.
- **Use Previous Run—**LoadBuilder's most time-consuming calculations when using the Nearest Node strategy are the spatial calculations that are performed to determine proximity between the meter elements and the node elements. When this box is checked, the proximity calculations that were generated from a previous run are used, thereby increasing the overall calculation performance.

• **Nearest Pipe**—Input Data—The following fields require data to be specified:

- **Pipe Layer**—Specify the line feature class or shapefile that contains the pipes that will be used to determine meter-to-pipe proximity. Note that the pipes in this layer must connect to the nodes contained in the **Node Layer**.
- **Pipe ID Field**—Specify the source database field that contains the unique identifying label data.

Note: ElementID is the preferred Pipe ID value because it is always unique to any given element.

- **Load Assignment**—Specify the method that will be used to distribute the metered loads that are assigned to the nearest pipe to the end nodes of said pipe. Options include:
  - **Equal Distribution**—This method assigns an equal portion of the total load assigned to a pipe to each of the pipe's end nodes.
  - **Distance Weighted**—This method assigns a portion of the total load assigned to a pipe based on the distance between the meter(s) and the nodes at the pipe ends. The closer a meter is to the node at the end of the pipe, the more load will be assigned to it.
  - **Closest Node**—This method assigns the entire total load assigned to the pipe end node that is closest to the meter.
  - **Farthest Node**—This method assigns the entire total load assigned to the pipe end node that is farthest from the meter.
- **Node Layer**—Specify the point feature class or shapefile that contains the nodes that will be used to determine node-to-pipe proximity. Note that the nodes in this layer must connect to the pipes contained in the **Pipes Layer**.

– **Node ID Field**—Specify the source database field that contains the unique identifying label data.

Note: ElementID is the preferred Junction ID value because it is always unique to any given element.

– **Use Previous Run**—LoadBuilder's most time-consuming calculations when using the Nearest Pipe strategy are the spatial calculations that are performed to determine proximity between the meter elements, the pipe elements, and the node elements. When this box is checked, the proximity calculations that were calculated from a previous run are used, thereby increasing the overall calculation performance.

– **Billing Meter Layer**—Specify the point or polyline feature class or shapefile that contains the geocoded billing meter data.

– **Billing Meter ID Field**—Billing Meter ID is used to identify the unique meter. When polylines are used to represent water consumption meters, multiple polylines (multiple records) may designate one actual meter, but each (record in the attribute Table) of the polylines contains the same consumption data with the same billing meter ID.

– **Load Type Field**—This field allows you to specify the source database field that contains load type data. Load Type is an optional classification that can be used to assign composite loads to nodes, which enables different behaviors, multipliers, and patterns to be applied in various situations. For example, possible load types may include Residential, Commercial, Industrial, etc. To make use of the Load Type classification, your source database must include a column that contains this data.

– **Polyline Distribution**—When a polyline meter layer is selected, this field will be activated. When multiple pipes are associated with (overlapped by) a polyline meter, the option chosen in this field determines the method that will be used to divide the polyline meter load among them. The available options are:

  - **Equal Distribution**—This option will distribute the load equally among the pipes associated with (overlapping) the meter.

  - **Proportional Distribution**—This option will divide the load proportionally according to the ratio of the length of pipe that is associated with (overlapping) the meter to the total length of the meter.

– **Usage Field**—Specify the source database field that contains usage data. The usage field in the source database must contain flow data. Also, use to select the unit associated with the usage field value.

- **Equal Flow Distribution**—Input Data—The following fields require data to be specified:
  - **Node Layer**—Specify the point feature class or shapefile that contains the nodes that the flow will be assigned to.
  - **Node ID Field**—Specify the source database field that contains identifying label data.

    Note: ElementID is the preferred Node ID value because it is always unique to any given element.

  - **Flow Boundary Layer**—Specify the polygon feature class that contains the flow monitoring meter data.
  - **Flow Field**—Specify the source database field that contains usage data. The usage field in the source database must contain flow data. Also, use to select the unit associated with the usage field value.
- **Proportional Distribution by Area**—Input Data—The following fields require data to be specified:
  - **Service Area Layer**—Specify the polygon feature class or shapefile that defines the service area for each node.
  - **Node ID Field**—Specify the source database field that contains the unique identifying label data.

    Note: ElementID is the preferred Junction ID value because it is always unique to any given element.

  - **Flow Boundary Layer**—Specify the polygon feature class or shapefile that contains the flow boundary data.
  - **Boundary Field**—Specify the source database field that contains the boundary label.
  - **Flow Field**—Specify the source database field that contains usage data. The usage field in the source database must contain flow data. Also, use to select the unit associated with the usage field value.
- **Proportional Distribution by Population**—Input Data—The following fields require data to be specified:
  - **Service Area Layer**—Specify the polygon feature class or shapefile that defines the service area for each node.
  - **Node ID Field**—Specify the source database field that contains the unique identifying label data.

Note: ElementID is the preferred Junction ID value because it is always unique to any given element.

- **Flow Boundary Layer**—Specify the polygon feature class or shapefile that contains the flow boundary data.
- **Boundary Field**—Specify the source database field that contains the boundary label.
- **Flow Field**—Specify the source database field that contains usage data. The usage field in the source database must contain flow data. Also, use to select the unit associated with the usage field value.
- **Population Layer**—Specify the polygon feature class or shapefile that contains population data.
- **Population Count Field**—Specify the source database field that contains population data.
- **Land Type Field**—Specify the source database field that contains land use type.

- **Unit Line**—Input Data—The following fields require data to be specified:
  - **Include known demands in results**—When this box is checked the Demand Alternative field is activated, allowing you to specify a demand alternative whose demands will be included in the results.
  - **Demand Alternative**—Select a demand alternative to use when the Include known demands in results box is checked.
  - **K Factor Field**—Specify the user-defined attribute field that contains K-Factor data. You can add the user-defined field to the project by clicking the ellipsis button and specifying a default K-Factor.
  - **Include**—Check the box next to each element type (junctions, tanks, and hydrants) you want included in the calculation.
  - **Unaccounted-for Demand by Selection Set Table**—This table allows you to assign unaccounted-for demands by selection set. Click the new button to add a row to the table, then choose a selection set (or Entire Network to include all applicable elements) and specify an unaccounted-for demand value. Highlight a row and click the Delete button to remove it.

- **Projection by Land Use**—Input Data—The following fields require data to be specified:
  - **Service Area Layer**—Specify the polygon feature class or shapefile that defines the service area for each node.
  - **Node ID Field**—Specify the source database field that contains the unique identifying label data.

Note: ElementID is the preferred Junction ID value because it is always unique to any given element.

- **Land Use Layer**—Specify the polygon feature class or shapefile that contains the land use data.
- **Land Type Field**—Specify the source database field that contains land use type.
- **Load Type and Load Density**—Use this table to assign load density values to the various load types contained within your land use layer.

- **Load Estimation by Population**—Input Data—The following fields require data to be specified:
  - **Service Area Layer**—Specify the polygon feature class or shapefile that defines the service area for each node.
  - **Node ID Field**—Specify the source database field that contains identifying label data.

Note: ElementID is the preferred Junction ID value because it is always unique to any given element.

  - **Population Layer**—Specify the polygon feature class or shapefile that contains the population data.
  - **Population Density Type Field**—Specify the source database field that contains the population density type data.
  - **Population Density Field**—Specify the source database field that contains population density data.
  - **Load Type and Load Density**—Use this table to assign load density values to the various load types contained within your population density layer.
- **Customer Meter Aggregation**—The following fields require data to be specified:
  - **Service Area Layer**—Specify the polygon feature class or shapefile that defines the service area for each demand node.
  - **Node ID Field**—Specify the source database field that contains identifying label data.

Note: ElementID is the preferred Junction ID value because it is always unique to any given element.

  - **Customer Meter Layer**—Specify the point feature class or shapefile that contains the customer meter data.
  - **Customer Meter ID Field**—Specify the source database field that contains identifying label data.

  - **Only Process Unassigned Customer Meters**—The engine will process only those customer elements that are not associated to any node or pipe.

- **Nearest Node**—The following fields require data to be specified:
  - **Node Layer**—Specify the feature class or shapefile that contains the nodes that the loads will be assigned to.
  - **Node ID Field**—Specify the source database field that contains identifying label data.

    Note: ElementID is the preferred Junction ID value because it is always unique to any given element.

  - **Customer Meter Layer**—Specify the point feature class or shapefile that contains the customer meter data.
  - **Customer Meter ID Field**—Specify the source database field that contains identifying label data.
  - **Only Process Unassigned Customer Meters**—The engine will process only those customer elements that are not associated to any node or pipe.
- **Nearest Pipe**—The following fields require data to be specified:
  - **Pipe ID Field**—Specify the source database field that contains the unique identifying label data.

    Note: ElementID is the preferred Pipe ID value because it is always unique to any given element.

  - **Load Assignment**—Specify the method that will be used to distribute the customer loads that are assigned to the nearest pipe to the end nodes of said pipe. Options include:
    - **Equal Distribution**—This method assigns an equal portion of the total load assigned to a pipe to each of the pipe's end nodes.
    - **Distance Weighted**—This method assigns a portion of the total load assigned to a pipe based on the distance between the meter(s) and the nodes at the pipe ends. The closer a meter is to the node at the end of the pipe, the more load will be assigned to it.
    - **Closest Node**—This method assigns the entire total load assigned to the pipe end node that is closest to the meter.
    - **Farthest Node**—This method assigns the entire total load assigned to the pipe end node that is farthest from the meter.
  - **Node Layer**—Specify the feature class or shapefile that contains the nodes that the loads will be assigned to.
  - **Node ID Field**—Specify the source database field that contains identifying label data.

Note: ElementID is the preferred Junction ID value because it is always unique to any given element.

- **Customer Meter Layer**—Specify the point feature class or shapefile that contains the customer meter data.
- **Customer Meter ID Field**—Specify the source database field that contains identifying label data.
- **Only Process Unassigned Customer Meters**—The engine will process only those customer elements that are not associated to any node or pipe.

Step 3: Calculation Summary

This step displays the Results Summary pane, which displays the total load, load multiplier, and hydraulic pattern associated with each load type in a tabular format. The number of entries listed will depend on the load build method and data types selected in Step 1.

Note: Different types of shapefiles may need to be created based on the loadbuilder method selected.

The Results Summary pane contains the following columns:

- **Load Type**—This column contains an entry for each load type contained within the database column specified in step one. (Examples include Residential, Commercial, Industrial, etc.)
- **Consumption**—This column displays the total load associated with each load type entry.
- **Multiplier**—This column displays the multiplier that is applied to each load type entry. Multipliers can be used to account for peak loads, expected future loads, or to reflect unaccounted-for-loads. This field can be edited.
- **Pattern**—This column displays the hydraulic pattern associated with each demand type entry. A different pattern can be specified using the menu contained within each cell of this column. New patterns cannot be created from this dialog box; see the Pattern manager help topic for more information regarding the creation of new patterns.

In addition to the functionality provided by the tabular summary pane, the following controls are also available in this step:

- **Global Multiplier**—This field allows you to apply a multiplier to all of the entries contained within the Results Summary Pane. Any changes are automatically reflected in the Total Load text field. Multipliers can be used to account for peak loads, expected future loads, or to reflect unaccounted-for-loads. The Global Multiplier should be used when the conditions relating to these considerations are identical for all usage types and elements.
- **Total Load**—This field displays an updated total of all of the entries contained within the Results Summary Pane, as modified by the local and global multipliers that are in effect.

Step 4: Results Preview

This step displays the calculated results in a tabular format. The table consists of the following information:

- **Node ID**—The unique identifying label assigned to all geodatabase elements by the GIS.
- **Label**—The unique identifying label assigned by Bentley WaterCAD V8*i* Modeler.
- **Load Type**—An optional classification that can be used to assign different behaviors, multipliers, and patterns in various situations. For example, possible load types may include Residential, Commercial, Industrial, etc. To make use of the Load Type classification, your source database must include a column that contains this data.
- **Pattern**—The type of pattern assigned to the node. The source database must include a column that contains this data.

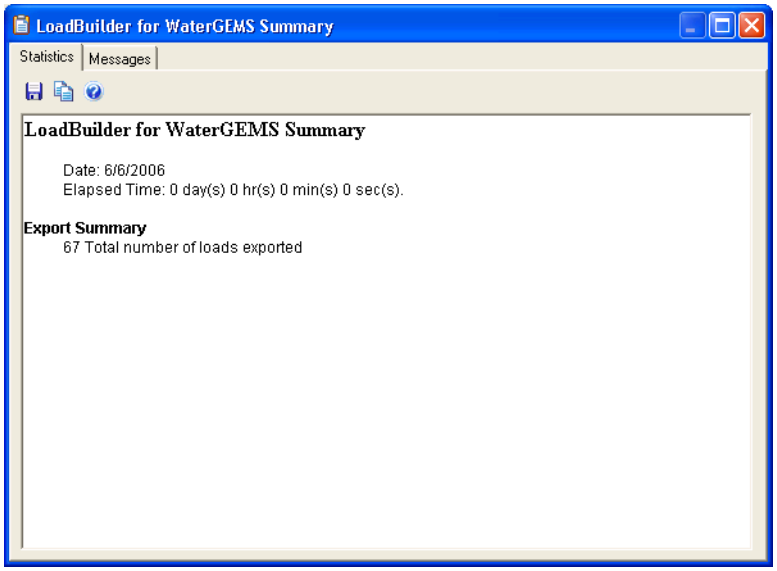
Step 5: Completing the LoadBuilder Wizard

In this step, the load build template is given a label and the results are exported to an existing or new load alternative. This step contains the following controls:

- **Label**—This field allows a unique label to be assigned to the load build template.
- **Override an Existing Alternative**—Choosing this option will cause the calculated loads to overwrite the loads contained within the existing load alternative that is selected.
- **Append to an Existing Alternative**—Choosing this option will cause the calculated loads to be appended to the loads contained within the existing load alternative that is selected. Loads within the existing alternative that are assigned to a specific node will not be overwritten by newly generated loads assigned to the same node; the new loads will be added to them.
- **New Alternative**—Choosing this option will cause the calculated loads to be applied to a new load alternative. Enter your text into this field. The Parent Alternative field will only be active when this option is selected.

LoadBuilder Run Summary

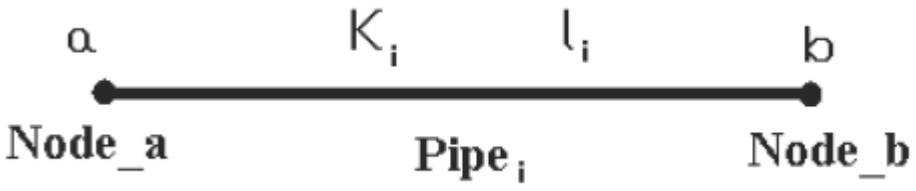
The LoadBuilder Run Summary dialog box details important statistics about the results of a completed LoadBuilder run, including the number of successfully added loads, file information, and informational and/or warning messages.



Unit Line Method

The Unit Line Flow Method divides the total demand in the system (or in a section of the system) into 2 parts: known demand (metered) and unknown demand (leakage and unmeasured user demand).

The following diagram shows a sample pipe. The known (metered) demands at *nodes a and b* are q_a and q_b respectively. The unknown demand is computed by considering if there are users on none, one, or both sides of the pipe. This is accounted for using the coefficient, K.



Where

l_i = length of $Pipe_i$

K_i = coefficient indicating the capability of Pipei to consume water

If there are no users on either side of the pipe (the pipe is only used to transfer water to another part of the system), then K is 0. If there are users along only one side of the pipe (for example, pipes along a river), K is 0.5. If both sides of the pipe supply water to users, K is 1.

The equations below are used to determine the total demands at nodes a and b:

$$Q_a = q_a + \frac{1}{2} \times \frac{Q_{totalunknown}}{\left(\sum_{j=1}^{n} K_j \times l_j \right)} \times \sum_{i=1}^{m} K_i \times l_i$$

$$Q_b = q_b + \frac{1}{2} \times \frac{Q_{totalunknown}}{\left(\sum_{j=1}^{n} K_j \times l_j \right)} \times \sum_{i=1}^{m} K_i \times l_i$$

Where

Q_a = the total demand at node *a*

Q_b = the total demand at node *b*

q_a = The known demand at node *a*

q_b = The known demand at node *b*

$Q_{total\ unknown}$ = Total real demand minus total known demand(for the network or selection set)

n = number of pipes in network (or selection set)

m = the number of pipes connected to node *a* or *b*

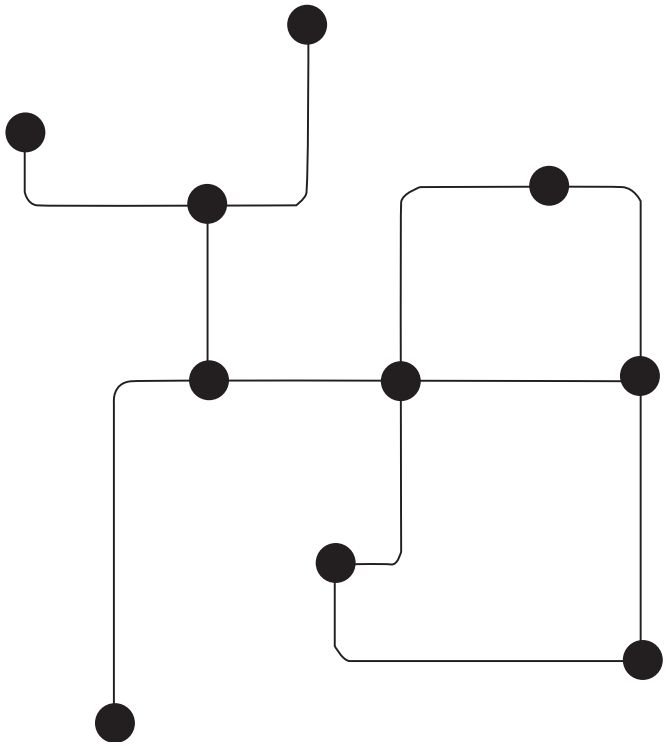
Generating Thiessen Polygons

A Thiessen polygon is a Voronoi Diagram that is also referred to as the Dirichlet Tessellation. Given a set of points, it defines a region around each point. A Thiessen polygon divides a plane such that each point is enclosed within a polygon and assigns the area to a point in the point set. Any location within a particular Thiessen polygon is nearer to that polygon's point than to any other point. Mathematically, a Thiessen is constructed by intersecting perpendicular bisector lines between all points.

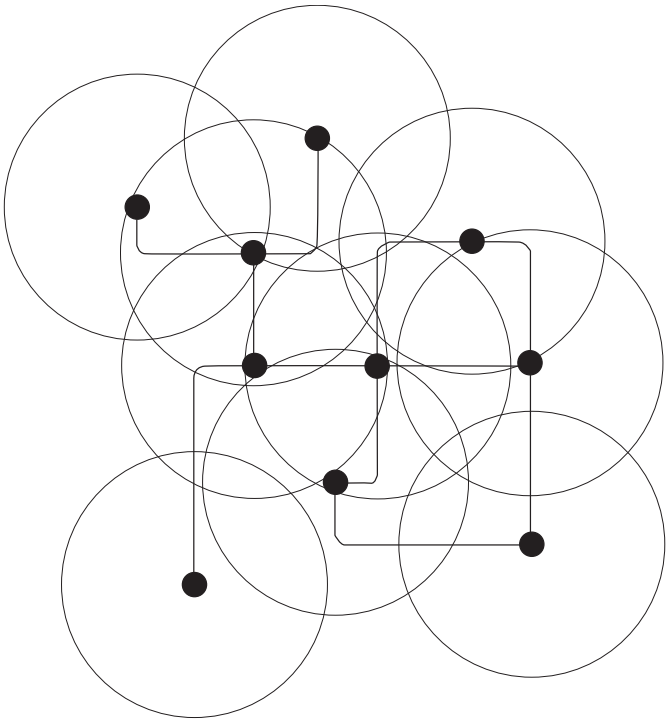
Thiessen polygon has many applications in different location-related disciplines such as business planning, community services, transportation and hydraulic/hydrological modeling. For water distribution modeling, the Thiessen Polygon Creator was developed to quickly and easily define the service areas of demand nodes. Since each customer within a Thiessen polygon for a junction is nearer to that node than any others, it is assumed that the customers within a particular Thiessen polygon are supplied by the same demand node.

The following diagrams illustrate how Thiessen polygons would be generated manually. The Thiessen Polygon Creator does not use this method, although the results produced by the generator are consistent with those that would be obtained using this method.

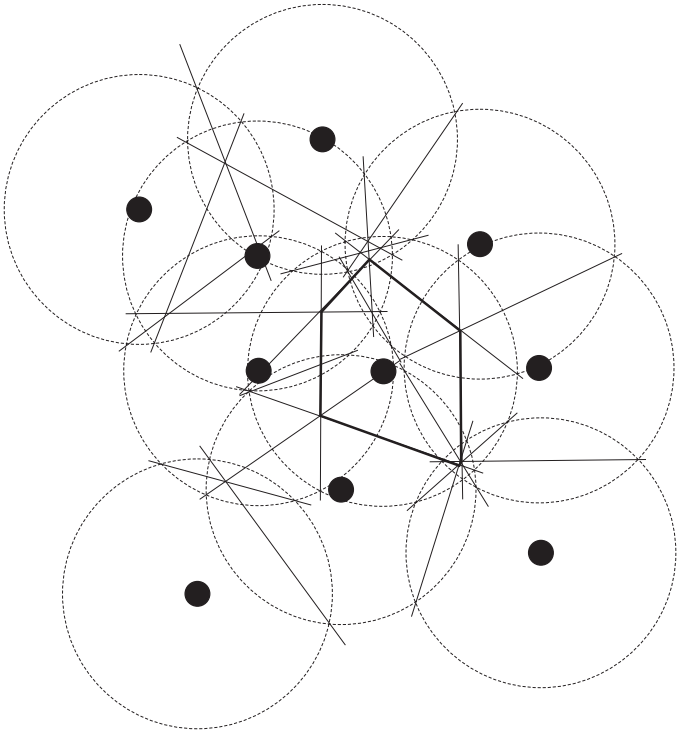
The first diagram shows a pipe and junction network.



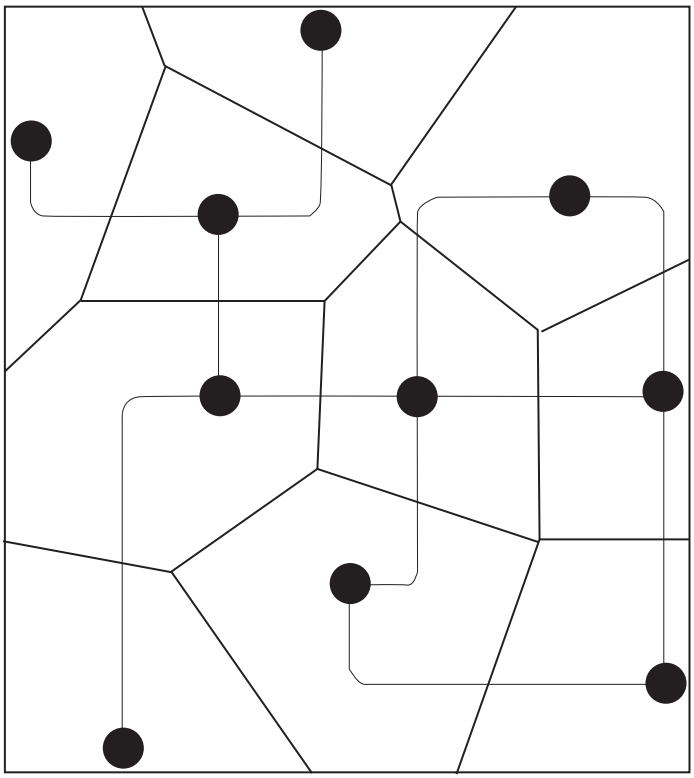
In the second diagram, the circles are drawn around each junction.



In the third diagram, bisector lines are added by drawing a line where the circles inter-join.



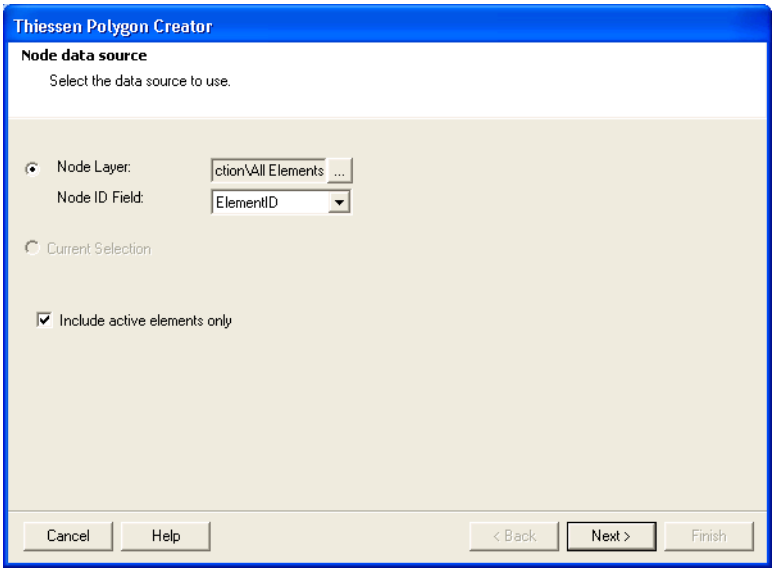
In the final diagram, the network is overlaid with the polygons that are created by connecting the bisector lines.



Thiessen Polygon Creator Dialog Box

The Thiessen Polygon Creator allows you to quickly create polygon layers for use with the LoadBuilder demand allocation module. This utility creates polygon layers that can be used as service area layers for the following LoadBuilder loading strategies:

- Billing Meter Aggregation
- Proportional Distribution By Area
- Proportional Distribution By Population
- Projection by Land Use
- Load Estimation by Population.



The Thiessen Polygon Creator dialog box consists of the following controls:

- **Node Data Source**—Select the data source to use.
  - **Node Layer**—This lists the valid point feature classes and shapefiles that Thiessen Polygon Creator can use.
  - **Current Selection**—Click if the current feature data set contains a previously created selection set.
  - **Include active elements only**—Click to activate.
  - **Selection**—This option allows you to create a selection on the fly for use with the Thiessen Polygon Creator. To use this option, use the ArcMap **Select Features** tool to select the point features that you want before opening the Thiessen Polygon Creator.
- **Buffering Percentage**—This percentage value is used for calculating the boundary for a collection of points. In order to make the buffer boundary big enough to cover all the points, the boundary is enlarged based upon the value entered in this field as it relates to the percentage of the area enclosed by drawing a polygon that connects the outermost nodes of the model.
- **Polygon Boundary Layer**—Select the boundary polygon feature class or shapefile, if one has already been created. A boundary is specified so that the outermost polygons do not extend to infinity.
- **Output File**—Specify the name of the shapefile that will be created.

Note: The Thiessen Polygon Creator is flexible enough to generate Thiessen polygons for unusual boundary shapes, such as borders with cutouts or holes that Thiessen polygons should not be created inside. To accomplish this, the boundary polygon must be created as one complex (multi-part) polygon. For more information about creating boundary polygon feature classes, see your ArcGIS documentation.

Creating Boundary Polygon Feature Classes

The Thiessen Polygon Creator requires a boundary to be specified around the area in which Thiessen Polygons will be created. This is to prevent the outside edge of the polygons along the perimeter of this area from extending to infinity. The generator can automatically create a boundary using the **Buffering Percentage** value, or it can use a previously created polygon feature class as the boundary.

A border polygon feature class can be created in ArcCatalog and edited in ArcMap.

To create a border feature class, you will need a Bentley WaterCAD V8*i* model that has had at least one scenario published as an ESRI feature data set. Then, follow these steps:

1. In the directory structure pane of ArcCatalog, right-click the Bentley WaterCAD V8*i* feature data set and select **New** > **Feature Class**.
2. A dialog box will open, prompting you to name the new feature class. Enter a name and click **Next**.
3. In the second step, you are prompted to select the database storage configuration. Do so, and click **Next**.
4. In the third step, click the **Shape** cell under the **Field Name** column, and ensure that the **Geometry Type** is **Polygon**. Click **Finish**.
5. In ArcMap, click the **Add Data** button and select your Bentley WaterCAD V8*i* feature dataset. 
6. Click the **Editor** button and select **Start Editing**. Ensure that the border feature class is selected in the **Target** drop-down list.
7. Draw a polygon around the point features (generally junctions) that you wish to be used to generate the polygons. When you are finished drawing the polygon, click **Editor...Stop Editing**. Choose **Yes** when prompted to save your edits.

The polygon feature class you just created can now be used as the boundary during Thiessen polygon generation. For more information about creating and editing feature classes, see your ArcGIS documentation.

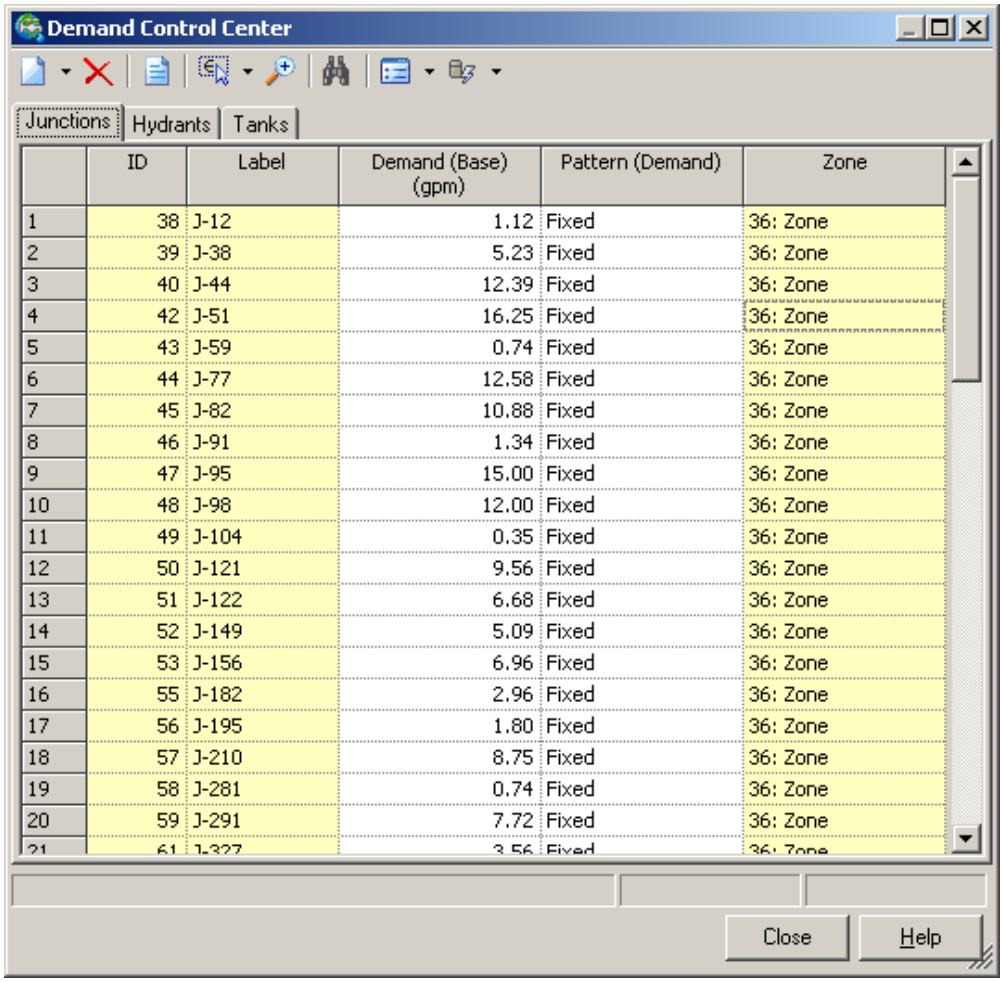
Demand Control Center

The Demand Control Center is an editor for manipulating all the demands in your water model. Using the Demand Control Center, you can add new demands, delete existing demands, or modify the values for existing demands using standard SQL select and update queries.

The Demand Control Center provides demand editing capabilities which can:

- open on all demand nodes, or subset of demand nodes,
- sort and filter based on demand criteria or zone,
- add, edit, and delete individual demands,
- global edit demands,
- provides access to statistics for the demands listed in the table,
- and filter elements based on selection set, attribute, predefined query, or zone.

In order to access the Demand Control Center go to Tools > Demand Control Center or click Demand Control. The Demand Control Center opens.



	ID	Label	Demand (Base) (gpm)	Pattern (Demand)	Zone
1	38	J-12	1.12	Fixed	36: Zone
2	39	J-38	5.23	Fixed	36: Zone
3	40	J-44	12.39	Fixed	36: Zone
4	42	J-51	16.25	Fixed	36: Zone
5	43	J-59	0.74	Fixed	36: Zone
6	44	J-77	12.58	Fixed	36: Zone
7	45	J-82	10.88	Fixed	36: Zone
8	46	J-91	1.34	Fixed	36: Zone
9	47	J-95	15.00	Fixed	36: Zone
10	48	J-98	12.00	Fixed	36: Zone
11	49	J-104	0.35	Fixed	36: Zone
12	50	J-121	9.56	Fixed	36: Zone
13	51	J-122	6.68	Fixed	36: Zone
14	52	J-149	5.09	Fixed	36: Zone
15	53	J-156	6.96	Fixed	36: Zone
16	55	J-182	2.96	Fixed	36: Zone
17	56	J-195	1.80	Fixed	36: Zone
18	57	J-210	8.75	Fixed	36: Zone
19	58	J-281	0.74	Fixed	36: Zone
20	59	J-291	7.72	Fixed	36: Zone
21	61	J-327	3.56	Fixed	36: Zone

The Demand Control Center toolbar includes the following:

New



Clicking this button opens a submenu containing the following commands:

- **Add Demand to Element**—Adds a row to the table, allowing you to assign a demand and demand pattern to the element that is currently highlighted in the list.
- **Add Demand**—Opens the Domain Element Search box, allowing you to select elements in the drawing pane and assign a demand and demand pattern to them.
- **Initialize Demands for All Elements**—Adds a row to the table for each element (each junction if executed on the Junction tab, each hydrant if executed on the Hydrant tab, etc.) in the model that does not currently have a demand assigned to it. The initialized rows will assign a Base Flow of 0 and a Fixed demand pattern to the associated elements.

Delete



Deletes an existing demand.

Report



Generates a demand report based on the contents of the table.

Create or Add to a Selection Set



Creates a new selection set containing the currently selected elements, adds currently selected elements to an existing selection set, or removes currently selected elements from a selection set.

Zoom Zooms to a specific element.



Find Opens the Domain Element Search editor.



Options Provides access to global sort and filter capabilities.



Query Opens a submenu allowing you to filter the table according to one of the following:



- **Selection Set**: The submenu contains a list of previously created selection sets. If you choose a selection set only those elements contained in that selection set will be displayed.
- **Attribute**: If this command is selected, the Query Builder opens, allowing you to diaply only those elements that meet the criteria of the query you create.
- **Predefined Queries**: The submenu contains a number of predefined queries grouped categorically. For more information about these queries, see Using the Network Navigator.

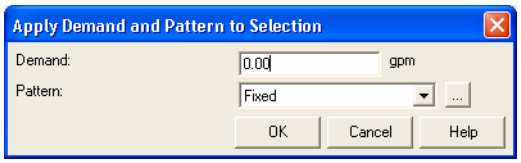
Note: To view statistics for the demands listed in the Demand Control Center, right-click the Demand column heading and select Statistics from the context menu.

Apply Demand and Pattern to Selection Dialog Box

This dialog allows you to assign a demand and demand pattern to the currently selected element or elements. The dialog appears after you have used the Add Demands command in the Demand Control Center or the Unit Demand Control Center and then selected one or more elements in the drawing pane. The dialog itself will vary depending on whether it was accessed from the Demand Control Center or the Unit Demand Control Center.

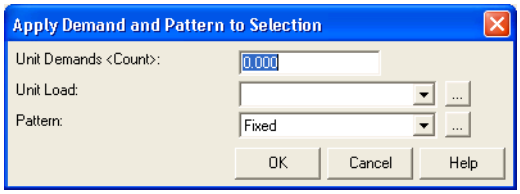
From the Demand Control Center

Enter a demand value in the Demand field, then choose a previously created pattern in the Pattern list, create a new pattern by clicking the ellipsis button to open the Patterns dialog, or leave the default value of Fixed if the demand does not vary over time.



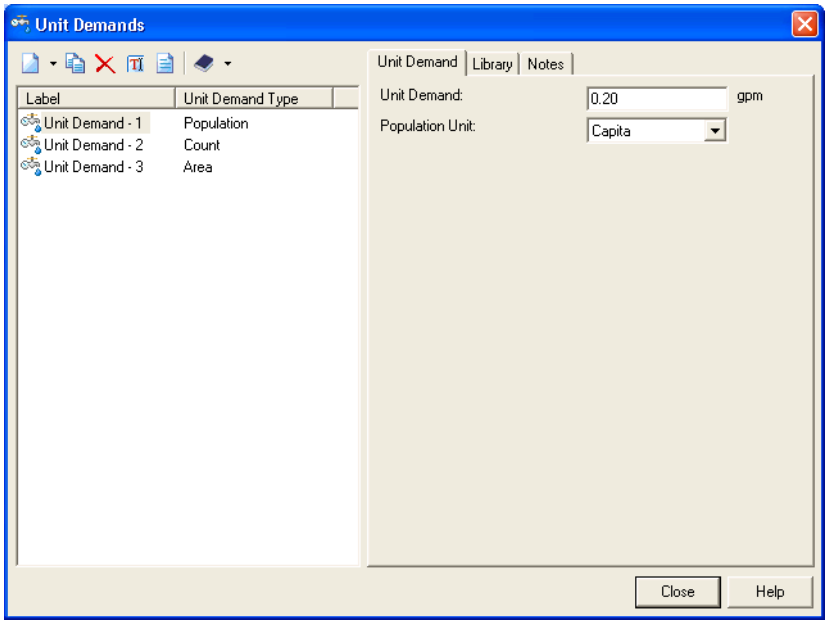
From the Unit Demand Control Center

Enter the number of individual unit demands in the Unit Demands <Count> field. Choose a previously defined unit load from the Unit Load list, or create a new one in the Unit Demands dialog by clicking the ellipsis button. Choose a previously created pattern in the Pattern list, create a new pattern by clicking the ellipsis button to open the Patterns dialog, or leave the default value of Fixed if the demand does not vary over time.



Unit Demands Dialog Box

The Unit Demands dialog box allows you to create unit-based demands that can later be added to model nodes.



A unit demand consists of a unit (person, area) multiplied by a unit demand (gal/capita/day, liters/sq m/day, cfs/acre). The units are assigned to node elements (like junctions) while the unit demands are created using the Unit Demands dialog box. If the unit demands are not assigned to nodes but to polygons in a GIS, then it is best to use LoadBuilder to import the loads.

There are two sections of the Unit Demands dialog box: the Unit Demands Pane on the left and the tab section on the right. The Unit Demands Pane is used to create, edit, and delete unit demands. This section contains the following controls:

New 	Creates a new unit demand. When you click the new button, a submenu opens containing the following choices: • Area—Creates a new Area-based unit demand. • Count—Creates a new Count-based unit demand. • Population—Creates a new Population-based unit demand.
Duplicate 	Copies the currently selected unit demand.
Delete 	Deletes the currently highlighted unit demand. You can hold down the Ctrl key while clicking on items in the list to select multiple entries at once.
Rename 	Renames the currently highlighted unit demand.
Report 	Generates a detailed report on the selected unit demand.
Synchronization Options 	Browses the Engineering Library, synchronizes to or from the library, imports from the library or exports to the library.

The tab section is used to define the settings for the unit demand that is currently highlighted in the unit demands list pane.

The following controls are available:

Unit Demand Tab

This tab consists of input data fields that allow you to define the unit demand. The available controls will vary depending on the type of unit demand being defined.

Population Unit Demand

- **Unit Demand**—Lets you specify the amount of demand required per population unit.
- **Population Unit**—Lets you specify the base unit used to define the population-based demand.

Count Unit Demand

- **Unit Demand**—Lets you specify the amount of demand required per count unit.
- **Count Unit**—Lets you specify the base unit used to define the unit-based demand.
- **Report Population Equivalent**—Checking this box enables the Population Equivalent field, letting you specify the equivalent population count per demand unit.
- **Population Equivalent**—When the Report Population Equivalent box is checked, this field lets you specify the equivalent population count per demand unit. For area based demands, this is essentially a population density, or population per unit area.

Area Unit Demand

- **Unit Demand**—Lets you specify the amount of demand required per area unit.
- **Area Unit**—Lets you specify the base unit used to define the area-based demand.
- **Report Population Equivalent**—Checking this box enables the Population Equivalent field, letting you specify the equivalent population count per demand unit.
- **Population Equivalent**—When the Report Population Equivalent box is checked, this field lets you specify the equivalent population count per demand unit. For area based demands, this is essentially a population density, or population per unit area.

Library Tab

This tab displays information about the unit demand that is currently highlighted in the Unit Demand list pane. If the unit demand is derived from an engineering library, the synchronization details can be found here. If the unit demand was created manually for this project, the synchronization details will display the message Orphan (local), indicating that the unit demand was not derived from a library entry.

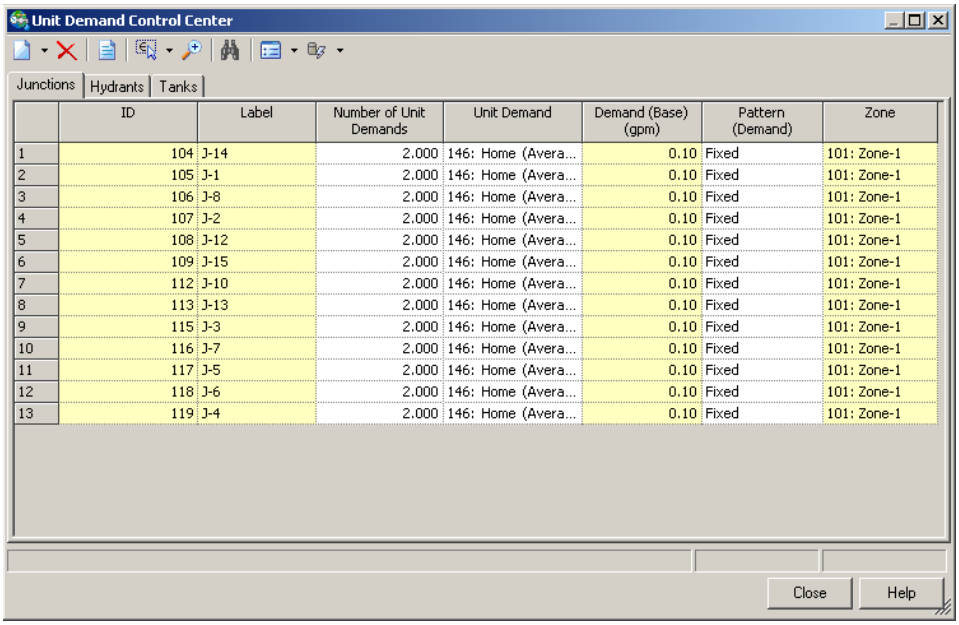
Notes Tab

This tab contains a text field that is used to type descriptive notes that will be associated with the unit demand that is currently highlighted in the Unit Demand list pane.

Unit Demand Control Center

The Unit Demand Control Center is an editor for manipulating all the unit demands in your water model. Using the Unit Demand Control Center, you can add new unit demands, delete existing unit demands, or modify the values for existing unit demands. You can also and filter elements based on demand criteria, pattern, or zone.

In order to access the Unit Demand Control Center go to Tools > Unit Demand Control Center or click the Unit Demand Control Center icon. The Unit Demand Control Center opens.



	ID	Label	Number of Unit Demands	Unit Demand	Demand (Base) (gpm)	Pattern (Demand)	Zone
1	104	J-14	2.000	146: Home (Avera...	0.10	Fixed	101: Zone-1
2	105	J-1	2.000	146: Home (Avera...	0.10	Fixed	101: Zone-1
3	106	J-8	2.000	146: Home (Avera...	0.10	Fixed	101: Zone-1
4	107	J-2	2.000	146: Home (Avera...	0.10	Fixed	101: Zone-1
5	108	J-12	2.000	146: Home (Avera...	0.10	Fixed	101: Zone-1
6	109	J-15	2.000	146: Home (Avera...	0.10	Fixed	101: Zone-1
7	112	J-10	2.000	146: Home (Avera...	0.10	Fixed	101: Zone-1
8	113	J-13	2.000	146: Home (Avera...	0.10	Fixed	101: Zone-1
9	115	J-3	2.000	146: Home (Avera...	0.10	Fixed	101: Zone-1
10	116	J-7	2.000	146: Home (Avera...	0.10	Fixed	101: Zone-1
11	117	J-5	2.000	146: Home (Avera...	0.10	Fixed	101: Zone-1
12	118	J-6	2.000	146: Home (Avera...	0.10	Fixed	101: Zone-1
13	119	J-4	2.000	146: Home (Avera...	0.10	Fixed	101: Zone-1

The Unit Demand Control Center toolbar includes the following:

New 	Add Demands opens the Domain Element Search dialog box, allowing you to search for the element to include. Once you've added an element, you can choose to Add Demand to Element, and the element that is selected is duplicated. Initialize Demands for All Elements adds all the demand elements to the control center.
Delete 	Deletes an existing unit demand.
Report 	Generates a unit demand report based on the contents of the table.
Create or Add to a Selection Set 	Creates a new selection set containing the currently selected elements, adds currently selected elements to an existing selection set, or removes currently selected elements from a selection set.

Zoom Zooms to a specific element.



Find Opens the Domain Element Search editor.



Options Provides access to global sort and filter capabilities.



Query Opens a submenu allowing you to filter the elements displayed based on a number of predefined queries. For more information about the .available queries, see Using the Network Navigator.



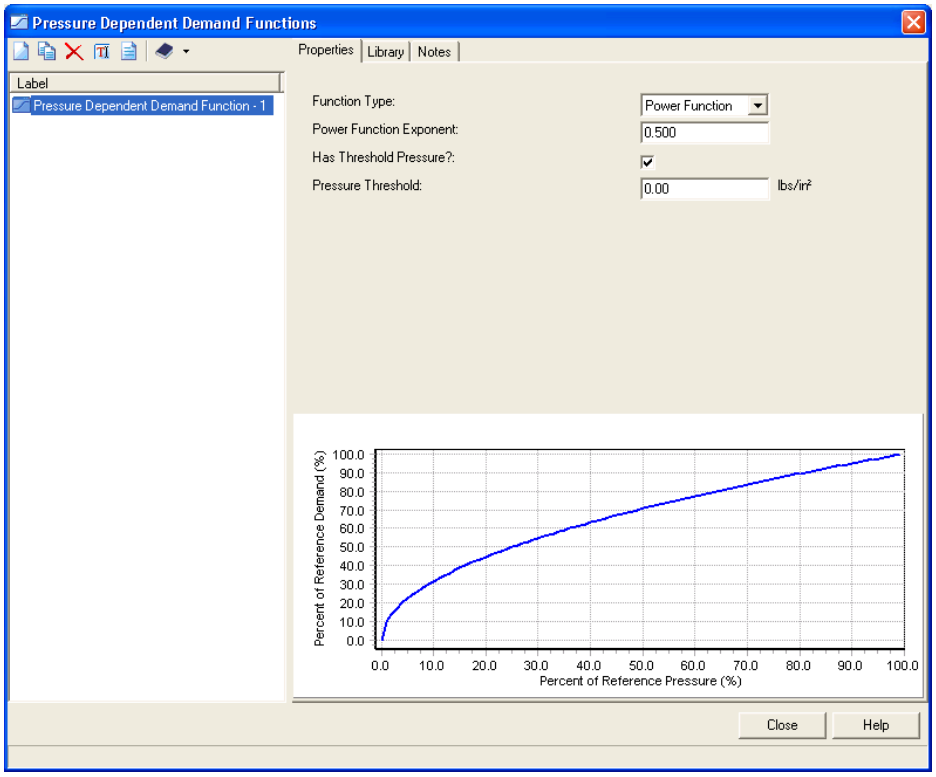
Note: To view statistics for the demands listed in the Unit Demand Control Center, right-click the Unit Demand or Demand (Base) column headings and select Statistics from the context menu.

Pressure Dependent Demands

Pressure Dependent Demands (PDD) allows you to perform hydraulic simulation by treating the nodal demand as a variable of nodal pressure. Using PDD you can perform hydraulic simulation for:

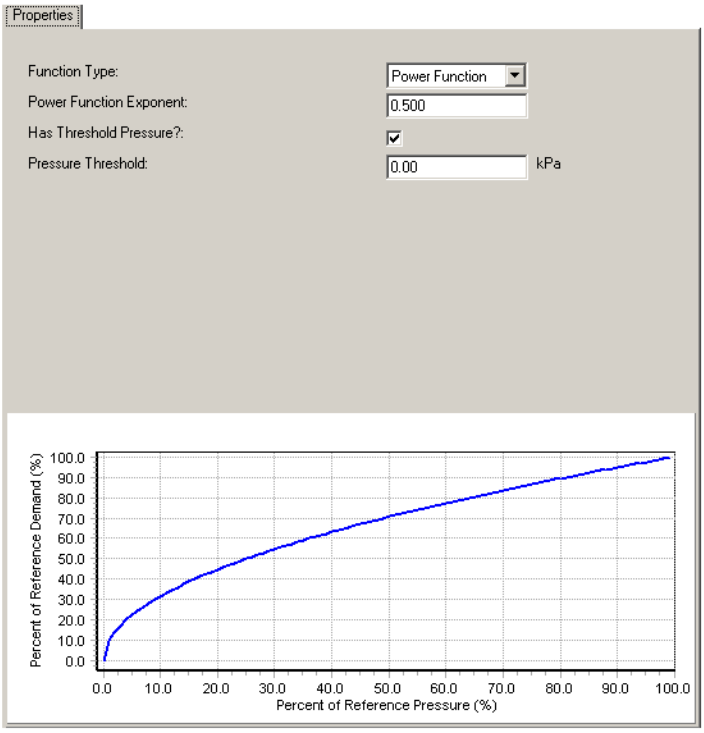
- Pressure dependent demand at a node or a set of nodes
- Combination of PDD and volume based demand
- Calculate the actual supplied demand at a PDD node and demand shortfall
- Present the calculated PDD and the associated results in a table and graph.

In order to access PDD choose Components > Pressure Dependent Demand Functions or click Pressure Dependent Demand Functions to open the Pressure Dependent Demand Functions dialog box.



New 	Creates a a new pressure dependent demand function.
Duplicate 	Copies the currently selected demand.
Delete 	Deletes an existing demand. You can hold down the Ctrl key while clicking on items in the list to select multiple entries at once.
Rename 	Renames an existing pressure dependent demand function.
Report 	Generates a pressure dependent demand report based on the selected demand.
Synchronization Options 	Browses the Engineering Library, synchronizes to or from the library, imports from the library or exports to the library.

Properties tab

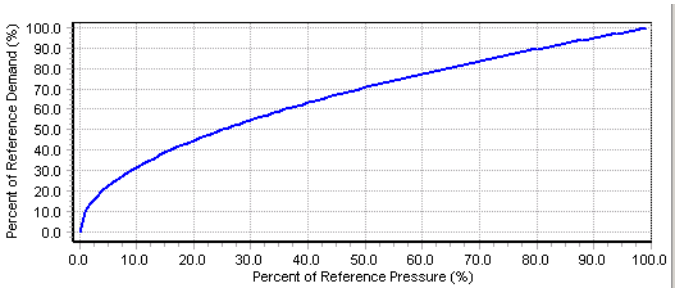


Function Type - Either Power Function or Piecewise Linear. Power Function is used to define the exponential relationship between the nodal pressure and demand. The ratio of actual supplied demand to reference demand is defined as a power function of the ratio of actual pressure to reference pressure.

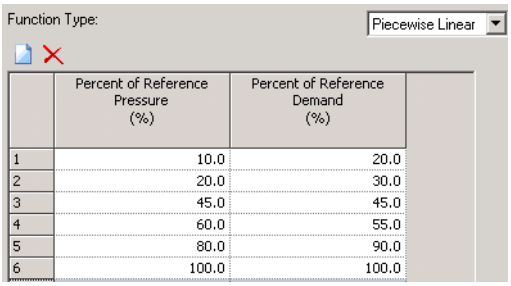
Power Function Exponent - The coefficient that defines the power function relationship between the demand ratio and pressure ratio.

Has Threshold Pressure? - Turn on to specify if a threshold pressure is to be input.

Pressure Threshold is the maximum pressure above which the demand is kept constant.



If the function type chosen is Piecewise Linear then the following opens.

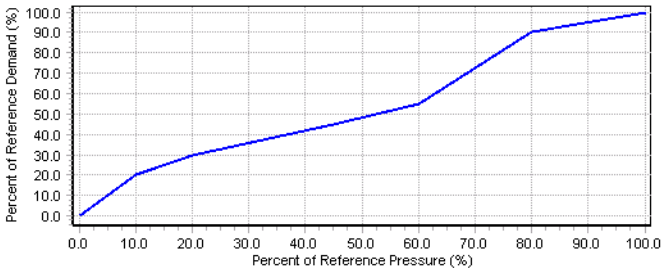


	Percent of Reference Pressure (%)	Percent of Reference Demand (%)
1	10.0	20.0
2	20.0	30.0
3	45.0	45.0
4	60.0	55.0
5	80.0	90.0
6	100.0	100.0

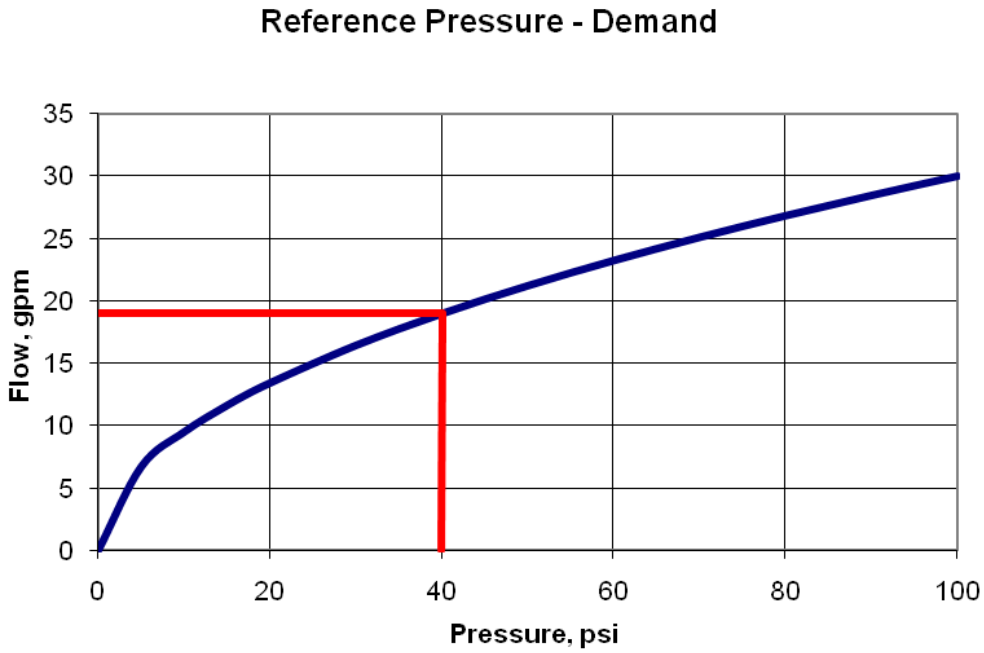
Piecewise Linear is a table of reference pressure percentage vs. reference demand percentage. The last entry value of reference pressure is the greatest that defines the threshold pressure. If the last pressure percentage is less than 100%, the threshold pressure is equal to the reference pressure. If the last pressure percentage is greater than 100%, the threshold pressure is the multiplication of the reference pressure with the greatest pressure percentage.

Percent of Reference Pressure % - defines the percentage of a nodal pressure to reference pressure.

Percent of Reference Demand - defines the percentage of a nodal demand to reference demand.



The **Reference Pressure** is the pressure at which the demands are fully met at a node. In the graph below, the demand assigned to the node is 18 gpm and the reference pressure is 40 psi. As the pressure deviates from 40 psi, the actual demand at the node changes in response to the pressure dependent demand curve (blue line).

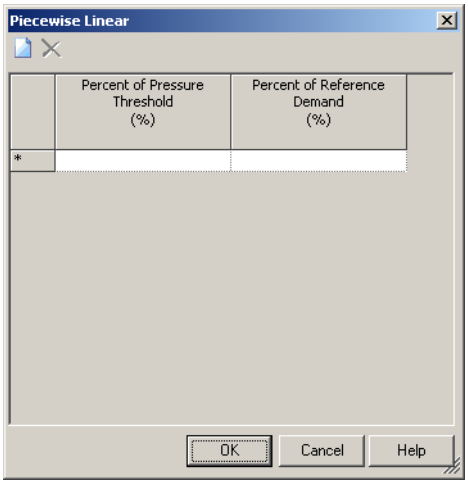


In some cases, there is an upper limit to the amount of water that will be used as pressure increases (users will throttle back their faucets). In this case the pressure at which demand is no longer a function of pressure is called the **Pressure Threshold**. In the graph below the pressure threshold is 50 psi.

The pressure threshold must be equal to or greater than the reference pressure. A reference pressure must be specified to use pressure dependent demand. The threshold pressure is optional. The user can optionally set the reference pressure to the threshold pressure. These values can be set globally or the global value can be overridden on a node by node basis.

Piecewise Linear Dialog Box

This dialog allows you define engineering library entries for Piecewise Linear Curves.



The following buttons are located above the curve points table on the left:

-  **New**—Creates a new row in the curve points table.
-  **Delete**—Deletes the currently highlighted row from the curve points table.

The curve points table contains the following columns:

- **Percent of Pressure Threshold**—defines the percentage of a nodal pressure to reference pressure.
- **Percent of Reference Demand**— defines the percentage of a nodal demand to reference demand.

Piecewise Linear is a table of reference pressure percentage vs. reference demand percentage. The last entry value of reference pressure is the greatest that defines the threshold pressure. If the last pressure percentage is less than 100%, the threshold pressure is equal to the reference pressure. If the last pressure percentage is greater than 100%, the threshold pressure is the multiplication of the reference pressure with the greatest pressure percentage.

8 Reducing Model Complexity with Skelebrator

Skeletonization

Skeletonization Example

Common Automated Skeletonization Techniques

Skeletonization Using Skelebrator

Using the Skelebrator Software

Backing Up Your Model

Skeletonization

Skeletonization is the process of selecting only the parts of the hydraulic network that have a significant impact on the behavior of the system for inclusion in a water distribution model. For example, including each individual service connection, valve, and every one of the numerous other elements that make up the actual network would be a huge undertaking for larger systems. The portions of the network that are not modeled are not ignored; rather, the effects of these elements are accounted for within the parts of the system that are included in the model.

A fully realized water distribution model can be an enormously complex network consisting of thousands of discrete elements, and not all of these elements are necessary for every application of the model. When elements that are extraneous to the desired purpose are present, the efficiency, usability, and focus of the model can be substantially affected, and calculation and display refresh times can be seriously impaired. In addition to the logistics of creating and maintaining a model that employs little or no skeletonization, a high level of detail might be unnecessary when incorporating all of these elements in the model and has no significant effect on the accuracy of the results that are generated.

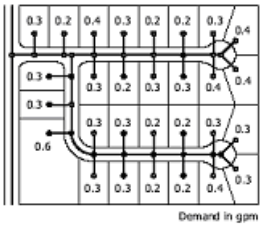
Different levels of skeletonization are appropriate depending on the intended use of the model. For an energy cost analysis, a higher degree of skeletonization is preferable and for fire flow and water quality analysis, minimal skeletonization is necessary. This means that multiple models are required for different applications. Due to this necessity, various automated skeletonization techniques have been developed to assist with the skeletonization process.

Automated Skeletonization includes:

- A generic skeletonization example.
- What automated skeletonizers generally do
- How Skelebrator approaches skeletonization
- Using the Skelebrator software.

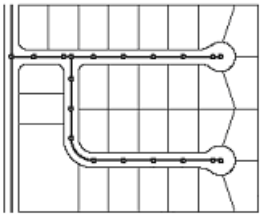
Skeletonization Example

The following series of diagrams illustrate various levels of skeletonization that can be applied. The diagram below shows a network subdivision before any skeletonization has been performed.



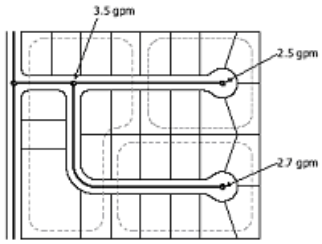
There is a junction at each service tap and a pipe and node at each house for a total of 48 junctions and 47 pipes within this subdivision.

To perform a low level of skeletonization, the nodes at each house could be removed along with the connecting pipes that tie in to the service line. The demands at each house would be moved to the corresponding service tap. The resulting network would now look like this:

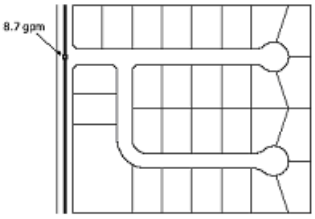


There are now 19 junctions and 18 pipes in the subdivision. The demands that were assigned to the junctions that were removed are moved to the nearest upstream junction. The only information that has been lost is the data at the service connections that were removed.

A further level of skeletonization is possible if you remove the service taps and model only the ends and intersections of the main pipes. In this case, re-allocating the demands is a bit more complex. The most accurate approximation can be obtained by associating the demands with the junction that is closest to the original demand junction (as determined by following the service pipe). In the following diagram, these service areas are marked with a dotted line.



To fully skeletonize this subdivision, the pipes and junctions that serve the subdivision can be removed, and the demands can be assigned to the point where the branch connects to the rest of the network, as shown in the following diagram:



As can be seen by this example, numerous levels of skeletonization can be applied; determining the extent of the skeletonization depends on the purpose of the model. At each progressive level of skeletonization, more elements are removed, thus the amount of available information is decreased. Deciding whether this information is necessary to the intended use of the model dictates the point at which the model is optimally skeletonized.

Common Automated Skeletonization Techniques

The following are descriptions of the skeletonization techniques that have been employed to achieve a level of automation of the skeletonization process. Generally, a combination of these techniques proves to be more effective than any one on its own.

Generic—Data Scrubbing

Data scrubbing is usually the first step of the skeletonization process. Some automated skeletonizers rely entirely on this reduction technique. (Data scrubbing is called Smart Pipe Removal in Skelebrator.) Data scrubbing consists of removing all pipes that meet user-specified criteria, such as diameter, roughness, or other attributes. Criteria combinations can also be applied, for example: “Remove all 2-inch pipes that are less than 200 feet in length.”

This step of skeletonization is especially useful when the model has been created from GIS data, since GIS maps generally contain much more information than is necessary for the hydraulic model. Examples of elements that are commonly included in GIS maps, but not necessarily in the distribution model, are service connections and isolation valves. Removing these elements generally has a negligible impact on the accuracy of the model, depending on the application for which the model is being used.

The primary drawback of this type of skeletonization is that there is generally no network awareness involved. No consideration of the hydraulic effects of a pipe’s removal is taken into account, so there is a large potential for errors to be made by inadvertent pipe removal or by causing network disconnections. (Bentley Systems Skelebrator does account for hydraulic effect.)

Generic—Branch Trimming

Branch trimming, also referred to as Branch Collapsing, is the process of removing short dead-end links and their corresponding junctions. Since pipes and junctions are removed by this process, you specify the criteria for both types of element. An important element of this skeletonization type is the reallocation of demands that are associated with junctions that are removed. The demand associated with a dead-end junction is assigned to the junction at the beginning of the branch.

Branch trimming is a recursive process; as dead-end pipes and junctions are removed, other junctions and pipes can become the new dead-ends—if they meet the trimming criteria, these elements may also be removed. You specify whether this process continues until all applicable branches have been trimmed or if the process should stop after a specified number of trimming levels.

Branch trimming is an effective skeletonization technique; dead-end junctions with no loading have no effect on the model, and dead end junctions that do have demands are accounted for at the point through which this flow would pass anyway (without skeletonization), so the hydraulic behavior of the network as a whole is unaffected.

A drawback to this type of skeletonization is that information and results cannot be obtained from non-existent elements. During water quality or fire flow analysis, information on these trimmed elements may be desired but unavailable. Having multiple models utilizing various levels of skeletonization is the solution to this potential issue.

Generic—Series Pipe Removal

Series pipe removal, also known as intermediate node removal or pipe merging, is the next skeletonization technique. It works by removing nodes that have only two adjacent pipes and merging these pipes into a single one. As with Branch trimming, any demands associated with the junctions being removed must be reallocated to nearby nodes, and generally a number of strategies for this allocation can be specified.

An evenly-distributed strategy divides the demand equally between the two end nodes of the newly merged pipe. A distance-weighted technique divides the demands between the two end nodes based on their proximity to the node being removed. These strategies can be somewhat limiting, and maintaining an acceptable level of network hydraulic precision while removing nodes and merging pipes is made more difficult with this restrictive range of choices.

Other criteria are also used to set the allowable tolerances for relative differences in the attributes of adjacent pipes and nodes. For example, an important consideration is the elevation difference between nodes along a pipe-merge candidate. If the junctions mark critical elevation information, this elevation (and by extension, pressure) data would be lost if this node attribute is not accounted for when the pipes are merged.

Another set of criteria would include pipe attributes. This information is needed to prevent pipes that are too different (as defined by the tolerance settings) hydraulically from being merged. It is important to compare certain pipe attributes before merging them to ensure that the hydraulic behavior will approximate the conditions before the merge. However, requiring that pipes have exactly matching criteria limits the number of elements that could potentially be removed, thus reducing the level of skeletonization that is possible.

In other words, although it is desirable for potential pipe merge candidates to have similar hydraulic attributes, substantial skeletonization is difficult to achieve if there are even very slight variances between the hydraulic attributes of the pipes, since an exact match is required. This process is, however, very good at merging pipes whose adjacent nodes have no demand and that have exactly the same attributes. Removing these zero-demand junctions and merging the corresponding pipes has no effect on the model's hydraulics, except for loss of pressure information at the removed junctions.

Series pipe removal is called Series Pipe Merging in Skelebrator.

Skeletonization Using Skelebrator

This section discusses the advantages and approach to performing skeletonization using Skelebrator.

Skelebrator—Smart Pipe Removal

The first step that Skelebrator performs is Smart Pipe Removal, which is an improved version of the data scrubbing technique. The main drawback of standard data scrubbing procedures is that they have no awareness of the effects that removing elements from the model will have on the calculated hydraulics. This can easily cause network disconnections and lead to a decrease in the accuracy of the simulated network behavior.

Skelebrator eliminates the possibility of inadvertent network disconnections caused by the data scrubbing technique. This is accomplished by utilizing a sophisticated network-walking algorithm. This algorithm marks pipes as safe to be removed if the removal of the pipe so marked would not invalidate, or disconnect, the network. For a pipe to be removed, it must:

- Meet the user-specified removal criteria
- Be marked safe for removal
- Not be marked as non-removable
- Not be connected to a non-removable junction (to prevent orphaning).

This added intelligence protects the model's integrity by eliminating the possibility of inadvertently introducing catastrophic errors during the model reduction process.

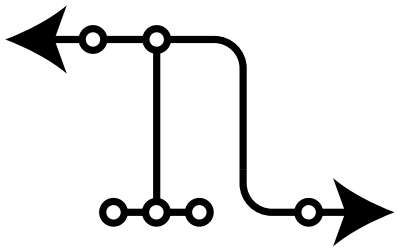
This innovation is not available in other automated skeletonization applications; a likely result of performing skeletonization without this intelligent safety net is the invalidation of the network caused by the removal of elements that are critical to the performance and accuracy of the model. At the very least, verifying that no important elements have been removed during this skeletonization step and re-creating any elements that have been erroneously removed can be a lengthy and error-prone process. These considerations are addressed automatically and transparently by the Skelebrator's advanced network traversal algorithm.

Skelebrator—Branch Collapsing

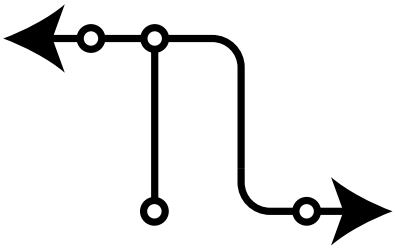
Branch Collapsing is a fundamental skeletonization technique; the improvements over the branch trimming that Skelebrator brings to the table are primarily a matter of flexibility, efficiency, and usability. The branch trimming method utilized by other automated skeletonization applications allows a limited range of removal criteria; in some cases, just elevation and length. Workarounds are required if another removal criteria is desired, resulting in more steps to obtain the desired results.

Conversely, Skelebrator innately provides a wide range of removal criteria, increasing the scope of this skeletonization step and eliminating the need for inefficient manual workarounds.

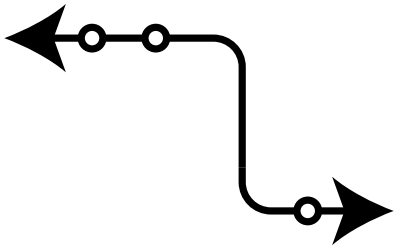
The following diagrams illustrate the results of Branch Collapsing.



Before Branch Collapsing



After One Branch Collapsing Iteration



After Two Branch Collapsing Iterations (Branch is Completely Removed)

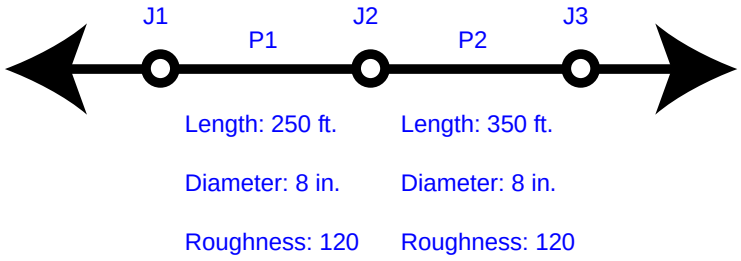
Skelebrator—Series Pipe Merging

The Skelebrator Series Pipe Merging technique overcomes the basic drawbacks to series pipe removal that were mentioned previously in two ways:

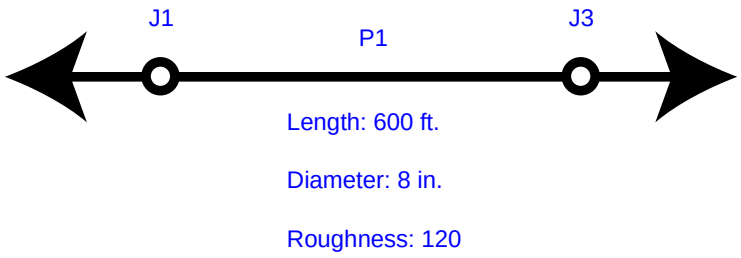
First, the demand reallocation strategies normally available for this step are not comprehensive enough, limiting you to choosing from an even demand distribution or a distance-weighted one. This limitation can hinder your ability to maintain an acceptable level of hydraulic parity.

To overcome this limitation, Skelebrator provides a greater range of demand reallocation strategies, including: Equally Distributed, Proportional to Existing Load (at the ends of the new pipe), Proportional to Dominant Criteria, and User Defined Ratio. Evenly Distributed divides the demand equally between the two end nodes of the newly merged pipe. The Proportional to Existing Load divides demand based on the amount of demand already associated with the end nodes. The Proportional to Dominant Criteria strategy can supply the distance-weighted option and allows other pipe attributes to be weighting factors as well (for example, roughness or diameter). The User-Defined Ratio option assigns the specified proportion of demand to the upstream junction and the remainder of the demand to the downstream one. These additional choices allow the proper simulation of a wider range of hydraulic behaviors.

Second, and more importantly, this technique is effective because it allows you to specify tolerances that determine if the pipes to be merged are similar enough that combining them into a single pipe will not significantly impact the hydraulic behavior of the network. This increases the number of potential merge candidates over requiring exact matches, thereby increasing the scope of skeletonization but affecting hydraulics, since differences in hydraulic properties are ignored.



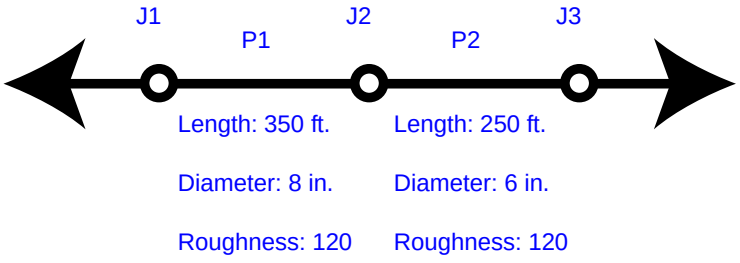
Before Series Pipe Merging (Exact Match Pipes)



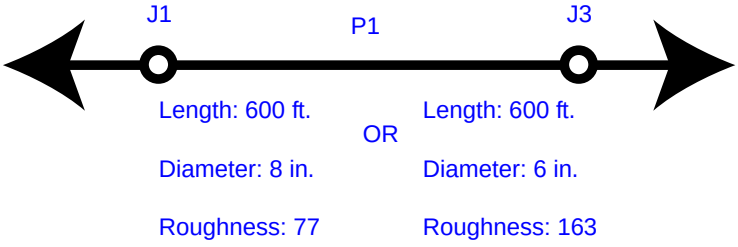
After Series Pipe Merging (Exact Match Pipes)

To counter the hydraulic effects of merging pipes with different hydraulic attributes, a unique hydraulic equivalency feature has been developed. This feature works by determining the combination of pipe attributes that will most closely mimic the hydraulic behavior of the pipes to be merged and applying these attributes to the newly merged pipe. By generating an equivalent pipe from two non-identical pipes, the number of possible removal candidates (and thus, the potential level of skeletonization) is greatly increased.

This hydraulic equivalency feature is integral to the application of a high degree of effective skeletonization, the goal of which is the removal of as many elements as possible without significantly impacting the accuracy of the model. Only Skelebrator implements this concept of hydraulic equivalency, breaking the barrier that is raised by other skeletonizers that only allow exactly matched pipes to be merged by this process.



Before Series Pipe Merging (Different Diameters)



After Series Pipe Merging (Using Skelebrator's Hydraulic Equivalency feature)

Tip: If you want to combine only pipes with the same hydraulic characteristics (i.e., diameter and roughness) then to a series pipe removal operation, add a pipe tolerance of 0.0 and a roughness tolerance of 0.0. Also make sure to deselect the Use Equivalent Pipes option.

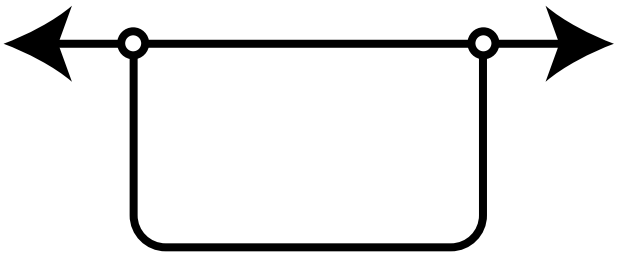
Skelebrator—Parallel Pipe Merging

Parallel Pipe Merging is the process of combining pipes that share the same two end nodes into a single hydraulically equivalent pipe. This skeletonization strategy relies on the hydraulic equivalency feature.

To merge parallel pipes, you specify which of the two pipes is the "dominant" one. The length of the dominant pipe becomes the length of the merged pipe, as does either the diameter or the roughness value of the dominant pipe. You specify which of the two attributes to retain (diameter or roughness) and the program determines what the value of the other attribute should be in order to maintain hydraulic equivalence.

For example, the dominant pipe has a diameter of 10 inches and a C factor of 120; one of these values is retained. The pipe that will be removed has a diameter of 6 inches and a C factor of 120. If the 10-inch diameter value is retained, the program performs hydraulic equivalence calculations to determine what the roughness of the new pipe should be in order to account for the additional carrying capacity of the parallel pipe that is being removed.

Because this skeletonization method removes only pipes and accounts for the effect of the pipes that are removed, the network hydraulics remain intact while increasing the overall potential for a higher level of skeletonization.



Before Parallel Pipe Merging



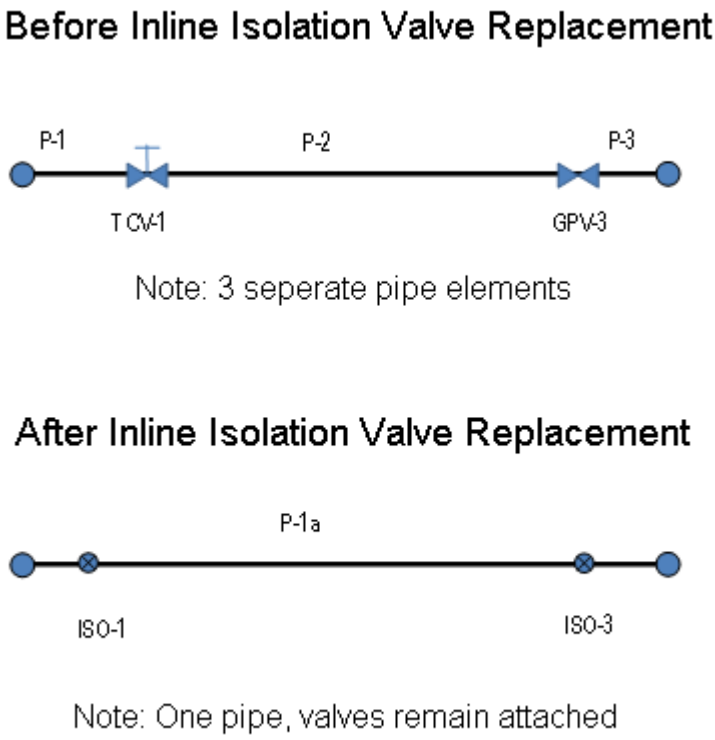
After Parallel Pipe Merging

Skelebrator—Inline Isolation Valve Replacement

In building a model from an external source such as a GIS, the GIS may be set up such that isolation valves split a pipe into two separate pipes. These isolation valves are usually imported into WaterGEMS as throttling control valves (TCV) or general purpose valves (GPV) with ModelBuilder. This is due to the fact that WaterGEMS isolation valves are attached to pipes and do not split them.

While models that split pipes with a TCV or GPV will run, they are usually about twice as large as one that models isolation valves as attached to a single pipe and not splitting pipes. In Skelebrator, it is possible to automatically convert all or a selection of valves into WaterGEMS isolation valves, and merge the pipes on either side of the

valve into a single pipe element. This process is shown graphically below. The pipes that are merged are treated the same as they are under the series pipe merging option except that the isolation valve element is maintained at its original location and can be used for segmentation.



See Inline Isolating Valve Replacement for details on using this option.

Skelebrator—Other Skelebrator Features

Skelebrator offers numerous other features that improve the flexibility and ease-of-use of the skeletonization process.

The Skeletonization Preview option allows you to preview the effects that a given skeletonization step, or method, will have on the model. This important tool can assist the modeler in finding potential problems with the reduced model before a single element is removed from it.

Before skeletonization is begun or between steps, you can use Skelebrator's protected element feature to manually mark any junctions or pipes as non-removable. Any pipes marked in this way will always be preserved by the Skelebrator, even if the elements meet the removal criteria of the skeletonization process in question. This option provides the modeler with an additional level of control as well as improving the flexibility of the process.

The ability of the Skelebrator to preserve network integrity by not removing elements that would cause the network to be invalidated is an important timesaving feature that can prevent this common error from happening. There may be circumstances, however, when you do not want or need this additional check, so this option can be switched off.

For the utmost control over the skeletonization process, you can perform a manual skeletonization. This feature allows you to step through each individual removal candidate. The element can then be removed or marked to be excluded from the skeletonization. You can save this process and choices you made and reuse them in an automatic skeletonization of the same model.

Skelebrator—Conclusion

With the overwhelming amount of data now available to the water distribution modeler, some degree of skeletonization is appropriate for practically every model, although the extent of the skeletonization varies widely depending on the intended purpose of the model. In light of this, it has become desirable to maintain multiple models of the same system, each for use in different types of analysis and design.

A model that has been minimally skeletonized serves as a water quality and fire flow analysis model, while energy cost estimating is performed using a model with a higher degree of skeletonization.

Creating a number of reduced models with varying levels of skeletonization can be a lengthy and tedious process, which is where the automated techniques described above demonstrate their value. To ensure that the skeletonization process produces a reduced model with the minimum number of elements necessary for the intended application while simultaneously maintaining an accurate simulation of network behavior, the automated skeletonization routine must be flexible enough to accommodate a wide variety of conditions.

Skelebrator provides an unmatched level of flexibility, providing numerous demand reallocation and element removal strategies. It alone, amongst automated skeletonizers, maximizes the potential level of skeletonization by introducing the concept of Hydraulic Equivalence, eliminating the limitation posed by exact attribute matching requirements. Another distinction is the advanced network walking algorithm employed by Skelebrator, which ensures that your model remains connected and valid, thereby greatly reducing the possibility for inadvertent element removal errors.

These features, and others such as the Skeletonization Preview and Manual Skeletonization, greatly expedite and simplify the process of generating multiple, special-purpose water distribution models, each skeletonized to the optimal level for their intended purpose.

Using the Skelebrator Software

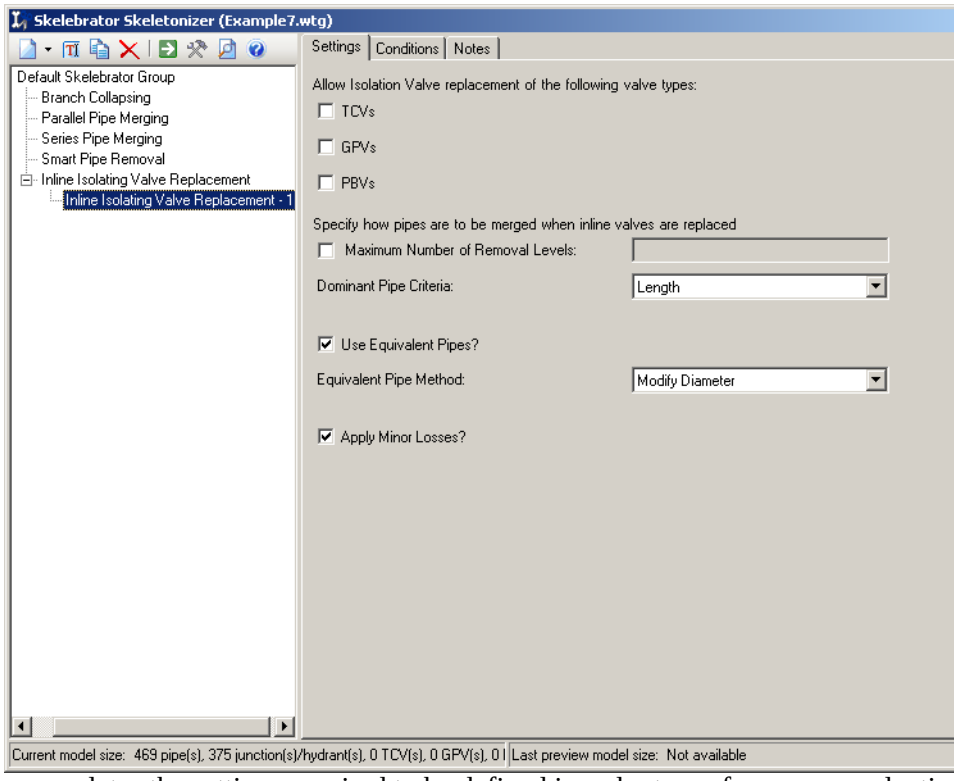
Skelebrator is available for use in Stand-Alone, MicroStation, and AutoCAD modes. Skelebrator has slightly different behavior and features in some environments. This section describes using the Skelebrator software.

When using Skelebrator, please note:

- We strongly recommended that you eliminate all scenarios other than the one to be skeletonized from a model prior to skeletonization.
- Skelebrator reduces a WaterCAD V8*i* model and applies its changes to the model's WaterCAD V8*i* datastore, which is contained within an .sqlite file. Skelebrator cannot view or make changes to a standard GIS geodatabase.
- To use Skelebrator with a GIS geodatabase, you must first use ModelBuilder to create a WaterCAD V8*i* datastore from the GIS data.
- To use Skelebrator with a CAD drawing, you must firstuse ModelBuilder to create a WaterCAD V8*i* datastore from the CAD file.

Skeletonizer Manager

Use Skelebrator's skeletonization manager to define how you are going to skeletonize your network. The basic unit in Skelebrator is an operation. An operation defines and



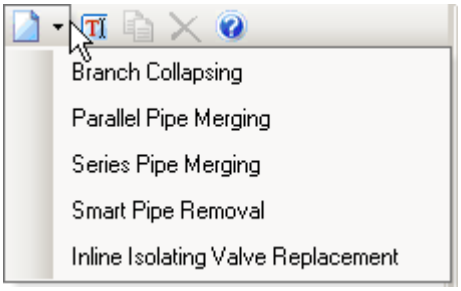
encapsulates the settings required to be defined in order to perform some reduction process on your hydraulic network. Skelebrator provides these types of operations that may be used to reduce the size of your model:

- **Branch Collapsing**
- **Parallel Pipe Merging**
- **Series Pipe Merging**
- **Smart Pipe Removal**
- **Inline Isolating Valve Replacement**

	
New 	Click **New** to add a skeletonization operation. This adds an operation for the option that is currently selected: Smart Pipe Removal, Branch Collapsing, Series Pipe Merging, or Parallel Pipe Merging. Skelebrator performs a single operation at a time. An operation consists of the strategy to use (Smart Pipe Removal, Branch Collapsing, etc.) and the settings and conditions specific to that operation.
Rename 	Click **Rename** to rename the currently selected operation.
Duplicate 	Click **Duplicate** to create a copy of the currently selected operation. You can rename and edit the copy as needed.
Delete 	Click **Delete** to remove the currently selected operations from the list.
Automatic 	To run automatic skeletonization and apply your skeletonization operations to your model. The run is executed using the selected operations. More than one operation can be selected.
Manual 	Click to manually run the skeletonization operation. Manual skeletonization allows you to conduct skeletonizations in a concise and controlled manner while viewing the pipes that will be removed and gives you the opportunity to protect some of those pipes on a real-time basis.
Print Preview 	Preview the results of your skeletonization.

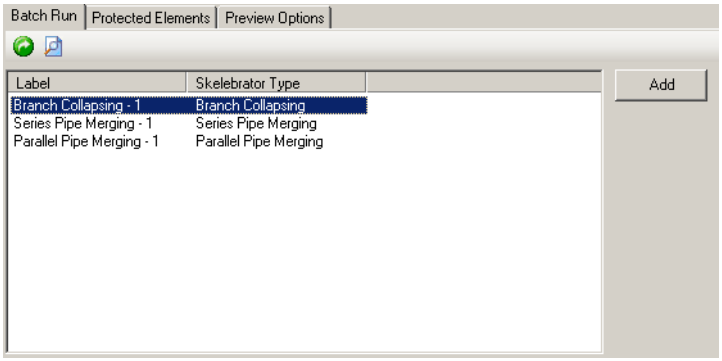
To use Skeletonizer Manager

1. Click the skeletonization technique you want to use: Branch Collapsing, Parallel Pipe Merging, Series Pipe Merging, Smart Pipe Removal.
2. Click **New** and select from the menu.

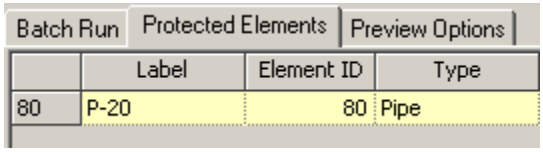


3. Type a new name or keep the default name.
4. Choose your Settings, Conditions, and add Notes.
5. Click on Default Skelebrator Group (the first in the list and it can be renamed).
6. Tabs for **Batch Run, Protected Elements, Preview Options** open:

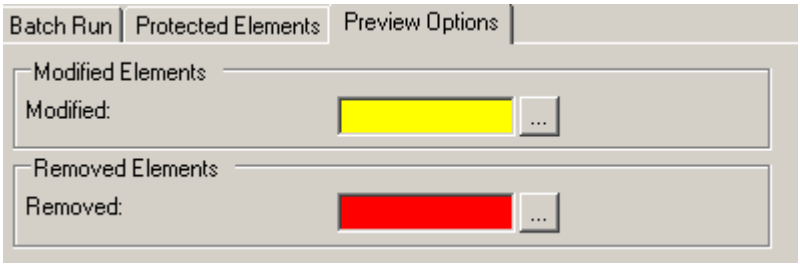
   Batch Run - Choose which of your defined skeletonization operations to run and in what order to run them. Use Batch Run if you want to run skeletonization operations for more than one option, for example, a combination of Smart Pipe Removal, Branch Collapsing, Series Pipe Merging, or Parallel Pipe Merging operations and where the order of applied operations is important.



   Protected Elements - Saved as references to the originally skeletonized model. Using the Skelebrator protected element settings with a different model is likely to result in different (and unintended) elements being protected from skeletonization. If you wish to re-run previously saved skeletonizations on the original model, save your Skelebrator setup with the original model or in a place with a name that shows that the export file belongs to that particular model.



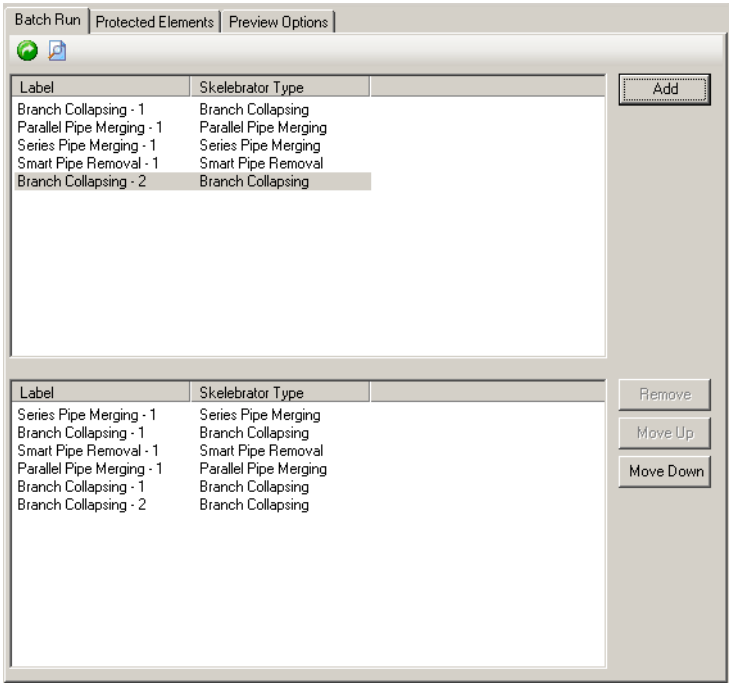
Preview Options - Review the effects of a skeletonization on your model without making any changes to or deletions from your model. Click the Ellipsis button to select a color from the color palette.



7. Click Close to exit the window.

Batch Run

When Default Skelebrator Group is highlighted, the Batch Run tab is opened with the Batch Run Manager in view. Use the Batch Run Manager to select the skeletonization strategies you want to use and the order to run them.

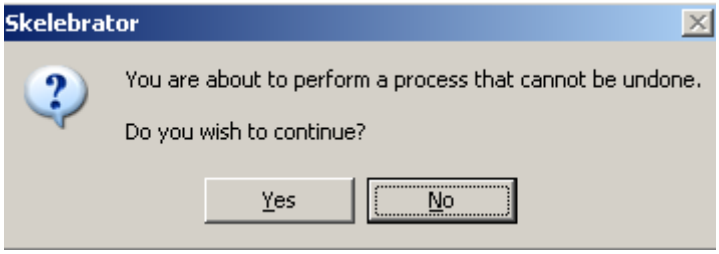


Operations appearing in the top window are the operations you have defined and which are available for use in a batch run. Any operations in this window may be selected for a batch run. The same operation can be selected multiple times.

To Use Batch Run

1. Select Default Skelebrator Group.
2. Select the Skeletonization strategies.
3. Click **Add** to add selected operations to the lower window. Any operations in the lower window are selected as part of the batch run. Use Remove, Move Up, and Move Down to manage the makeup and order of the operations in the batch run list.
4. Click **Batch Run**  to start an automatic skeletonization using the operations you have defined in your batch run or click Preview  to preview the results of the operations you have defined in your batch run prior to running it.

5. The following message opens:



   Click Yes to continue.

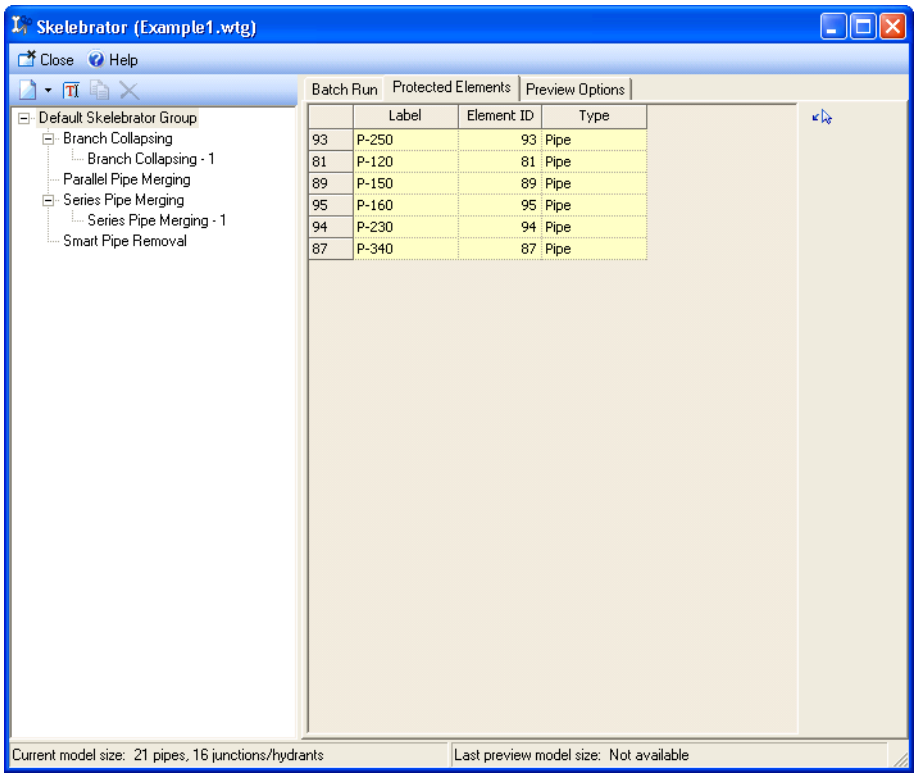
6. Results of the batch run show in the drawing pane.

Note: **The batch run manager does not become available until at least one Skelebrator operation is added.**

All operations selected into the lower window of the batch run manager dialog box will be executed during a batch run. There is no need to select (highlight) the operations before running them. Conversely, selecting only some operations in this window does not mean only those operations will be run.

Protected Elements Manager

The Protected Elements Manager provides a way of making certain elements in your model immune to skeletonization. Use this feature to mark important elements in your model as not skeletonizable. Note that only pipes and junctions may be protected from skeletonization since all other node elements (valves, pumps, tanks, reservoirs, and all WaterCAD V8*i* elements) are already immune to skeletonization. (TCVs are the noted exception to this rule and may be treated as junctions, if selected, during Series Pipe Merging.)



Selecting Elements from Skelebrator

This section describes how to use the selection tools to create Skelebrator-specific selection sets.

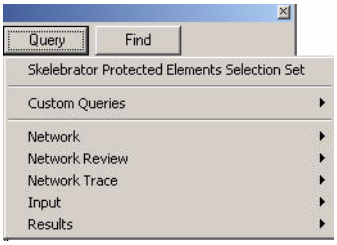
In order to select elements from the Skelebrator user interface

1. Open the **Example1** model which is included with WaterCAD V8*i*.
2. Go to Tools > Skelebrator Skeletonizer.
3. Click on the Protected Elements tab and click Select. The Skelebrator window closes and a Select toolbar opens:

	
Done 	Used when you are finished with the element selection process.
Add 	Used to process elements that are being added. As the elements are selected they change to the default color.
Remove 	Used to remove elements, not to delete them. When the remove button is selected, anytime you select a selection set menu item (see below) or execute a query (see below), the results will be removed from the selection. For example, if you were to have the remove button selected and created a custom query for pipes (see below for details) and had no definition (clicking OK in the Query Builder without any SQL statement defined), it would remove all pipes from the selection.

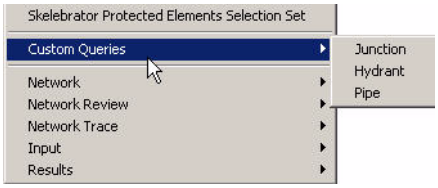
	
Select By Polygon 	Allows you to draw a polygon. All elements within the polygon will be selected.
Query 	Opens a submenu containing various query options.
Find 	Used for a Domain Element Search to run the query.
Clear 	Used to clear the entire selection. You will be prompted to verify if you want to clear the entire selection.

4. Click Query and the following menu opens:



The first item listed is a selection set which is automatically created by Skelebrator. When you select a selection set menu item, the IDs are retrieved and applied to the selection. Only valid elements are selected.

The Custom Queries menu will contain menu items that allow you to create custom, non-persisting queries for the valid elements.



Since this menu only contains custom queries for valid elements, any results passed back from the query execution will be applied to the selection. In this example only junctions and pipes can be selected so you can only create custom queries for junctions and pipes.

The next set of menus are for the available queries. The queries are processed in the following order: Project, Shared, and Predefined. Each menu item for the queries represents the equivalent folder in the query manager View > Queries.

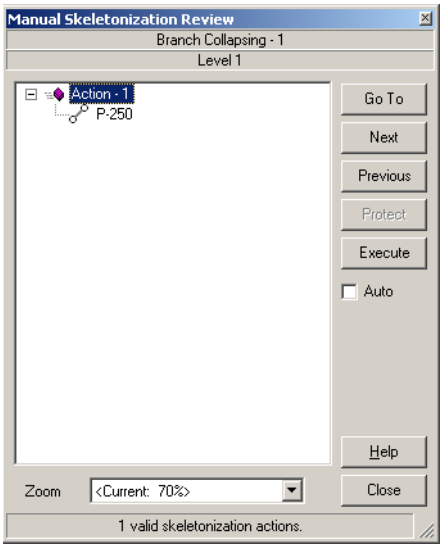
5. Click **FIND** to open the Domain Element Search window. Click  to get results for pipes and junctions. You can only select one row at a time. In order to make your selection, select the row and click OK. If the element is not already selected, it will be selected.

   Note: In order to cancel the selection, click on the x.

Manual Skeletonization

If you click the Manual Skeletonization button, the Manual Skeletonization Review dialog box opens. The manual skeletonization review dialog box lists the proposed skeletonization actions for the particular skeletonization process selected. The contents of the action list window (to the left of the buttons) will vary depending on the type of operation being run. For Smart Pipe Removal and Branch Collapsing, each Skelebrator action will have one pipe associated with it, whereas Series and Parallel

Pipe Merging will have two pipes associated with each action. For Smart Pipe Removal, when network integrity is enforced, the contents of the action list are updated, after every executed action, to reflect only valid actions, after each action is performed.



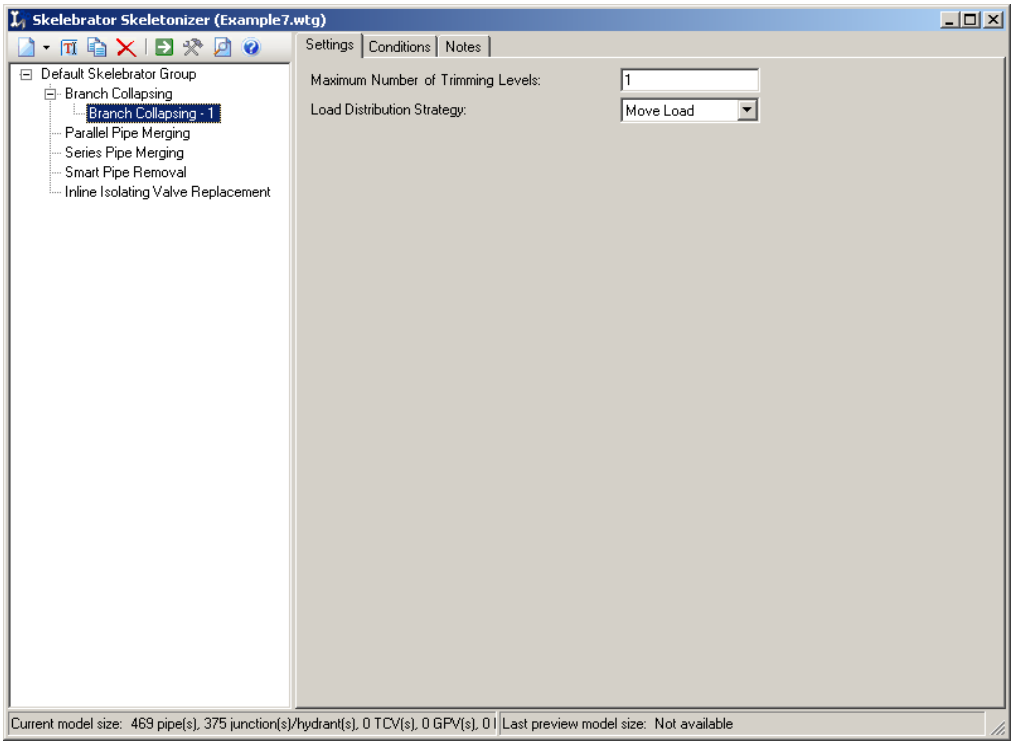
- **Go To**—Select an element in the element window and click Go To to jump to the element in WaterCAD V8*i*. WaterCAD V8*i* displays the element at the level of zoom you selected in the Zoom drop-down list.
- **Next**—Click Next to preview the next element in the Manual Skeletonization Review dialog box.
- **Previous**—Click Previous to preview the previous element to the one you have selected in the Manual Skeletonization Review dialog box.
- **Protect**—Click Protect to protect the selected element. Protected elements cannot be deleted from the network by skeletonization. In a Series or Parallel Pipe Merging operation, protecting one pipe in an action will mean that the action will not be able to be executed. The remaining un-protected pipe will not be skeletonized during this skeletonization level; however, it is not precluded from subsequent skeletonization levels unless it also is protected.
- **Execute**—Click Execute to run Skelebrator only for the selected Skelebrator action. In the case of Smart Pipe Removal and Branch Collapsing, the associated pipe will be removed from the model and associated loads redistributed as specified. Additionally, for branch collapsing, one junction will be removed. For Series Pipe Merging, two pipes and one junction will be removed, associated loads redistributed as specified and an equivalent pipe added as a replacement, if the option is selected. Otherwise, the properties of the dominant pipe will be used to create a new pipe. For Parallel Pipe Merging, one pipe will be removed and the remaining pipe will be updated to the hydraulic equivalent, if you selected hydraulic equivalency.

- **Auto Next?**—Select this check box if you wish for Skelebrator to immediately advance to the next pipe element in the action list. This is the equivalent of clicking Execute then clicking Next immediately afterwards.
- **Close**—Click Close to exit the Manual Skeletonization Review dialog box. Any remaining actions listed will not be executed.
- **Zoom**—Select a Zoom at which you want to display elements you preview using Go To, Previous, and Next.

Branch Collapsing Operations

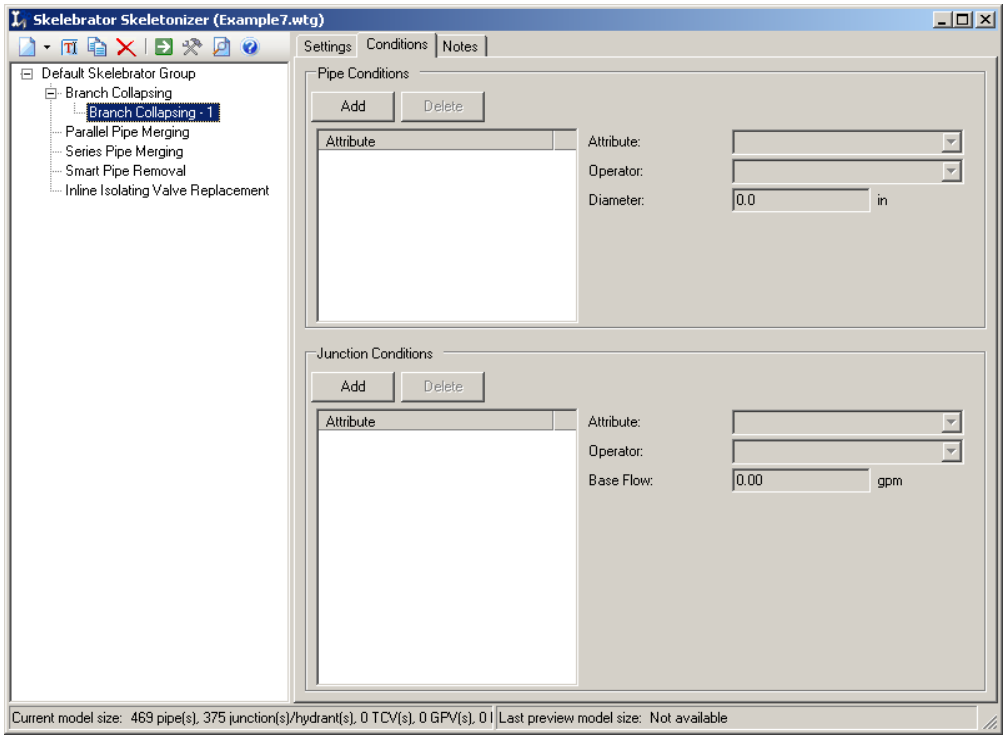
When you add or edit a Branch Collapsing operation, the Branch Collapsing Operation Editor dialog box opens. Branch Collapsing operations have two sets of parameters, Settings and Conditions.

1. Click the **Settings** tab to edit settings.



- **Maximum Number of Trimming Levels**—Set the maximum number of trimming levels you want to allow. In Branch Collapsing, a single trimming level run to completion would trim every valid branch in the model back by one pipe link. Two trimming levels would trim every valid branch back two pipe links and so on.
- **Load Distribution Strategy**—Select what you want to do with the hydraulic load on the sections you trim. The choices are **Don't Move Load**, which means that the demands are no longer included in the model, or **Move Load**, which means transfer the demands to the upstream node.

2. Click **Conditions** to edit or create conditions.



3. Click **Add** to add conditions. You can add pipe and/or junction conditions. You can add more than one condition.

4. Or, select an existing condition and click **Edit** to modify a selected condition. You can add and edit Junction and Pipe Conditions.

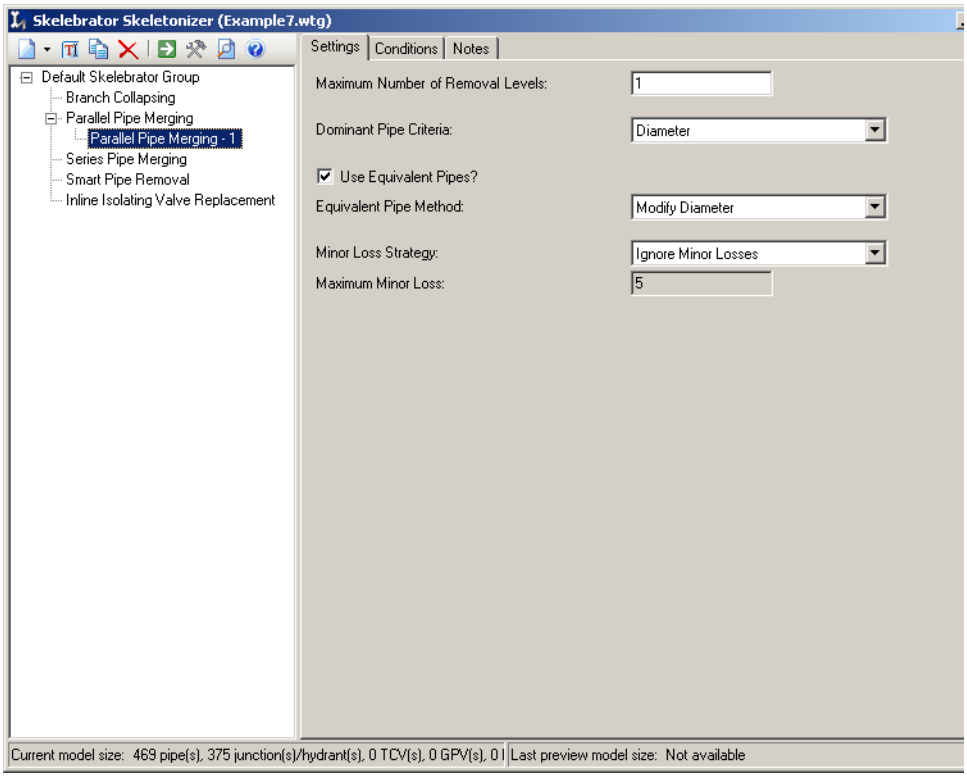
   You can set select parameters that determine which pipes are included in the skeletonizing process in the Conditions tab. In Branch Collapsing, the junctions referred to (in junction conditions) are the two end junctions of the pipe being trimmed. Tolerances can also be defined for junctions. Tolerances work by limiting the pipes skeletonized only to the ones that have the specified attribute within the specified tolerance. For example, in Branch Collapsing a tolerance on junction elevation of 3 feet would limit skeletonization to pipes that had both end junctions with an elevation within three feet of each other.

Parallel Pipe Merging Operations

Note: In Stand-Alone mode, you can assign prefixes and/or suffixes to pipes and junctions created during Parallel Pipe Merging operations by using the Element Labeling feature.

For instance, to assign a prefix of "sk" to all pipes that are merged using the Parallel Pipe Merging operation, open the Element Labeling dialog box and enter "sk" before the "P-" in the Prefix field of the Pressure Pipe row. Any pipes merged during the Parallel Pipe Merging will now be labeled "skP-1"," skP-2", etc.

When you add or edit a Parallel Pipe Merging operation, the Parallel Pipe Merging Operation Editor controls become active in the control pane on the right.



Operations have two sets of parameters, Settings and Conditions.

1. Click **Settings** to edit or create settings.
2. Click **Add** to add a new pipe condition.
3. Or, select a condition and click **Edit** to change its parameters.

The condition editor allows you to set select parameters that determine which pipes are included in the skeletonization process.

Maximum Number of Removal Levels—Set the maximum number of removal levels you want to allow. In the context of Parallel Pipe Merging a single removal level will merge two parallel pipes. Consider a case where there exists 4 pipes in parallel. It would take 3 removal levels to merge all 4 pipes into a single pipe. In the first removal level, two pipes are merged leaving three pipes. In the second level another two pipes are merged leaving only two pipes. The last two pipes are merged into a single pipe in the third removal level. Unless you have a large degree of parallel pipes in your model, one or two levels of Parallel Pipe Merging will generally be all that is necessary to merge the majority of parallel pipes in your system.

Dominant Pipe Criteria—Select the criteria by which Skelebrator determines the dominant pipe. The dominant pipe is the pipe whose properties are retained as appropriate. For example, when merging a 6-in. pipe and an 8-in. pipe, if diameter is selected as the dominant pipe criteria then the larger diameter pipe (e.g., 8-in.) will provide the properties for the new pipe. That is, the 8-in. pipe's diameter, roughness, bulk reaction rate, etc., will be used for the new pipe.

Use Equivalent Pipes—Select **Use Equivalent Pipe** if you want Skelebrator to adjust remaining pipes to accommodate the removal of other pipes in series.

Equivalent Pipe Method—Select whether you wish to modify the dominant pipe roughness or the dominant pipe diameter for the equivalent pipe calculations.

- Modify Diameter
- Modify Roughness.

If modify diameter is selected, the new pipe's roughness is kept constant and the diameter adjusted such that the head loss through the pipe remains constant. Conversely, if modify roughness is selected, the new pipe's diameter is kept constant and the roughness adjusted such that the head loss through the pipe remains constant.

> **Note: When using Darcy-Weisbach for the friction method, Modify Diameter is the only available selection since calculated equivalent roughness can be invalid (negative) in some circumstances.**

Minor Loss Strategy—If your network models minor losses, select what you want Skelebrator to do with them.

- **Use Ignore Minor Losses** if you want to ignore any minor losses in parallel pipes. Resulting merged pipes will have a minor loss of 0.
- **Use Skip Pipe if Minor Loss > Max** to protect from skeletonization any pipes that have a higher minor loss than a value you set for the Maximum Minor Loss.
- **Use 50/50 Split** to apply 50% of the sum of the minor losses from the parallel pipes to the replacement pipe that Skeletonizer uses.

Maximum Minor Loss—If you select Skip Pipe if Minor Loss > Max from the Minor Loss Strategy drop-down list, any pipes with a minor loss value greater than the value you set will not be removed by Skelebrator.

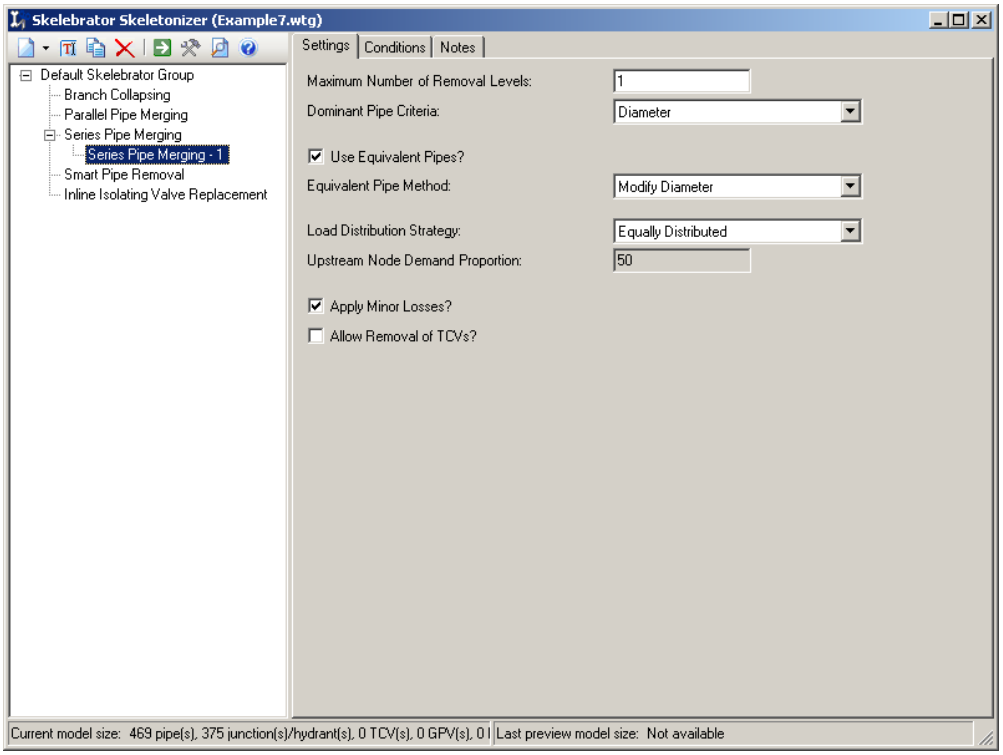
Series Pipe Merging Operations

Note: In Stand-Alone mode, you can assign prefixes and/or suffixes to pipes and junctions created during Series Pipe Merging operations by using the Element Labeling feature.

For instance, to assign a prefix of "sk" to all pipes that are merged using the Series Pipe Merging operation, open the Element Labeling dialog box and enter "sk" before the "P-" in the Prefix field of the Pressure Pipe row. Any pipes merged during the Series Pipe Merging will now be labeled "skP-1"," skP-2", etc. Remember to reinstate the original prefixes/suffixes after skeletonization has been performed.

When you add or edit a Series Pipe Merging operation, the Series Pipe Merging Operation Editor dialog box opens. Operations have two sets of parameters, Settings and Conditions.

1. Click the **Settings** tab to edit settings.



– **Maximum Number of Removal Levels**—Select the number of levels of pipes that get removed per iteration of the Series Pipe Merging operation. The maximum number of removal levels is 50. This is because in the absence of any other limiting factors (conditions, protected elements, non-removable nodes, etc.) one series pipe removal iteration will effectively halve the number of pipes. A second iteration will again halve the number of pipes, and so on. Therefore, 50 is the practical limit for removal levels.

– **Dominant Pipe Criteria**—Select the criteria by which Skelebrator determines the dominant pipe. The dominant pipe is the pipe whose properties are retained as appropriate. For example, when merging a 6-in. pipe and an 8-in. pipe, if diameter is selected as the dominant pipe criteria then the larger diameter pipe (e.g., 8-in.) will provide the properties for the new pipe. That is, the 8-in. pipe's diameter, roughness, bulk reaction rate, etc. will be used for the new pipe.

– **Use Equivalent Pipes**—Select **Use Equivalent Pipe** if you want Skelebrator to adjust the merged pipe properties as such to attain equivalent hydraulics as the two merged pipes.

– **Equivalent Pipe Method**—Select whether you wish to modify the dominant pipe roughness or the dominant pipe diameter for the equivalent pipe calculations.

  - Modify Diameter
  - Modify Roughness.

  If modify diameter is selected, the new pipe's roughness is kept constant and the diameter adjusted such that the head loss through the pipe remains constant. Conversely, if modify roughness is selected the new pipe's diameter is kept constant and the roughness adjusted such that the head loss through the pipe remains constant.

Note: When using Darcy-Weisbach for the friction method, Modify Diameter is the only available selection since calculated equivalent roughness can be invalid (negative) in some circumstances.

– **Load Distribution Strategy**—Select how you want the load distributed from junctions that are removed.

  - **Equally Distributed** puts 50% of the load on the starting and ending junctions of the post-skeletonized pipe.
  - **Proportional to Dominant Criteria** assigns loads proportional to the attribute used to select the dominant pipe. For example, if diameter is the dominant attribute and one pipe is 6-in., while the other is 8-in. (14-in. total length), 8/14 of the load will go to the upstream node, while 6/14 will go to the downstream node.

Note: For the length attribute, load assignment is inversely proportional, such that the closest junction gets the majority of the demand.

- **Proportional to Existing Load** maintains the pre-skeletonization load proportions.
- **User-Defined Ratio** allows you to specify the percentage of the load applied to the upstream node in the post-skeletonized pipe.

Note: If either of the uncommon nodes of the two pipes being merged are not junction nodes, then the selected load distribution strategy is ignored and all load is moved to the junction node. If both uncommon nodes are not junctions, then skeletonization is only carried out if the common junction node has zero demand.

– **Upstream Node Demand Proportion**—Set a user-defined load distribution percentage. Set the percentage of the node demand that you want applied to the upstream node adjacent to the removed sections. This parameter is only available if you select User Defined in the Load Distribution Strategy drop-down list. Upstream in this context relates to the physical topology of the pipe and its nodes and may not correspond to the direction of flow in either the pre-skeletonized or post-skeletonized pipe.

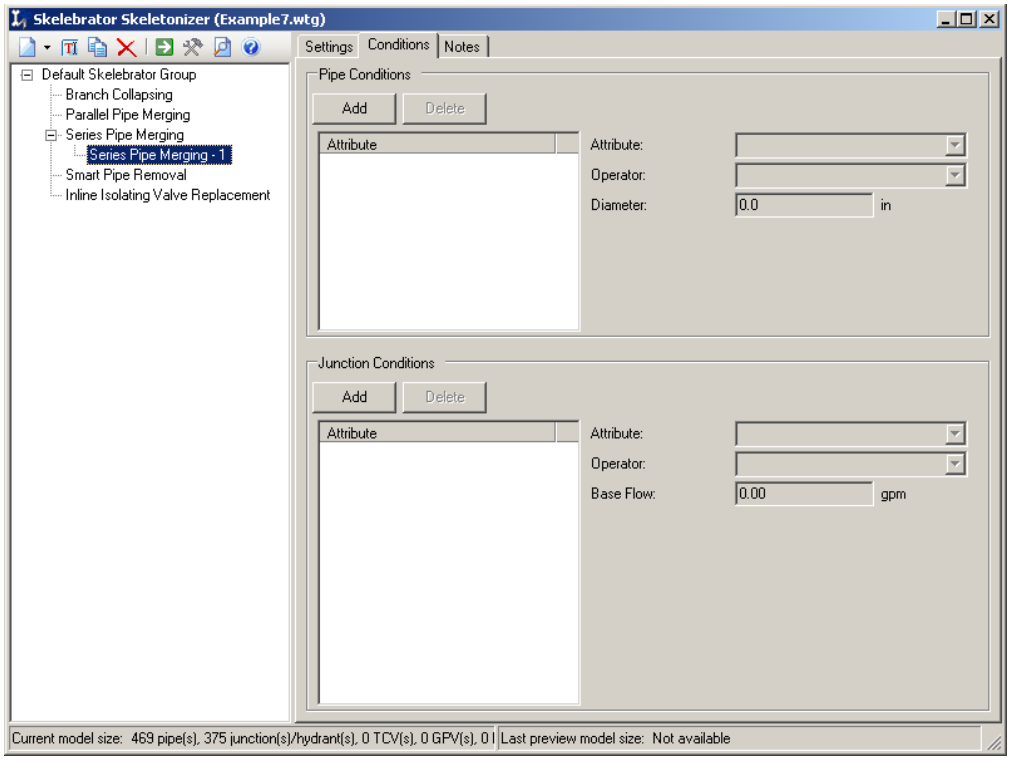
Note: The resulting pipe from a Series Pipe Merging operation is routed in the same direction as the dominant pipe. Therefore, upstream and downstream nodes relate to the topological direction of the dominant pipe. If check valves are present, then the resulting pipe is routed in the direction of the pipe that contains the check valve. If check valves are present in both pipes and those pipes oppose each other then skeletonization is not performed.

– **Apply Minor Losses**—Select **Apply Minor Losses** if you wish for Skelebrator to preserve any minor losses attached to the pipes in your network. For Series Pipe Merging the minor losses for the original pipes are summed and added to the resulting pipe. If this option is not selected then the minor loss of the resulting pipe will be set to zero.

Tip: To combine only pipes with the same hydraulic characteristics (i.e., diameter and roughness), create a Series Pipe Removal Operation and click the Conditions tab. Then, add a pipe tolerance condition of 0.0 and a roughness tolerance condition of 0.0. Also, make sure to deselect the *Use Equivalent Pipes* check box.

– **Allow Removal of TCVs**—Activate this option by checking the box to allow Skelebrator to remove TCVs during the Series Pipe Merging operation.

2. Click **Conditions** to edit or create conditions.



a. Click **Add** to add conditions. You can add pipe and/or junction conditions. You can add more than one condition.

b. Or, select an existing condition and click **Edit** to modify a selected condition. You can add and edit Junction and Pipe Conditions.

Note: In the case where not all nodes connected to the two pipes are junctions, tolerances are only evaluated based upon the junction type nodes. For example, if a tolerance of 5gpm was defined this would not invalidate the merging of two pipes that had one uncommon node that was a pump, for example. The tolerance condition would be evaluated based only upon the two junction type nodes.

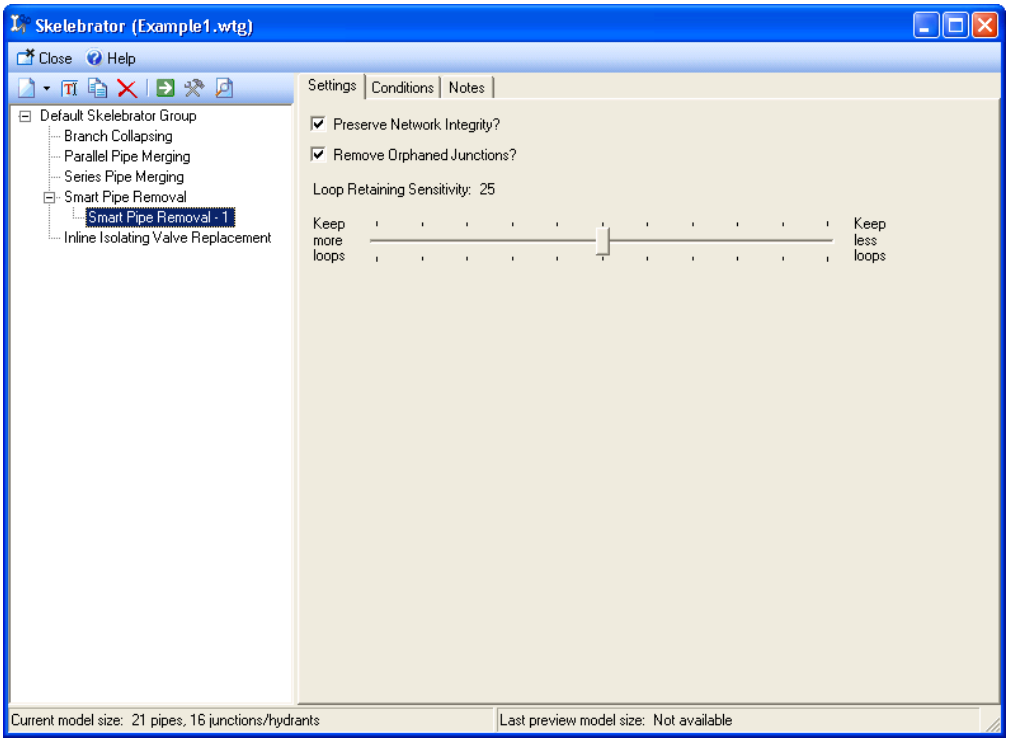
The Pipe Condition Editor allows you to set select parameters that determine which pipes are included in the skeletonizing process. Tolerances can also be specified for both pipe and junction conditions.

In the context of series pipe merging, pipe tolerances are calculated between the specified attribute of the two pipes to be merged. For example, a tolerance on diameter of 2-in. means that only pipes within a range of 2-in. diameter of each other will be merged (i.e., a 6-in. and an 8-in. pipe would be merged, an 8-in. and a 12-in. pipe would not).

In the context of series pipe merging, junction tolerances are calculated on all present junctions. If all three nodes are junctions, then all three junctions will be used to evaluate the tolerance. For example, a tolerance of 10 ft. on elevation would mean that the two pipes would not be merged unless all of the three junctions had an elevation within 10 ft. of each other.

Smart Pipe Removal Operations

When you add or edit a removal operation, the Smart Pipe Removal Operation Editor dialog box opens. Removal operations have two sets of parameters, Settings and Conditions.



Note: We recommend that Smart Pipe Removal be performed with conditions defined. At the very least, a limiting condition placed on pipe diameter should be used. Smart Pipe Removal is designed to allow removal of small diameter pipes (including those that form parts of loops) and thus it is recommended that smart pipe removal be used with a condition that limits the scope to only remove small diameter pipes.

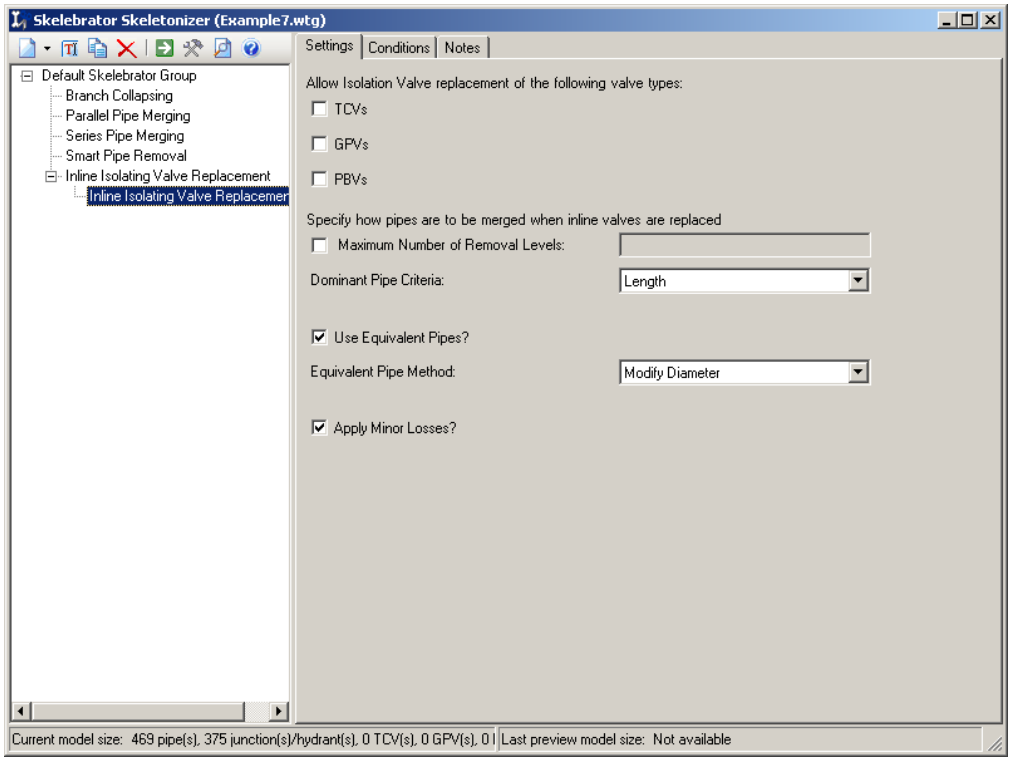
1. Click the **Settings** tab to edit settings.
   - **Preserve Network Integrity**—Select **Preserve Network Integrity** if you want Skelebrator to ensure the topological integrity of your network will not be broken by a removal operation. All non-junction node elements (valves, tanks, pumps and reservoirs) will remain connected to the network, and the network will not be disconnected by Skelebrator. Total system demand will be preserved. Any junctions marked as non-removable will also remain connected to the network.
   - **Remove Orphaned Nodes**—Select **Remove Orphaned Nodes** if you want Skelebrator to find and automatically remove any nodes left disconnected from the network after removal operations. (Orphaned or disconnected nodes are solitary nodes no longer connected to any pipes. By virtue of the nature of pipe removal, junctions can be left disconnected.) Note that Skelebrator does not remove any orphaned nodes that were orphaned prior to skeletonization. This option is not available if the preserve network integrity is not selected. If you leave this option unchecked, your model will contain junctions not physically connected to the hydraulic network, which will result in warning messages when you run your model.
   - **Loop Retaining Sensitivity**—Adjust the loop retaining sensitivity in order to control how sensitive the pipe removal algorithm is to retaining loops in your model. The lower the setting is, and in the absence of any other limiting conditions, the higher number of loops will be retained in your model (i.e., loops are less likely to be broken). Conversely, a higher setting will favor retaining less loops in your model. Use this setting in tandem with Skelebrator's preview feature to get a feel for the effect of the various settings. This option is only available if you have selected the **Preserve Network Integrity** option.
2. Click **Conditions** to edit or create pipe conditions. You can add more than one condition.
3. Click **Add** to add pipe conditions. You can add more than one condition.
4. Or, select an existing condition and click **Edit** to modify a selected condition.

The condition editor allows you to define pipe conditions that determine which pipes are included in the Smart Pipe Removal process. It is acceptable to define an operation that has no conditions (the default). In this case no pipes will be excluded from the skeletonization based on any of their physical attributes alone.

Inline Isolating Valve Replacement

In many GIS models, isolating valves split pipes into two segments, creating large numbers of redundant pipes that affect model performance and unnecessarily increase model complexity. This feature allows you easily remove the isoation valves, merge the adjacent pipe segments, and assign new isolation valve elements to the newly created pipes.

When you add or edit an Inline Isolating Valve Replacement operation, the Inline Isolating Valve Replacement Operation Editor dialog box opens. Operations have two sets of parameters, Settings and Conditions.



The Settings tab consists of the following controls:

- **Allow Isolation Valve replacement of the following valve types**: Check the boxes for each of the valve types (TCV, PBV, GPV) that you want Skelebrator to replace with isolation valves.
- **Maximum Number of Removal Levels**: Set the maximum number of pipe segments to remove for each isolation valve in the original model.

- **Dominant Pipe Criteria**: Select the criteria by which Skelebrator determines the dominant pipe (the one that will be kept after the operation). The dominant pipe is the pipe whose properties are retained as appropriate. For example, when merging a 6-in. pipe and an 8-in. pipe, if diameter is selected as the dominant pipe criteria then the larger diameter pipe (e.g., 8-in.) will provide the properties for the new pipe. That is, the 8-in. pipe's diameter, roughness, bulk reaction rate, etc., will be used for the new pipe
- **Use Equivalent Pipes**: Select Use Equivalent Pipe if you want Skelebrator to adjust remaining pipes to accommodate the removal of other pipes in series.
- **Equivalent Pipe Method**: Select whether you wish to modify the dominant pipe roughness or the dominant pipe diameter for the equivalent pipe calculations.
- **Apply Minor Losses**: When this box is checked minor losses associated with the newly created valve will be applied.

Conditions and Tolerances

Conditions and Tolerances are used in Skelebrator to define the scope of Skelebrator operations. They consist of an attribute (e.g., diameter), an operator (e.g., less than) and a unitized value (e.g., 6 inches). These values together define the effect of the condition. The examples just listed when combined into a condition would reduce the scope of an operation to only skeletonizing pipes with a diameter less than 6 inches.

A condition is able to be assessed based on a single element type, regardless of topology. It is possible to assess whether pipes meet the specified condition of diameter less than 6 inches without knowing the pipes' location in the hydraulic model. Tolerances, however, are different. They are assessed based on the ensuing topology, and thus, the meaning of a tolerance varies depending on Skelebrator operation type. Additionally, the tolerance operator is not available when it doesn't make sense. For example, it does not make sense to define a pipe tolerance for Smart Pipe Removal since only a single pipe is being considered at a time. An example of a valid tolerance is for Branch Collapsing where a junction tolerance can be specified between the two end junctions of the pipe.

Conditions and tolerances are cumulative. That is with every additional condition, the number of pipes able to be skeletonized will be reduced. Setting conflicting conditions such as diameter < 6-in. and diameter > 8-in. will result in no pipes being able to be skeletonized since conditions are joined with the logical AND operator. It is not possible to specify OR conditions or tolerances.

It is possible to specify no conditions for a particular operation. In that case all pipes are valid for skeletonization based on their physical attributes.

However, conditions and tolerances are not the only elements that determine whether a pipe will be skeletonized. For a pipe to be skeletonized it has to meet all of the following criteria:

- Be valid in terms of the network topology with respect to the particular skeletonization operation. That is, during Branch Reduction the pipe has to be part of a branch. Any pipes whose topology dictates they are not part of a branch will not be skeletonized.
- Must not be an element that is inactive as part of a topological alternative. All inactive topological elements are immune to skeletonization.
- Must not be referenced by a logical control, simple control, or calibration observed data set.
- Must not be connected to a VSP control node or the trace node for WQ analysis.
- Must not be a user-protected element.
- Must meet all user defined conditional and tolerance criteria.

Pipe Conditions and Tolerances

Click **Add** to add conditions. You can add more than one condition.

Attribute—Select the Attribute that you want to use to determine which pipes to skeletonize. These include:

- Bulk Reaction Rate
- Diameter
- Has Check Valve
- Installation Year
- Length
- Material
- Minor Loss Coefficient
- Roughness
- Wall Reaction Rate.

Operator—Select an operator that defines the relationship between the attribute you select and the value you select for that attribute. For example, if you select an attribute of Diameter, an operator of Less Than, and a value of 6 in., then any pipes with less than a 6-in. diameter are valid for skeletonization. Depending on operation type, Tolerance may also be an option for operator. When using a tolerance, a tolerance (as opposed to a condition) is defined. For example, in the context of Series Pipe Merging where two pipes are being merged, a tolerance of 2-in. diameter means that those pipes will only be merged if their diameters are within 2-in. of each other.

Value—The label, units, and appropriate value range depend on the attribute you select.

Junction Conditions and Tolerances

You can set selective parameters that determine which junctions are included in Branch Collapsing, Parallel Pipe Merging and Series Pipe Merging operations. Click Add to activate.

Attribute—Select the Attribute that you want to use to determine which junctions to trim. These include:

- Base Flow
- Elevation
- Emitter Coefficient.

Operator—Select an operator that defines the relationship between the attribute you select and the value you select for that attribute. For example, if you select an attribute of Base Demand, an operator of Less Than, and a value of 50 gpm, any pipes with end nodes with a base demand less than 50 gpm are valid for skeletonization.

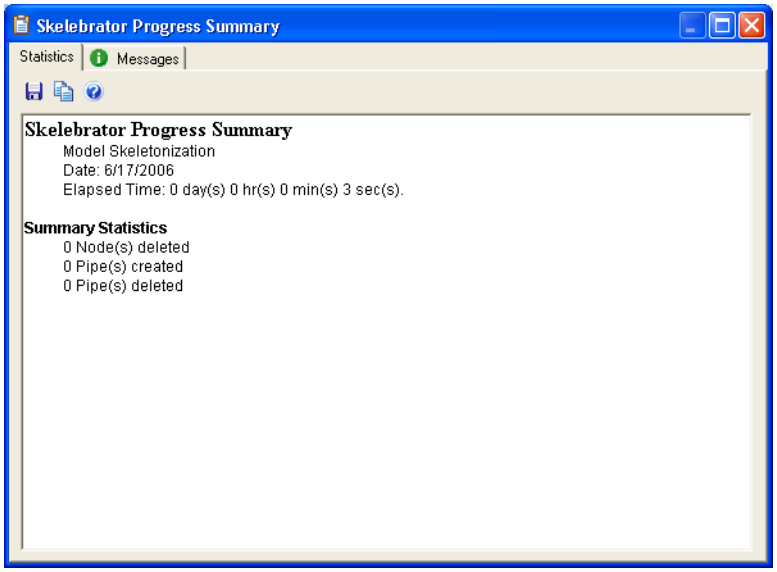
Value—The label, units, and appropriate value range depend on the attribute you select.

Junction tolerances are only evaluated against junctions. For example, if two series pipes are to be merged but their common node is a pump, any defined junction tolerance is evaluated based on the two end nodes only.

Where only one junction exists, as may be the case when allowing skeletonization of TCVs, tolerance conditions are not evaluated and do not limit the scope of the skeletonization.

Skelebrator Progress Summary Dialog Box

This dialog box opens following the successful completion of an automatic skeletonization operation. The text pane provides information concerning the operation that was performed, including the model name, date, the length of time the operation took to run, and the number of elements that were modified.



Click the Save Statistics button on the Statistics tab to save the summary to a text file. Click the Copy Statistics button to copy the summary to the Windows clipboard. The Messages tab displays warning, error, and success messages as applicable.

Backing Up Your Model

In ArcGIS (ArcCatalog or ArcMap), there is no ability to undo your changes after they have been made. Skelebrator makes transactions against the GEMS database without the ability to rollback those changes. From within WaterCAD V8*i*, changes can be undone on a global level by not saving the model after skeletonizing. However, any changes made prior to skelebration will also be lost if this method of avoiding committing skeletonization changes is used.

Making a copy of your model up front will ensure that you can always get back to your original model if problems occur.

Note: We strongly recommended that you first make a copy of your model as a safe guard before proceeding with Skelebration.

Skeletonization and Scenarios

Skelebrator is designed to skeletonize a single scenario at a time. Specifically, skelebrator modifies information in the set of alternatives (topological, demand, physical etc.) that are referred to by the currently selected scenario. It follows that any other scenarios that refer to these alternatives in some way can also potentially be modified by skeletonization but most likely in an undesirable and inconsistent way, since skeletonization only works on the data in the alternatives referenced by the currently active scenario.

For example, a second scenario that references all the same alternatives as the scenario being skeletonized except for, say, the demand alternative, will itself be seemingly skeletonized (its topological and physical alternatives, etc. are modified) except that the values of demands in its local demand records have no way of being factored into the skeletonization process. Due to this, demands may actually be lost since pipes that were deleted (e.g., dead ends) did not have their local demands relocated upstream. Relocated demands will represent the result of merging the demands in the parent alternative and not those of the child alternative where local records are present.

Due to the behavior of skeletonization with respect to scenarios and alternatives and to save possible confusion after skeletonization, it is very strongly recommended that you eliminate all other scenarios (other than the one to be skeletonized) from the model prior to skeletonization. Some exceptions, however, exist to this recommendation and may provide some additional flexibility to those users who have a strong desire to skeletonize multiple scenarios. In general, it is strongly recommended that multiple scenario skeletonization be avoided.

A multiple scenario model can be successfully skeletonized only if all of the following conditions are met:

- All scenarios all belong to the same parent-child hierarchy
- The scenario being selected for skeletonization must contain only parent (base) alternatives
- All elements that reference local records in any child alternative are protected from skeletonization.

As a simple example, consider a model with two scenarios, Base and Fire Flow. The Base scenario references a set of parent (base) alternatives, and the Fire Flow scenario references all the same alternatives, except for the demand alternative, where it references a child alternative of the Base scenario demand alternative, with local records at

junctions A-90 and A-100 which are to model the additional flow at the fire flow junctions. This model meets all of the above 3 conditions and thus skeletonization of this model can be conducted successfully for all scenarios in the model, but only if all of the following skeletonization rules are adhered to:

- The Base scenario is always selected for skeletonization
- The elements associated with local demand records (i.e., junctions A-90 and A-100 in our example) are protected from skeletonization using the Skelebrator element protection feature.

The reason the base scenario (a) must be selected for skeletonization is so that only parent (base) alternatives are modified by skeletonization. This is so that changes made to alternatives propagate down the parent-child hierarchy. If skeletonization was to occur on a scenario that referenced child alternatives, then the changes made to the scenario will not propagate back up the parent-child hierarchy and would result in incorrect results.

The reason for the element protections (b) is to limit the scope of skeletonization to the data common to both scenarios. That is, any model elements that possess any local records in any referenced child alternative are excluded from the skeletonization since the differences in properties between the child and parent alternatives cannot be resolved in a skeletonization process that acts for all intents and purposes on a single scenario. This idiom can be extended to other alternative types besides the demand alternative.

Note: Before you use Skelebrator, we strongly recommended that you eliminate from your model all scenarios other than the one to be skeletonized.

Importing/Exporting Skelebrator Settings

Skeletonization settings can be saved and restored by using Skelebrator's import/export feature. This feature allows all skeletonization settings to be retained and reused later on the same computer or on different computers as required.

In addition to saving skelebrator operations and batch run settings, protected element information is saved. Ideally, this information should be stored only with the model that it pertains to, because it only makes sense for that model, but that limitation would prevent skelebrator settings to be shared between different projects or users. The caveat of allowing protected element information to be saved in a file that is separate to the original model and thus be able to be shared between users, is that the situation is created whereby importing a .SKE file that was created with another model can result in meaningless protected element information being imported in the context of the new model.

However, your protected element information will probably be valid if you import a skelebrator .SKE file that was created using the same original model, or a model that is closely related to the original. The reason for this is that protected element information is stored in a .SKE file by recording the element's GEMS IDs from the GEMS database. For the same or closely related models, the same pipes and junctions will still have the same GEMS IDs and so, will remain correctly protected.

Protected element behavior for imported files is not guaranteed because a potential problem arises when elements that were deleted from the model were previously marked as protected and where the following three things have happened in order:

1. Modeling elements (pipes, junctions) have been deleted from the model.
2. The model database is compacted (thus making available the IDs of deleted elements for new ones).
3. New elements (pipes, junctions) have been added to the model after compaction, potentially using IDs of elements that have been deleted earlier.

From the above steps, it is possible that the IDs of new pipe or junction elements are the same as previously protected and deleted elements, thereby causing the new elements to be protected from skeletonization when they should not necessarily be protected.

Even though the above protected-element behavior is conservative by nature, it is recommended that you review protected element information after importing a .SKE file to make sure that it is correct for your intended skeletonization purposes.

Note: We strongly recommended that you review protected element settings when importing a .SKE file that was created using a different model.

Skeletonization and Active Topology

Skeletonization occurs on only active topology but considers all topology. That is, any inactive topology of a model is unable to be skeletonized but is not outright ignored for skeletonization purposes. This fact can be used to perform spatial skeletonization. For example, if you only wish to skeletonize a portion of your model, you can temporarily deactivate the topology you wish to be immune to skeletonization, remembering of course, to reactivate it after you have completed the skeletonization process. Any points where inactive topology ties in to the active topology will not be compromised. To better explain this, consider two series pipes that are not merged by series pipe removal. Under most circumstances two series pipes that meet the following conditions will be skeletonized:

- Meet topological criteria (e.g., that the two pipes are in series and have a common node that is legal to remove, i.e., not a tank, reservoir, valve or pump)

- Meet all conditional and tolerance based criteria
- Are not protected from skeletonization
- Have a common node that is not protected from skeletonization
- Have no simple control or logical control references
- Have no calibration references including to the junctions they are routed between
- Are routed between nodes that are free of references from variable speed pumps (VSPs)
- Are routed between nodes that are free from Water Quality (WQ) trace analysis references
- Are routed between nodes that represent at least one junction, if the common node is a loaded junction (so the load can be distributed)
- Do not have opposing check valves.

The two series pipes still may not be skeletonized if any inactive topology could be affected by the execution of the skeletonization action. For example, if the two series pipes have an additional but inactive pipe connected to their common node, and if the series pipe removal action was allowed to proceed, the common node would be removed from the model, and the inactive topology would become invalid. This is prevented from occurring in Skelebrator.

Scenarios and Alternatives

9

[Understanding Scenarios and Alternatives](#)

[Scenario Example - A Water Distribution System](#)

[Scenarios](#)

[Alternatives](#)

Understanding Scenarios and Alternatives

Scenarios and alternatives allow you to create, analyze, and recall an unlimited number of variations of your model. In Bentley WaterCAD V8*i* , scenarios contain alternatives to give you precise control over changes to the model.

Scenario management can dramatically increase your productivity in the "What If?" areas of modeling, including calibration, operations analysis, and planning.

Advantages of Automated Scenario Management

In contrast to editing or copying data, automated scenario management using inheritance gives you significant advantages:

- A single project file makes it possible to generate an unlimited number of "What If?" conditions without becoming overwhelmed with numerous modeling files and separate results.
- The software maintains the data for all the scenarios in a single project so it can provide you with powerful automated tools for directly comparing scenario results where any set is available at any time.
- The Scenario/Alternative relationship empowers you to mix and match groups of data from existing scenarios without having to re-declare any data.
- You do not have to re-enter data if it remains unchanged in a new alternative or scenario, avoiding redundant copies of the same data. It also enables you to correct a data input error in a parent scenario and automatically update the corrected attribute in all child scenarios.

These advantages may not seem compelling for small projects, however, as projects grow to hundreds or thousands of network elements, the advantages of true scenario inheritance become clear. On a large project, being able to maintain a collection of base and modified alternatives accurately and efficiently can be the difference between evaluating optional improvements or ignoring them.

A History of What-If Analyses

The history of what-if analyses can be divided into two periods: Distributed Scenarios and Self Contained Scenarios.

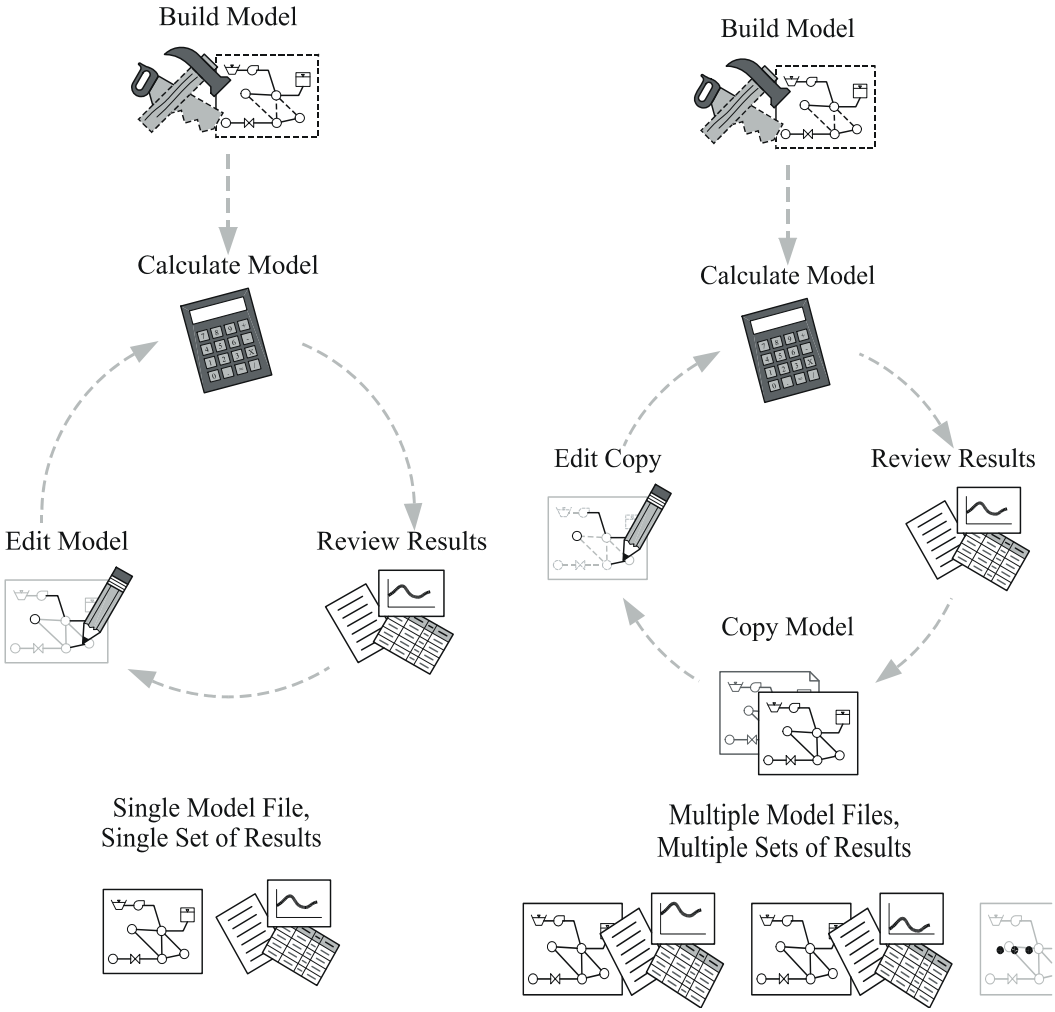
Distributed Scenarios

Traditionally, there have only been two possible ways of analyzing the effects of change on a software model:

- Change the model, recalculate, and review the results
- Create a copy of the model, edit that copy, calculate, and review the results.

Although either of these methods may be adequate for a relatively small system, the data duplication, editing, and re-editing become very time-consuming and error-prone as the size of the system and the number of possible conditions increase. Also, comparing conditions requires manual data manipulation, because all output must be stored in physically separate data files.

Distributed Scenarios



Self-Contained Scenarios

Effective scenario management tools need to meet these objectives:

- Minimize the number of project files the modeler needs to maintain.
- Maximize the usefulness of scenarios through easy access to things such as input and output data, and direct comparisons.
- Maximize the number of scenarios you can simulate by mixing and matching data from existing scenarios (data reuse).

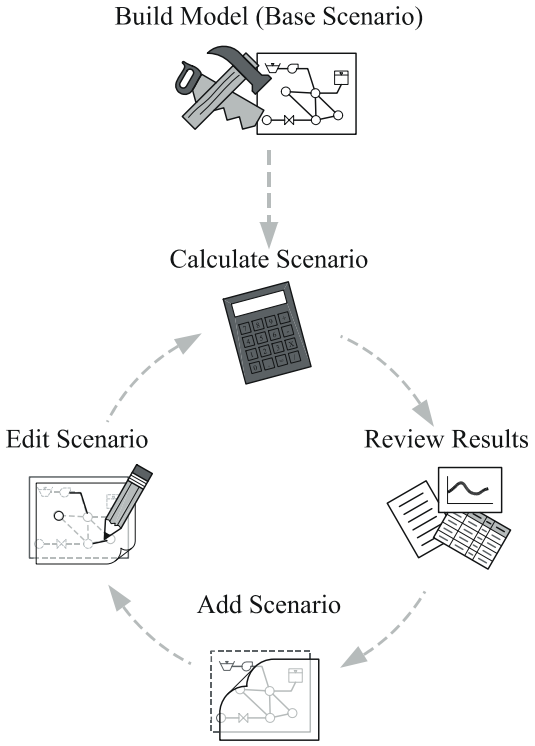
- Minimize the amount of data that needs to be duplicated to consider conditions that have a lot in common.

The scenario management feature in WaterCAD V8*i* successfully meets all of these objectives. A single project file enables you to generate an unlimited number of What If? conditions; edit only the data that needs to be changed and quickly generate direct comparisons of input and results for desired scenarios.

The Scenario Cycle

The process of working with scenarios is similar to the process of manually copying and editing data but without the disadvantages of data duplication and troublesome file management. This process allows you to cycle through any number of changes to the model, without fear of overwriting critical data or duplicating important information. It is possible to directly change data for any scenario, but an audit trail of scenarios can be useful for retracing the steps of a calibration series or for understanding a group of master plan updates.

Figure 9-1: Manual Scenarios

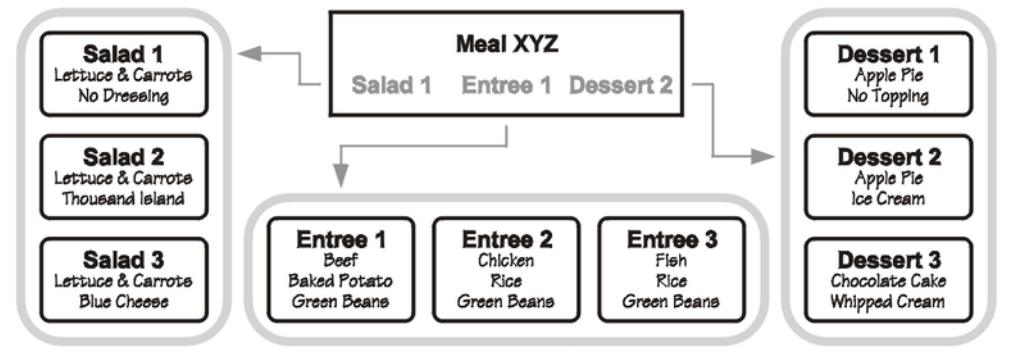


Scenario Attributes and Alternatives

- **Attribute**—An attribute is a fundamental property of an object and is often a single numeric quantity. For example, the attributes of a pipe include diameter, length, and roughness.
- **Alternative**—An alternative holds a family of related attributes so pieces of data that you are most likely to change together are grouped for easy referencing and editing. For example, a physical properties alternative groups physical data for the network's elements, such as elevations, sizes, and roughness coefficients.
- **Scenario**—A scenario has a list of referenced alternatives (which hold the attributes) and combines these alternatives to form an overall set of system conditions that can be analyzed. This referencing of alternatives enables you to easily generate system conditions that mix and match groups of data that have been previously created. Scenarios do not actually hold any attribute data—the referenced alternatives do.

A Familiar Parallel

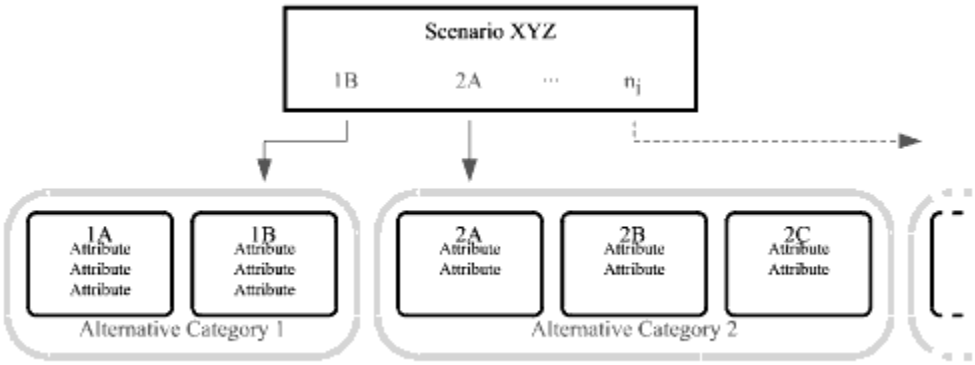
Although the structure of scenarios may seem a bit difficult at first, if you have ever eaten at a restaurant, you should be able to understand the concept. A meal (scenario) is comprised of several courses (alternatives), which might include a salad, an entrée, and a dessert. Each course has its own attributes. For example, the entrée may have a meat, a vegetable, and a starch. Examining the choices, we could present a menu as in the following figure:



The restaurant does not have to create a new recipe for every possible meal (combination of courses) that could be ordered. They can just assemble any meal based on what the customer orders for each alternative course. Salad 1, Entrée 1, and Dessert 2 might then be combined to define a complete meal.

Generalizing this concept, we see that any scenario references one alternative from each category to create a big picture that can be analyzed. Different types of alternatives may have different numbers and types of attributes, and any category can have an unlimited number of alternatives to choose from.

Generic Scenario Anatomy



Inheritance

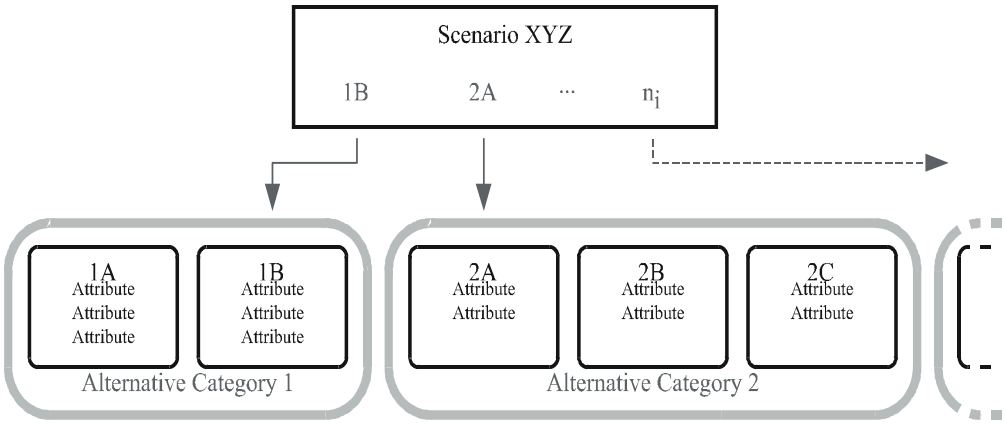
The separation of scenarios into distinct alternatives (groups of data) meets one of the basic goals of scenario management: maximizing the number of scenarios you can develop by mixing and matching existing alternatives. Two other primary goals have also been addressed: a single project file is used, and easy access to input data and calculated results is provided in numerous formats through the intuitive graphical interface.

In order to meet the objective of minimizing the amount of data that needs to be duplicated, and in order to consider conditions that have a lot of common input, you use inheritance.

In the natural world, a child inherits characteristics from a parent. This may include such traits as eye-color, hair color, and bone structure.

Overriding Inheritance

A child can override inherited characteristics by specifying a new value for that characteristic. These overriding values do not affect the parent and are therefore considered local to the child. Local values can also be removed at any time, reverting the characteristic to its inherited state. The child has no choice in the value of his inherited



attributes, only in local attributes.

For example, a child has inherited the attribute of blue eyes from his parent. If the child puts on a pair of green tinted contact lenses to hide his natural eye color, his natural eye color is overridden locally, and his eye color is green. When the tinted lenses are removed, the eye color reverts to blue, as inherited from the parent.

Dynamic Inheritance

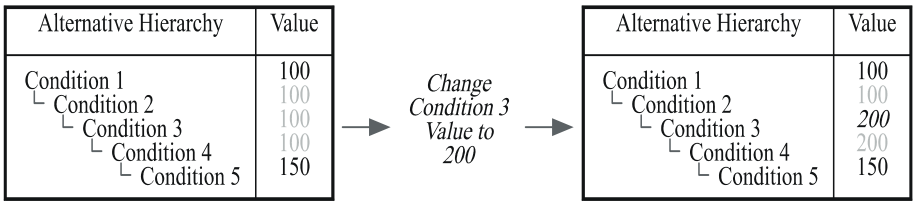
Dynamic inheritance does not have a parallel in the genetic world. When a parent's characteristic is changed, existing children also reflect the change. Using the eye-color example, this would be the equivalent of the parent changing eye color from blue to brown and the children's eyes instantly inheriting the brown color also. Of course, if the child has already overridden a characteristic locally, as with the green lenses, his eyes will remain green until the lenses are removed. At this point, his eye color will revert to the inherited color, now brown.

This dynamic inheritance has remarkable benefits for applying wide-scale changes to a model, fixing an error, and so on. If rippling changes are not desired, the child can override all of the parent's values, or a copy of the parent can be made instead of a child.

Local and Inherited Values

Any changes that are made to the model belong to the currently active scenario and the alternatives that it references. If the alternatives happen to have children, those children will also inherit the changes unless they have specifically overridden that attribute. The following figure demonstrates the effects of a change to a mid-level alternative. Inherited values are shown as gray text, local values are shown as black text.

A Mid-level Hierarchy Alternative Change



Minimizing Effort through Attribute Inheritance

Inheritance has an application every time you hear the phrase, "just like x except for y." Rather than specifying all of the data from x again to form this new condition, we can create a child from x and change y appropriately. Now we have both conditions with no duplicated effort.

We can even apply this inheritance to our restaurant analogy as follows. Inherited values are shown as gray text, local values are shown as black text.

Note: Salad 3 could inherit from Salad 2, if we prefer: "Salad 3 is just like Salad 2, except for the dressing."

Salad Alternative Hierarchy	Attribute: Vegetables	Attribute: Dressing
Salad 1	Lettuce & Carrots	No Dressing
├ Salad 2	Lettuce & Carrots	Thousand Island
└ Salad 3	Lettuce & Carrots	Blue Cheese

- "Salad 2 is just like Salad 1, except for the dressing."
- "Salad 3 is just like Salad 1, except for the dressing."

Note: **If the vegetable of the day changes (from green beans to peas), only Entrée 1 needs to be updated, and the other entrées will automatically inherit the vegetable attribute of "Peas" instead of "Green Beans."**

Entree Alternative Hierarchy	Attribute: Meat	Attribute: Starch	Attribute: Vegetable
Entree 1 └ Entree 2 └ Entree 3	Beef Chicken Fish	Baked Potato Rice Rice	Green Beans Green Beans Green Beans

- "Entrée 2 is just like Entrée 1, except for the meat and the starch."
- "Entrée 3 is just like Entrée 2, except for the meat."

Note: **Dessert 3 has nothing in common with the other desserts, so it can be created as a "root" or base alternative. It does not inherit its attribute data from any other alternative.**

Dessert Alternative Hierarchy	Attribute: Bakery Item	Attribute: Topping
Dessert 1 └ Dessert 2 Dessert 3	Apple Pie Apple Pie Chocolate Cake	No Topping Ice Cream Whipped Cream

- "Dessert 2 is just like Dessert 1, except for the topping."

Minimizing Effort through Scenario Inheritance

Just as a child alternative can inherit attributes from its parent, a child scenario can inherit which alternatives it references from its parent. This is essentially the phrase “just like x except for y”, but on a larger scale.

Using the meal example, consider a situation where you go out to dinner with three friends. The first friend orders a meal and the second friend orders the same meal with a different dessert. The third friend orders a different meal and you order the same meal with a different salad.

The four meal scenarios could then be presented as follows (inherited values are shown as gray text, local values are shown as black text).

- "Meal 2 is just like Meal 1, except for the dessert." The salad and entrée alternatives are inherited from Meal 1.
- "Meal 3 is nothing like Meal 1 or Meal 2." A new base or root is created.

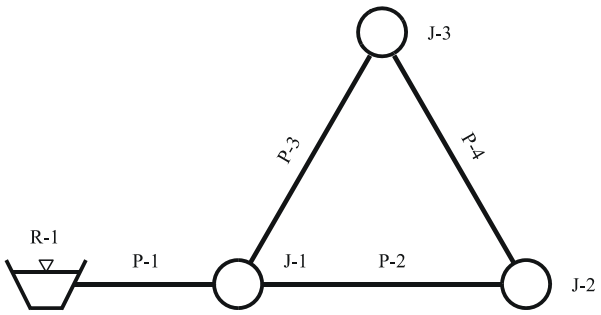
Meal Scenario Hierarchy	Salad Alternative	Entree Alternative	Dessert Alternative
Meal 1	Salad 1	Entree 2	Dessert 3
└ Meal 2	Salad 1	Entree 2	Dessert 1
Meal 3	Salad 3	Entree 3	Dessert 2
└ Meal 4	Salad 2	Entree 3	Dessert 2

- "Meal 4 is just like Meal 3, except for the salad." The entrée and dessert alternatives are inherited from Meal 3.

Scenario Example - A Water Distribution System

A water distribution system where a single reservoir supplies water by gravity to three junction nodes.

Example Water Distribution System



Although true water distribution scenarios include such alternative categories as initial settings, operational controls, water quality, and fire flow, the focus here is on the two most commonly changed sets of alternatives: demands and physical properties. Within these alternatives, the concentration will be on junction baseline demands and pipe diameters.

Building the Model (Average Day Conditions)

During model construction, only one alternative from each category is going to be considered. This model is built with average demand calculations and preliminary pipe diameter estimates. You can name the scenario and alternatives, and the hierarchies look like the following (showing only the items of interest):

Demand Alternative Hierarchy	J-1	J-2	J-3
Average Day	*100 gpm*	*500 gpm*	*100 gpm*

Physical Alternative Hierarchy	P-1	P-2	P-3	P-4
Preliminary Pipes	*8 inches*	*6 inches*	*6 inches*	*6 inches*

Scenario Hierarchy	Demand Alternative	Physical Alternative
Avg. Day	*Average Day*	*Preliminary Pipes*

Analyzing Different Demands (Maximum Day Conditions)

In this example, the local planning board also requires analysis of maximum day demands, so a new demand alternative is required. No variation in demand is expected at J-2, which is an industrial site. As a result, the new demand alternative can inherit J-2's demand from Average Day while the other two demands are overridden.

Demand Alternative Hierarchy	J-1	J-2	J-3
Average Day └ *Maximum Day*	100 gpm *200 gpm*	500 gpm 500 gpm	100 gpm *200 gpm*

Now we can create a child scenario from Average Day that inherits the physical alternative but overrides the selected demand alternative. As a result, we get the following scenario hierarchy:

Scenario Hierarchy	Demand Alternative	Physical Alternative
Avg. Day └ *Max. Day*	Average Day *Maximum Day*	Preliminary Pipes Preliminary Pipes

Since no physical data (pipe diameters) have been changed, the physical alternative hierarchy remains the same as before.

Another Set of Demands (Peak Hour Conditions)

Based on pressure requirements, the system is adequate to supply maximum day demands. Another local regulation requires analysis of peak hour demands with slightly lower allowable pressures. Since the peak hour demands also share the industrial load from the Average Day condition, Peak Hour can be inherited from Average Day. In this instance, Peak Hour could also inherit from Maximum Day.

Demand Alternative Hierarchy	J-1	J-2	J-3
Average Day ├ Maximum Day └ *Peak Hour*	100 gpm 200 gpm *250 gpm*	500 gpm 500 gpm 500 gpm	100 gpm 200 gpm *250 gpm*

Another scenario is also created to reference these new demands, as shown below:

Scenario Hierarchy	Demand Alternative	Physical Alternative
Avg. Day ├ Max. Day └ *Peak*	Average Day Maximum Day *Peak Hour*	Preliminary Pipes Preliminary Pipes Preliminary Pipes

No physical data was changed, so the physical alternatives remain the same.

Correcting an Error

This analysis results in acceptable pressures until it is discovered that the industrial demand is not actually 500 gpm—it is 1,500 gpm. However, due to the inheritance within the demand alternatives, only the Average Day demand for J-2 needs to be updated. The changes effect the children. After the single change is made, the demand hierarchy is as follows:

Demand Alternative Hierarchy	J-1	J-2	J-3
Average Day ├ Maximum Day └ Peak Hour	100 gpm 200 gpm 250 gpm	*1,500 gpm* 1,500 gpm 1,500 gpm	100 gpm 200 gpm 250 gpm

Notice that no changes need to be made to the scenarios to reflect these corrections. The three scenarios can now be calculated as a batch to update the results.

When these results are reviewed, it is determined that the system does not have the ability to adequately supply the system as it was originally thought. The pressure at J-2 is too low under peak hour demand conditions.

Analyzing Improvement Suggestions

To counter the headloss from the increased demand load, two possible improvements are suggested:

- A much larger diameter is proposed for P-1 (the pipe from the reservoir). This physical alternative is created as a child of the Preliminary Pipes alternative, inheriting all the diameters except P-1's, which is overridden.
- Slightly larger diameters are proposed for all pipes. Since there are no commonalities between this recommendation and either of the other physical alternatives, this can be created as a base (root) alternative.

These changes are then incorporated to arrive at the following hierarchies:

Physical Alternative Hierarchy	P-1	P-2	P-3	P-4
Preliminary Pipes	8 inches	6 inches	6 inches	6 inches

Scenario Hierarchy	Demand Alternative	Physical Alternative
Avg. Day	Average Day	Preliminary Pipes
├ Max. Day	Maximum Day	Preliminary Pipes
└ Peak	Peak Hour	Preliminary Pipes
├ *Peak, Big P-1*	Peak Hour	*Larger P-1*
└ *Peak, All Big Pipes*	Peak Hour	*Larger All Pipes*

This time the demand alternative hierarchy remains the same since no demands were changed. The two new scenarios (Peak, Big P-1, Peak, All Big Pipes) can be batch run to provide results for these proposed improvements.

Finalizing the Project

It is decided that enlarging P-1 is the optimum solution, so new scenarios are created to check the results for average day and maximum day demands. Notice that this step does not require handling any new data. All of the information to be modeled is already present in the alternatives.

Scenario Hierarchy	Demand Alternative	Physical Alternative
Avg. Day	Average Day	Preliminary Pipes
├ Max. Day	Maximum Day	Preliminary Pipes
│ └ *Max. Day, Big P-1*	Maximum Day	*Larger P-1*
├ Peak	Peak Hour	Preliminary Pipes
│ ├ Peak, Big P-1	Peak Hour	Larger P-1
│ └ Peak, All Big Pipes	Peak Hour	Larger All Pipes
└ *Avg. Day, Big P-1*	Average Day	*Larger P-1*

Also note that it would be equally effective in this case to inherit the Avg. Day, Big P-1 scenario from Avg. Day (changing the physical alternative) or to inherit from Peak, Big P-1 (changing the demand alternative). Max. Day, Big P-1 could inherit from either Max. Day or Peak, Big P-1.

Neither the demand nor physical alternative hierarchies were changed in order to run the last set of scenarios, so they remain the same.

Demand Alternative Hierarchy	J-1	J-2	J-3
Average Day ├ Maximum Day └ Peak Hour	100 gpm 200 gpm 250 gpm	1,500 gpm 1,500 gpm 1,500 gpm	100 gpm 200 gpm 250 gpm

Physical Alternative Hierarchy	P-1	P-2	P-3	P-4
Preliminary Pipes └ *Larger P-1* *Larger All Pipes*	8 inches *18 inches* *12 inches*	6 inches 6 inches *12 inches*	6 inches 6 inches *12 inches*	6 inches 6 inches *12 inches*

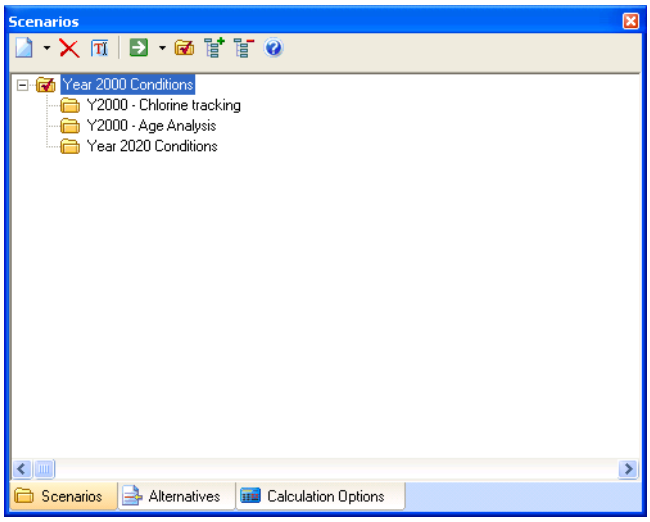
Scenarios

A Scenario contains all the input data (in the form of Alternatives), calculation options, results, and notes associated with a set of calculations. Scenarios let you set up an unlimited number of “What If?” situations for your model, and then modify, compute, and review your system under those conditions.

You can create an unlimited number of scenarios that reuse or share data in existing alternatives, submit multiple scenarios for calculation in a batch run, switch between scenarios, and compare scenario results—all with a few mouse clicks.

Scenarios Manager

The Scenario Manager allows you to create, edit, and manage an unlimited number of scenarios. There is one built-in default scenario—the Base scenario. If you want, you only have to use this one scenario. However, you can save yourself time by creating additional scenarios that reference the alternatives needed to perform and recall the results of each of your calculations.



The Scenario Manager consists of a hierarchical tree view and a toolbar. The tree view displays all of the scenarios in the project. If the Property Editor is open, clicking a scenario in the list causes the alternatives that make up the scenario to open. If the Property Editor is not open, you can display the alternatives and scenario information by selecting the desired scenario and right-clicking on Properties.

New Scenario 	Opens a submenu containing the following commands: • **Child Scenario**—creates a new Child scenario from the currently selected Base scenario. • **Base Scenario**—creates a new Base scenario.
Delete 	Removes the currently selected scenario, greyed out on the menu bar when Base Scenario is active.
Rename 	Renames the currently selected scenario.
Compute Scenario 	Opens a submenu containing the following command: • **Scenario**—calculates the currently selected scenario. • Hierarchy—Lets you calculate the entire currently highlighted branch—the Base scenario and all Child scenarios currently associated with it. • Children—Lets you calculate all of the Child scenarios associated with the currently highlighted scenario. • Batch Run—Lets you run a user-defined group of scenarios at once.

Make Current 	Causes the currently selected scenario to become the active one and displays it in the drawing pane.
Expand All 	Opens all scenarios within all folders in the list.
Collapse All 	Closes all of the folders in the list.
Help 	Displays online help for the Scenario Manager.

Note: When you delete a scenario, you are not losing data records because scenarios never actually hold calculation data records (alternatives do). The alternatives and data records referenced by that scenario exist until you explicitly delete them. By accessing the Alternative Manager, you can delete the referenced alternatives and data records.

Base and Child Scenarios

There are two types of scenarios:

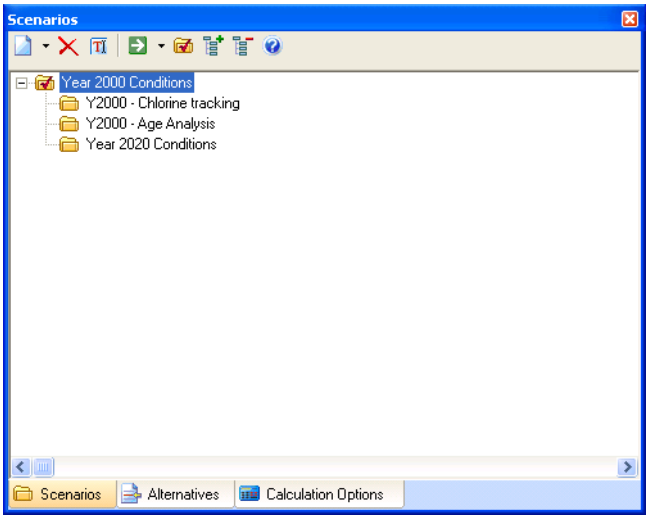
- **Base Scenarios**—Contain all of your working data. When you start a new project, you begin with a default base scenario. As you enter data and calculate your model, you are working with this default base scenario and the alternatives it references.
- **Child Scenarios**—Inherit data from a base scenario or other child scenarios. Child scenarios allow you to freely change data for one or more elements in your system. Child scenarios can reflect some or all of the values contained in their parent. This is a very powerful concept, giving you the ability to make changes in a parent scenario that will trickle down through child scenarios, while also giving you the ability to override values for some or all of the elements in child scenarios.

Creating Scenarios

You create new scenarios in the Scenario Manager. A new scenario can be a Base scenario or a Child scenario.

To create a new scenario

1. Select **Analysis** > **Scenarios** to open the Scenario Manager, or click  .



2. Click **New** and select whether you want to create a Base Scenario or a Child Scenario. When creating a Child scenario, you must first select the scenario from which the child is derived in the Scenario Manager tree view.

   By default, a new scenario comprises the Base Alternatives associated with each alternative type.

3. Double-click the new scenario to edit its properties in the Property Editor.

4. Close when finished.

Editing Scenarios

Scenarios can be edited in two places:

- The **Scenario Manager** lists all of the project's scenarios in a hierarchical tree format and displays the Base/Child relationship between them.
- The **Property Editor** displays the alternatives that make up the scenario that is currently selected in the Scenario Manager, along with the scenario label, any notes associated with the scenario, and the calculation options profile that is used when the scenario is calculated.

To edit a scenario

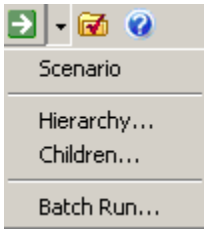
1. Select **Analysis** > **Scenarios** to open the Scenario Manager, or click  .
2. Double-click the scenario you want to edit to display its properties in the Properties Editor.
3. You can then edit the Scenario Label, Notes, Alternatives, and Calculation Options.
4. When finished, close the editor.

Running Multiple Scenarios at Once (Batch Runs)

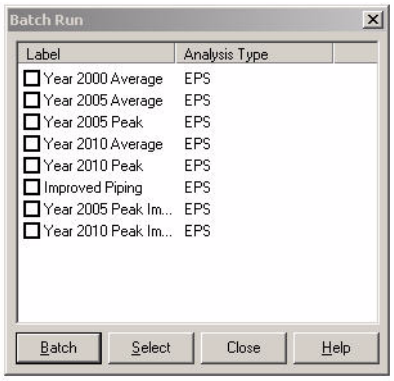
Performing a batch run allows you to set up and run calculations for multiple scenarios at once. This is helpful if you want to perform a large number of calculations or manage a group of smaller calculations as a set. It can be run at any time. The list of selected scenarios for the batch run remain with your project until you change it.

To perform a batch run

1. Select **Analysis** > **Scenarios** to open the Scenario Manager, or click  .
2. Click to open the **Compute** list and then select **Batch Run**. This will open the



Batch Run Editor.



3. Check the scenarios you want to run, then click **Batch**.

4. A **Please Confirm** dialog box opens to confirm running the selected scenarios as a batch. Click **Yes** to run.
5. When the batch is completed an **Information** box opens. Click **OK**.
6. Select a calculated scenario from the Scenario toolbar list to see the results throughout the program.

Note: When the batch run is completed, the scenario that was current stays current, even if it was not calculated.

Batch Run Editor Dialog Box

The Batch Run Editor dialog box contains the following controls:



Batch	Start the batch run of the selected scenarios.
Select	Display a menu containing the following commands: • **Select All**-Select all scenarios listed. • **Clear Selection**-Clear all selected scenarios.
Close	Close the Batch Run Editor dialog box.
Help	Display context-sensitive help for the Batch Run Editor dialog box.

Alternatives

Alternatives are the building blocks behind scenarios. They are categorized data sets that create scenarios when placed together. Alternatives hold the input data in the form of records. A record holds the data for a particular element in your system.

Scenarios are composed of alternatives as well as other calculation options, allowing you to compute and compare the results of various changes to your system. Alternatives can vary independently within scenarios and can be shared between scenarios.

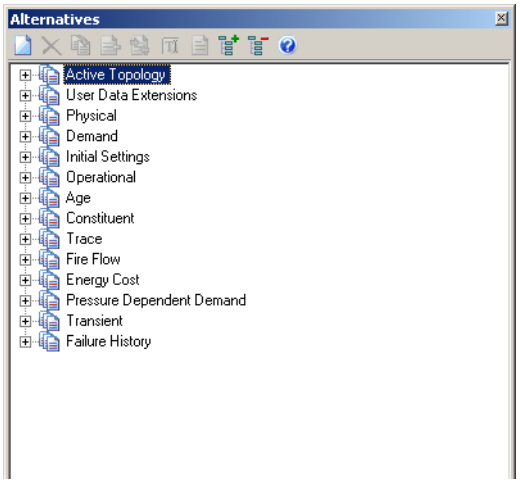
Scenarios allow you to specify the alternatives you want to analyze. In combination with scenarios, you can perform calculations on your system to see the effect of each alternative. Once you have determined an alternative that works best for your system, you can permanently merge changes from the preferred alternative to the base alternative.

When you first set up your system, the data that you enter is stored in the various base alternative types. If you want to see how your system behaves, for example, by increasing the diameter of a few select pipes, you can create a child alternative. You can make another child alternative with even larger diameters and another with smaller diameters. The number of alternatives that can be created is unlimited.

Note: WaterGEMS, WaterCAD, and HAMMER all use the same file format (.wtg). Because of this interoperability, some alternatives are exposed within a product even though that data is not used in that product (data in the Transient Alternative is not used by WaterGEMS, data in the Water Quality, Energy Cost, Flushing, etc. alternatives is not used in HAMMER, etc.).

Alternatives Manager

The Alternative Manager allows you to create, view, and edit the alternatives that make up the project scenarios. The dialog box consists of a pane that displays folders for each of the alternative types which can be expanded to display all of the alternatives for that type and a toolbar.

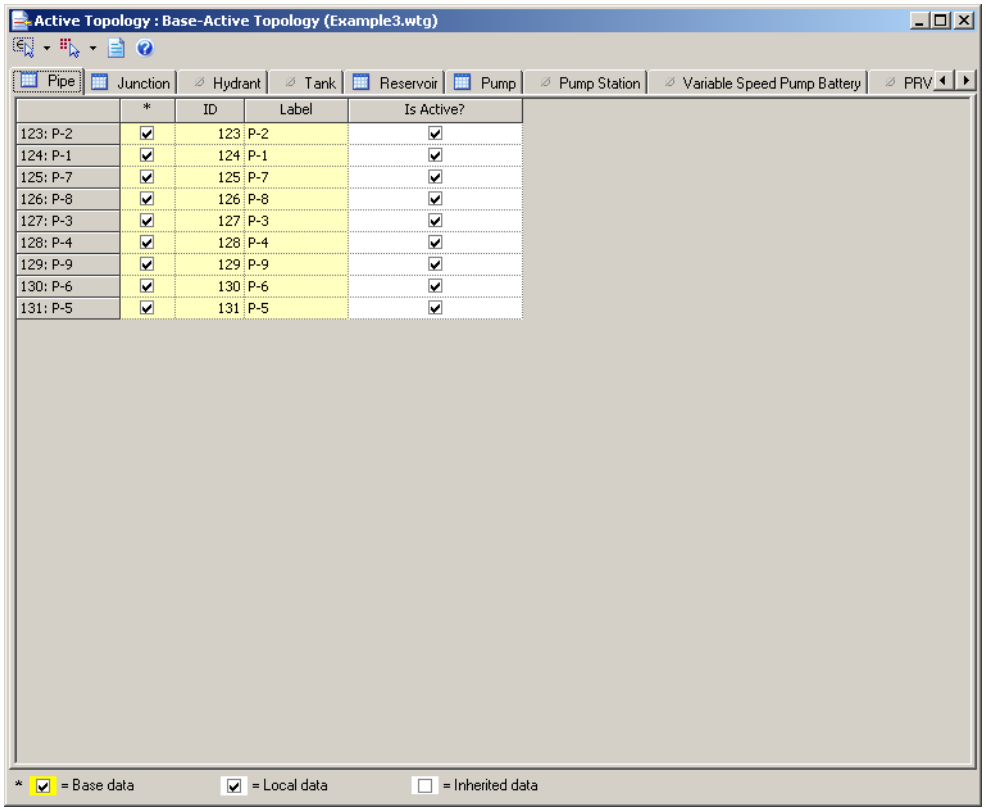


The toolbar consists of the following

New 	Creates a new Alternative.
Delete 	Deletes the currently selected alternative.
Duplicate 	Creates a copy of the currently selected alternative.
Open 	Opens the Alternative Editor dialog box for the currently selected alternative.
Merge Alternative 	Moves all records from one alternative to another.
Rename 	Renames the currently selected alternative.
Report 	Generates a report of the currently selected alternative.
Expand All 	Displays the full alternative hierarchy.
Collapse All 	Collapses the alternative hierarchy so that only the top-level nodes are visible.
Help 	Displays online help for the Alternative Manager.

Alternative Editor Dialog Box

This dialog box presents in tabular format the data that makes up the alternative being edited. Depending on the alternative type, the dialog box contains a separate tab for each element that possesses data contained in the alternative.



The Alternative Editor displays all of the records held by a single alternative. These records contain the values that are active when a scenario referencing this alternative is active. They allow you to view all of the changes that you have made for a single alternative. They also allow you to eliminate changes that you no longer need.

There is one editor for each alternative type. Each type of editor works similarly and allows you to make changes to a different aspect of your system. The first column contains check boxes, which indicate the records that have been changed in this alternative.

If the check box is selected, the record on that line has been modified and the data is local, or specific, to this alternative.

If the check box is cleared, it means that the record on that line is inherited from its higher-level parent alternative. Inherited records are dynamic. If the record is changed in the parent, the change is reflected in the child. The records on these rows reflect the corresponding values in the alternative's parent.

When the editor has tabs for various element types, you can determine whether the alternative contains data for that element type by the icon next to the element type ; if it is highlighted  , the alternative contains data for that element type. If the element type is not used in the current model the tab is marked with an icon  .

.

Note: **As you make changes to records, the check box automatically becomes checked. If you want to reset a record to its parent's values, clear the corresponding check box.**

Many columns support Global Editing (see Globally Editing Data), allowing you to change all values in a single column. Right-click a column header to access the Global Edit option.

The check box column is disabled when you edit a base alternative.

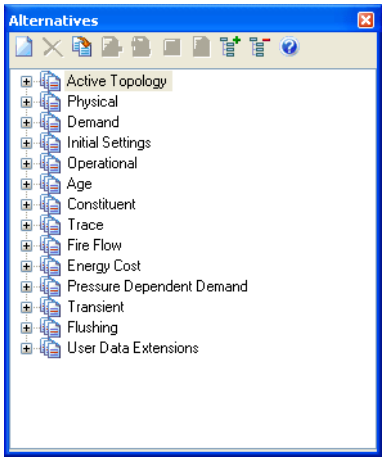
Base and Child Alternatives

There are two kinds of alternatives: Base alternatives and Child alternatives. Base alternatives contain local data for all elements in your system. Child alternatives inherit data from base alternatives, or even other child alternatives, and contain data for one or more elements in your system. The data within an alternative consists of data inherited from its parent and the data altered specifically by you (local data).

Remember that all data inherited from the base alternative are changed when the base alternative changes. Only local data specific to a child alternative remain unchanged.

Creating Alternatives

New alternatives are created in the Alternative Manager dialog box. A new alternative can be a Base scenario or a Child scenario. Each alternative type contains a Base alternative in the Alternative Manager tree view.



To create a new Alternative

1. Select **Analysis** > **Alternatives** to open the Alternative Manager, or click  .
2. To create a new Base alternative, select the type of alternative you want to create, then click the **New** button.
3. To create a new Child alternative, right-click the Base alternative from which the child will be derived, then select **New** > **Child Alternative** from the menu.
4. Double-click the new alternative to edit its properties.
5. Click Close when finished.

Editing Alternatives

You edit the properties of an alternative in its own alternative editor. The first column in an alternative editor contains check boxes, which indicate the records that have been changed in this alternative.

- If the box is checked, the record on that line has been modified and the data is local, or specific, to this alternative.
- If the box is not checked, it means that the record on that line is inherited from its higher-level parent alternative. Inherited records are dynamic. If the record is changed in the parent, the change is reflected in the child. The records on these rows reflect the corresponding values in the alternative's parent.

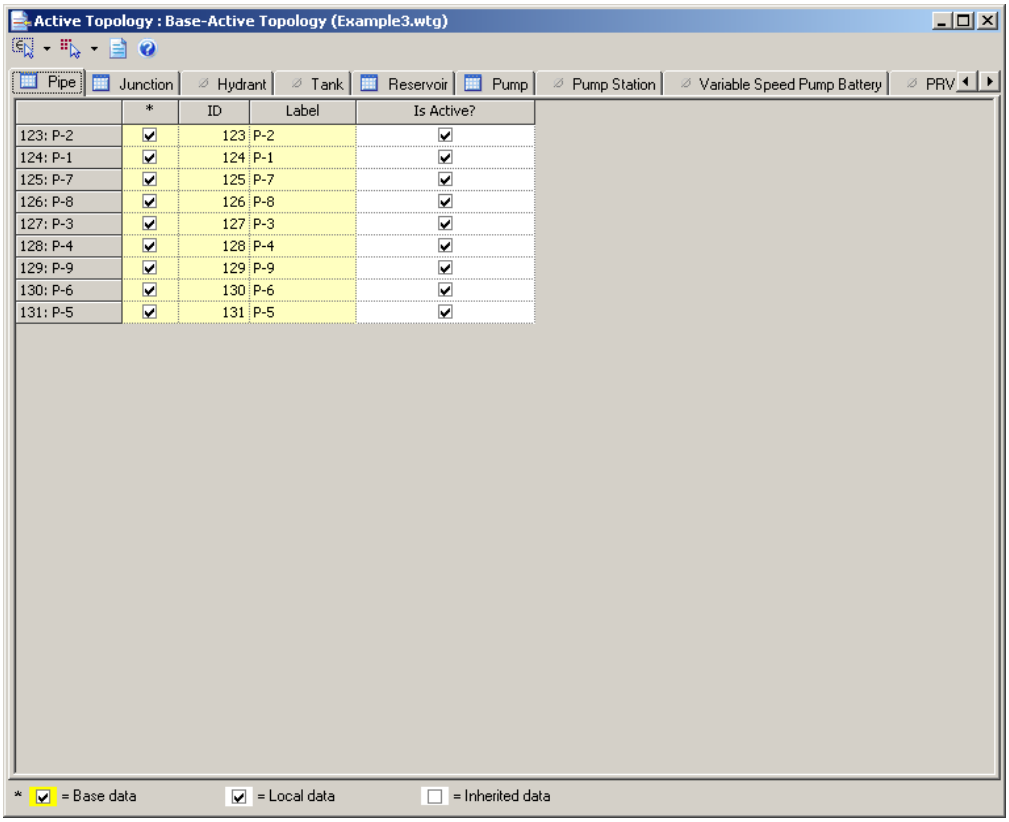
To edit an existing alternative, you can use one of two methods:

- Double-click the alternative to be edited in the Alternative Manager or
- Select the alternative to be edited in the Alternative Manager and click Edit 

In either case, the Alternative Editor dialog box for the specified alternative opens, allowing you to view and define settings as desired.

Active Topology Alternative

The Active Topology Alternative allows you to temporarily remove areas of the network from the current analysis. This is useful for comparing the effect of proposed construction and to gauge the effectiveness of redundancy that may be present in the system.



For each tab, the same setup applies—the tables are divided into four columns. The first column displays whether the data is Base or Inherited, the second column is the element ID, the third column is the element Label, and the fourth column allows you to choose whether or not the corresponding element is Active in the current alternative.

To make an element Inactive in the current alternative, clear the check box in the **Is Active?** column that corresponds to that element's Label.

The following buttons are available:

Selection Set



Opens a submenu containing the following options:

- **Create Selection Set**—Allows you to create a new selection set.
- **Add to Selection Set**—Adds all of the elements in the current tab of the alternative to a previously created selection set that you specify.
- **Remove from Selection Set**——Removes all of the elements in the current tab of the alternative from a previously created selection set that you specify.

Select in Drawing



Opens a submenu containing the following options:

- **Select in Drawing**—Selects the elements in the current tab of the alternative in the drawing pane.
- **Add to Current Selection**—Adds all of the elements in the current tab of the alternative to the group of elements that are currently selected in the Drawing Pane.
- **Remove from Current Selection**—Removes the elements in the current tab of the alternative from the group of elements that are currently selected in the Drawing Pane.
- **Select Within Current Selection**—Selects the element or elements that are both in the current tab of the alternative and are already selected in the Drawing Pane.

Report



Generates a report containing the data within the current alternative.

Help



Opens the online help.

Creating an Active Topology Child Alternative

When creating an active topology child alternative, you may notice that the elements added to the child scenario become available in your model when the base scenario is the current scenario.

To create an active topology alternative so that the elements added to the child scenario do not show up as part of the base scenario

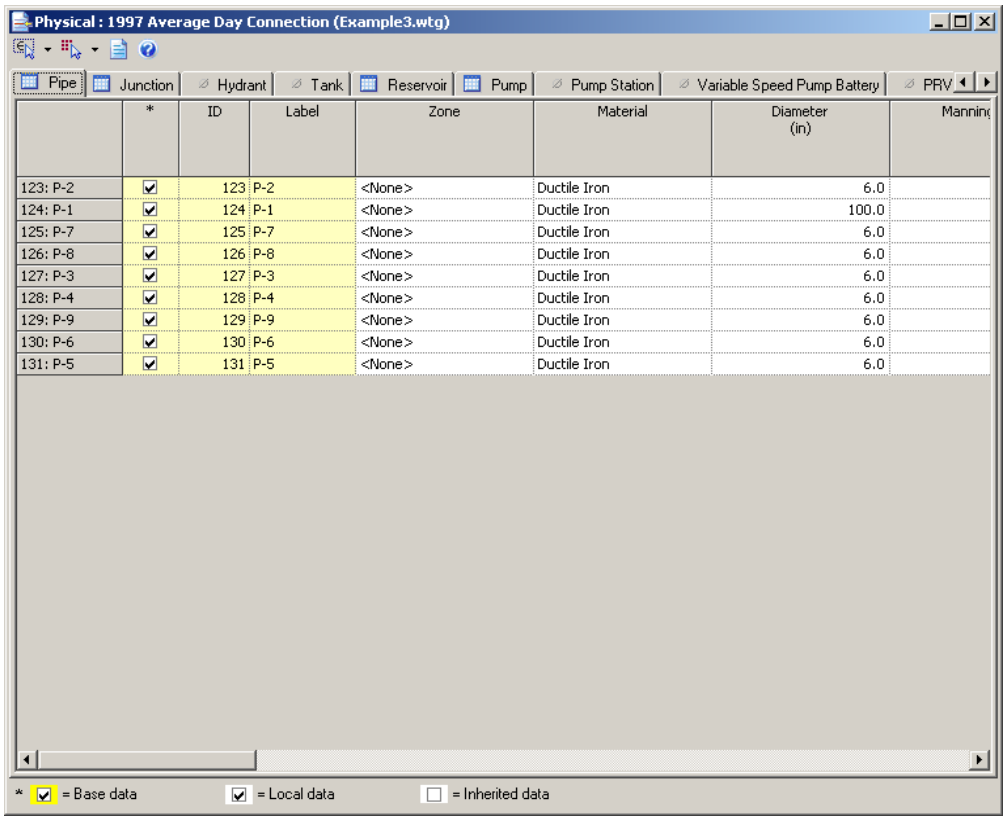
1. Create a new WaterCAD V8*i* project.
2. Open the Property Editor.
3. Open the Scenario Manager and make sure the Base scenario is current (active).
4. Create your model by adding elements in the drawing pane.
5. Create a new child scenario and a new child active topology alternative:
    a. In the Scenario Manager, click the **New** button and select **Child Scenario** from the submenu.
    b. The new Child Scenario is created and can be renamed.
    c. In the Alternatives Manager, open **Active Topology,** select the **Base Active Topology**, right-click to select **New**, then **Child Alternative**.
    d. Rename the new Child Alternative.
6. In the Scenario Manager, select the new child scenario then click **Make Current** to make the child scenario the current (active) scenario.
7. Add new elements to your model. These elements will be active only in the new child alternative.
8. To verify that this worked:
    a. In the Scenario Manager, select the base scenario then click **Make Current** to make the base scenario the current (active) scenario. The new elements are shown as inactive (they are grayed out in the drawing pane).
    b. In the Scenario Manager, select the new child scenario then click **Make Current** to make the child scenario the current (active) scenario. The new elements are shown as active.

Note: If you add new elements in the base scenario, they will show up in the child scenario.

Physical Alternative

One of the most common uses of a water distribution model is the design of new or replacement facilities. During design, it is common to try several physical alternatives in an effort to find the most cost effective solution. For example, when designing a replacement pipeline, it would be beneficial to try several sizes and pipe materials to find the most satisfactory combination.

Each type of network element has a specific set of physical properties that are stored in a physical properties alternative.To access the Physical Properties Alternative select **Analysis** > **Alternatives** and select Physical Alternative.



	*	ID	Label	Zone	Material	Diameter (in)	Mannin
123: P-2	☑	123	P-2	<None>	Ductile Iron	6.0	
124: P-1	☑	124	P-1	<None>	Ductile Iron	100.0	
125: P-7	☑	125	P-7	<None>	Ductile Iron	6.0	
126: P-8	☑	126	P-8	<None>	Ductile Iron	6.0	
127: P-3	☑	127	P-3	<None>	Ductile Iron	6.0	
128: P-4	☑	128	P-4	<None>	Ductile Iron	6.0	
129: P-9	☑	129	P-9	<None>	Ductile Iron	6.0	
130: P-6	☑	130	P-6	<None>	Ductile Iron	6.0	
131: P-5	☑	131	P-5	<None>	Ductile Iron	6.0	

The Physical Alternative editor for each element type is used to create various data sets for the physical characteristics of those elements.

The following buttons are available:

Selection Set



Opens a submenu containing the following options:

- **Create Selection Set**—Allows you to create a new selection set.
- **Add to Selection Set**—Adds all of the elements in the current tab of the alternative to a previously created selection set that you specify.
- **Remove from Selection Set**——Removes all of the elements in the current tab of the alternative from a previously created selection set that you specify.

Select in Drawing



Opens a submenu containing the following options:

- **Select in Drawing**—Selects the elements in the current tab of the alternative in the drawing pane.
- **Add to Current Selection**—Adds all of the elements in the current tab of the alternative to the group of elements that are currently selected in the Drawing Pane.
- **Remove from Current Selection**—Removes the elements in the current tab of the alternative from the group of elements that are currently selected in the Drawing Pane.
- **Select Within Current Selection**—Selects the element or elements that are both in the current tab of the alternative and are already selected in the Drawing Pane.

Report



Generates a report containing the data within the current alternative.

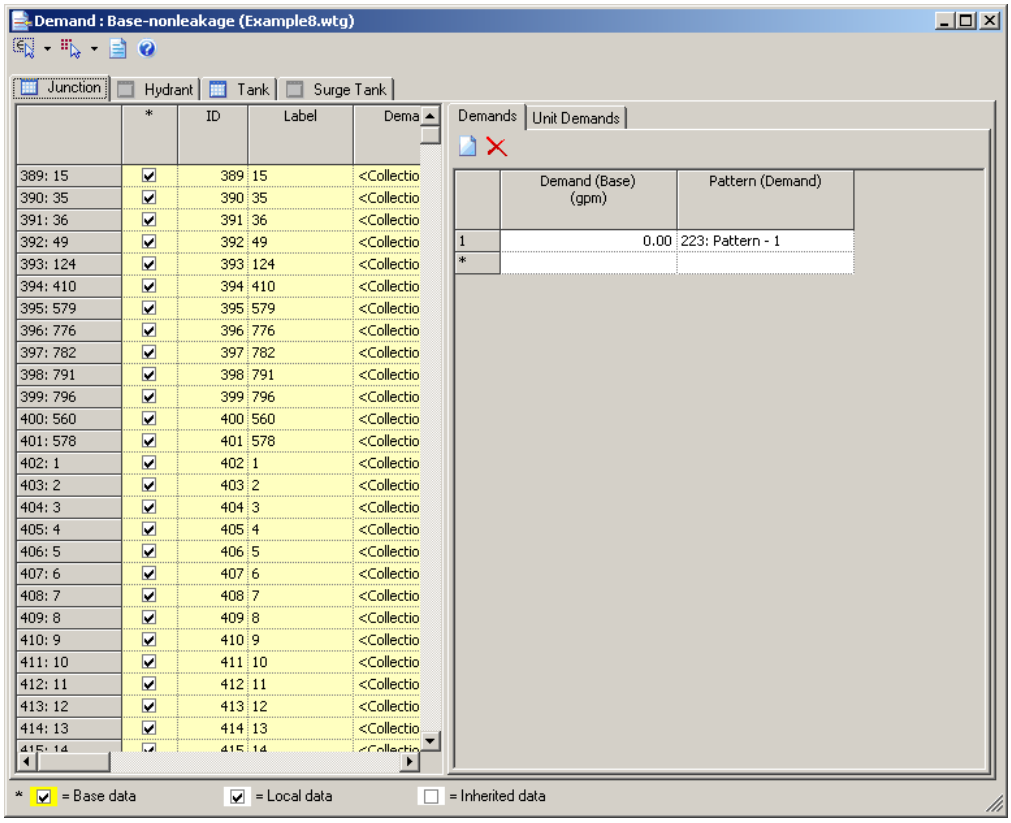
Help



Opens the online help.

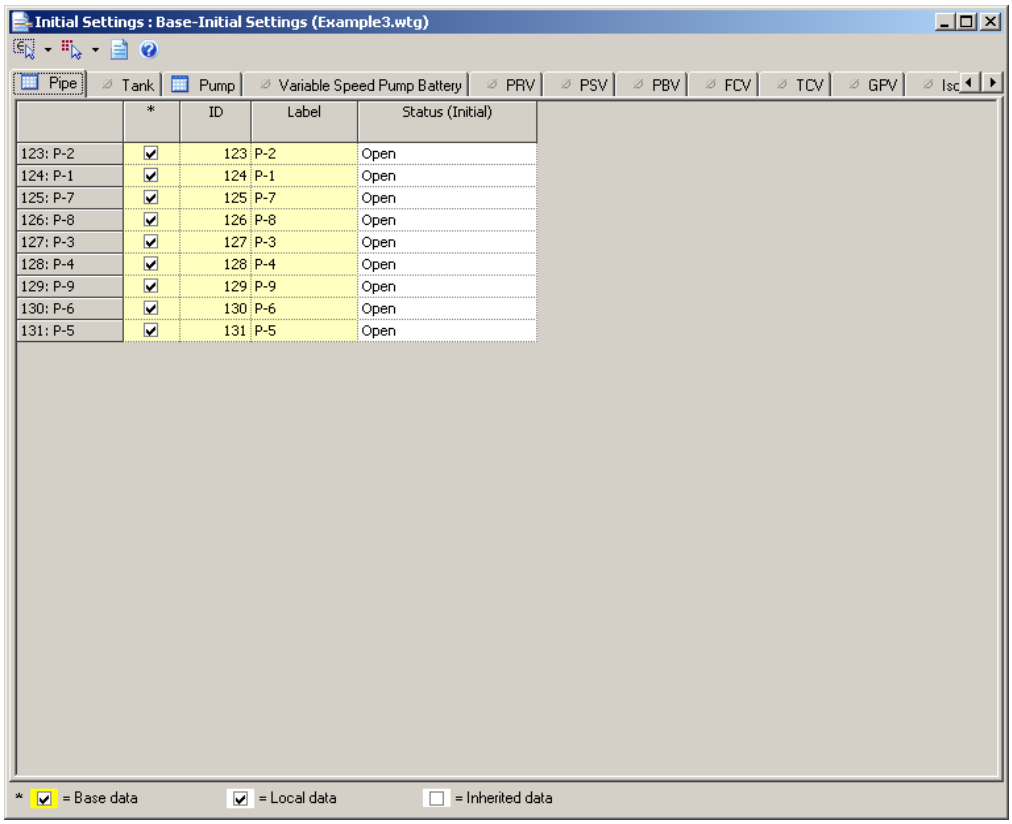
Demand Alternatives

The demand alternative allows you to model the response of the pipe network to different sets of demands, such as the current demand and the demand of your system ten years from now.



Initial Settings Alternative

The Initial Settings Alternative contains the data that set the conditions of certain types of network elements at the beginning of the simulation. For example, a pipe can start in an open or closed position and a pump can start in an on or off condition.



	*	ID	Label	Status (Initial)
123: P-2	☑	123	P-2	Open
124: P-1	☑	124	P-1	Open
125: P-7	☑	125	P-7	Open
126: P-8	☑	126	P-8	Open
127: P-3	☑	127	P-3	Open
128: P-4	☑	128	P-4	Open
129: P-9	☑	129	P-9	Open
130: P-6	☑	130	P-6	Open
131: P-5	☑	131	P-5	Open

The following buttons are available:

Selection Set



Opens a submenu containing the following options:

- **Create Selection Set**—Allows you to create a new selection set.
- **Add to Selection Set**—Adds all of the elements in the current tab of the alternative to a previously created selection set that you specify.
- **Remove from Selection Set**——Removes all of the elements in the current tab of the alternative from a previously created selection set that you specify.

Select in Drawing



Opens a submenu containing the following options:

- **Select in Drawing**—Selects the elements in the current tab of the alternative in the drawing pane.
- **Add to Current Selection**—Adds all of the elements in the current tab of the alternative to the group of elements that are currently selected in the Drawing Pane.
- **Remove from Current Selection**—Removes the elements in the current tab of the alternative from the group of elements that are currently selected in the Drawing Pane.
- **Select Within Current Selection**—Selects the element or elements that are both in the current tab of the alternative and are already selected in the Drawing Pane.

Report



Generates a report containing the data within the current alternative.

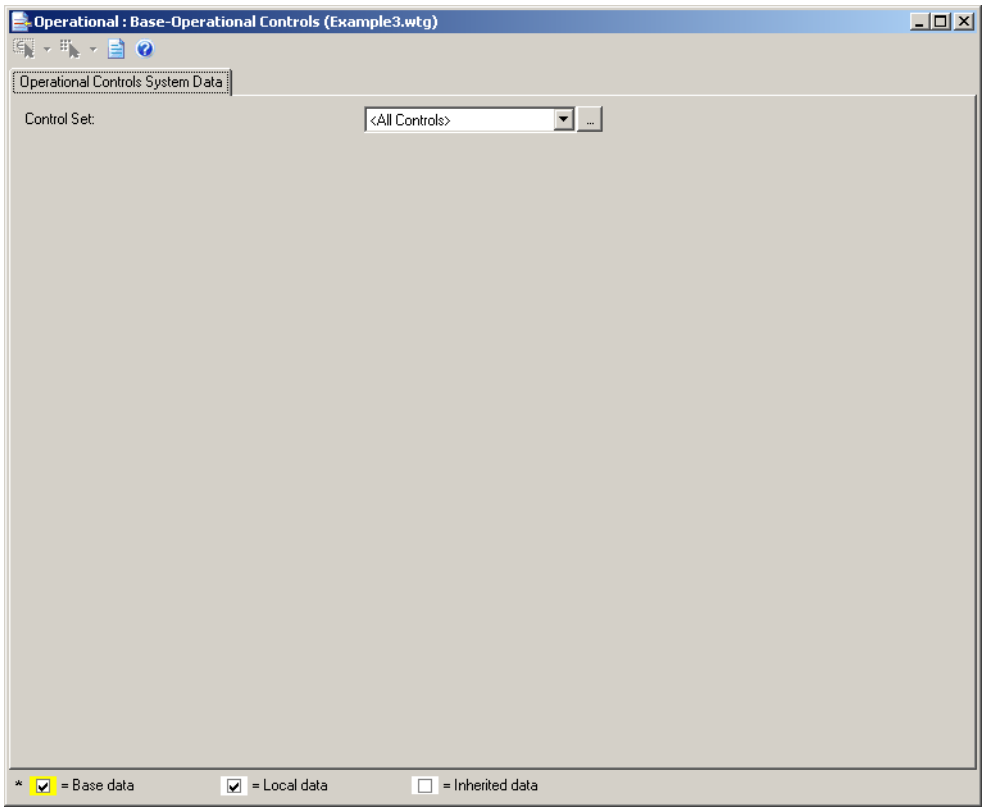
Help



Opens the online help.

Operational Alternatives

The Operational Alternative is where you can specify controls on pressure pipes, pumps, as well as valves.



The Operational Controls alternative allows you to create, modify and manage both logical controls and logical control sets.

The following buttons are available:

Selection Set



Opens a submenu containing the following options:

- **Create Selection Set**—Allows you to create a new selection set.
- **Add to Selection Set**—Adds all of the elements in the current tab of the alternative to a previously created selection set that you specify.
- **Remove from Selection Set**——Removes all of the elements in the current tab of the alternative from a previously created selection set that you specify.

Select in Drawing



Opens a submenu containing the following options:

- **Select in Drawing**—Selects the elements in the current tab of the alternative in the drawing pane.
- **Add to Current Selection**—Adds all of the elements in the current tab of the alternative to the group of elements that are currently selected in the Drawing Pane.
- **Remove from Current Selection**—Removes the elements in the current tab of the alternative from the group of elements that are currently selected in the Drawing Pane.
- **Select Within Current Selection**—Selects the element or elements that are both in the current tab of the alternative and are already selected in the Drawing Pane.

Report



Generates a report containing the data within the current alternative.

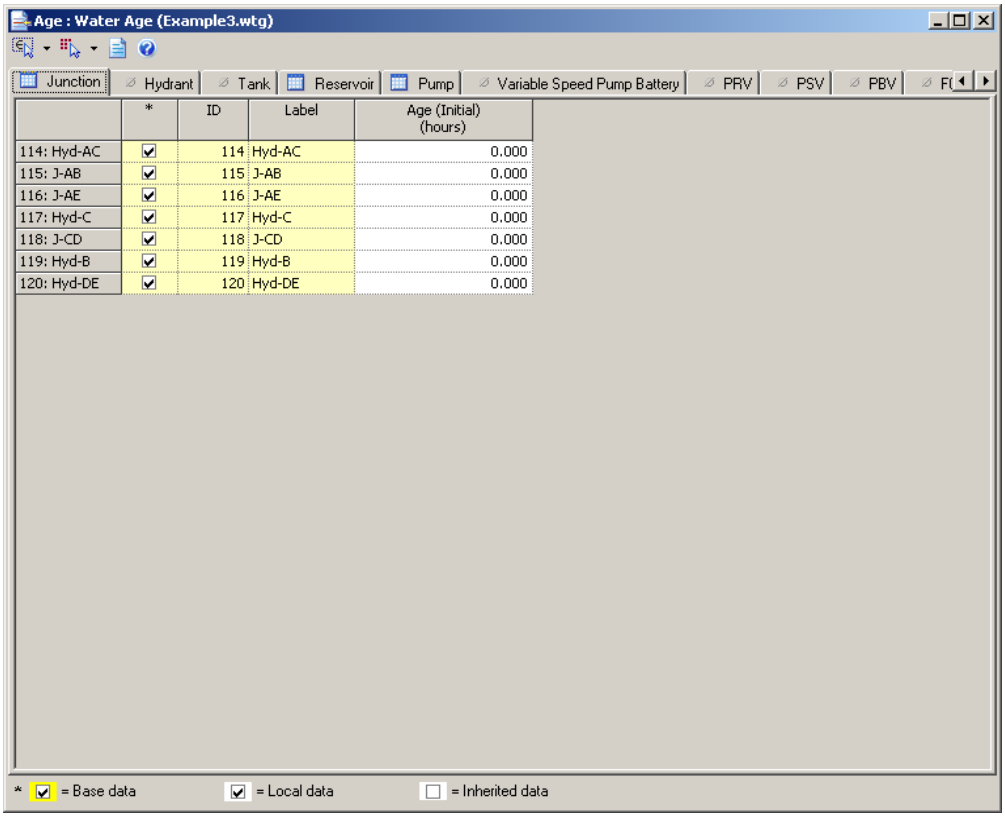
Help



Opens the online help.

Age Alternatives

The Age Alternative is used when performing a water quality analysis for modeling the age of the water through the pipe network. This alternative allows you to analyze different scenarios for varying water ages at the network nodes.



	*	ID	Label	Age (Initial) (hours)
114: Hyd-AC	☑	114	Hyd-AC	0.000
115: J-AB	☑	115	J-AB	0.000
116: J-AE	☑	116	J-AE	0.000
117: Hyd-C	☑	117	Hyd-C	0.000
118: J-CD	☑	118	J-CD	0.000
119: Hyd-B	☑	119	Hyd-B	0.000
120: Hyd-DE	☑	120	Hyd-DE	0.000

The following buttons are available:

Selection Set



Opens a submenu containing the following options:

- **Create Selection Set**—Allows you to create a new selection set.
- **Add to Selection Set**—Adds all of the elements in the current tab of the alternative to a previously created selection set that you specify.
- **Remove from Selection Set**——Removes all of the elements in the current tab of the alternative from a previously created selection set that you specify.

Select in Drawing



Opens a submenu containing the following options:

- **Select in Drawing**—Selects the elements in the current tab of the alternative in the drawing pane.
- **Add to Current Selection**—Adds all of the elements in the current tab of the alternative to the group of elements that are currently selected in the Drawing Pane.
- **Remove from Current Selection**—Removes the elements in the current tab of the alternative from the group of elements that are currently selected in the Drawing Pane.
- **Select Within Current Selection**—Selects the element or elements that are both in the current tab of the alternative and are already selected in the Drawing Pane.

Report



Generates a report containing the data within the current alternative.

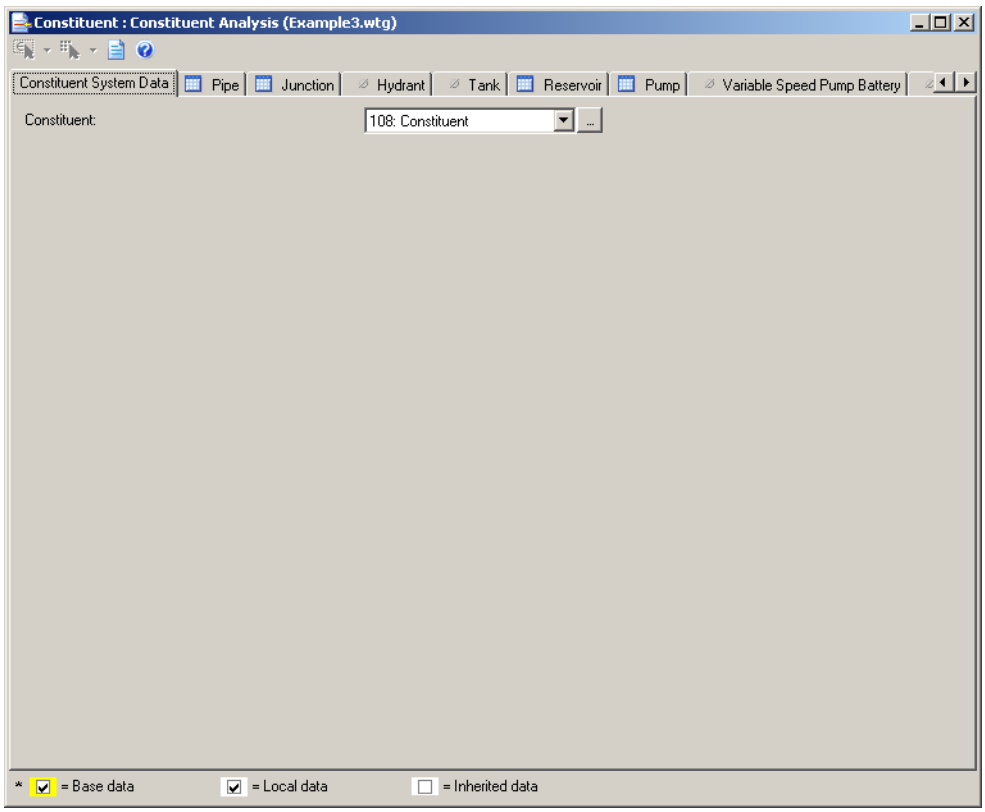
Help



Opens the online help.

Constituent Alternatives

The Constituent Alternative contains the water quality data used to model a constituent concentration throughout the network when performing a water quality analysis.



Selecting a constituent from the Constituent drop-down list provides default values for table entries. This software provides a user-editable library of constituents for maintaining these values, which may be accessed by clicking the **Ellipsis (...)** next to the Constituent menu.

The following attributes can be defined in the Constituent alternative:

- **Concentration (Initial)** - The concentration at the associated node at the start of an EPS run.
- **Concentration (Base)** - The concentration of the inflow into the system at the associated node. If there is no inflow, then this flow does not affect constituent concentration.
- **Mass Rate (Base)** - The mass per unit time injected at a node when the constituent source type is set to "Mass Rate".

- **Constituent Source Type** - there are four ways in which you can specify a constituent entering a system:
  - A concentration source fixes the concentration of any external inflow entering the network, such as flow from a reservoir or from a negative demand placed at a junction.
  - A mass booster source adds a fixed mass flow to that entering the node from other points in the network.
  - A flow paced booster source adds a fixed concentration to that resulting from the mixing of all inflow to the node from other points in the network.
  - A setpoint booster source fixes the concentration of any flow leaving the node (as long as the concentration resulting from all inflow to the node is below the setpoint).
- **Pattern (Constituent)** - The name of the constituent pattern created under Component > Patterns that the constituent will follow. The default value is "Fixed".
- **Is Constituent Source?** - This attribute should be set to True if the element is to be a source in the scenario. Setting it to False will turn off the source even if there are values defined for Concentration (Base) or Mass Rate (Base).

The following buttons are available:

Selection Set



Opens a submenu containing the following options:

- **Create Selection Set**—Allows you to create a new selection set.
- **Add to Selection Set**—Adds all of the elements in the current tab of the alternative to a previously created selection set that you specify.
- **Remove from Selection Set**——Removes all of the elements in the current tab of the alternative from a previously created selection set that you specify.

Select in Drawing



Opens a submenu containing the following options:

- **Select in Drawing**—Selects the elements in the current tab of the alternative in the drawing pane.
- **Add to Current Selection**—Adds all of the elements in the current tab of the alternative to the group of elements that are currently selected in the Drawing Pane.
- **Remove from Current Selection**—Removes the elements in the current tab of the alternative from the group of elements that are currently selected in the Drawing Pane.
- **Select Within Current Selection**—Selects the element or elements that are both in the current tab of the alternative and are already selected in the Drawing Pane.

Report



Generates a report containing the data within the current alternative.

Help



Opens the online help.

Constituents Manager Dialog Box

The Constituents manager allows you to:

- Create new Constituents for use in Water Quality Analysis
- Define properties for newly created constituents
- Edit properties for existing constituents.

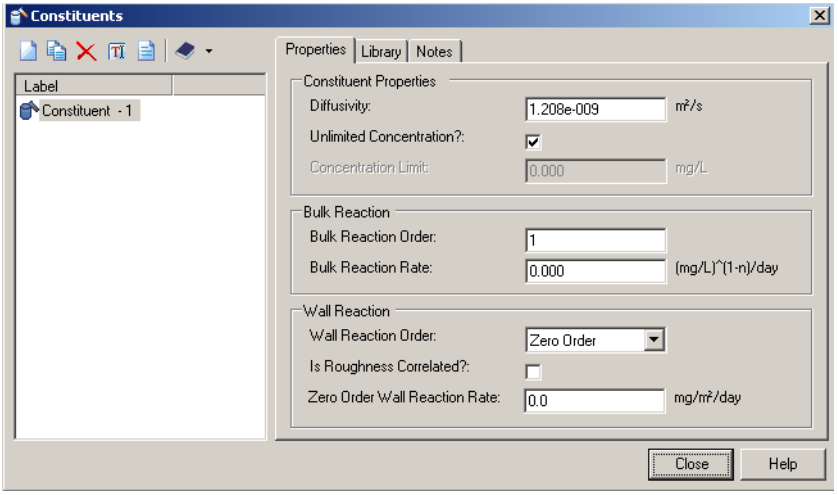
To open the Constituents manager

Choose Components > Constituents

or

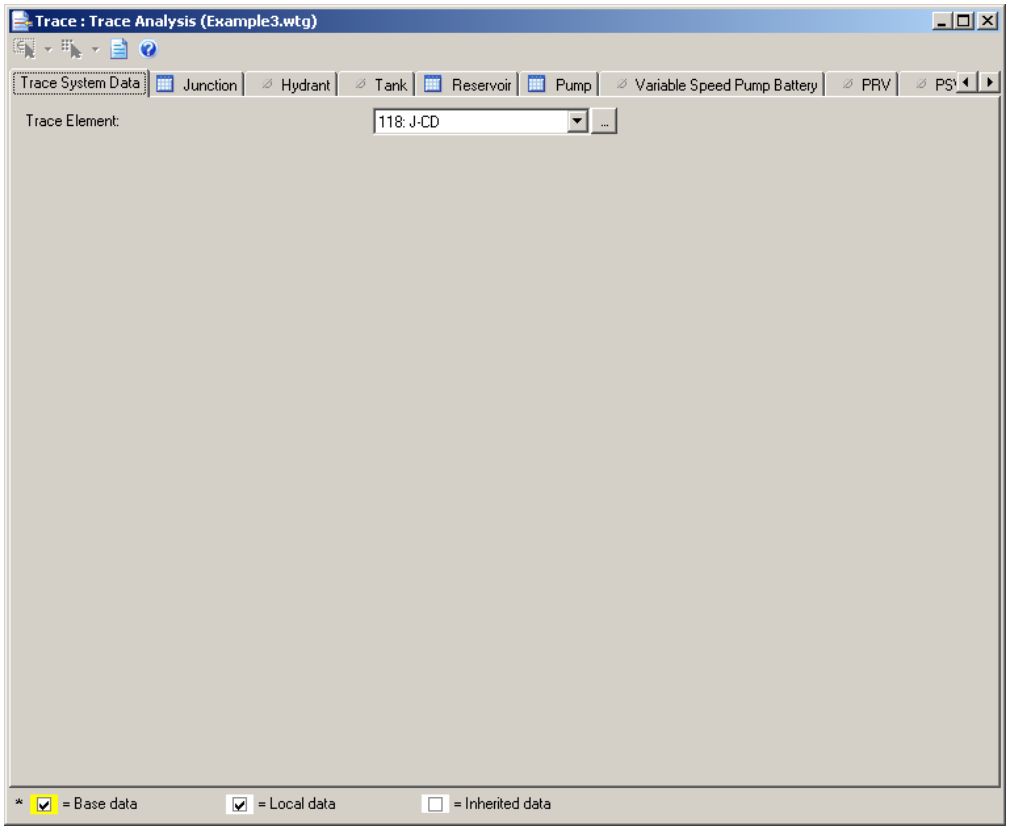
Click the Constituents icon  from the Components toolbar.

The Constituents manager opens.



Trace Alternative

The Trace Alternative is used when performing a water quality analysis to determine the percentage of water at each node coming from a specified node. The Trace Alternative data includes a Trace Node, which is the node from which all tracing is computed.



The following buttons are available:

Selection Set



Opens a submenu containing the following options:

- **Create Selection Set**—Allows you to create a new selection set.
- **Add to Selection Set**—Adds all of the elements in the current tab of the alternative to a previously created selection set that you specify.
- **Remove from Selection Set**——Removes all of the elements in the current tab of the alternative from a previously created selection set that you specify.

Select in Drawing



Opens a submenu containing the following options:

- **Select in Drawing**—Selects the elements in the current tab of the alternative in the drawing pane.
- **Add to Current Selection**—Adds all of the elements in the current tab of the alternative to the group of elements that are currently selected in the Drawing Pane.
- **Remove from Current Selection**—Removes the elements in the current tab of the alternative from the group of elements that are currently selected in the Drawing Pane.
- **Select Within Current Selection**—Selects the element or elements that are both in the current tab of the alternative and are already selected in the Drawing Pane.

Report



Generates a report containing the data within the current alternative.

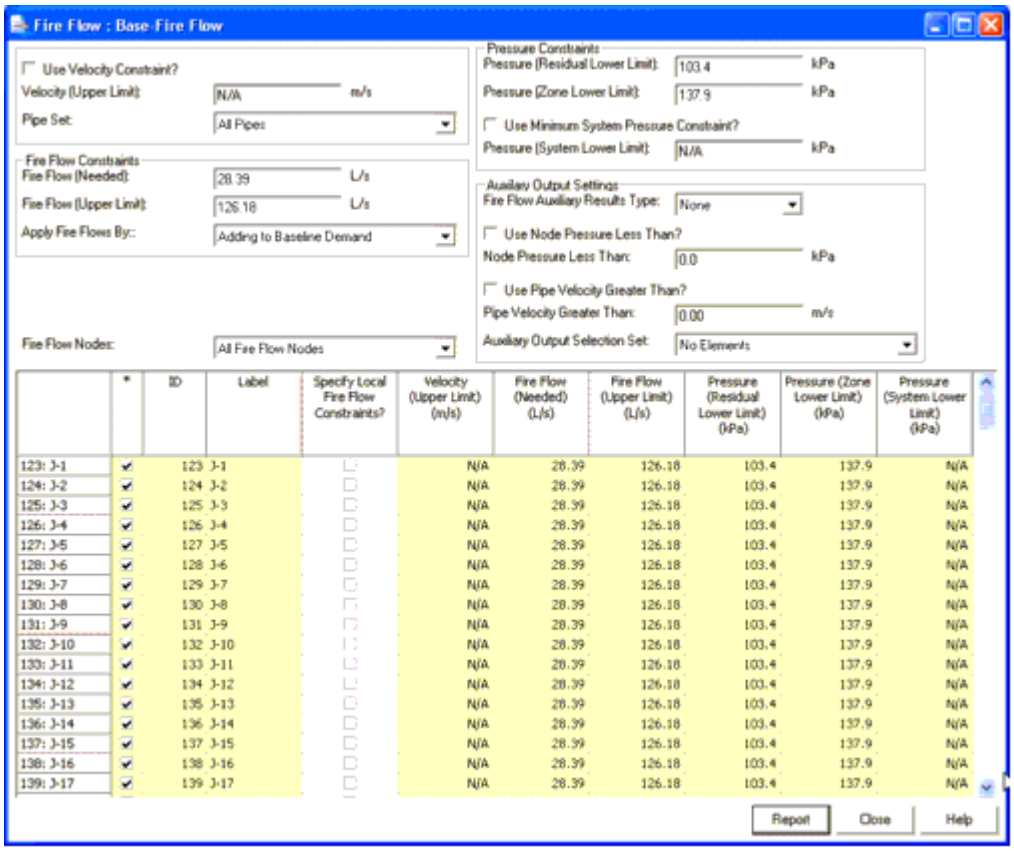
Help



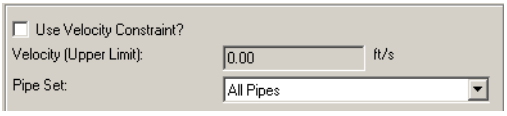
Opens the online help.

Fire Flow Alternative

The Fire Flow Alternative contains the input data required to perform a fire flow analysis. This data includes the set of junction nodes for which fire flow results are needed, the set of default values for all junctions included in the fire flow set, and a record for each junction node in the fire flow set.

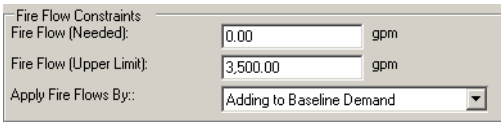


The Fire Flow Alternative window is divided into sections which contain different fields to create the fire flow.



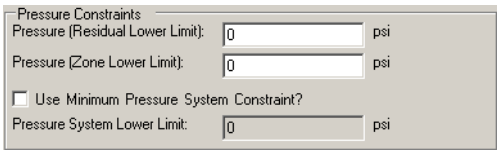
Use Velocity Constraint?	If set to true, then a velocity constraint can be specified for the node.

Velocity (Upper Limit)	Specifies the maximum velocity allowed in the associated set of pipes when drawing out fire flow from the selected node.
Pipe Set	The set of pipes associated with the current node where velocities are tested during a fire flow analysis.

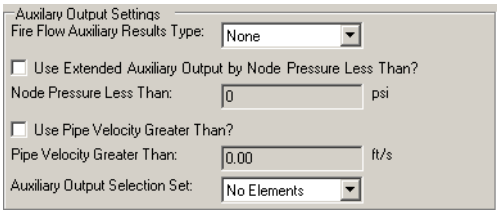


Fire Flow (Needed)	Flow rate required at the junction to meet fire flow demands. This value will be added to the junction's baseline demand or it will replace the junction's baseline demand, depending on the default setting for applying fire flows.
Fire Flow (Upper Limit)	Maximum allowable fire flow that can occur at a withdrawal location. This value will prevent the software from computing unrealistically high fire flows at locations such as primary system mains, which have large diameters and high service pressures. This value will be added to the junction's baseline demand or it will replace the junction's baseline demand, depending on the default setting for applying fire flows.
Apply Fire Flows By	There are two methods for applying fire flow demands. The fire flow demand can be added to the junction's baseline demand, or it can completely replace the junction's baseline demand. The junction's baseline demand is defined by the Demand Alternative selected for use in the Scenario along with the fire flow alternative.

Fire Flow Nodes	 A selection set that defines the fire flow nodes to be subject to a fire flow analysis. The selection set must be a concrete selection set (not query based) and must include the junctions and hydrants that need to be analyzed. Any non-junction and hydrant elements in the selection set are ignored.



Pressure (Residual Lower Limit)	Minimum residual pressure to occur at the junction node. The program determines the amount of fire flow available such that the residual pressure at the junction node does not fall below this target pressure.
Pressure (Zone Lower Limit)	Minimum pressure to occur at all junction nodes within a zone. The model determines the available fire flow such that the minimum zone pressures do not fall below this target pressure. Each junction has a zone associated with it, which can be located in the junction's input data. If you do not want a junction node to be analyzed as part of another junction node's fire flow analysis, move it to another zone.
Use Minimum System Pressure Constraint?	Check whether a minimum pressure is to be maintained throughout the entire pipe system.
Pressure System Lower Limit	Minimum pressure allowed at any junction in the entire system as a result of the fire flow withdrawal. If the pressure at a node anywhere in the system falls below this constraint while withdrawing fire flow, fire flow will not be satisfied.



Fire Flow Auxiliary Results Type	This setting controls whether the fire flow analysis will save "auxiliary results" (a snap shot result set of the fire flow analysis hydraulic conditions) for no fire flow nodes, just the failing fire flow nodes, if any, or all fire flow nodes. For every fire flow node that attracts auxiliary results a separate result set (file) is created. When enabling this setting be conscious of the number of fire flow nodes in your system and the potential disk space requirement. Enabling this option also will slow down the fire flow analysis due to the need to create the additional results sets. Note: The base result set includes hydraulic results for the actual fire flow node and also for the pipes that connect to the fire flow node. The results stored are for the hydraulic conditions that are experienced during the actual fire flow analysis (i.e., under fire flow loading). No other hydraulic results are stored unless the auxiliary result set is "extended" by other options listed below.
Use Extended Auxiliary Output by Node Pressure Less Than?	Defines whether to include in the stored fire flow auxiliary results, results for nodes that fall below a defined pressure value. Such nodes might indicate low pressure problems under the fire flow conditions.
Node Pressure Less Than?	Specifies the number.
Use Pipe Velocity Greater Than?	Defines whether to include in the stored fire flow auxiliary results, results for pipes that exceed a defined velocity value. Such pipes might indicate bottle necks in the system under the fire flow conditions.
Pipe Velocity Greater Than?	Specifies the number.
Auxiliary Output Selection Set	This selection set is used to force any particular elements of interest (e.g., pumps, tanks) into a fire flow node's auxiliary result set, irrespective of the hydraulic result at that location. Said another way this option defines which elements to always include in the fire flow auxiliary result set for each fire flow node that has auxiliary results.

Fire Flow System Data

Each fire flow alternative has a set of default parameters that are applied to each junction in the fire flow set. When a default value is modified, you will be prompted to decide if the junction records that have been modified from the default should be updated to reflect the new default value.

	*	ID	Label	Specify Local Fire Flow Constraints?	Velocity (Upper Limit) (ft/s)	Fire Flow (Needed) (gpm)	Fire Flow (Upper Limit) (gpm)	Pressure (Residual Lower Limit) (psi)	Pressure (Zone Lower Limit) (psi)	Pressure (System Low Limit) (psi)
J-1	☑	20	J-1	☐	0.00	0.00	3,500.00	0	0	
J-2	☑	22	J-2	☐	0.00	0.00	3,500.00	0	0	
J-3	☑	24	J-3	☐	0.00	0.00	3,500.00	0	0	
J-4	☑	26	J-4	☐	0.00	0.00	3,500.00	0	0	
J-5	☑	28	J-5	☐	0.00	0.00	3,500.00	0	0	
J-6	☑	30	J-6	☐	0.00	0.00	3,500.00	0	0	
J-7	☑	32	J-7	☐	0.00	0.00	3,500.00	0	0	
J-8	☑	34	J-8	☐	0.00	0.00	3,500.00	0	0	
J-9	☑	36	J-9	☐	0.00	0.00	3,500.00	0	0	
J-10	☑	38	J-10	☐	0.00	0.00	3,500.00	0	0	

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Specify Local Fire Flow Constraints?	Select this check box to allow input different from the global values. When you select this check box, the fields in that row turn from yellow (read-only) to white (editable).
Velocity (Upper Limit)	Specify the maximum velocity allowed in the associated set of pipes when drawing out fire flow from the selected node.
Fire Flow (Needed)	Flow rate required at a fire flow junction to satisfy demands.
Fire Flow Upper Limit	Maximum allowable fire flow that can occur at a withdrawal location. It will prevent the software from computing unrealistically high fire flows at locations such as primary system mains, which have large diameters and high service pressures.

Column	Description
Pressure (Residual Lower Limit)	Minimum residual pressure to occur at the junction node. The program determines the amount of fire flow available such that the residual pressure at the junction node does not fall below this target pressure.
Pressure (Zone Lower Limit)	Minimum pressure to occur at all junction nodes within a zone. The model determines the available fire flow such that the minimum zone pressures do not fall below this target pressure. Each junction has a zone associated with it, which can be located in the junction's input data. If you do not want a junction node to be analyzed as part of another junction node's fire flow analysis, move it to another zone.
Pressure (System Lower Limit)	Minimum pressure to occur at all junction nodes within the system.

Filter Dialog Box

The Filter dialog box lets you specify your filtering criteria. Each filter criterion is made up of three items:

- **Column**—The attribute to filter.
- **Operator**—The operator to use when comparing the filter value against the data in the specific column (operators include: =, >, >=, <, <=, < >).
- **Value**—The comparison value.

Any number of criteria can be added to a filter. Multiple filter criteria are implicitly joined with a logical AND statement. When multiple filter criteria are defined, only rows that meet all of the specified criteria will be displayed. A filter will remain active for the associated table until the filter is reset.

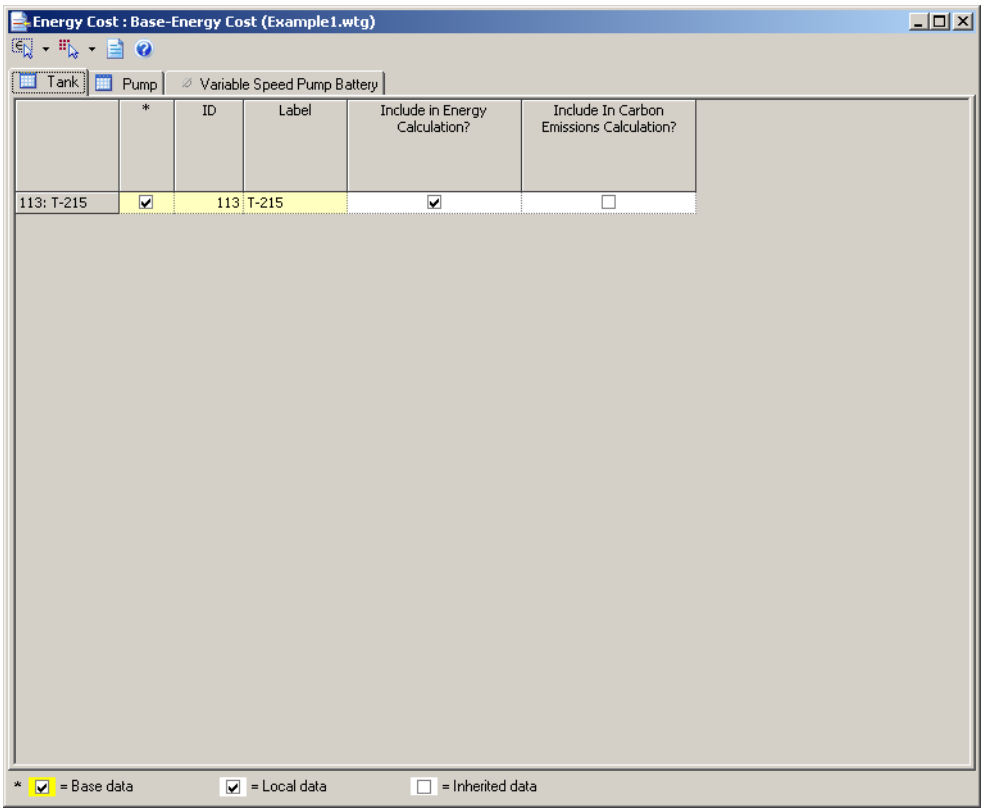
The status pane at the bottom of the Table window always shows the number of rows displayed and the total number of rows available (e.g., 10 of 20 elements displayed). When a filter is active, this message will be highlighted.

Energy Cost Alternative

The Energy Cost Alternative Manager is where the user can select the elements to be included in the energy cost analysis. The energy cost alternative is used when it is necessary to perform multiple energy analyses with alternative pricing or for pumping stations in different parts of the system.

All pumps, tanks, and variable speed pump batteries are included in the analysis by default. However, you can override this by unchecking the box labeled Include in Energy Calculation?

You can also set which energy price functions to use with each element. This function can also be done within the Energy Cost manager..



The following buttons are available:

Selection Set



Opens a submenu containing the following options:

- **Create Selection Set**—Allows you to create a new selection set.
- **Add to Selection Set**—Adds all of the elements in the current tab of the alternative to a previously created selection set that you specify.
- **Remove from Selection Set**——Removes all of the elements in the current tab of the alternative from a previously created selection set that you specify.

Select in Drawing



Opens a submenu containing the following options:

- **Select in Drawing**—Selects the elements in the current tab of the alternative in the drawing pane.
- **Add to Current Selection**—Adds all of the elements in the current tab of the alternative to the group of elements that are currently selected in the Drawing Pane.
- **Remove from Current Selection**—Removes the elements in the current tab of the alternative from the group of elements that are currently selected in the Drawing Pane.
- **Select Within Current Selection**—Selects the element or elements that are both in the current tab of the alternative and are already selected in the Drawing Pane.

Report



Generates a report containing the data within the current alternative.

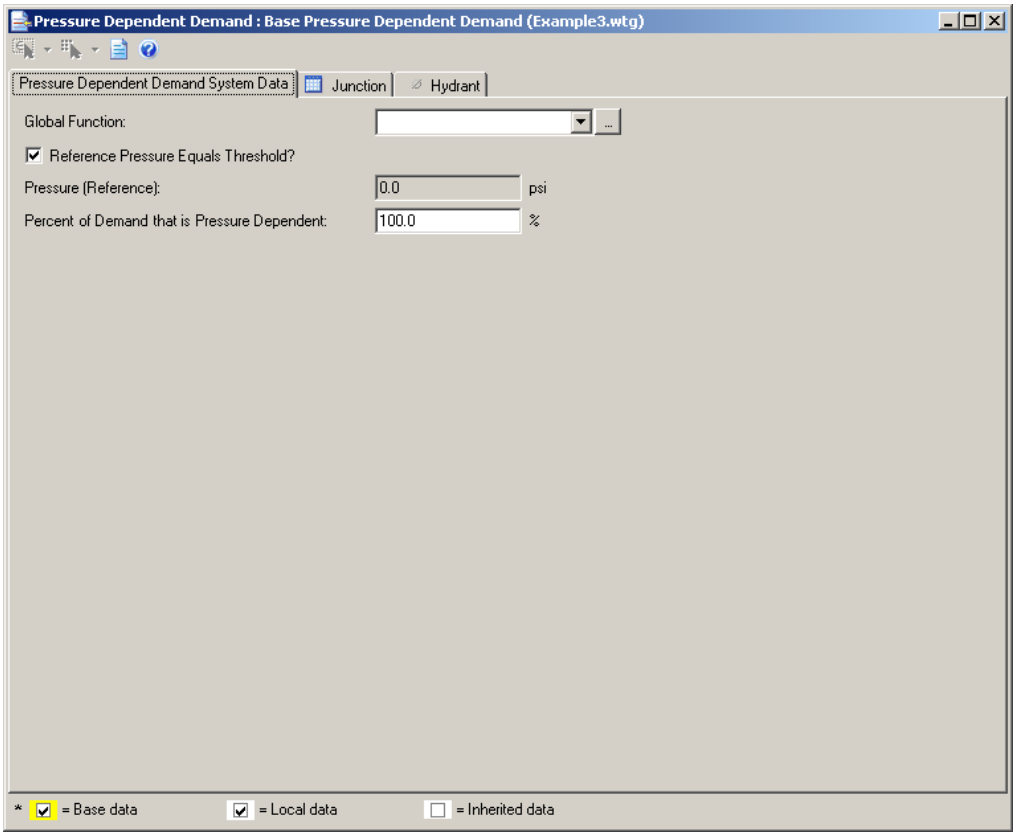
Help



Opens the online help.

Pressure Dependent Demand Alternative

The Pressure Dependent Demand Alternative allows a pressure dependent demand function to be used.



The following buttons are available:

Selection Set



Opens a submenu containing the following options:

- **Create Selection Set**—Allows you to create a new selection set.
- **Add to Selection Set**—Adds all of the elements in the current tab of the alternative to a previously created selection set that you specify.
- **Remove from Selection Set**——Removes all of the elements in the current tab of the alternative from a previously created selection set that you specify.

Select in Drawing



Opens a submenu containing the following options:

- **Select in Drawing**—Selects the elements in the current tab of the alternative in the drawing pane.
- **Add to Current Selection**—Adds all of the elements in the current tab of the alternative to the group of elements that are currently selected in the Drawing Pane.
- **Remove from Current Selection**—Removes the elements in the current tab of the alternative from the group of elements that are currently selected in the Drawing Pane.
- **Select Within Current Selection**—Selects the element or elements that are both in the current tab of the alternative and are already selected in the Drawing Pane.

Report



Generates a report containing the data within the current alternative.

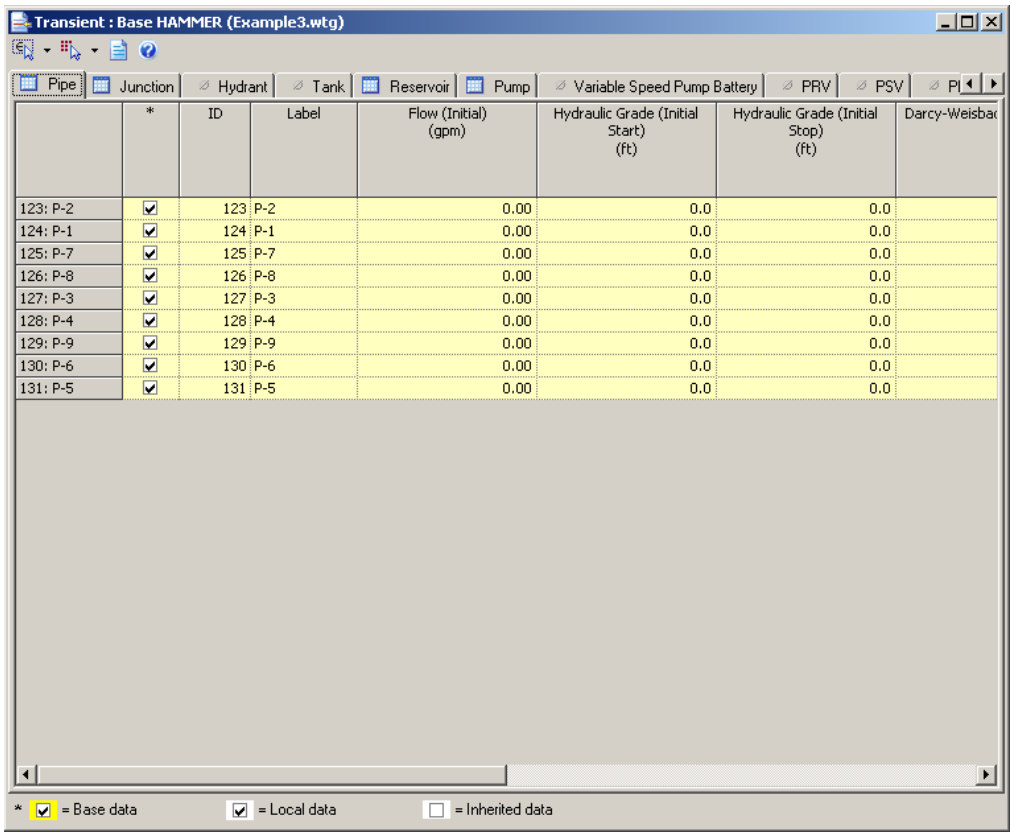
Help



Opens the online help.

Transient Alternative

The Transient Alternative allows you to edit and view data that is used for WaterCAD V8*i* transient calculations. There is a tab for each element type, each containing the WaterCAD V8*i* specific attributes for that element type.



	*	ID	Label	Flow (Initial) (gpm)	Hydraulic Grade (Initial Start) (ft)	Hydraulic Grade (Initial Stop) (ft)	Darcy-Weisbac
123: P-2	☑	123	P-2	0.00	0.0	0.0	
124: P-1	☑	124	P-1	0.00	0.0	0.0	
125: P-7	☑	125	P-7	0.00	0.0	0.0	
126: P-8	☑	126	P-8	0.00	0.0	0.0	
127: P-3	☑	127	P-3	0.00	0.0	0.0	
128: P-4	☑	128	P-4	0.00	0.0	0.0	
129: P-9	☑	129	P-9	0.00	0.0	0.0	
130: P-6	☑	130	P-6	0.00	0.0	0.0	
131: P-5	☑	131	P-5	0.00	0.0	0.0	

The following buttons are available:

Selection Set



Opens a submenu containing the following options:

- **Create Selection Set**—Allows you to create a new selection set.
- **Add to Selection Set**—Adds all of the elements in the current tab of the alternative to a previously created selection set that you specify.
- **Remove from Selection Set**——Removes all of the elements in the current tab of the alternative from a previously created selection set that you specify.

Select in Drawing



Opens a submenu containing the following options:

- **Select in Drawing**—Selects the elements in the current tab of the alternative in the drawing pane.
- **Add to Current Selection**—Adds all of the elements in the current tab of the alternative to the group of elements that are currently selected in the Drawing Pane.
- **Remove from Current Selection**—Removes the elements in the current tab of the alternative from the group of elements that are currently selected in the Drawing Pane.
- **Select Within Current Selection**—Selects the element or elements that are both in the current tab of the alternative and are already selected in the Drawing Pane.

Report



Generates a report containing the data within the current alternative.

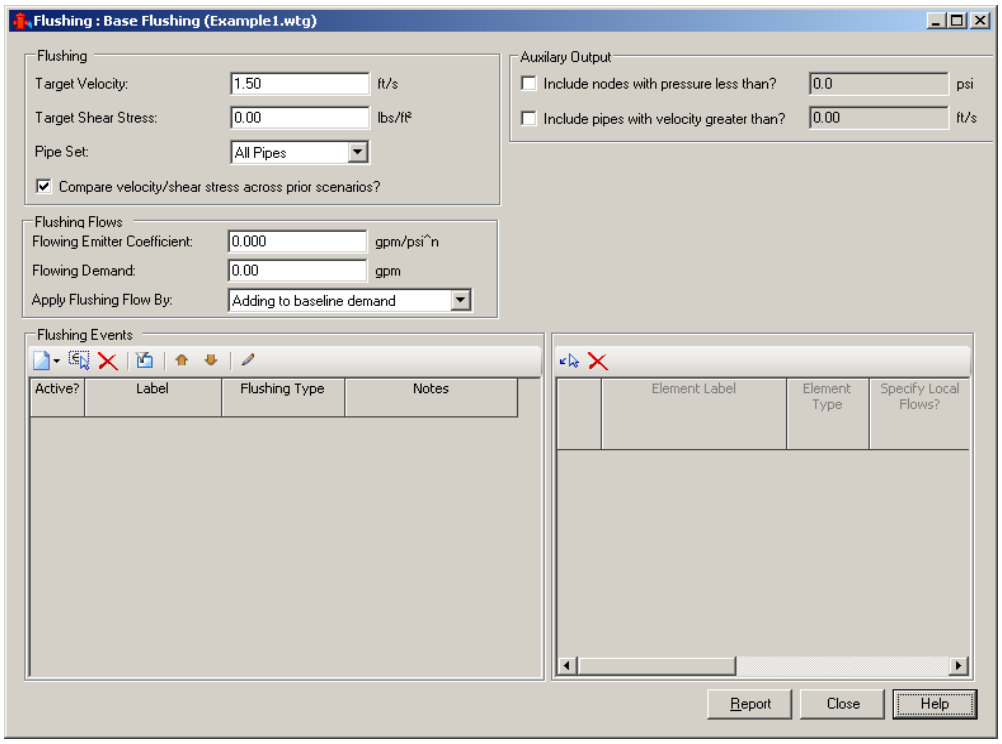
Help



Opens the online help.

Flushing Alternative

The flushing alternative allows you to define flushing events and the conditions of a flushing analysis.



The alternative consists of the following controls:

Target Velocity: Pipes with a calculated velocity exceeding this value will be considered flushed.

Target Shear Stress: Pipes with a calculated shear stress exceeding this value will be considered flushed.

Pipe Set: Set of pipes which will be evaluated with regard to whether they reached target velocity (Default is All Pipes although the user can specify a previously created Selection Set in the drop down menu.)

Compare velocity/shear stress across prior scenarios?: If checked, each run will set all the Velocity (Maximum Flushing) results to 0 ft/s and Shear Stress (Maximum Flushing) to 0 lbs/ft^2 at the start of the run (Scenario). If unchecked, it will base the Velocity (Maximum Flushing) and Shear Stress (Maximum Flushing) on all of the

existing scenarios for which results are available since the last time a run was made with the box checked. If the user is evaluating all pipes at once, it is best to check this box. If the user is building up a flushing program through a number of scenarios using different areas, then it is best to uncheck the box.

Flowing Emitter Coefficient: Emitter coefficient to be used globally for hydrants. This value can be overridden for individual nodes on the next tab.

Flowing Demand: Instead of specifying an emitter coefficient, the user can directly specify the flow in flow units. The user should generally not specify non-zero values for both emitter coefficient and flowing demand as this can double count the hydrant flow.

Apply Flushing Flow By: Describes whether the flushing discharge is added to or replaces the normal demand. The default value is Adding to Baseline demand.

Report on Minimum Pressure?: If box is checked, flushing will not allow the pressure to drop below a predefined value specified by the user. Caution: there may be some nodes (e.g. suction side of pump) than have habitual low pressure and will prevent flushing from working).

Include nodes with pressure less than?: If checked, flushing runs will save the nodes that dropped below some minimum pressure during any flush. These can be reviewed as a check to see if flushing will adversely affect customer pressure. Unlike the constraint listed above, flushing will still occur but low pressures will be noted.

Include pipes with velocity greater than?: If checked, for any event velocity data on which pipes exceeded some velocity are saved, This need not be the same velocity as the target velocity specified above. All pipes that are in the “Pipe Set” are automatically included in the auxiliary results regardless of their velocity."

The right side of the dialog contains a list of flushing events that have been specified in the Conventional or Unidirectional tabs. You can exclude an event from the alternative when during a run by unchecking the "Is Active?" box next to that event.

The Conventional and Unidirectional tabs allow you to define flushing events as follows:

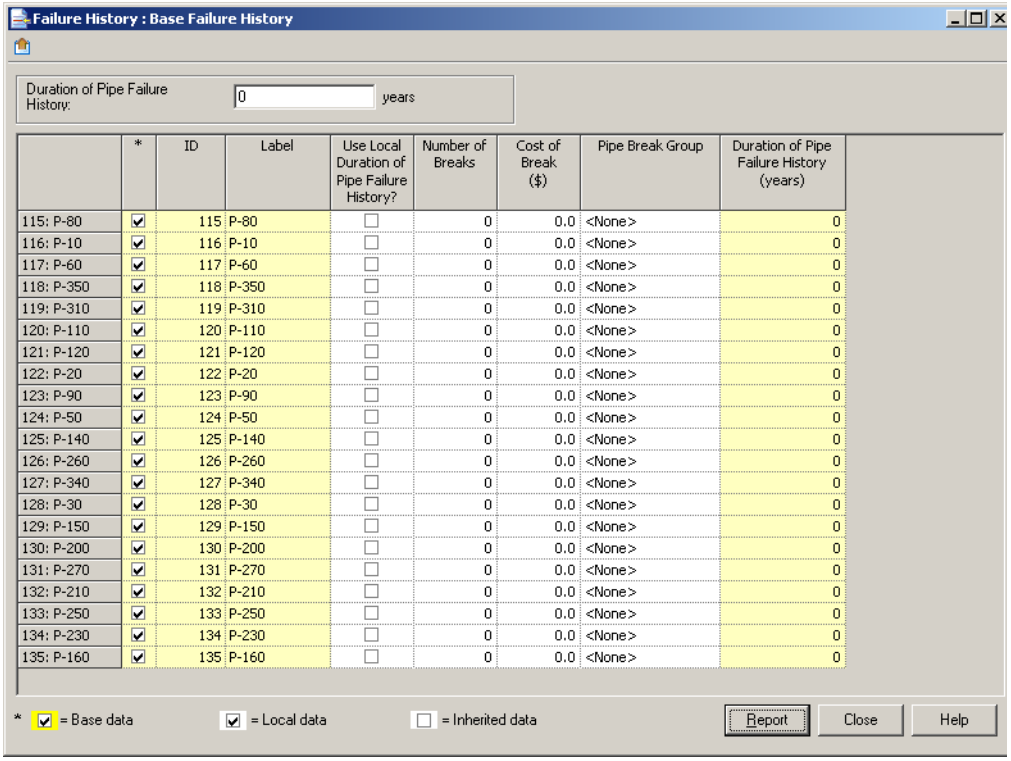
Conventional flushing events are defined in the Conventional tab of the flushing alternative. The user can add a flushing event by clicking the New button (leftmost button) on top of the flushing tab. This will create a new flushing event that the user can label. By clicking on the ellipse which appears when the "Element ID" is selected, the user can select the element (junction node or hydrant) to be flowed. If the user also checks the box under the "Is Local?" column, the user can override the global values for Emitter Coefficient or Hydrant Flow.

Unidirectional flushing events are more complex and therefore additional information is required to describe the event. To create an event, the user selects the

new button (Leftmost button on top row of the Unidirectional dialog). From this button, the user can either add a flushing event or add elements to an existing flushing event.

Failure History Alternative

The Failure History alternative allows you to edit data associated with pipe break analysis.



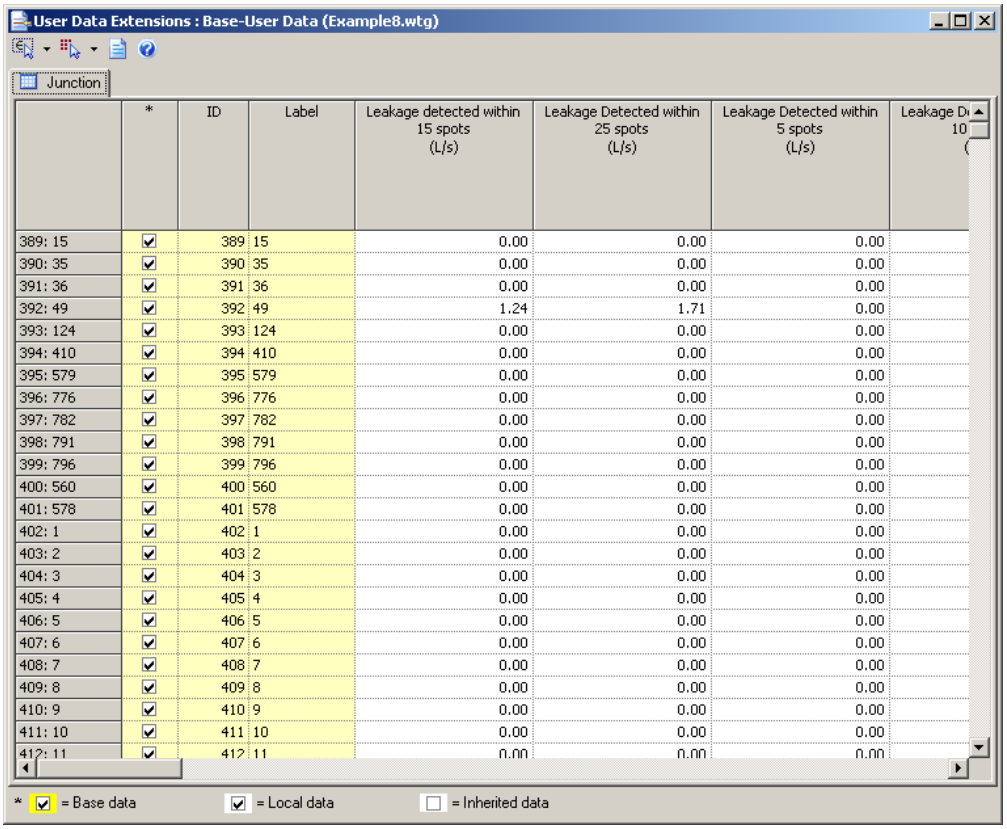
	*	ID	Label	Use Local Duration of Pipe Failure History?	Number of Breaks	Cost of Break ($)	Pipe Break Group	Duration of Pipe Failure History (years)
115: P-80	☑	115	P-80	☐	0	0.0	<None>	0
116: P-10	☑	116	P-10	☐	0	0.0	<None>	0
117: P-60	☑	117	P-60	☐	0	0.0	<None>	0
118: P-350	☑	118	P-350	☐	0	0.0	<None>	0
119: P-310	☑	119	P-310	☐	0	0.0	<None>	0
120: P-110	☑	120	P-110	☐	0	0.0	<None>	0
121: P-120	☑	121	P-120	☐	0	0.0	<None>	0
122: P-20	☑	122	P-20	☐	0	0.0	<None>	0
123: P-90	☑	123	P-90	☐	0	0.0	<None>	0
124: P-50	☑	124	P-50	☐	0	0.0	<None>	0
125: P-140	☑	125	P-140	☐	0	0.0	<None>	0
126: P-260	☑	126	P-260	☐	0	0.0	<None>	0
127: P-340	☑	127	P-340	☐	0	0.0	<None>	0
128: P-30	☑	128	P-30	☐	0	0.0	<None>	0
129: P-150	☑	129	P-150	☐	0	0.0	<None>	0
130: P-200	☑	130	P-200	☐	0	0.0	<None>	0
131: P-270	☑	131	P-270	☐	0	0.0	<None>	0
132: P-210	☑	132	P-210	☐	0	0.0	<None>	0
133: P-250	☑	133	P-250	☐	0	0.0	<None>	0
134: P-230	☑	134	P-230	☐	0	0.0	<None>	0
135: P-160	☑	135	P-160	☐	0	0.0	<None>	0

Failure History Alternative Properties

Column	Description
ID	Displays the unique identifier for each element in the alternative.
Label	Displays the label for each element in the alternative.
Use Local Duration of Pipe Failure History?	When this box is checked the value entered in the corresponding **Duration of Pipe Failure History** column will override the duration set in the **Length of Pipe Break History** field.
Number of Breaks	The number of pipe breaks in the duration of the pipe's failure history.
Cost of Break	The cost of each break in the duration of the pipe's failure history.
Pipe Break Group	The pipe break group to which the associated pipe belongs.
Duration of Pipe Failure History	The local duration of the pipe failure history. This column becomes editable for pipes that have the Use Local Duration of Pipe Failure History? box checked.

User Data Extensions

The User Data Alternative allows you to edit the data defined in the User Data Extension command for each of the network element types. The User Data Alternative editor contains a tab for each type of network element and is project specific.



	*	ID	Label	Leakage detected within 15 spots (L/s)	Leakage Detected within 25 spots (L/s)	Leakage Detected within 5 spots (L/s)	Leakage D 10
389: 15	☑	389	15	0.00	0.00	0.00	
390: 35	☑	390	35	0.00	0.00	0.00	
391: 36	☑	391	36	0.00	0.00	0.00	
392: 49	☑	392	49	1.24	1.71	0.00	
393: 124	☑	393	124	0.00	0.00	0.00	
394: 410	☑	394	410	0.00	0.00	0.00	
395: 579	☑	395	579	0.00	0.00	0.00	
396: 776	☑	396	776	0.00	0.00	0.00	
397: 782	☑	397	782	0.00	0.00	0.00	
398: 791	☑	398	791	0.00	0.00	0.00	
399: 796	☑	399	796	0.00	0.00	0.00	
400: 560	☑	400	560	0.00	0.00	0.00	
401: 578	☑	401	578	0.00	0.00	0.00	
402: 1	☑	402	1	0.00	0.00	0.00	
403: 2	☑	403	2	0.00	0.00	0.00	
404: 3	☑	404	3	0.00	0.00	0.00	
405: 4	☑	405	4	0.00	0.00	0.00	
406: 5	☑	406	5	0.00	0.00	0.00	
407: 6	☑	407	6	0.00	0.00	0.00	
408: 7	☑	408	7	0.00	0.00	0.00	
409: 8	☑	409	8	0.00	0.00	0.00	
410: 9	☑	410	9	0.00	0.00	0.00	
411: 10	☑	411	10	0.00	0.00	0.00	
412: 11	☑	412	11	0.00	0.00	0.00	

The following buttons are available:

Selection Set



Opens a submenu containing the following options:

- **Create Selection Set**—Allows you to create a new selection set.
- **Add to Selection Set**—Adds all of the elements in the current tab of the alternative to a previously created selection set that you specify.
- **Remove from Selection Set**——Removes all of the elements in the current tab of the alternative from a previously created selection set that you specify.

Select in Drawing



Opens a submenu containing the following options:

- **Select in Drawing**—Selects the elements in the current tab of the alternative in the drawing pane.
- **Add to Current Selection**—Adds all of the elements in the current tab of the alternative to the group of elements that are currently selected in the Drawing Pane.
- **Remove from Current Selection**—Removes the elements in the current tab of the alternative from the group of elements that are currently selected in the Drawing Pane.
- **Select Within Current Selection**—Selects the element or elements that are both in the current tab of the alternative and are already selected in the Drawing Pane.

Report



Generates a report containing the data within the current alternative.

Help



Opens the online help.

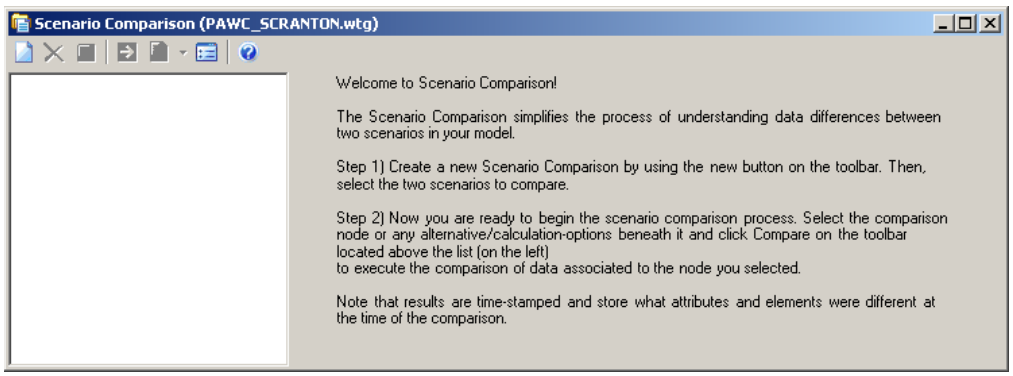
Scenario Comparison

The scenario comparison tool enables you to compare input values between any two scenarios to identify differences quickly. While WaterGEMS/CAD/ users have previously had the capability to open a child scenario or alternative and compare it with its parent, this tool greatly extends that capability in that you can compare any two scenarios or alternatives (not necessarily parent-child) and very easily detect differences.

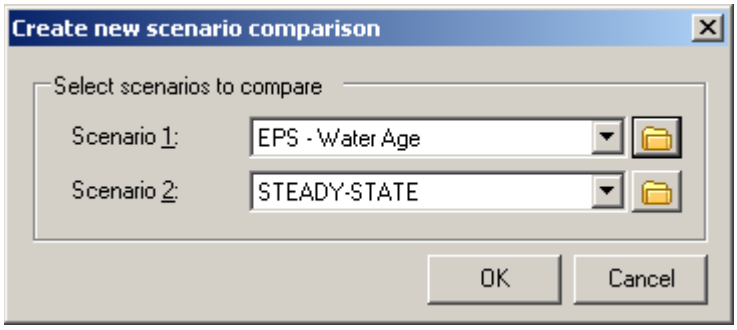
The scenario comparison tool can be started by picking Tools > Scenario Comparison

or by selecting the Scenario Comparison button from the toolbar  . If the button is not visible, it can be added using the "Add or Remove Buttons" drop down from the Tools toolbar (see Customizing WaterCAD V8i Toolbars and Buttons).

On first opening the scenario comparison tool, the dialog below opens which gives an overview of the steps involved in using the tool. Pick the New button (leftmost).

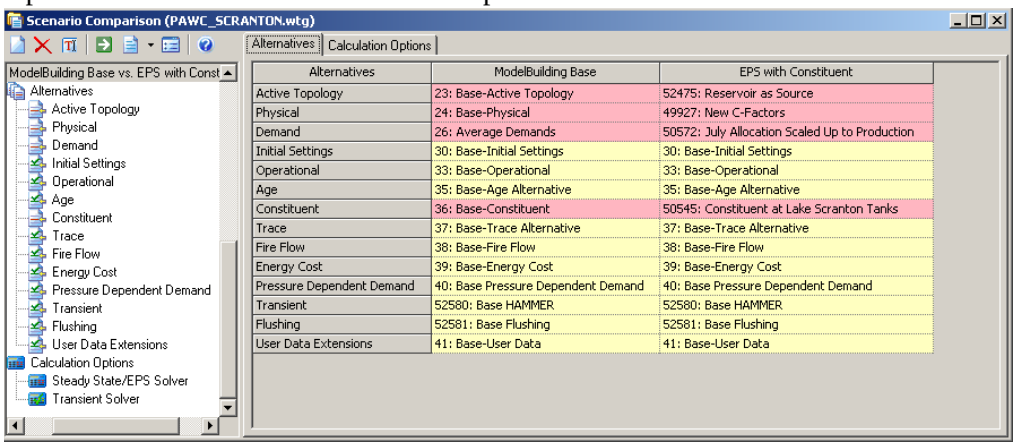


This opens a dialog which allows you to select which two scenarios will be compared.



The scenario manager button next to each selection gives you the ability to see the tree view of scenarios. Chose OK to begin the scenario comparison tool. This initially displays a list of alternatives and calculation options, with the ones with identical properties displayed with a yellow background and those with different properties displayed with a pink background. The background color can be changed from pink to any other color by selecting the sixth button from the left and then selecting the desired color.

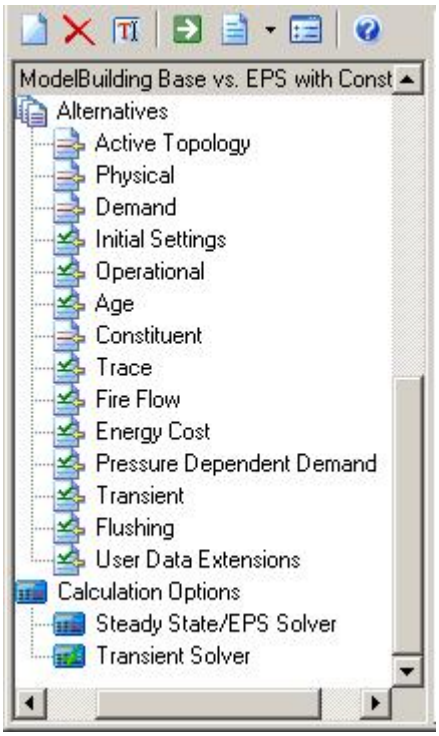
The dialog below shows that the Active Topology, Physical, Demand and Constituent alternatives are different between the scenarios. There is a second tab for Calculation Options which shows if the calculation options are different between scenarios.



Alternatives	ModelBuilding Base	EPS with Constituent
Active Topology	23: Base-Active Topology	52475: Reservoir as Source
Physical	24: Base-Physical	49927: New C-Factors
Demand	26: Average Demands	50572: July Allocation Scaled Up to Production
Initial Settings	30: Base-Initial Settings	30: Base-Initial Settings
Operational	33: Base-Operational	33: Base-Operational
Age	35: Base-Age Alternative	35: Base-Age Alternative
Constituent	36: Base-Constituent	50545: Constituent at Lake Scranton Tanks
Trace	37: Base-Trace Alternative	37: Base-Trace Alternative
Fire Flow	38: Base-Fire Flow	38: Base-Fire Flow
Energy Cost	39: Base-Energy Cost	39: Base-Energy Cost
Pressure Dependent Demand	40: Base Pressure Dependent Demand	40: Base Pressure Dependent Demand
Transient	52580: Base HAMMER	52580: Base HAMMER
Flushing	52581: Base Flushing	52581: Base Flushing
User Data Extensions	41: Base-User Data	41: Base-User Data

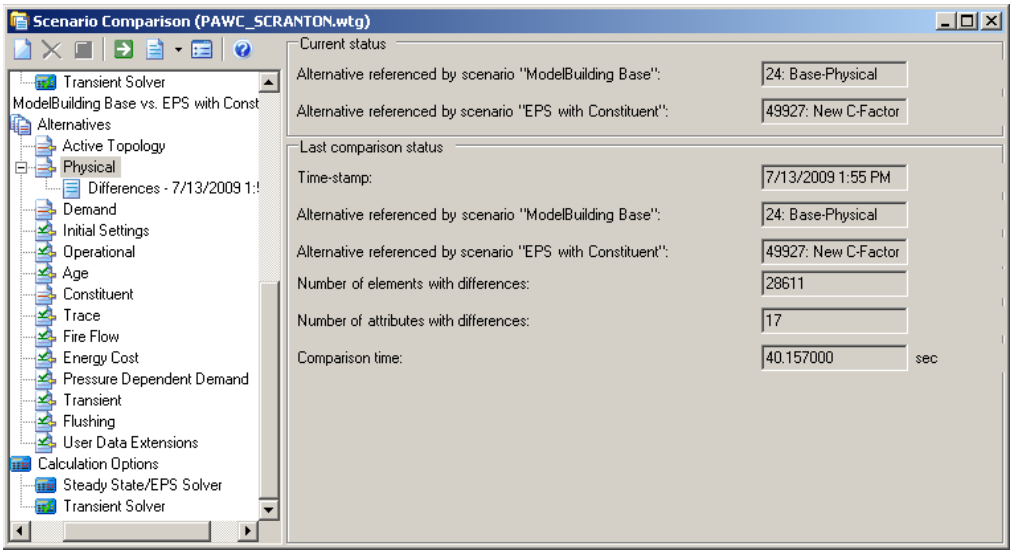
This display can also be copied to the clipboard using the Copy button.

The alternatives that have differences are also shown in the left pane with a red mark as opposed to the green check indicating that there are no differences.

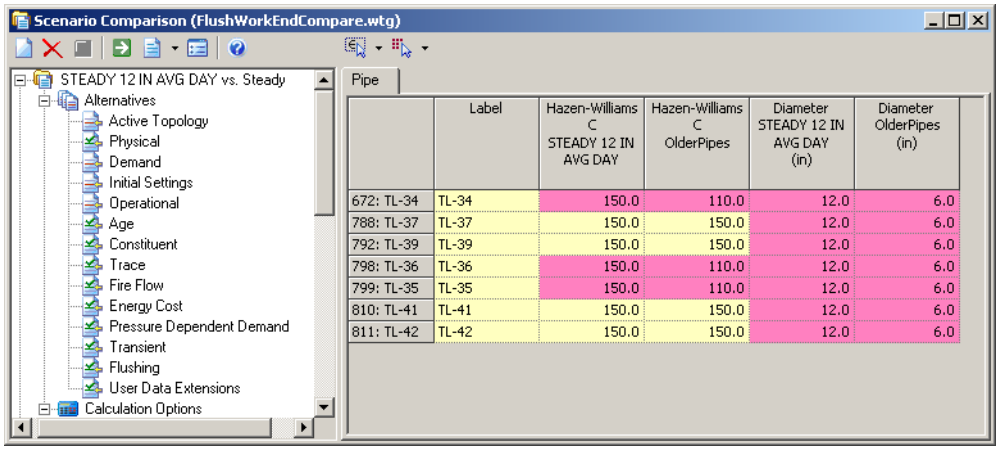


To obtain more detailed information on differences, highlight one of the alternatives and select the green and white Compute arrow at the top of pane (fourth button).

This initially returns a summary of the comparison which indicates the time when the comparison was run, which scenarios were involved and number of elements and attributes for which there were differences.



By picking "Differences" in the left pane for the alternative of interest, you can view the differences. In this display, only the elements and properties that are different are shown with a pink background. In the example below, only 7 pipes had their diameters changed and only 3 of those had difference C-factors. There are separate tables for each element type that had differences.

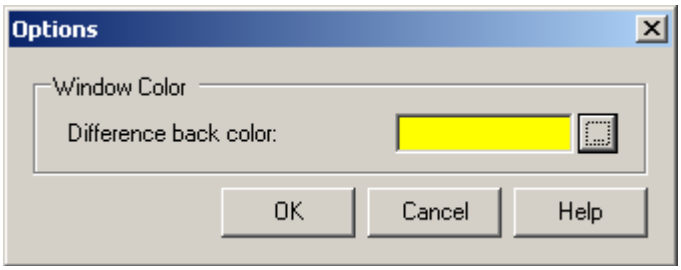


	Label	Hazen-Williams C STEADY 12 IN AVG DAY	Hazen-Williams C OlderPipes	Diameter STEADY 12 IN AVG DAY (in)	Diameter OlderPipes (in)
672: TL-34	TL-34	150.0	110.0	12.0	6.0
788: TL-37	TL-37	150.0	150.0	12.0	6.0
792: TL-39	TL-39	150.0	150.0	12.0	6.0
798: TL-36	TL-36	150.0	110.0	12.0	6.0
799: TL-35	TL-35	150.0	110.0	12.0	6.0
810: TL-41	TL-41	150.0	150.0	12.0	6.0
811: TL-42	TL-42	150.0	150.0	12.0	6.0

Using the buttons on top of the right pane, when Differences is selected, you can create a selection set of the elements with differences or highlight those elements in the drawing. This is very useful for finding elements with differences in a large model.

Scenario Comparison Options Dialog Box

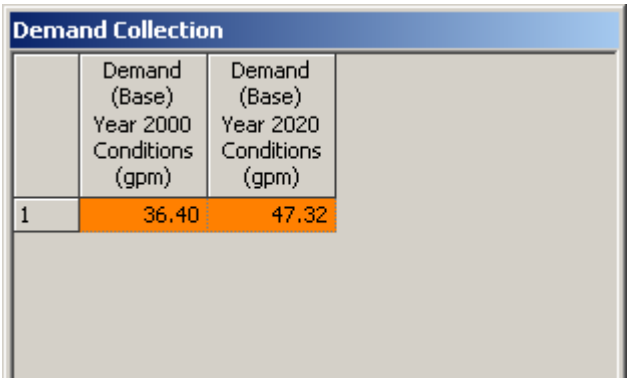
This dialog box allows you to select the color used to highlight differences between the scenarios being compared in the Scenario Comparison tool.



To choose another color, click the ellipsis button, select the new color from the palette, and click OK.

Scenario Comparison Collection Dialog Box

Some of the Differences types (such as Demand) may include collections of data (multiple demands within a single Demand Collection). By clicking the ellipsis button next to one of these collections you can open this dialog, which displays a table that breaks down the collection by the individual pieces of data.



Modeling Capabilities 10

Model and Optimize a Distribution System

Steady-State/Extended Period Simulation

Global Demand and Roughness Adjustments

Check Data/Validate

Calculate Network

Using the Totalizing Flow Meter

System Head Curves

Flow Emitters

Parallel VSPs

Fire Flow Analysis

Water Quality Analysis

Criticality Analysis

Calculation Options

Patterns

Controls

Active Topology

External Tools

SCADAConnect Overview

Model and Optimize a Distribution System

Bentley WaterCAD V8*i* provides modeling capabilities, so that you can model and optimize practically any distribution system aspect, including the following operations:

- Hydraulic Analysis
  - Perform a steady-state analysis for a snapshot view of the system, or perform an extended-period simulation to see how the system behaves over time.
  - Use any common friction method: Hazen-Williams, Darcy-Weisbach, or Manning's methods.
  - Take advantage of scenario management to see how your system reacts to different demand and physical conditions, including fire and emergency usage.
  - Control pressure and flow completely by using flexible valve configurations. You can automatically control pipe, valve, and pump status based on changes in system pressure (or based on the time of day). Control pumps, pipes, and valves based on any pressure junction or tank in the distribution system.
  - Perform automated fire flow analysis for any set of elements and zones in the network.
  - Calibrate your model manually, or use the Darwin Calibrator.
  - Generate capital and energy-cost estimates.
  - Compute system head curves.
- Water Quality Analysis
  - Track the growth or decay of substances (such as chlorine) as they travel through the distribution network.
  - Determine the age of water anywhere in the network.

Identify source trends throughout the system.Modeling capabilities include:

- Steady-State/Extended Period Simulation
- Global Demand and Roughness Adjustments
- Check Data/Validate
- Calculate Network
- Flow Emitters
- Parallel VSPs
- Fire Flow Analysis
- Water Quality Analysis

- Calculation Options
- Patterns
- Controls
- Active Topology

Steady-State/Extended Period Simulation

Bentley WaterCAD V8*i* gives the choice between performing a steady-state analysis of the system or performing an extended-period simulation over any time period.

Steady-State Simulation

Note:

Steady-state analyses determine the operating behavior of the system at a specific point in time or under steady-state conditions (flow rates and hydraulic grades remain constant over time). This type of analysis can be useful for determining pressures and flow rates under minimum, average, peak, or short term effects on the system due to fire flows.

For this type of analysis, the network equations are determined and solved with tanks being treated as fixed grade boundaries. The results that are obtained from this type of analysis are instantaneous values and may or may not be representative of the values of the system a few hours, or even a few minutes, later in time.

Extended Period Simulation (EPS)

When the variation of the system attributes over time is important, an extended period simulation is appropriate. This type of analysis allows you to model tanks filling and draining, regulating valves opening and closing, and pressures and flow rates changing throughout the system in response to varying demand conditions and automatic control strategies formulated by the WaterCAD V8*i*.

While a steady-state model may tell whether the system has the capability to meet a certain average demand, an extended period simulation indicates whether the system has the ability to provide acceptable levels of service over a period of minutes, hours, or days. Extended period simulations (EPSes) can also be used for energy consumption and cost studies, as well as water quality modeling.

Data requirements for extended period simulations are greater than for steady-state runs. In addition to the information required by a steady-state model, you also need to determine water usage Patterns, more detailed tank information, and operational rules for pumps and valves.

The following additional information is required only when performing Extended Period Simulation, and therefore is not enabled when Steady-State Analysis has been specified.

- **Start Time**—Select the clock time at which the simulation begins.
- **Duration**—Specify the total duration of an extended period simulation.
- **Hydraulic Time Step**—Select the length of the calculation time step.
- **Override Reporting Time Step?**—Set to true if you want the Reporting Time Step to differ from the Hydraulic Time Step.
- **Reporting Time Step**—Data will be presented at every reporting time step. The reporting time step should be a multiple of the hydraulic time step.

Note: If you run an Extended Period Simulation, you can generate graphs of the elements in the results by right-clicking an element and selecting Graph.

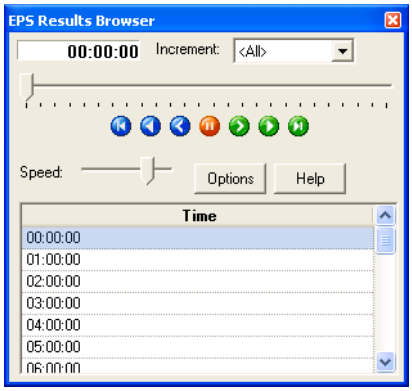
Note: Each of the parameters needed for an extended period analysis has a default value. You will most likely want to change the values to suit your particular analysis.

Occasionally the numerical engine will not converge during an extended period analysis. This is usually due to controls (typically based on tank elevations) or control valves (typically pressure regulating valves) toggling between two operational modes (on/off for pump controls, open/closed for pipe controls, active/closed for valves). When this occurs, try adjusting the hydraulic time step to a smaller value. This will minimize the differences in boundary conditions between time steps, and may allow for convergence.

Time Browser

The Time Browser dialog box is where you can change the currently displayed time step and animate the main drawing pane.

Choose Analysis > Time Browser to open the dialog box.



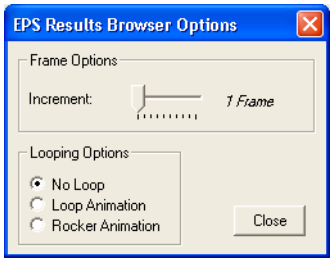
The dialog box contains the following controls:

Time Display	Shows the current time step that is displayed in the drawing pane.
Time Slider	Manually moves the slider representing the currently displayed time step along the bar, which represents the full length of time that the scenario encompasses.
Go to start	Sets the currently displayed time step to the beginning of the simulation.
Play backward	Sets the currently displayed time step from the end to the beginning.
Step backward	Returns the currently displayed time step to the previous time step.
Pause/Stop	Stops the animation. Restarts it again with another click.
Step	Advances the currently displayed time step to the following time step.
Play	Advances the currently displayed time step from beginning to end.
Go to end	Sets the currently displayed time step to the end of the simulation.
Speed Slider	Controls the length of the delay between time steps during animations.

Options	Opens the Time Browser Options dialog box where Increments and Looping Options can be set.
Help	Opens online help.
Time Step Pane	Lists each time step in the simulation. Clicking a time step sets it as current.

Time Browser Options

This dialog box is where you define the animation settings that are applied when the drawing pane is animated. Click Options from Time Browser.



It contains the following controls:

Frame Options

Increment — Controls the smoothness of the animation. Each time step in a scenario counts as one animation frame. Use this slider to specify the number of frames that are skipped for each step in the animation. For example, if there are time steps every 3 minutes in the scenario and the slider is set at 3 frames, each step in the animation represents 9 minutes of scenario time when you click the Play button.

Looping Options

No Loop — Stops the animation at the end of the simulation, if selected.

Loop Animation — Restarts the animation automatically, if selected. When this option is selected, the animation reaches the end of the simulation and then restarts from the beginning.

Rocker Animation — Restarts the animation automatically in reverse. When this option is selected, the animation reaches the end of the simulation and then plays the simulation in reverse. When the beginning of the simulation is reached, the animation advances towards the end again and will do so continually.

Steady State Run

This feature allows you to obtain a hydraulic steady state from the data in your WaterCAD V8*i* model. When the **Steady** button is selected in the "Type of Run" area of the **Run** dialog box, the model data is sent to the steady state solver so it can begin the calculations. If errors are encountered, the steady state solver will show a dialog box with a list of messages. Prior to a steady state run:

- **Steady State Options**—The parameters that control the steady state hydraulic computations are similar to those in WaterCAD V8*i*. They can be modified using the Tools > Project Options menu command and clicking the Steady State tab:
  - **Steady State Trials** is set for maximum accuracy by default. We recommend you not modify this setting. This is similar to the setting in WaterCAD V8*i*.
  - **Steady State Accuracy** is set for maximum accuracy by default. We recommend you not modify this setting. This is similar to the setting in WaterCAD V8*i*.
  - **Pump Curves Linear Mode** is either **True** or **False**. If **True**, the steady state solver uses linear interpolation to estimate the curve if the solution lies between points entered in the pump table. This method is consistent with the transient solver in WaterCAD V8*i*.
  - **Friction Method** is either **Hazen-Williams** (for which the Friction Coefficient is a C factor) or **Darcy-Weisbach**. Selecting **Darcy-Weisbach** will display both the Darcy-Weisbach *f* (for the Friction Coefficient) and the Roughness Height in the Drawing Pane. Roughness Height is only used for a steady state run and typical values are available from the material library.
- **Element Data for Steady State**—Some fields in the Drawing Pane are only required for a steady state run, as described by tooltips. If some information required by the steady state solver is missing, WaterCAD V8*i* will display a **Warning Message** dialog prompting for additional data or an **Error Message** dialog with instructions on how to proceed. Typically, error messages are related to problems in the network topology, such as a pump or valves not being connected on both its intake and discharge sides.

Calculate Network

The following steps need to be completed before performing hydraulic calculations for a network.

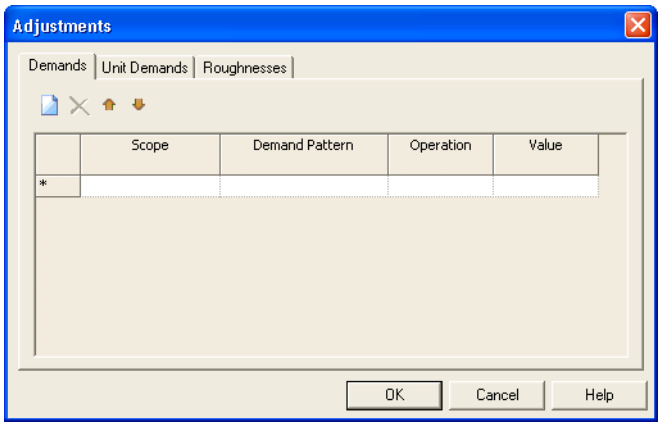
1. Click the **Analysis** toolbar and select **Calculation Options**.
2. In the **Calculation Options** dialog, double-click **Base Calculation Options** or create a new one and double-click it. This will open the **Properties** viewer.
3. In the Properties viewer, set the **Time Analysis Type** to **Steady-State** or **Extended Period**. If Extended Period is selected, then specify the starting time, the duration, and the time step to be used.
4. Optionally, in Extended Period mode, you may perform a Water Quality Analysis. Set the **Calculation Type** to **Age**, **Constituent** or **Trace**.
5. Optionally, in Steady-State mode, you may also perform a Fire Flow Analysis. Change the **Calculation Type** to **Fire Flow**.
6. Optionally, in the Adjustments section, you may modify the demand, unit demand, or roughness values of your entire network for calibration purposes. If Demand Adjustments, Unit Demand Adjustments, or Roughness Adjustments are set to Active in the Calculation Option properties and adjustments have been specified, the active adjustments will be used. This does not permanently change the value of the input data, but allows you to experiment with different calibration factors until you find the one that causes your calculation results to most closely correspond with your observed field data.
7. Optionally, verify and/or adjust the settings in Hydraulics section to change the general algorithm parameters used to perform Hydraulic and Water Quality calculations.
8. Click **Validate**  to ensure that your input data does not contain errors.
9. Click **Compute**  to start the calculations.

Global Demand and Roughness Adjustments

Demand and Roughness Adjustments based on observed data are an important part of the development of hydraulic and water quality models. It is a powerful feature for tweaking the two most commonly used parameters during model calibration: junction demands and pipe roughness.

One of the first steps performed during a calculation is the transformation of the input data into the required format for the numerical analysis engine. If Demand Adjustments, Unit Demand Adjustments, or Roughness Adjustments are set to Active in the Calculation Option properties and adjustments have been specified, the active adjustments will be used during this transformation. This does not permanently change the value of the input data but allows you to experiment with different adjustment factors until you find the one that causes your calculation results to most closely correspond with your observed field data.

For example, assume node J-10 has two demands, a 100 gpm fixed pattern demand and a 200 gpm residential pattern demand, for a total baseline demand of 300 gpm. If you enter a demand adjustment multiplier of 1.25, the input to the numerical engine will be 125 gpm and 250 gpm respectively, for a total baseline demand of 375 gpm at node J-10. If you use the Set operation to set the demands to 400, the demand will be adjusted proportionally to become 133 and 267 gpm, for a total baseline of 400 gpm. In addition, if a junction has an inflow of 100 gpm (or a demand of -100 gpm), and the adjustment operation Set demand of 200 gpm, then the inflow at that junction will be -200 gpm (equivalent to a demand of 200 gpm).



The Adjustments dialog is divided into three tabs, each containing a table of adjustments and controls to control the data within the table. These controls are as follows:

- **New**—Adds a new adjustment to the table.
- **Delete**—Removes the currently highlighted adjustment from the table.
- **Shift Up**—Adjustments are executed in the order they appear in the table. This button shifts the currently highlighted adjustment up in the table.

- **Shift Down**—Adjustments are executed in the order they appear in the table. This button shifts the currently highlighted adjustment down in the table.

The tables contained within the tabs are as follows:

- **Demands**—Use this adjustment tab to temporarily adjust the individual demands at all junction nodes in the system that have demands for the current scenario or a subset of junctions contained within a previously created selection set. The Demands adjustment table contains the following columns:
  - **Scope**—Use this field to specify the elements that the adjustment will be applied. Choose <Entire Network> to apply the adjustment to every demand node, or choose a subset of nodes by selecting one of the previously created selection sets from the list.
  - **Demand Pattern**—Use this field to specify the demands to which the adjustment will be applied. Choose <All Base Demands> to perform the adjustment on every base demand in the model. Choose Fixed to perform the adjustment on only those nodes with a Fixed demand pattern. Choose one of the demand patterns in the list to apply the adjustment to only the specified pattern.
  - **Operation**—Choose the operation to be performed in the adjustment using the value specified in the Value column.
  - **Value**—Type the value for the adjustment.

- **Unit Demands**—Use this adjustment tab to temporarily adjust the unit demands at all junction nodes in the system that have demands for the current scenario, or a subset of junctions contained within a previously created selection set.
  - **Scope**—Use this field to specify the elements that the adjustment will be applied. Choose <Entire Network> to apply the adjustment to every node with a unit demand, or choose a subset of nodes by selecting one of the previously created selection sets from the list.
  - **Unit Demand**—Use this field to specify the unit demands to which the adjustment will be applied. Choose <All Unit Demands> to perform the adjustment on every unit demand in the model. Choose one of the unit demands in the list to apply the adjustment to only the specified unit demand.
  - **Operation**—Choose the operation to be performed in the adjustment using the value specified in the Value column.
  - **Value**—Type the value for the adjustment.

- **Roughnesses**—Use this adjustment tab to temporarily adjust the roughness of all pipes in the distribution network or a subset of pipes contained within a previously defined selection set.

- **Scope**—Use this field to specify the elements that the adjustment will be applied. Choose <Entire Network> to apply the adjustment to every pipe, or choose a subset of pipes by selecting one of the previously created selection sets from the list.
- **Operation**—Choose the operation to be performed in the adjustment using the value specified in the Value column.
- **Value**—Type the value for the adjustment.

Check Data/Validate

This feature allows you to validate your model against typical data entry errors, hard to detect topology problems, and modeling problems. When the Validate box is checked, the model validation is automatically run prior to calculations. It can also be run at any time by clicking Validate  . The process will produce either a dialog box stating No Problems Found or a Status Log with a list of messages.

The validation process will generate two types of messages. A warning message means that a particular part of the model (i.e., a pipe's roughness) does not conform to the expected value or is not within the expected range of values. This type of warning is useful but not fatal. Therefore, no corrective action is required to proceed with a calculation. Warning messages are often generated as a result of a topographical or data entry error and should be corrected. An error message, on the other hand, is a fatal error, and the calculation cannot proceed before it is corrected. Typically, error messages are related to problems in the network topology, such as a pump or valve not being connected on both its intake and discharge sides.

Note: **In earlier versions of the software, it was possible to create a topological situation that was problematic but was not checked for in the network topology validation. The situation could be created by morphing a node element such as a junction, tank, or reservoir into a pump or valve. This situation is now detected and corrected automatically, but it is strongly recommended that you verify the flow direction of the pump or valve in question. If you have further questions or comments related to this, please contact Bentley Support.**

Warning messages related to the value of a particular attribute being outside the accepted range can often be corrected by adjusting the allowable range for that attribute.

The check data algorithm performs the following validations:

- **Network Topology**—Checks that the network contains at least one boundary node, one pipe, and one junction. These are the minimum network requirements. It also checks for fully connected pumps and valves and that every node is reachable from a boundary node through open links.
- **Element Validation**—Checks that every element in the network is valid for the calculation. For example, this validation ensures that all pipes have a non-zero length, a non-zero diameter, a roughness value that is within the expected range, etc.

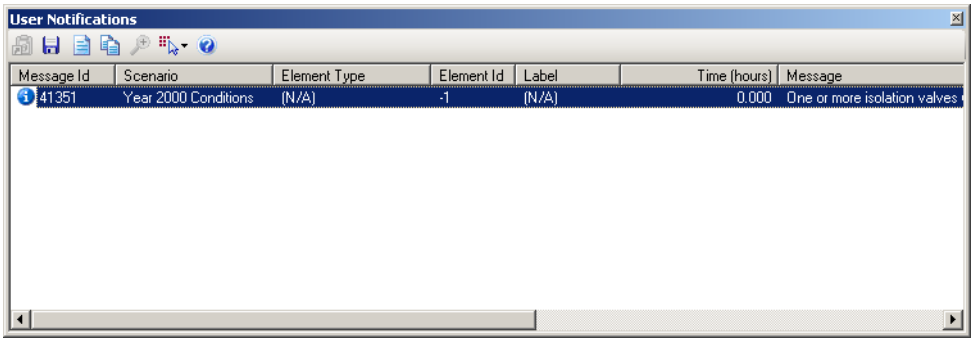
User Notifications

User notifications are messages about your model. These messages can warn you about potential issues with your model, such as slopes that might be too steep or elements that slope in the wrong direction. These messages also point you to errors in your model that prevent Bentley WaterCAD V8*i* from solving your model.

The User Notifications dialog box displays warnings and error messages that are turned up by Bentley WaterCAD V8*i* 's validation routines. If the notification references a particular element, you can zoom to that element by either double-clicking the notification, or right-clicking it and selecting the Zoom To command.

- Warnings are denoted by an orange icon and do not prevent the model from calculating successfully.
- Errors are denoted by a red icon, and the model will not successfully calculate if errors are found.

The User Notifications dialog box consists of a toolbar and a tabular view containing a list of warnings and error messages.



The toolbar consists of the following buttons:

Details



Displays the User Notification Details dialog box, which includes information about any warning or error messages.

Save



Saves the user notifications as a comma-delimited .csv file. You can open the .csv file in Microsoft Excel or Notepad.

Report



Displays a User Notification Report.

Copy



Copies the currently highlighted warning or error message to the Windows clipboard.

Zoom To



If the warning or error message is related to a specific element in your model, click this button to center the element in question in the drawing pane.

Select In Drawing



Click this button to select all of the elements that the associated message applies to in the drawing pane.

Help



Displays online help for User Notifications.

User Notifications displays warnings and error messages in a tabular view. The table includes the following columns:

Message ID

The message ID associated with the corresponding message.

Scenario

The scenario associated with the corresponding message. This column will display “Base” unless you ran a different scenario.

Element Type	The element type associated with the corresponding message.
Element ID	The element ID associated with the corresponding message.
Label	If the notification is caused by a specific element, this column displays the label of the element associated with the corresponding message.
Message	The description associated with the corresponding message.
Time (hours)	If the user notification occurred during a specific time step, it is displayed. Otherwise, this column is left blank.
Source	The validation routine that triggered the corresponding message.

To view user notifications

1. Compute your model. If there are any.
2. If needed, open the User Notification manager by going to **Analysis > User Notifications** <F8>.
3. Or, if the calculation fails to compute because of an input error, when your model is finished computing, Bentley WaterCAD V8*i* prompts you to view user notifications to validate the input data.

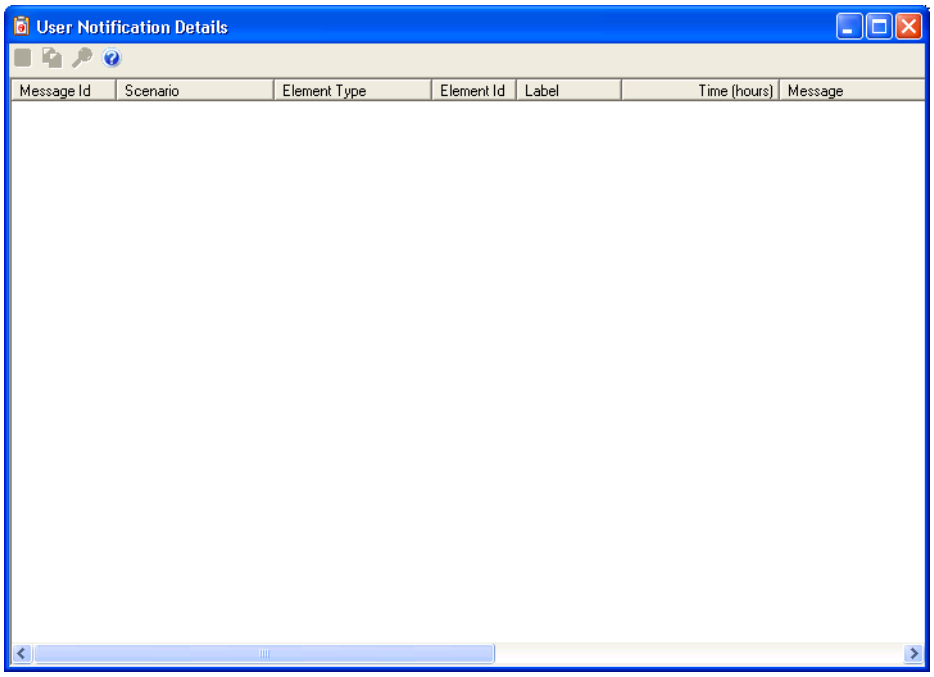
   You must fix any errors identified by red circles before Bentley WaterCAD V8*i* can compute a result.

   Errors identified by orange circles are warnings that do not prevent the computation of the model.
4. In the User Notifications manager, if a notification pertains to a particular element, you can double-click the notification to magnify and display the element in the center of the drawing pane.
5. Use the element label to identify the element that generates the error and use the user notification message to edit the element's properties to resolve the error.

User Notification Details Dialog Box

This dialog lists the elements that are referred to by a time-sensitive user notification message. In the User Notification dialog, there is a time column that displays the time-step during which time-sensitive messages occur. These messages will say “during this time-step” or “for this time-step”, and do not display information about the referenced element or elements. Double-clicking one of these messages in the User Notifications dialog opens the User Notification Details dialog, which does provide information about the referenced element(s).

You can double-click messages in the User Notification Details dialog to zoom the drawing pane view to the referenced element.



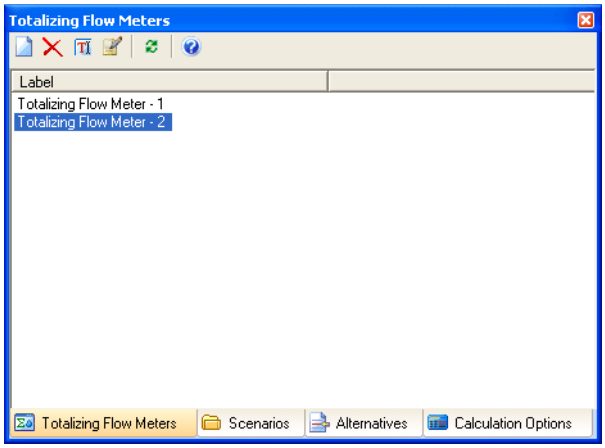
Using the Totalizing Flow Meter

Totalizing flow meters allow you to view results of the total volume going through your model for a specific selection of elements.

Totalizing Flow Meters Manager Dialog

The Totalizing Flow Meter manager consists of the following controls:

New	Create a new totalizing flow meter.
Delete	Delete the selected totalizing flow meter.
Rename	Rename the label for the current totalizing flow meter.
Edit	Open the totalizing flow meter editor.
Refresh	Recompute the volume of the current totalizing flow meter.
Help	Opens the online help for totalizing flow meter.



To create a new Totalizing Flow Meter

1. Click Compute. (EPS settings must be on in order to utilize this feature.)
2. From the Analysis Menu click Totalizing Flow Meters.

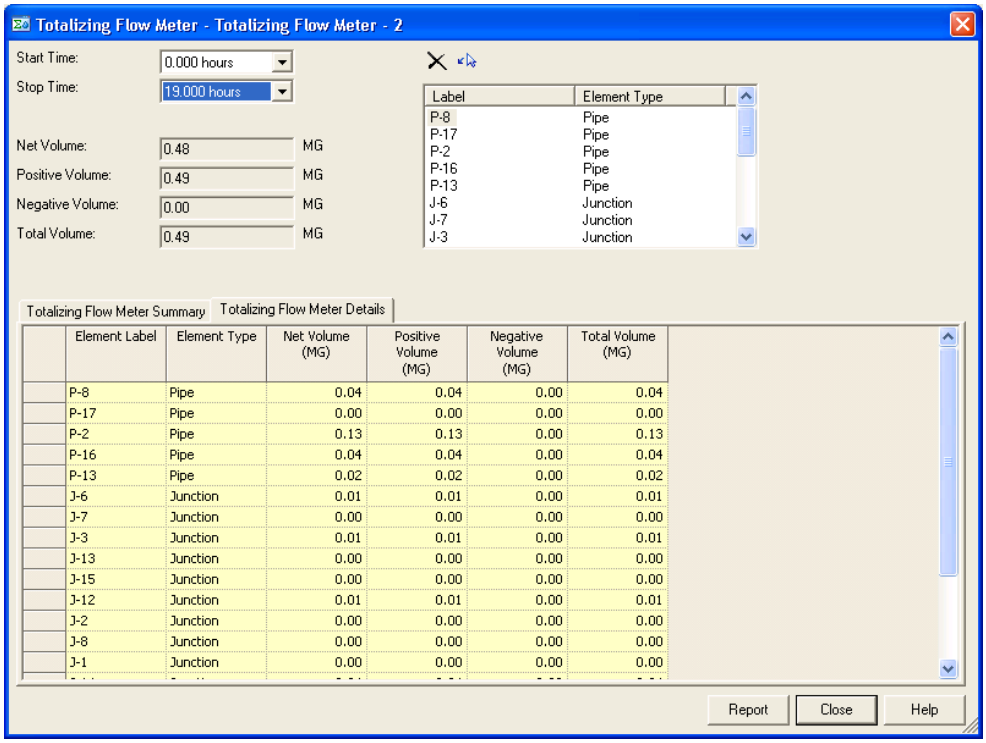


3. Click New which will open up the Select box.
4. Select the elements to be calculated or click the Query box then click Done.

Totalizing Flow Meter Editor Dialog

The Totalizing Flow Meter editor allows you to:

- Define settings for new or existing flow meters
- Display the calculated results for the current flow meter settings.



The **Totalizing Flow Meter Summary** tab displays the totals for each element type.

The **Totalizing Flow Meter Details** tab displays results for each individual element.

To define flow meter settings

1. Set Start and Stop times. Once selected, the results are automatically updated.
2. Click the Report button to run a report or click Close.

To remove elements from the Totalizing Flow Meter definition

Highlight the element to be removed in the list and click the Delete button above the list pane.

To add elements to the Totalizing Flow Meter definition

1. Click the Select From Drawing button above the element list pane.
2. In the Drawing View, click the element or elements to be added.
3. Click the Done button in the Select dialog.

System Head Curves

The purpose of a pump is to overcome elevation differences and head losses due to pipe friction and fittings. The amount of head the pump must add to overcome elevation differences is dependent on system characteristics and topology (and independent of the pump discharge rate), and is referred to as static head. Friction and minor losses, however, are highly dependent on the rate of discharge through the pump. When these losses are added to the static head for a series of discharge rates, the resulting plot is called a system head curve.

Pumps are designed to lift water from one elevation to another, while overcoming the friction and minor losses associated with the piping system. To correctly size a pump, one must understand the static head (elevation differences) and dynamic head (friction and minor losses) conditions under which the pump is expected to operate. The static head will vary due to changes in reservoir or tank elevations on both the suction and discharge sides of the pump, and the dynamic head is dependent on the rate of discharge through the pump.

System head curves are a useful tool for visualizing the static and dynamic head for varying rates of discharge and various static head conditions. The system head curve is a graph of head vs. flow that shows the head required to move a given flow rate through the pump and into the distribution system.

System Head Curves in Closed Systems

The theory behind system head curves is that there is a tank or reservoir on both the suction and discharge side of the pump for which the curve is developed. In the case of closed (dead end) systems, there is no reservoir or tank downstream of the pump. The demands must always be exactly met. In order to develop a system head curve for such a pipe network, it is necessary to account for the relationship between usage and pressure. Therefore the network demands must be represented by pressure dependent demands.

To develop a system head cure for such a network, the demands must be set to pressure dependent demands (PDD) and there must be no threshold pressure set for demands. This is done by:

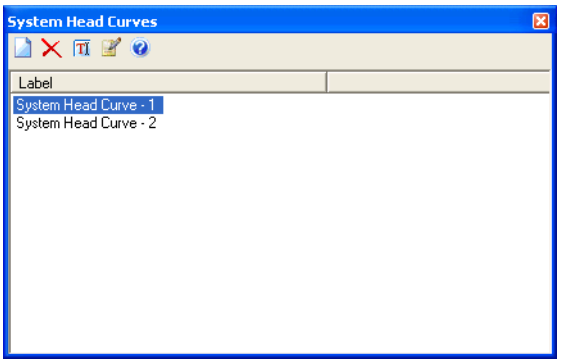
1. Defining a PDD function in Components > Pressure Dependent Demand.
2. Setting up a PDD alternative assigning the PDD functions and making certain that "Reference Pressure Equals Threshold?" is unchecked.
3. Setting the Calculation Option "Use Pressure Dependent Demand" to True.

Check the model to make sure it runs correctly before creating the System Head Curve.

System Head Curves Manager Dialog

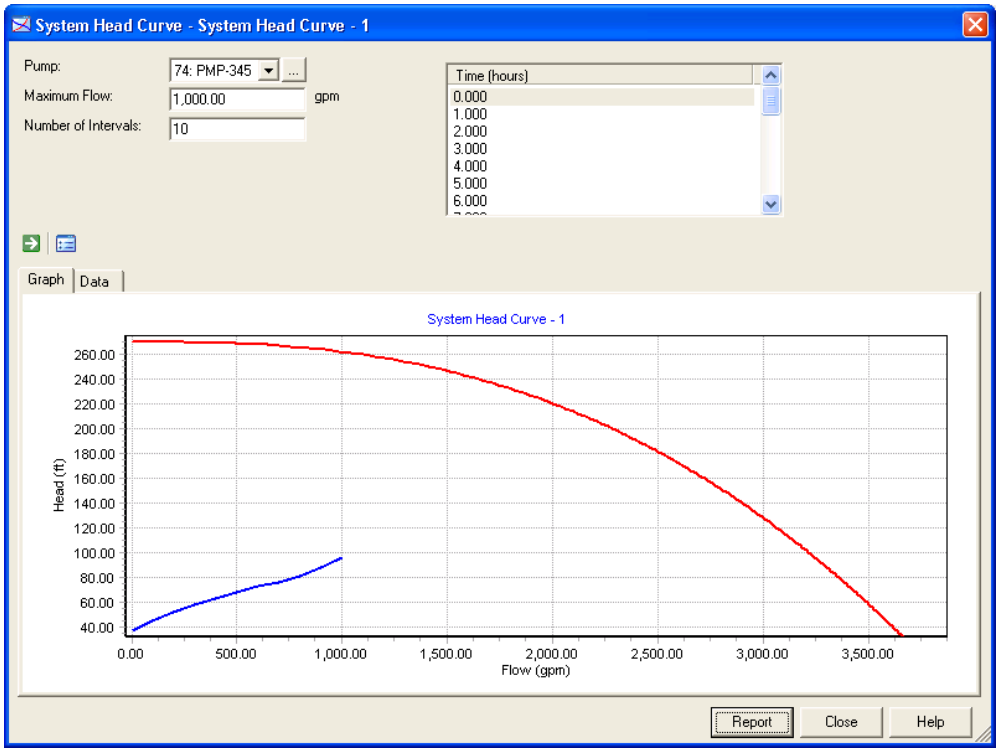
The System Head Curves manager allows you to create, edit, and manager system head curves. It consists of the following controls:

New	Create a new system head curve.
Delete	Delete the selected system head curve.
Rename	Rename the label for the current system head curve.
Edit	Open the system head curve editor.
Help	Open the online help for system head curves.



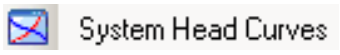
System Head Curve Editor Dialog

The System Head Curve editor allows you to define and calculate a graph of head vs. flow that shows the head required to move a given flow rate through the selected pump and into the distribution system.



To create a new System Head Curve Definition

1. Click Compute. (EPS settings must be on in order to utilize this feature.)
2. From the Analysis Menu click System Head Curves.



3. Click New which will open the System Head Curve editor.

   The System Head Curves Editor is where you can specify the settings of System Head Curve Definition. You can also compute and view the system head curve for a specific timestep.

4. Choose the pump that will be used for the system head curve from the **Pump** pull-down menu, or click the ellipsis and click the pump to be used in the drawing pane.
5. Type a value for **Maximum Flow** and **Number of Intervals**.

6. Choose a time step in the **Time (hours)** column.
7. Click **Compute** to calculate the results for the specified time step.
8. View the results as a graph or data.
9. Click Report to view the report.
10. Click Close to exit the System Head Curve editor.

Note: You can select more than one time step for the system head curve calculation by holding down the <Ctrl> key and clicking each time step that you want to calculate.

Post Calculation Processor

The Post Calculation Processor allows you to perform statistical analysis for an element or elements on various results obtained during an extended period simulation calculation.

The results of the Post Calculation Processor analysis are then displayed in a previously defined user defined field. To learn more about user defined fields see User Data Extensions.

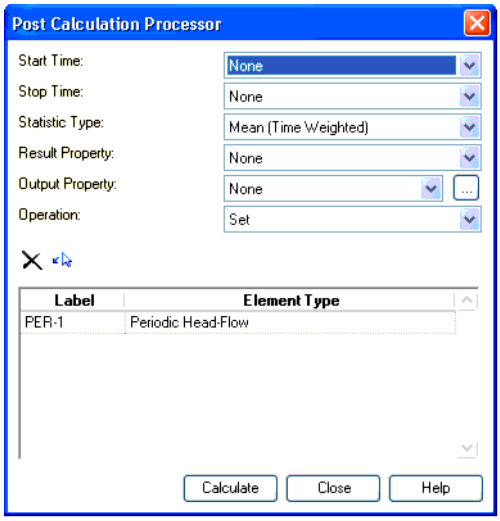
The Post Calculation Processor dialog consists of the following controls:

Start Time	Specify the start time for the period of time that will be analysed.
Stop Time	Specify the stop time for the period of time that will be analysed.
Statistic Type	Choose the type of statistical analysis to perform.
Result Property	Choose the calculated result that will be analysed for the selected element(s).
Output Property	Choose the user-defined data extension where the results of the analysis will be stored.

Operation — Choose an operation to determine how to apply the calculation result to the output field. For example Set will enter the result of the analysis to the field without modification, Add will enter the sum of any current value in the output field and the calculated result, and so on.

Remove Element — Removes the element that is currently selected in the table.

Select From Drawing — Allows you to select additional elements from the drawing pane and add them to the table.



Flow Emitters

Flow Emitters are devices associated with junctions that model the flow through a nozzle or orifice. In these situations, the demand (i.e., the flow rate through the emitter) varies in proportion to the pressure at the junction raised to some power. The constant of proportionality is termed the discharge coefficient. For nozzles and sprinkler heads, the exponent on pressure is 0.5 and the manufacturer usually states the value of the discharge coefficient as the flow rate in gpm through the device at a 1 psi pressure drop.

Emitters are used to model flow through sprinkler systems and irrigation networks. They can also be used to simulate leakage in a pipe connected to the junction (if a discharge coefficient and pressure exponent for the leaking crack or joint can be estimated) and compute a fire flow at the junction (the flow available at some minimum residual pressure). In the latter case, one would use a very high value of the discharge coefficient (e.g., 100 times the maximum flow expected) and modify the junction's elevation to include the equivalent head of the pressure target.

When both an emitter and a normal demand are specified for a junction, the demand that Bentley WaterCAD V8*i* reports in its output results includes both the normal demand and the flow through the emitter.

The flow through an emitter is calculated as:

$$Q = kP^n$$

Where

Q is flow.

k is the emitter coefficient and is a property of the node.

P is pressure.

n is the emitter exponent and is set globally in the calculation options for the run; it is dimensionless but affects the units of k. The default value for n is 0.5 which is a typical value for an orifice.

Parallel VSPs

Variable speed pumps (VSPs) can be modeled in parallel. This allows you to model multiple VSPs operated at the same speed at one pump station. To model this, a VSP is chosen as a “lead VSP”, which will be the primary pump to deliver the target head. If the lead VSP cannot deliver the target head while operating at maximum speed, then the second VSP will be triggered on and the VSP calculation will determine the common speed for both VSPs. If the target head cannot be delivered while operating both VSPs at the maximum speed, then another VSP will be triggered on until the target head is met with all the available VSPs.

All VSPs that are turned on are operated at the same speed. VSPs are to be turned off if they are not required due to a change in demand. If all standby VSPs are running at the maximum speed but still cannot deliver the target head, the VSPs are translated into fixed speed pumps.

To correctly apply the VSP feature to multiple variable speed pumps in parallel, the following criteria must be met:

1. Parallel VSPs must be controlled by the same target node;
2. Parallel VSPs must be controlled by the same target head;
3. Parallel VSPs must have the same maximum relative speed factors;
4. Parallel VSPs must be identical, namely the same pump curve.
5. Parallel VSPs must share common upstream and downstream junctions within 3 nodes (inclusive) of the pumps in order for them to be recognized as parallel VSPs.

   If there are more than 3 nodes between the pumps and their common node, upstream and downstream, the software will treat them as separate VSPs. Since separate VSPs cannot target the same control node, this will result in an error message.

Fire Flow Analysis

One of the goals of a water distribution system is to provide adequate capacity to fight fires. Bentley WaterCAD V8*i* 's powerful fire flow analysis capabilities can be used to determine if the system can meet the fire flow demands while maintaining various pressure constraints. Fire flows can be computed for a single node, a group of selected nodes, or all nodes in the system. A complete fire flow analysis can comprise hundreds or thousands of individual flow solutions—one for each junction selected for the fire flow analysis.

Fire flows are computed at user-specified locations by iteratively assigning demands and computing system pressures. The program calculates a steady-state analysis for each node in the Fire Flow Alternative. At each node, it begins by running a Steady-State analysis to ensure that the fire flow constraints that have been set can be met without withdrawing Fire Flow from any of the nodes. If the constraints are met in this initial run, the program then begins iteratively assigning the Needed Fire Flow demands at each of the nodes, and checking to ensure that the constraints are met. The program then runs another set of Steady State analyses, this time either adding the Maximum Fire Flow (as set in the Fire Flow Upper Limit input box of the Fire Flow Alternative) to whatever normal demands are required at that node, or replacing the normal demands. In either case, the program checks the residual pressure at that node, the Minimum Zone Pressure, and, if applicable, the Minimum System Pressure. If the Fire Flow Upper Limit can be delivered while maintaining the various pressure constraints, that node will satisfy the Fire Flow constraints. If one or more of the pressure constraints is not met while attempting to withdraw the Fire Flow Upper Limit, the program will iteratively assign lesser demands until it finds the maximum flow that can be provided while maintaining the pressure constraints. If a node is not providing the Fire Flow Upper Limit, it is because the Residual Pressure at that node, the Minimum Zone Pressure, or the Minimum System Pressure constraints are not met while attempting to withdraw the Fire Flow Upper Limit (or the maximum number of iterations has been reached). If a node completely fails to meet the Fire Flow constraints, it is because the network is unable to deliver the Needed Fire Flow while still meeting the pressure constraints.

After the program has gone through the above process for each node in the Fire Flow Analysis, it runs a final Steady-State calculation that does not apply Fire Flow demands to any of the junctions. This provides a baseline of calculated results that can then be compared to the Fire Flow conditions, which can be determined by viewing the results presented on the Fire Flow tab of the individual junction editors, or in the Fire Flow Tabular Report. The baseline pressures are the pressures that are modeled under the standard steady-state demand conditions in which fire flows are not exerted.

Tip: **All parameters defining a fire flow analysis, such as the residual pressure or the minimum zone pressure, are explained in detail in the Fire Flow Alternative (see** Fire Flow Alternative**)and in the Fire Flow tab topics.**

An online Tutorial on Fire Flow can be found by selecting the Help > Tutorials menu.

To perform a Fire Flow analysis

1. Change the **Calculation Type** calculation option to **Fire Flow (see** Calculation Options**)**.
2. Open the **Alternatives** manager (Cick the Analysis menu and select Alternatives).
3. Double click on **Base-Fire Flow** to open the Fire Flow Alternative editor.
4. Define the needed fireflow, fireflow upper limit, pressure constraints and the fire flow nodes selection set.
5. After all necessary fields have been entered, close the Fire Flow Alternative and Aternatives manager and click Compute  .
6. Open the Fire Flow Results Browser  . Only the elements that were specified in the selection set will be color coded.

Fire Flow Results

After performing a fire flow analysis, calculation results are available for each junction node in the fire flow selection set. These results can be viewed in the predefined Fire Flow Report (in tabular format).

	*	ID	Label	Specify Local Fire Flow Constraints?	Velocity Upper Limit (ft/s)	Needed Fire Flow (gpm)	Fire Flow Upper Limit (gpm)	Pressure Residual Lower Limit (lbs/in²)	Pressure Zone Lower Limit (lbs/in²)	Press System Lim (lbs/
58: J-120	☑	58	J-120	☐	0.00	0.00	0.00	20.00	20.00	
59: J-110	☑	59	J-110	☐	0.00	0.00	0.00	20.00	20.00	
60: J-170	☑	60	J-170	☐	0.00	0.00	0.00	20.00	20.00	
61: J-190	☑	61	J-190	☐	0.00	0.00	0.00	20.00	20.00	
62: J-30	☑	62	J-30	☐	0.00	0.00	0.00	20.00	20.00	
63: J-130	☑	63	J-130	☐	0.00	0.00	0.00	20.00	20.00	
64: J-100	☑	64	J-100	☐	0.00	0.00	0.00	20.00	20.00	
65: J-60	☑	65	J-60	☐	0.00	0.00	0.00	20.00	20.00	
66: J-40	☑	66	J-40	☐	0.00	0.00	0.00	20.00	20.00	
67: J-10	☑	67	J-10	☐	0.00	0.00	0.00	20.00	20.00	
68: J-140	☑	68	J-140	☐	0.00	0.00	0.00	20.00	20.00	
69: J-50	☑	69	J-50	☐	0.00	0.00	0.00	20.00	20.00	
70: J-210	☑	70	J-210	☐	0.00	0.00	0.00	20.00	20.00	

The results can also be viewed by clicking Report.

36: Base-Fire Flow, Fire Flow System Data Alternative Report

Fire Flow		
Needed Fire Flow	0.00	gpm
Fire Flow Upper Limit	0.00	gpm
Pressure Residual Lower Limit	20.00	lbs/in²
Pressure Zone Lower Limit	20.00	lbs/in²
Use Minimum Pressure System Constraint	False	
Pressure System Lower Limit	0.00	lbs/in²
Use Velocity Constraint	False	
Velocity Upper Limit	0.00	ft/s
Apply Fire Flows By:	Adding to Baseline Demand	
Pipe Set	All Pipes	
Fire Flow Nodes	All Fire Flow Nodes	
Category		
Use Pipe Velocity Greater Than?	False	
Use Extended Auxiliary Output by Node Pressure Less Than?	False	
Pipe Velocity Greater Than	0.00	ft/s
Node Pressure Less Than	0.00	lbs/in²
Auxiliary Output Selection Set	No Elements	
Generic		
Fire Flow Auxiliary Results Type	All Nodes	
Fire Flow Auxiliary Results Type	All Nodes	

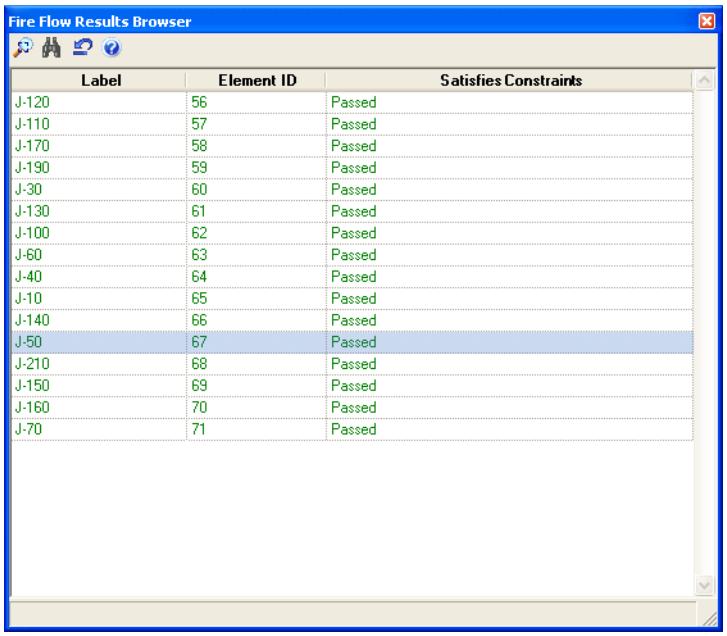
You can also choose to have the program save "auxiliary results" (a snap shot result set of the fire flow analysis hydraulic conditions) for no fire flow nodes, just the failing fire flow nodes, if any, or all fire flow nodes. For every fire flow node that attracts auxiliary results a separate result set (file) is created. When enabling this setting be conscious of the number of fire flow nodes in your system and the potential disk space requirement.

Enabling this option also will slow down the fire flow analysis due to the need to create the additional results sets. Note: The base result set includes hydraulic results for the actual fire flow node and also for the pipes that connect to the fire flow node. The results stored are for the hydraulic conditions that are experienced during the actual fire flow analysis (i.e., under fire flow loading). No other hydraulic results are stored unless the auxiliary result set is "extended" by other options listed below.

Fire Flow Results Browser

The Fire Flow Results Browser allows you to quickly jump to fire flow nodes and display the results of fire flow analysis at the highlighted node. it also allows you to view Fire Flow Auxiliary results (a snap shot result set of the fire flow analysis hydraulic conditions), if the **Fire Flow Auxiliary Results Type** option is set to Failed Nodes or All Nodes. Auxiliary results can also be displayed using the Fire Flow Node FlexTable (see FlexTables) and Element Symbology (see Annotating Your Model).

Go to Analysis > Fire Flow Results Browser or click  .



Label	Element ID	Satisfies Constraints
J-120	56	Passed
J-110	57	Passed
J-170	58	Passed
J-190	59	Passed
J-30	60	Passed
J-130	61	Passed
J-100	62	Passed
J-60	63	Passed
J-40	64	Passed
J-10	65	Passed
J-140	66	Passed
J-50	67	Passed
J-210	68	Passed
J-150	69	Passed
J-160	70	Passed
J-70	71	Passed

Zoom to see results of the specific element  .

To find a specific element, click the Find button  .

Reset to Standard Steady State Results  .Click to override the selection set and apply results to all elements in the model. Reset will also occur when you close Fire Flow Results Browser.

Not Getting Fire Flow at a Junction Node

Perform the following checks if you are not getting expected fire flow results:

- Check the Available Fire Flow. If it is lower than the Needed Fire Flow, the fire flow conditions for that node are not satisfied. Therefore, Satisfies Fire Flow Constraints is false.

- Check the Calculated Residual Pressure. If it is lower than the Residual Pressure Constraint, the fire flow condition for that node is not satisfied. Therefore, Satisfies Fire Flow Constraints is false.
- Check the Calculated Minimum Zone Pressure. If it is lower than the Minimum Zone Pressure Constraint, the fire flow condition for that node is not satisfied. Therefore, Satisfies Fire Flow Constraints is false.
- If you checked the box for Minimum System Pressure Constraint in the Fire Flow Alternative dialog box, check to see if the Calculated Minimum System Pressure is lower than the set constraint. If it is, Satisfies Fire Flow Constraints is false.

Note: If you are not concerned about the pressure of a node that is NOT meeting the Minimum Zone Pressure constraint, move this node to another zone. Now, the node will not be analyzed as part of the same zone.

Water Quality Analysis

The following Water Quality Analysis parameters are available for user configuration:

- **Age Tolerance**—If the difference between two parcels of water is equal to or less than the value specified in this field, the parcels are considered to be of equal age.
- **Constituent Tolerance**—If the difference between two parcels of water is equal to or less than the value specified in this field, the parcels are considered to possess an equal concentration of the associated constituent.
- **Trace Tolerance**—If the difference between two parcels of water is equal to or less than the value specified in this field, the parcels are considered to be within the same percentile.
- **Set Quality Time Step**—Check this box if you want to manually set the water quality time step. By default, this box is not checked and the water quality time step is computed internally by the numerical engine.
- **Quality Time Step**—Time interval used to track water quality changes throughout the network. By default, this value is computed by the numerical engine and is equivalent to the smallest travel time through any pipe in the system.

- Age Analysis
- Constituent Analysis
- Trace Analysis

Note: If you run a Water Quality Analysis, you can generate graphs of the elements in the results by right-clicking an element and selecting Graph.

Age Analysis

An age analysis determines how long the water has been in the system and is more of a general water quality indicator than a measurement of any specific constituent. To configure for an age analysis:

Note: Water quality analysis can only be performed for extended period simulations.

1. Click the **Analysis** menu and select **Calculation Options**.
2. In the **Calculation Options** manager, click the **New** button  to create a new calculation option definition.
3. Change the **Calculation Type** to **Age**.
4. Specify the **Calculation Times** and the **Age Tolerance**. Optionally, specify Hydraulics, Adjustments, and/or Calculation Flag settings. **Close** the **Calculation Options** dialog.
5. Assuming you have not already set up an Age alternative for this scenario (including defining the trace node), go to the Alternatives tab, click the **Ellipsis (...)** or **New** button next to the Age choice list, and add or edit an Age alternative. To edit an existing alternative (see Age Alternatives), click the **Edit** button. Enter the appropriate data, and click **Close**. Rename the alternative to give it a descriptive name. To add a new alternative, click the **Add** button. Enter a descriptive name into the New Alternative dialog box and click **OK**. Enter the appropriate data into the Age Alternative Editor and click **Close**. Back in the Alternatives tab, choose the desired alternative from the Age Alternative choice list.
6. Click the **Compute** button  .

Constituent Analysis

A constituent is any substance, such as chlorine and fluoride, for which the growth or decay can be adequately described through the use of a bulk reaction coefficient and a wall reaction coefficient. A constituent analysis determines the concentration of a constituent at all nodes and links in the system. Constituent analyses can be used to determine chlorine residuals throughout the system under present chlorination schedules, or can be used to determine probable behavior of the system under proposed chlorination schedules. To configure for a constituent analysis:

Note: Water quality analysis can only be performed for extended period simulations.

1. Click the **Analysis** menu and select **Calculation Options**.
2. In the **Calculation Options** manager, click the **New** button  to create a new calculation option definition.
3. Change the **Calculation Type** to **Constituent**.
4. Specify the **Calculation Times** and the **Constituent Tolerance**. Optionally, specify Hydraulics, Adjustments, and/or Calculation Flag settings. **Close** the **Calculation Options** dialog.
5. Assuming you have not already set up a Constituent alternative for this scenario (including the selection of the constituent), go to the Alternatives tab, click the **Ellipsis (...)** or **New** button next to the Constituent scroll-down list, and add or edit a Constituent alternative (for more information, see Constituent Alternatives). To edit an existing alternative, click the **Edit** button. Enter the appropriate data, and click **Close**. Rename the alternative to give it a descriptive name. To add a new alternative, click the **Add** button. Enter a descriptive name into the New Alternative dialog box and click **OK**. Enter the appropriate data into the Constituent Alternative Editor and click **Close**. Specify the Constituent, which is defined in the Constituent Library and accessed by clicking the **Ellipsis (...)** button. Back in the Alternatives tab, choose the desired alternative from the Constituent Alternative choice list.
6. Click the **Compute** button  .

Trace Analysis

A trace analysis determines the percentage of the water at all nodes and links in the system. The source is designated as a specific node in the system and is called the trace node. In systems with more than one source, it is common to perform multiple trace analyses using the various trace nodes in successive analyses. The source node and initial traces are specified in the Trace Alternative dialog box (for more information, see Trace Alternative). To configure for a trace analysis:

Note: Water quality analysis can only be performed for extended period simulations.

1. Click the **Analysis** menu and select **Calculation Options**.
2. In the **Calculation Options** manager, click the **New** button  to create a new calculation option definition.
3. Change the **Calculation Type** to **Trace**.
4. Specify the **Calculation Times** and the **Trace Tolerance**. Optionally, specify Hydraulics, Adjustments, and/or Calculation Flag settings. **Close** the **Calculation Options** dialog.
5. Assuming you have not already set up a Trace alternative for this scenario (including defining the trace node), go to the Alternatives tab, click the **Ellipsis (...)** or **New** button next to the Trace choice list, and add or edit a trace alternative. Specify the trace node to be used for this analysis and provide the appropriate data. Back in the Alternatives tab, choose the desired alternative from the Trace Alternative choice list.
6. Click the **Compute** button  .

Modeling for IDSE Compliance

Under the US EPA's Stage 2 Disinfectant by-product Rule, utilities are required to identify locations in their water distribution systems that are likely to have high concentrations of disinfectant by-products such as Trihalomethanes and Haloacetic acids. Both of these are associated with high water age.

In general the easiest and most beneficial way to comply with the EPA regulations is to conduct a system specific study and the most expedient way of doing this is to construct a calibrated, detailed extended period simulation model which can identify locations in the system with high water age. The details of the requirements for such a model are provided in “System Specific Study Using a Distribution System Hydraulic Model” available at:

http://www.epa.gov/safewater/disinfection/stage2/compliance.html

Bentley WaterCAD V8*i* can be used to comply with these regulations. Special tools have been added to assist in IDSE (Initial Distribution System Evaluation) studies. They are described below:

The utility must demonstrate that it has a well calibrated model.

From the regulations:

“A description of all calibration activities undertaken (or to be undertaken). This must include, if calibration is complete,

- A graph of predicted tank levels versus measured tank levels for the storage facility with the highest residence time in each pressure zone.
- A time series graph of water age results for the storage facility with the highest residence time in your system showing predictions for the entire EPS simulation period (i.e. from time zero until the time it takes for the model to reach a consistently repeating pattern of residence time).”

The graphing tools for displaying field observations alongside of model results have been improved for Select Upgrade 1 to make it easier to import field data using copy/paste commands from data sources such as spreadsheets and data base files.

To prepare graphs of field observations vs. model predictions for tanks level and system flows:

1. Create an EPS model run for the selected scenario and calculate it
2. Graph the property of interest
3. Click the small drop down arrow to the right of the third button on the graph options dialog and select **Observed Data**.
4. Import time series data field observations from SCDA systems, data loggers or manual data entries in the **Observed Data** dialog box. For more information on using the Observed Data dialog box, see Observed Data Dialog Box.

Field imported data will display as discrete points while model data will display as continuous cures. Once the data are imported, the user can view the comparison between field and model data to determine if the model is adequately calibrated or if additional work is required.

The utility's model used in an IDSE study must contain at least 50% of the pipe length in the real system and at least 75% of the pipes volume.

EPA regulations require:

- At least 50 percent of total pipe length in the distribution system.
- At least 75 percent of the pipe volume in the distribution system.
- All 12-inch diameter and larger pipes.
- All 8-inch diameter and larger pipes that connect pressure zones, mixing zones from different sources, storage facilities, major demand areas, pumps, and control valves, or are known or expected to be significant conveyors of water.
- All 6-inch diameter and larger pipes that connect remote areas of a distribution system to the main portion of the system or are known or expected to be significant conveyors of water.
- All storage facilities, with controls or settings applied to govern the open/closed status of the facility that reflect standard operations.
- All active pump stations, with realistic controls or settings applied to govern their on/off status that reflect standard operations.
- All active control valves or other system features that could significantly affect the flow of water through the distribution system (e.g., interconnections with other systems, pressure reducing valves between pressure zones).

A table providing information on the total length of pipe and volume of water in the model is available by clicking the **Report** menu and selecting **Pressure Pipe Inventory**. This inventory can be printed using the **Print Preview** button at the top of the display or copied to the clipboard for use in other documents by highlighting all columns and hitting **CTRL-C**. If the columns are so wide that the wrapping of the columns does not look attractive, the user can resize the column widths by grabbing the edges of the column and sliding the border to a desired position.

Below is an example of one such table:

Pressure Pipe

Inventory - EPS Age

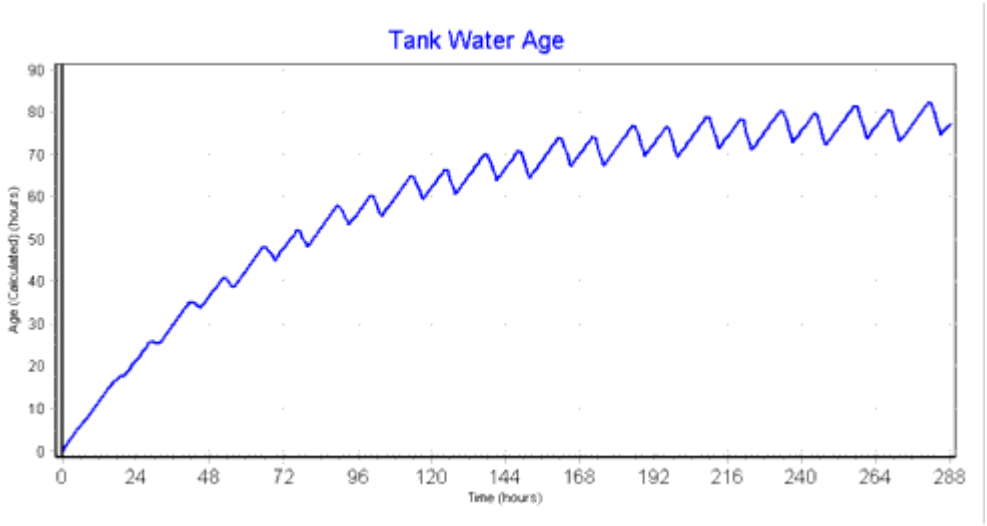
Diameter (in)	Ductile Iron (ft)	Cast iron (ft)	All Materials (ft)	Volume (gal)
1.0	45	0	45	
2.0	524	0	524	
3.0	299	0	299	
4.0	239,979	0	239,979	
6.0	1,007,785	0	1,007,785	
8.0	987,602	217	987,819	
9.0	39	0	39	
10.0	105,856	202	106,058	
12.0	363,000	268	363,268	
14.0	11,080	0	11,080	
16.0	125,446	0	125,446	
18.0	24,570	0	24,570	
20.0	4,330	0	4,330	
24.0	52,681	0	52,681	
30.0	11,636	0	11,636	
36.0	14,799	0	14,799	
42.0	6,220	0	6,220	
48.0	5,650	0	5,650	
All Diameters	2,961,540	687	2,962,227	1

idse.wtgBentley Systems, Inc. Haestad Methods Bentley WaterGENS V8 XM Edition [08
Solution Center

9/28/2006 27 Siemon Company Drive Suite 200 W
Page 1 of 1

The utility must be able to calculate, display and perform statistics on water age.

This is done by setting up an EPS run for a long duration (e.g. one week). The user then selects "Age" as the calculation type in the calculation options. The duration of the run should be sufficiently long such that the water age is not continuing to increase in the system at the end of the run. Selecting a good initial water age for the tanks can reduce the length of time required to reach a recurring pattern.



The user also needs the ability to calculate some statistics after an water age EPS run to include average water age at each element between hours a and b.

Average water age over the final 24 hours of an EPS run can be calculated using the **Post Calculation Processor** which can be found under the **Analysis** menu.

An example is shown below. To determine the average water age at all junctions for the last 24 hour of, for instance, a 144 hour run, set the following values:

- **Start time**: 120
- **Stop Time**: 144
- **Statistic Type**: Mean (Time weighted)
- **Results Property (field)**: Age (Calculated)
- **Output Property (field)**: AveAge
- **Operation**: Set

Then use the browser above the bottom pane to select all the junctions for which average age is to be calculated. It's recommended to create a selection set with the elements desired before entering the Post Calculation Processor.

Mean (Time weighted) takes into account the fact that not all time steps are of the same size.

Result property (field) means that the Age (Calculated) property (attribute) in the model will be used to determine the average age

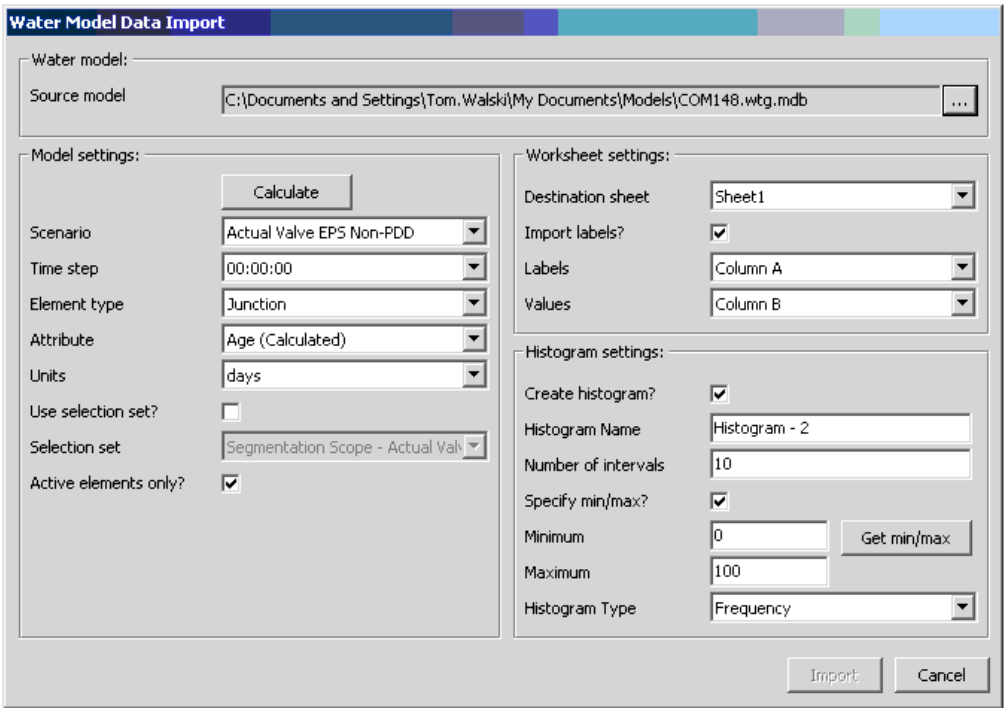
Output property (field) means that the resulting average age for each selected element will be placed in a user defined property (field) called AveAve. . Instructions on establishing a user defined output property (field) can be found under User Data Extensions Dialog Box.

Once the average age property has been determined for each element, it is possible to color, annotate, contour or perform other Bentley WaterCAD V8*i* operations on that property as with any other user defined property. The user can sort on this property (attribute) in FlexTables and determine the median. This helps the user comply with the portion of the regulation that states:

“Average residence time is the average age of water delivered to customers in a distribution system. Average residence time is not simply one-half the maximum residence time. Ideally, it should be a flow-weighted or population-weighted estimate. The model results for water age/DBP concentration can be used to determine the average residence time for your system. One option for doing this is to list the water age/DBP concentration results in ranked order for the entire system...”

A histogram plot sorts the water age results into groups and shows the percentage of nodes with water ages falling within the given range.

A histogram can be created using a WaterObjects.NET feature which enables the user to utilize the graphing capability of Excel to create the histogram. The user starts Excel and if Bentley WaterCAD V8*i* was loaded correctly, picks Bentley WaterCAD V8*i* > Import Data and will then enter a browser titled "Please select a Water Model." The user browses to the file corresponding to the model under consideration. The screen below opens. (If model results have not been calculated for the base scenario for the model the user will be asked if a calculation is desired.)



The fields in this dialog are described below for the case of creating a IDSE histogram.

- **Source model**: Full path name of model file
- **Scenario**: Name of Scenario to be imported
- **Time step**: Time step to be imported (value of average age is same for any time step)
- **Element type**: Average age is calculated at junctions
- **Property (attribute)**: Average age for this case but any property (attribute) can be imported
- **Use selection set**: check if user only wants to import a subset of junctions
- **Select set**: name of selection set if previous box is checked
- **Active elements only**: Check if inactive elements are to be ignored which is usually the case

The second group of settings refers to the Excel spreadsheet file:

- **Destination sheet**: Select existing sheet name
- **Import label**: Only needed if spreadsheet calculation involve knowing the element label
- **Labels**: Column in which labels are placed
- **Values**: Column in which values of selected property (attribute) are placed

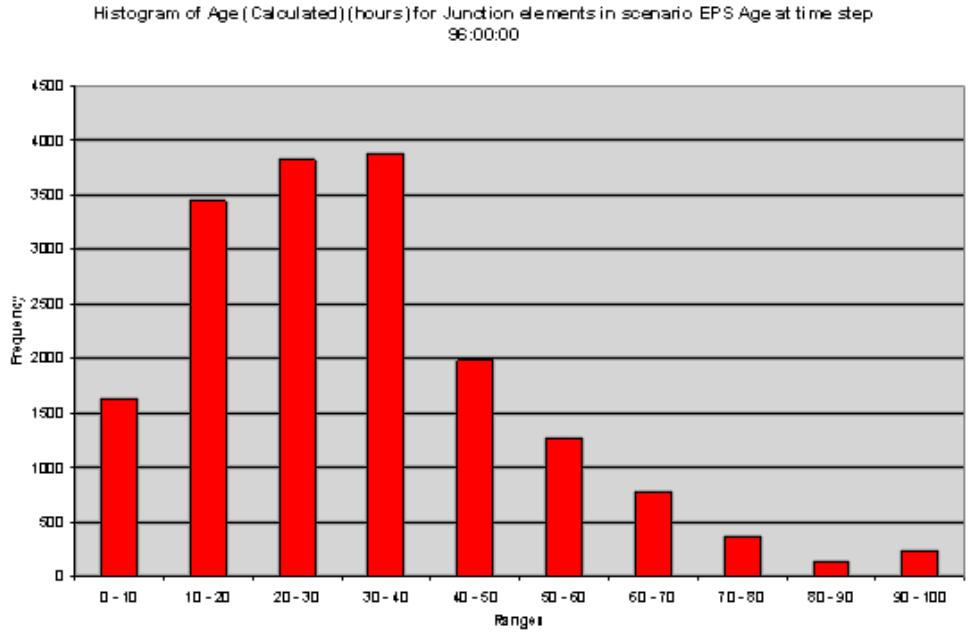
The next group of settings refers to the Histogram to be created:

- **Create histogram**: Check if histogram is desired
- **Histogram Name**: Name of worksheet in which histogram is placed
- **Number of intervals**: Number of bars in histogram
- **Specify min/max?**: If checked, user can override default values of ranges (recommended)
- **Minimum**: Minimum value of lowest interval
- **Maximum**: Maximum value of highest interval

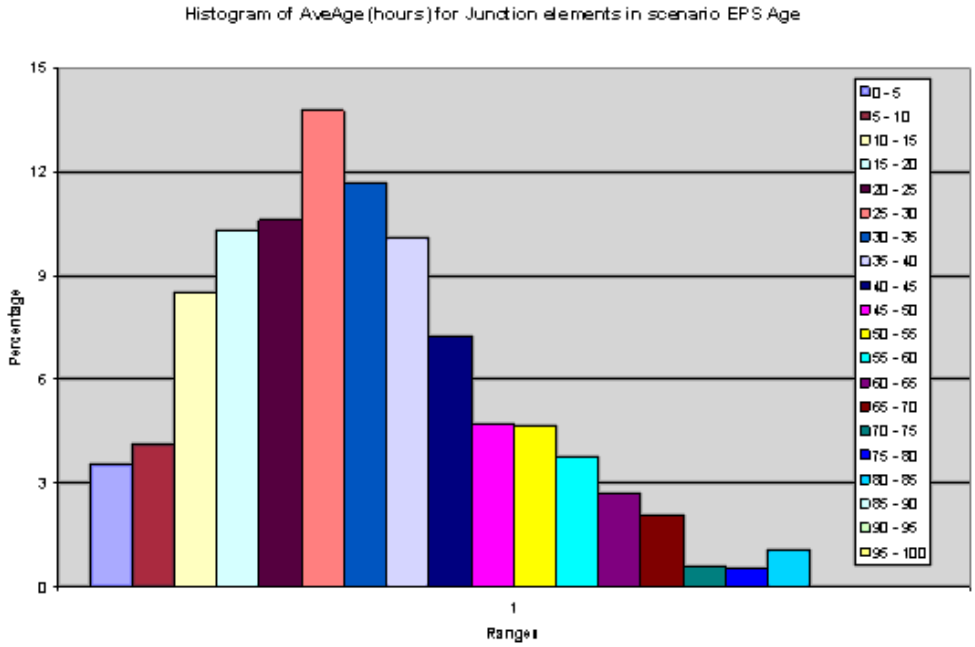
  Note: The "Get min/max" button will populate the Minimum and Maximum boxes and act as defaults. (The Minimum and maximum fields enable the user to create histograms which have round number a breakpoints instead of the default ranges which can be on the order of 18.34-24.67.)

- **Histogram type**: The vertical axis can be labeled by number of points (Junction elements) in each interval or percentage of point in each interval.

The **Import** button begins the importing of values from the model file into the spreadsheet and creates the histogram if that box is checked. The final histogram will look like the one below for 10 intervals with Frequency selected.

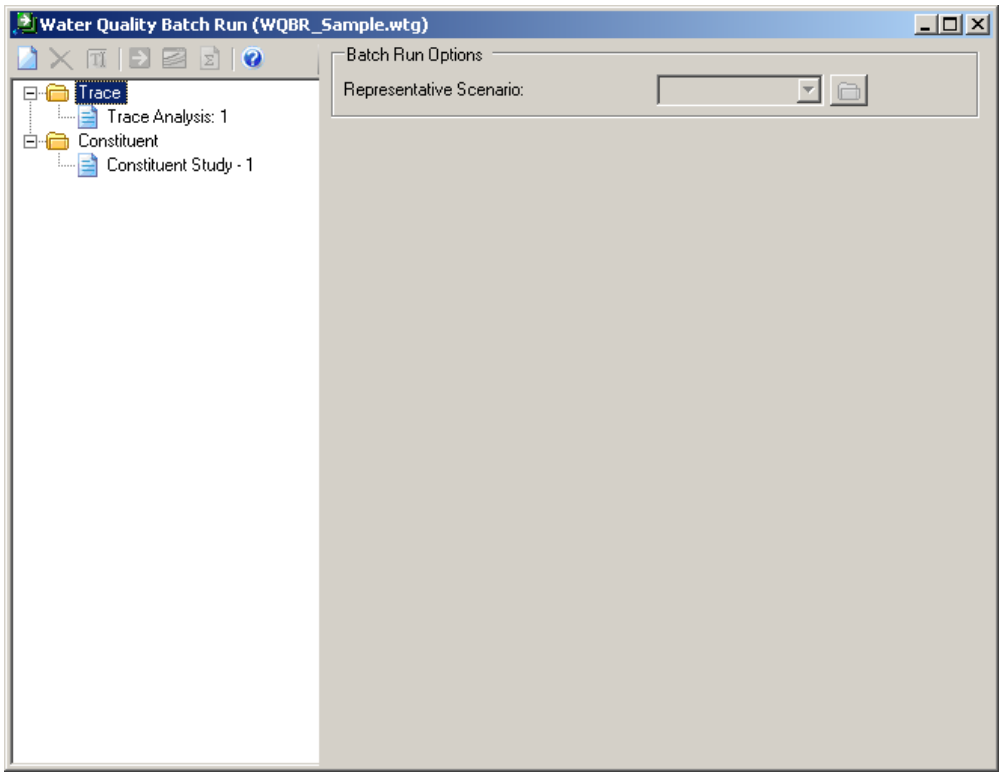


Here is an example with a large number of intervals and percentage of points as the axis.



Water Quality Batch Run

The Water Quality Batch Run feature allows you to perform a combined Water Quality Trace (see Trace Analysis) or Constituent (see Constituent Analysis) analysis. You can then use the provided reporting tools to graph the combined effects of each type of analysis on various parts of your system, or to review system-wide tabular statistics reports.



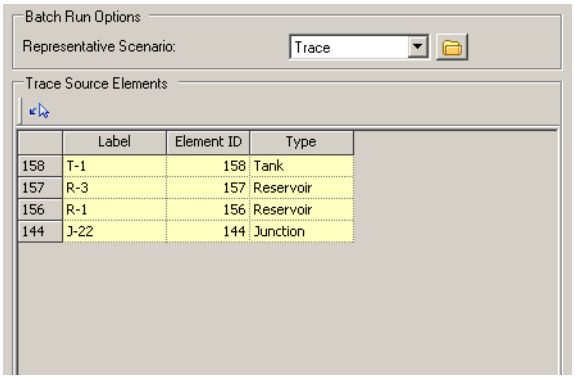
The dialog consists of a list pane on the left that displays all of the trace and constituent batch analyses in the project along with the following controls:

	**New**	Creates a new Trace or Constituent analysis. Highlight the folder for the type of analysis you want to create and click New.
	**Delete**	Deletes the trace or constituent analysis that is currently highlighted in the list pane.
	**Rename**	Allows you to enter a new label for the trace or constituent analysis that is currently highlighted in the list pane.
	**Compute**	Calculates the trace or constituent analysis that is currently highlighted in the list pane.
	**Graph**	After an analysis has been computed, this button opens the Graph Elemenrt Selection dialog, allowing you to select the elements to graph.
	**Statistics Table**	Opens the Water Quality Batch Run Statistics Table dialog, which displays statistics for each node and pipe.
	**Help**	Opens the online help.

The controls available in the right side of the dialog change depending on whether a Trace or Constituent analysis is highlighted in the list pane.

Trace Analysis

When a Trace analysis is highlighted in the list pane the right side of the dialog will look like this:



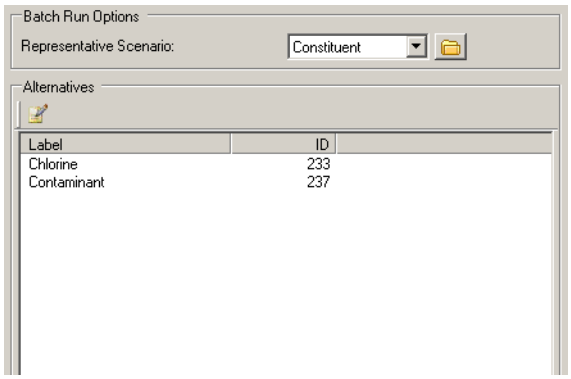
The following controls are available:

- **Representative Scenario**: Choose the scenario that represents the state of the system you would like to analyze. Select the scenario from the list or click the Scenarios button to open the Scenarios dialog and select the desired scenario from the tree view.
- **Select Elements**: Click this button to return to the drawing pane to select the trace source elements that will be used for the analysis.
- **Source Element Table**: This table lists the selected trace source elements that will be used in the analysis. The element Label, Element ID, and Element Type are displayed for each trace source element.

Note: If water passes through an inflow node, even it if came from another source, it is treated as having originated at this source. Therefore, the trace will essentially be double counted. The solution to this problem, when inflow source nodes are involved is to place them on a short stub where water from other sources will not flow through them.

Constituent Analysis

When a Constituent analysis is highlighted in the list pane the right side of the dialog will look like this:

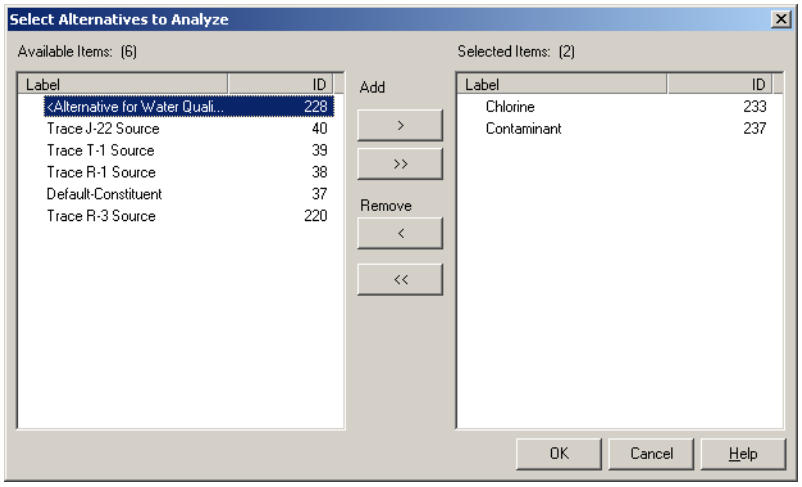


The following controls are available:

- **Representative Scenario**: Choose the scenario that represents the state of the system you would like to analyze. Select the scenario from the list or click the Scenarios button  to open the Scenarios dialog and select the desired scenario from the tree view.
- **Select Alternatives to Analyze**: Opens the Select Alternatives to Analyze dialog (see Select Alternatives to Analyze Dialog Box), allowing you to choose which alternatives will be used in the Constituent analysis.
- **Alternatives Pane**: This area displays the alternatives to be analyzed.

Select Alternatives to Analyze Dialog Box

This dialog allows you to select the alternatives to be used in a Constituent Water Quality Batch Run.



The following controls are available:

Available Items

Contains all of the alternatives that are available for the constituent analysis. The Available Items list is located on the left side of the dialog box.

Selected Items

Contains all of the alternatives that will be included in the constituent analysis. The Selected Items list is located on the left side of the dialog box.

Add and Remove Buttons

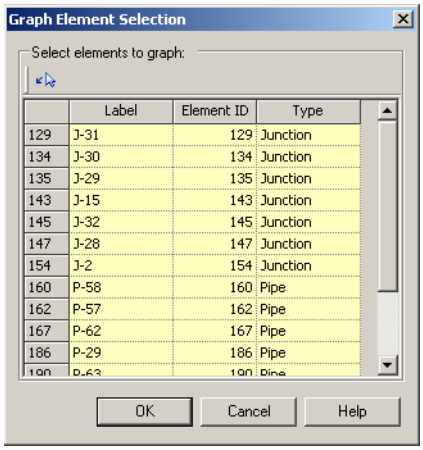


Select or clear alternatives to be used in the table. The Add and Remove buttons are located in the center of the dialog box.

- [>] Adds the selected alternative from the Available Items list to the Selected Items list.
- [>>] Adds all of the alternatives in the Available Items list to the Selected Items list.
- [<] Removes the selected alternative from the Selected Items list.
- [<<] Removes all alternatives from the Selected Items list.

Graph Element Selection Dialog Box

This dialog allows you to select the elements for which results will be displayed after a Water Quality Batch Run Trace analysis.

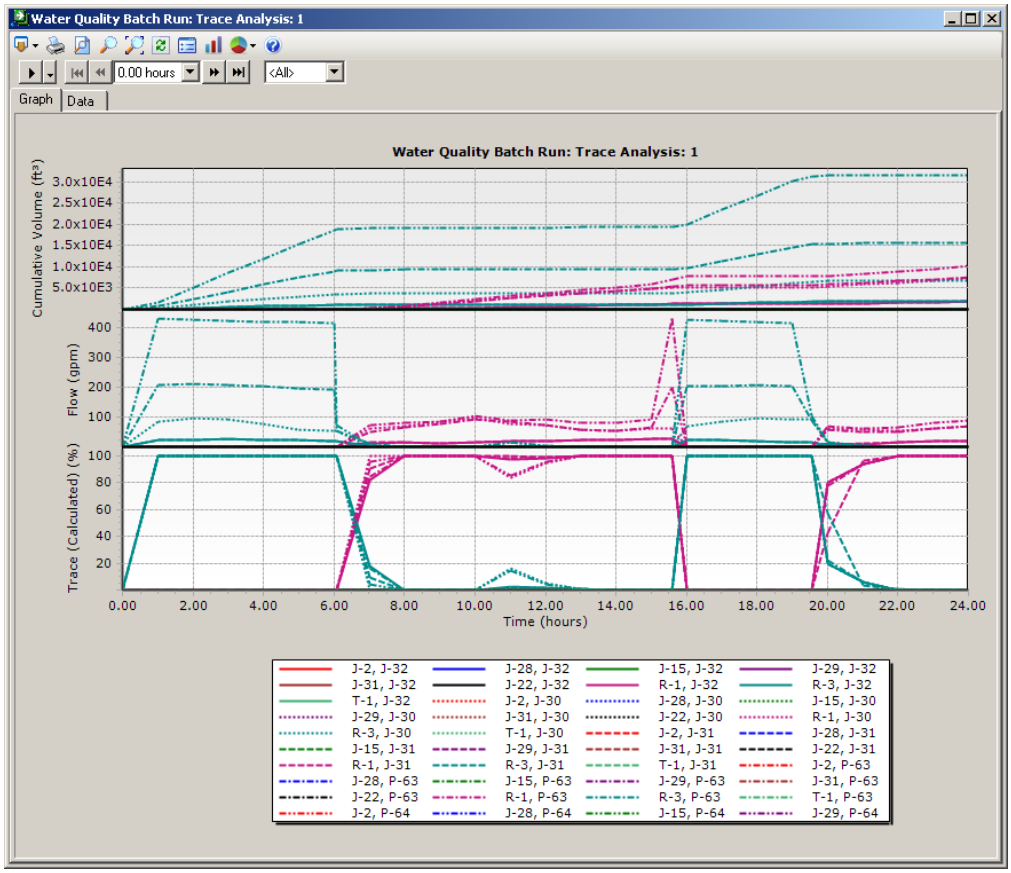


Click the Select From Drawing button  to return to the drawing pane, allowing you to select the elements that will be included in the graph. Pipes and Nodes can both be selected.

Once elements have been selected, the dialog will display a table containing the Label, ID, and Element Type of all of the selected elements. Click OK to open the graph.

Graph Viewer Dialog Box

Using the Graph Viewer dialog box you can view the results of the water quality batch run.



The following controls are available:

Graph Tab

Export



Allows you to export the Graph Image and/or the Graph Data.

Print



Prints the current view in the graph display pane.

Print Preview



Opens the Print Preview dialog box to view the current image and change the print information.

Activate/ Deactivate Zoom



Zooms in on a section of the graph. When the tool is toggled on, you can zoom in on any area of the graph by clicking on the chart to the left of the area to be zoomed, holding the mouse button, then dragging the mouse to the right (or the opposite extent of the area to be magnified) and releasing the mouse button when the area to be zoomed has been defined.
To zoom back out, click and hold the mouse button, drag the mouse in the opposite direction (right to left), and release the mouse button.

Zoom Extents



Zooms out so that the entire graph is displayed.

Refresh



Updates the graph to reflect any changes in the underlying data.

Graph Options



Allows you to modify the graph settings. See Chart Options Dialog Box for more details.

Bar Chart



Displays the graph data as a series of bar charts

Pie Charts



Displays the graph data for a selected junction as a series of pie charts. When this button is clicked a menu containing the available junctions will appear, allowing you to select the desired one.

Help Opens the online help.



Time (VCR) Controls Evaluate plots over time.

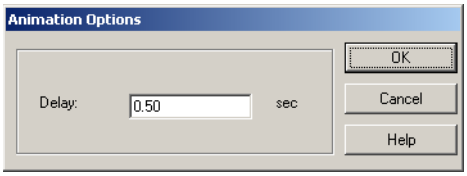
- If you click Play, a vertical line moves across the graph and the Time field increments. Click the down arrow button next to the play button to open the Animation Options dialog.
- If you click Go to Start, the Time resets to zero and the vertical line that marks time resets to the left edge of the Graph display.
- If you click Step Backwards the time goes back one increment.
- You can seelct a specific time using the time display pulldown menu.
- If you click Step Forwards the time moves forward one increment.
- If you click Go to End, the Time goes to the last time step and the vertical line that marks time resets to the right edge of the Graph display.
- Change the increment of the animation time step using the right-most menu.

Data Tab

Data Table The Data tab displays the data that make up the graphs. If there is more than one item plotted, the data for each plot is provided.
You can copy and paste the data from this tab to the clipboard for use in other applications, such as Microsoft Excel.
To select an entire column or row, click the column or row heading. To select the entire contents of the Data tab, click the heading cell in the top-left corner of the tab. Use <Ctrl+C> and <Ctrl+V> to paste your data. The column and row headings are not copied.

Animation Options Dialog Box

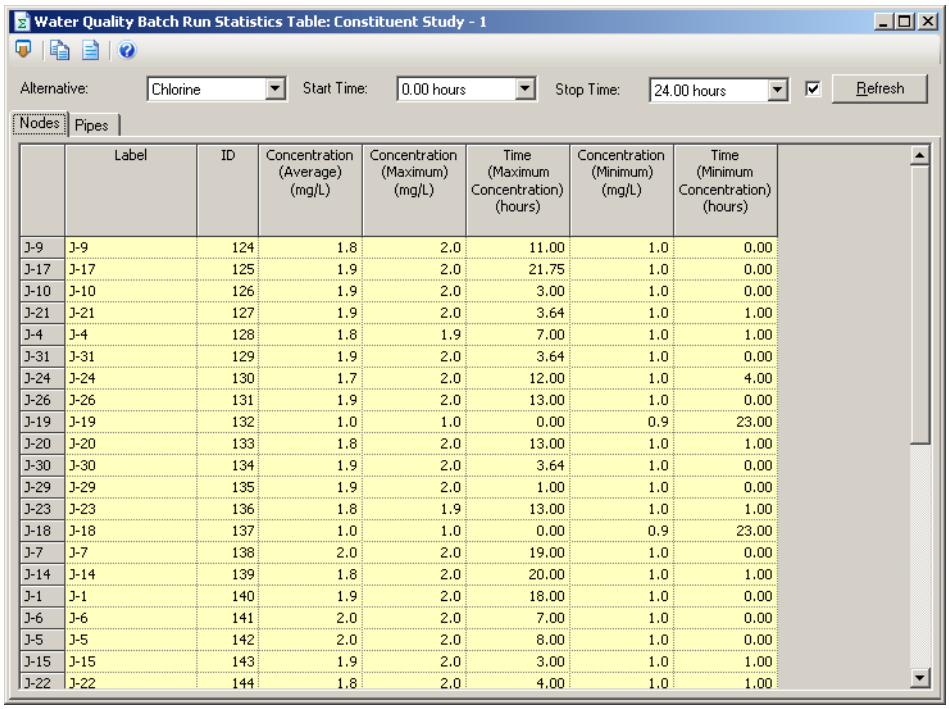
This dialog allows you to set the animation delay for the Graph Viewer Dialog Box.



The Delay value is the amount of time between frames of animation when the Play button in the Graph Viewer Dialog Box is clicked.

Statistics Table Dialog Box

This dialog displays calculated statistics for a water quality batch run.



	Label	ID	Concentration (Average) (mg/L)	Concentration (Maximum) (mg/L)	Time (Maximum Concentration) (hours)	Concentration (Minimum) (mg/L)	Time (Minimum Concentration) (hours)
J-9	J-9	124	1.8	2.0	11.00	1.0	0.00
J-17	J-17	125	1.9	2.0	21.75	1.0	0.00
J-10	J-10	126	1.9	2.0	3.00	1.0	0.00
J-21	J-21	127	1.9	2.0	3.64	1.0	1.00
J-4	J-4	128	1.8	1.9	7.00	1.0	1.00
J-31	J-31	129	1.9	2.0	3.64	1.0	0.00
J-24	J-24	130	1.7	2.0	12.00	1.0	4.00
J-26	J-26	131	1.9	2.0	13.00	1.0	0.00
J-19	J-19	132	1.0	1.0	0.00	0.9	23.00
J-20	J-20	133	1.8	2.0	13.00	1.0	1.00
J-30	J-30	134	1.9	2.0	3.64	1.0	0.00
J-29	J-29	135	1.9	2.0	1.00	1.0	0.00
J-23	J-23	136	1.8	1.9	13.00	1.0	1.00
J-18	J-18	137	1.0	1.0	0.00	0.9	23.00
J-7	J-7	138	2.0	2.0	19.00	1.0	0.00
J-14	J-14	139	1.8	2.0	20.00	1.0	1.00
J-1	J-1	140	1.9	2.0	18.00	1.0	0.00
J-6	J-6	141	2.0	2.0	7.00	1.0	0.00
J-5	J-5	142	2.0	2.0	8.00	1.0	0.00
J-15	J-15	143	1.9	2.0	3.00	1.0	1.00
J-22	J-22	144	1.8	2.0	4.00	1.0	1.00

The dialog shows the statistics in a tabular format divided into a Nodes tab and a Pipes tab, along with the following controls:

-  **Export for Color Coding**: Opens the Export for Color Coding dialog, allowing you to specify the scenario and fields to export for use with the color coding feeature.

-  **Copy**: Copies the statistic table to the clipboard for use in an external application.
-  **Report**: Generates a preformatted report containing the data in the statistics table.
-  **Help**: Opens the online help.
- **Alternative/Trace Node**: For Trace Analysis, this field allows you to select the Trace Node for which statistics are calculated. For Constituent Analysis, this field allows you to select the constituent scenario.
- **Start Time**: Allows you to select the start time for the statistics calculations.
- **Stop Time**: Allows you to select the stop time for the statistics calculations.
- **Auto-Refresh:** When this box is checked, the results in the table will automatically be refreshed when changes are made.
- **Refresh**: Refreshes the results displayed in the table to reflect changes made in this dialog.

Multi-Species Water Quality Analysis

Using constituent analysis, a user can track a single constituent through a water distribution system, provided the constituent behaved according to one of the kinetic models (e.g. first order decay). However, some constituents cannot be modeled this way because they are involved in significant multi-species reactions or their kinetics do not fit one of the existing models. To handle these cases, WaterGEMS provides Multi-species Analysis, based on the EPANET-MSX model with a WaterGEMS user interface.

In order to use multi-species analysis in WaterGEMS, in addition to the usual EPS input, the user must provide:

- Generic options such as water quality time step size
- Species name and mass units
- Reaction rate coefficients either globally or pipe by pipe
- Rate and equilibrium equations for pipes and tanks
- Sources of each species and relevant time patterns
- Initial concentration of species
- Reporting options

These values are entered as text in the Multi-species Analysis Setup and Multi-Species Model Configuration using the syntax described below. For a multi-species scenario, the calculation options are set to Multi-species analysis (see Multi-species Analysis Setup on page 10-805 and Multi-species Model Configuration on page 10-815). The distinction between Setup and Configuration is that setup contains values describing kinetics and can be used with any model while configuration contains values for sources and overrides that are specific to the current model.

The data for a multi-species analysis is divided into sections as described below:

[TITLE] adds a descriptive title to the data set

[OPTIONS] sets the values of computational options

[SPECIES] names the chemical species being analyzed

[COEFFICIENTS] names the parameters and constants used in chemical rate and equilibrium expressions

[TERMS] defines intermediate terms used in chemical rate and equilibrium expressions

[PIPES] supplies the rate and equilibrium dynamics in pipes expressions that govern species

[TANKS] supplies the rate and equilibrium dynamics in storage tanks expressions that govern species

[SOURCES] identifies input sources (i.e., boundary conditions) for selected species

[QUALITY] supplies initial conditions for selected species throughout the network

[PARAMETERS] allows parameter values to be assigned on a pipe by pipe basis

[PATTERNS] defines time patterns used with input sources

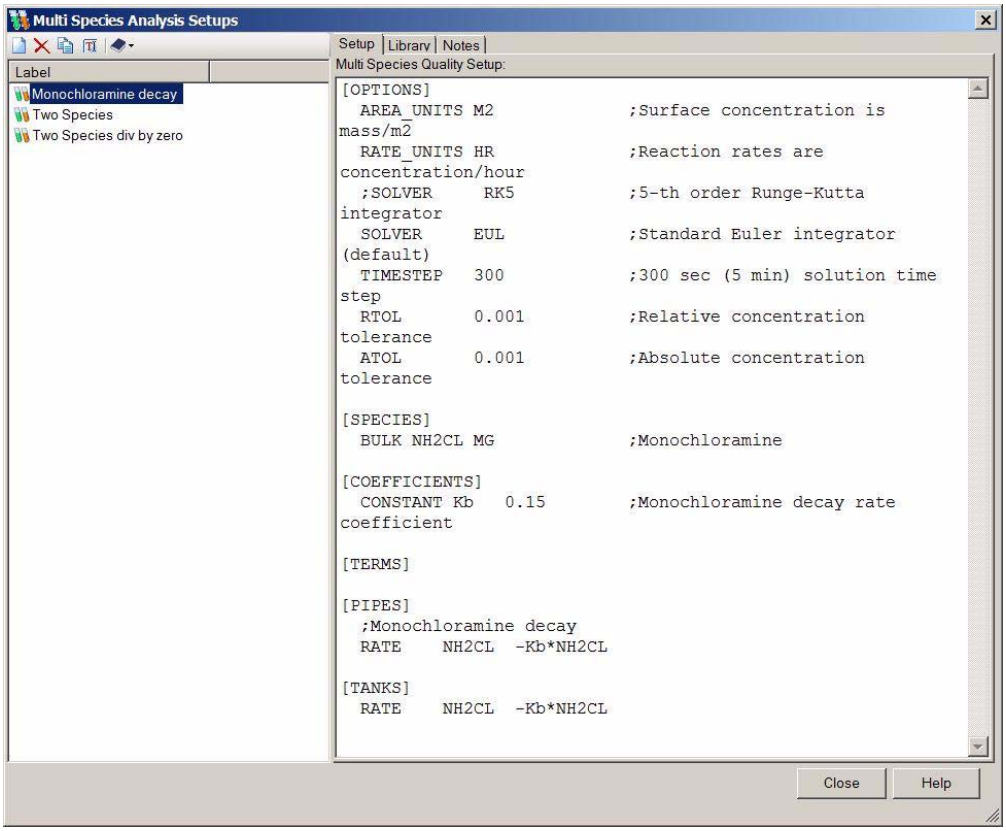
[REPORT] specifies reporting options

The values for most of these sections are entered under model setup while the sections on Sources, Quality and Parameters are model specific and are described under model configuration.

This help topic provides the steps to run a multi-species analysis in WaterGEMS. For more detailed background and theory, the user is referred to the EPANET-MSX manual.

Setup

To enter the data for a multi-species run, the user must create a multi-species setup by selecting Component > Multi-Species Analysis Setups. Pick New to create a setup.

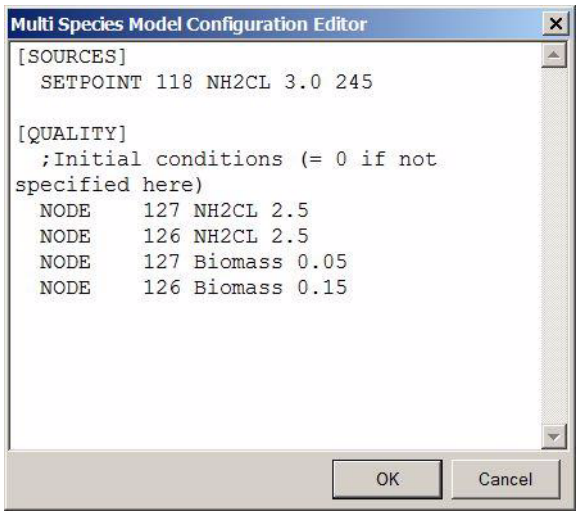


The user can also Delete, Copy, Rename or Import/Export setups from a library.

The exact syntax for the setup is provided in help topic Multi-species Analysis Setup on page 10-805.

Configuration

Within the calculation options, the user identifies model specific properties for a Multi-species analysis.



The exact syntax for the setup is provided in help topic Multi-species Analysis Setup on page 10-805.

Calculation Options

To run a multi-species analysis, the user must set the Calculation Type in the Calculation Options to Multi-Species Analysis.

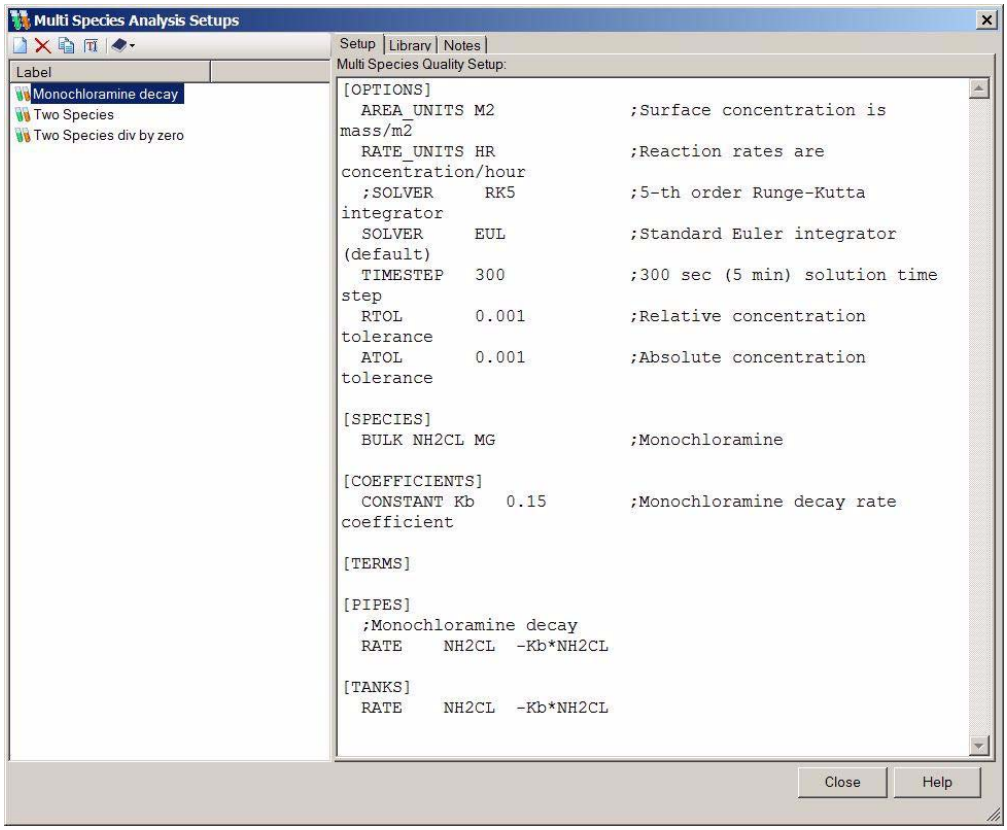
This opens up a category at the bottom of the calculation options where the user Multi-Species Setup to use from the list of those created above and the Model Configuration for this model.

Results

The results of a multi-species analysis run are available using the property grid, graphing, annotation, color coding just as any other WaterGEMS results.

Multi-species Analysis Setup

Multi-species Analysis Setup can be reached using Component > Multi-Species Analysis Setups. Once a setup has been created, it can be referenced in the calculation options for any scenario.



There are nine sections in specifying a multi-species setup. Information after a semicolon is considered a comment and is ignored during execution. Each of the sections is described below.

[TITLE]

Purpose: Attaches a descriptive title to the problem being analyzed.

Format: A single line of text.

Remarks: The [TITLE] section is optional.

[OPTIONS]

Purpose: Defines various simulation options.

Formats:

AREA_UNITS FT2/M2/CM2

RATE_UNITS SEC/MIN/HR/DAY

SOLVER EUL/RK5/ROS2

COUPLING FULL/NONE

TIMESTEP seconds

ATOL value

RTOL value

Definitions:

AREA_UNITS sets the units used to express pipe wall surface area where:

FT2 = square feet

M2 = square meters

CM2 = square centimeters

The default is FT2.

RATE_UNITS is the units in which all reaction rate terms are expressed. The default units are hours (HR). (Note that in the EPANET-MSX manual, this keyword is incorrectly labeled TIME_UNITS."

SOLVER is the choice of numerical integration method used to solve the reaction system where:

EUL = standard Euler integrator

RK5 = Runge-Kutta 5th order integrator

ROS2 = 2nd order Rosenbrock integrator. The default solver is EUL.

COUPLING determines to what degree the solution of any algebraic equilibrium equations is coupled to the integration of the reaction rate equations. If coupling is NONE then the solution to the algebraic equations is only updated at the end of each integration time step. With FULL coupling, the updating is done whenever a new set of values for the rate-dependent variables in the reaction rate expressions

is computed. This can occur at several intermediate times during the normal integration time step when using the RK5 and ROS2 integration methods. Thus the FULL coupling option is more accurate, but can require significantly more computation time. The default is FULL coupling.

TIMESTEP is the time step, in seconds, used to integrate the reaction system. The default time step is 300 seconds (5 minutes).

ATOL is the default absolute tolerance used to determine when two concentration levels of a species are the same. It applies to all species included in the model. Different values for individual species can be set in the [SPECIES] section of the input (see below). If no ATOL option is specified then it defaults to 0.01 (regardless of species concentration units). RTOL is a default relative accuracy level on a species' concentration used to adjust time steps in the RK5 and ROS2 integration methods. It applies to all species included in the model. Different values for individual species can be set in the [SPECIES] section of the input (see below). If no RTOL option is specified then it defaults to 0.001.

[SPECIES]

Purpose: Defines each chemical species being simulated.

Formats:

BULK name units (Atol Rtol)

WALL name units (Atol Rtol)

Definitions:

name: species name

units: species mass units

Atol: optional absolute tolerance that overrides the global value set in the [OPTIONS] section

Rtol: optional relative tolerance that overrides the global value set in the [OPTIONS] Section

Remarks:

The first format is used to define a bulk water (i.e., dissolved) species while the second is used for species attached (i.e., adsorbed) to the pipe wall.

Bulk species are measured in concentration units of mass units per liter while wall species are measured in mass units per unit area.

Any units can be used to represent species mass. The user is responsible for including any necessary unit conversion factors when specifying chemical reaction and equilibrium expressions that involve several species with different mass units.

Values for both Atol and Rtol must be provided to override the default tolerances.

Examples:

[SPECIES]

;Bulk chlorine in mg/L with default tolerances

BULK CL2 MG

;Bulk biomass in ug/L with specific tolerances

BULK Xb UG 0.0001 0.01

;Attached biomass in ug/area with specific tolerances

WALL Xa UG 0.0001 0.01

[COEFFICIENTS]

Purpose: Defines parameters and constants that are used in the reaction/equilibrium chemistry model.

Formats:

PARAMETER: name value

CONSTANT: name value

Definitions:

name: coefficient's identifying name

value: global value of the coefficient.

Remarks:

PARAMETER is a coefficient whose value can be changed on a pipe by pipe (or tank by tank basis (see the [PARAMETERS] section below) while a CONSTANT coefficient maintains same value throughout the pipe network.

Examples:

[COEFFICIENTS]

;Kb can vary by pipe

PARAMETER Kb 0.1

;Kw is fixed for all pipes

CONSTANT Kw 1.5

[TERMS]

Purpose: Defines mathematical expressions that are used as intermediate terms in the expressions for the chemical reaction/equilibrium model.

Formats:

termID: expression

Definitions:

termID: identifying name given to the term

expression: any well-formed mathematical expression involving species, parameters, constants, hydraulic variables or other terms.

Remarks:

Terms can be used to simplify reaction rate or equilibrium expressions that would otherwise be unwieldy to write all on one line or have the same terms repeated in several different rate/equilibrium equations. The definition and use of TERMS, when those terms are common and appear in multiple rate or equilibrium expressions, may speed computation because the common term expression requires only one evaluation.

Hydraulic variables consist of the following reserved names:

D: pipe diameter (feet or meters)

Q: pipe flow rate (flow units)

U: pipe flow velocity (ft/sec or m/sec)

Re: flow Reynolds number

Us: pipe shear velocity (ft/sec or m/sec)

Ff: Darcy-Weisbach friction factor

Av: Surface area per unit volume (area units/L)

Examples:

[TERMS]

;A mass transfer coefficient

Kf 1.2e-4*Re^0.88/D

;A reaction term

a1 k1*HOCL*NH3

[PIPES]

Purpose: Supplies the rate and equilibrium expressions that govern species dynamics in pipes.

Formats:

EQUIL: specieID expression

RATE: specieID expression

FORMULA: specieID expression

Definitions:

specieID: a species identifier

expression: any well-formed mathematical expression involving species, parameters, constants, hydraulic variables or terms.

Remarks:

There should be one expression supplied for each species defined in the model.

The allowable hydraulic variables were defined above in the description of the [TERMS] section.

The EQUIL format is used for equilibrium expressions where it is assumed that the expression supplied is being equated to zero. Thus formally there is no need to supply the name of a species, but requiring one encourages the user to make sure that all species are accounted for.

The RATE format is used to supply the equation that expresses the rate of change of the given species with respect to time as a function of the other species in the model.

The FORMULA format is used when the concentration of the named species is a simple function of the remaining species.

Examples:

```
[PIPES]
;Bulk chlorine decay
RATE CL2 -Kb*CL2
;Adsorption equilibrium between Cb in bulk and Cw on wall
EQUIL Cw Cmax*k*Cb / (1 + k*Cb) - Cw
;Conversion between biomass (X) and cell numbers (N)
FORMULA N log10(X*1.0e6)
;Bulk C formation plus non-equilibrium sorption between C and Cs
;Using hydraulic variable Av [Area-Units/Liter]
RATE C K*C - Av*(K1*(Smax-Cs)*C - K2*Cs)
;Equivalent sorption model, using 1/hydraulic radius = 4/D
;Assumes area units are FT2 and diameter in FT
;CFPL is TERM equal to FT3/Liter, thus (4*CFPL/D) == Av
RATE C K*C - (4*CFPL/D)*(K1*(Smax-Cs)*C - K2*Cs)
```

[TANKS]

Purpose: Supplies the rate and equilibrium expressions that govern species dynamics in storage tanks.

Formats:

EQUIL: specieID expression

RATE: specieID expression

FORMULA: specieID expression

Definitions:

specieID: a species identifier

expression: any well-formed mathematical expression involving species, parameters, constants, or terms.

Remarks:

[TANKS] section is always required when a model contains both bulk and wall species, even when there are no tanks in the pipe network. If the model contains only bulk species, then this section can be omitted if the reaction expressions within tanks are the same as within pipes.

There should be one expression supplied for each bulk species defined in the model. By definition, wall species do not exist within tanks.

Hydraulic variables are associated only with pipes and cannot appear in tank expressions.

The EQUIL format is used for equilibrium expressions where it is assumed that the expression supplied is being equated to zero. Thus, formally, there is no need to supply the name of a species but doing so allows one to make sure that all species are accounted for.

The RATE format is used to supply the equation that expresses the rate of change of the given species with respect to time as a function of the other species in the model.

The FORMULA format is used when the concentration of the named species is a simple function of the remaining species.

Examples: See the examples listed for the [PIPES] section.

[PATTERNS]

Purpose: Defines time patterns used to vary external source strength over time.

Formats:

name: multiplier multiplier ...

Definitions:

name: an identifier assigned to the time pattern

multiplier: a multiplier used to adjust a baseline value

Remarks:

Use one or more lines for each time pattern included in the model.

If extending the list of multipliers to another line remember to begin the line with the pattern name.

All patterns share the same time period interval as defined in the [TIMES] section of the EPANET input file being used in conjunction with the EPANET-MSX input file.

Each pattern can have a different number of time periods.

When the simulation time exceeds the pattern length the pattern wraps around to its first period.

Examples:

[PATTERNS]

;A 3-hour injection pattern over a 24 hour period

;(assuming a 1-hour pattern time interval is in use)

P1 0.0 0.0 0.0 0.0 1.0 1.0

P1 1.0 0.0 0.0 0.0 0.0 0.0

P1 0.0 0.0 0.0 0.0 0.0 0.0

P1 0.0 0.0 0.0 0.0 0.0 0.0

[REPORT]

Purpose: Describes the contents of the output report produced from a simulation.

Formats:

NODES ALL

NODES: node1 node2 ...

LINKS ALL

LINKS: link1 link2 ...

SPECIE:S speciesID YES/NO (precision)

STATISTIC NONE / AVERAGE / MINIMUM / MAXIMUM / RANGE

FILE: filename

PAGESIZE: lines

Definitions:

node1,

node2, etc. a list of nodes whose results are to be reported

link1,

link2, etc. a list of links whose results are to be reported

specieID the name of a species to be reported on precision number of decimal places used to report a species' concentration filename the name of a file to which the report will be written lines the number of lines per page to use in the report.

Remarks:

Use as many NODES and LINKS lines as it takes to specify which locations get reported.

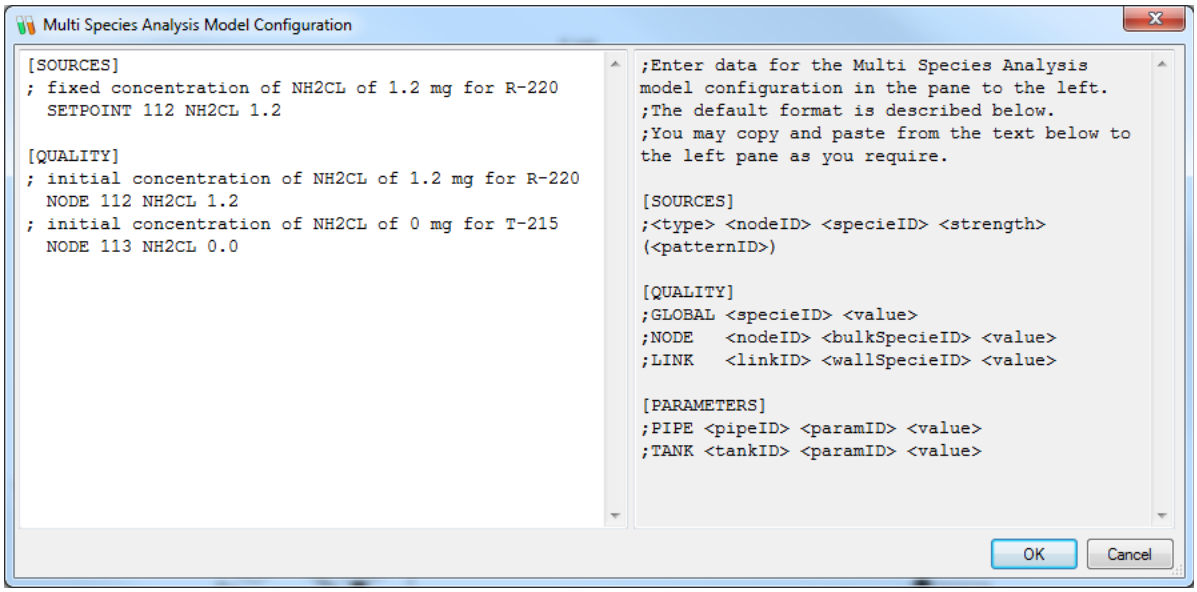
The default is not to report results for any nodes or links.

Use the SPECIES line to specify which species get reported and at what precision. The default is to report all species at two decimal places of precision.

The FILE line is used to have the report written to a specific file. If not provided the report will be written to the same file used for reporting program errors and simulation status.

Multi-species Model Configuration

Multi-species model configuration consists of data for multi-species analysis that is dependent on the model (as opposed to setup data which is independent of the model). The configuration is entered under calculation options. Picking the ellipse next to Model Configuration opens the dialog where model configuration data are specified. The semi-colon is used to designate a comment. Text is not case sensitive (e.g. SOLVER and solver are treated the same), but coefficients are (e.g. Kb and kb are different coefficients).



Descriptions of the individual sections of the data are given below:

[SOURCES]

Purpose: Defines the locations where external sources of particular species enter the pipe network.

Formats:

sourceType nodeID specieID strength (patternID)

Definitions:

sourceType: either MASS, CONCEN, FLOWPACED, or SETPOINT

nodeID: the ID label of the network node where the source is located

specieID: a bulk species identifier

strength: the baseline mass inflow rate (mass/minute) for MASS sources or concentration (mass/L) for all other source types

patternID: the name of an optional time pattern that is used to vary the source strength over time.

Remarks:

Use one line for each species that has non-zero source strength.

Only bulk species can enter the pipe network, not wall species.

The definitions of the different source types conform to those used in the original EPANET program are as follows:

MASS type source adds a specific mass of species per unit of time to the total flow entering the source node from all connecting pipes.

CONCEN type source sets the concentration of the species in any external source inflow (i.e., a negative demand) entering the node. The external inflow must be established as part of the hydraulic specification of the network model.

FLOWPACED type source adds a specific concentration to the concentration that results when all inflows to the source node from its connecting pipes are mixed together.

SETPOINT type source fixes the concentration leaving the source node to a specific level as long as the mixture concentration of flows from all connecting pipes entering the node is less than the set point concentration.

If a time pattern is supplied for the source, it must be one defined in the [PATTERNS] section of the MSX file, not a pattern from the associated EPANET input file.

Examples:

[SOURCES]

;Inject 6.5 mg/minute of chemical X into Node N1
;over the period of time defined by pattern PAT1
MASS N1 X 6.5 PAT1
;Maintain a 1.0 mg/L level of chlorine at node N100
SETPOINT N100 CL2 1.0

[QUALITY]

Purpose: Specifies the initial concentrations of species throughout the pipe network.

Formats:

GLOBAL: specieID concen
NODE: nodeID specieID concen
LINK: linkID specieID concen

Definitions:

specieID: a species identifier
nodeID: a network node ID label
linkID: a network link ID label
concen: a species concentration

Remarks:

Use as many lines as necessary to define a network's initial condition.

Use the GLOBAL format to set the same initial concentration at all nodes (for bulk species) or within all pipes (for wall species).

Use the NODE format to set an initial concentration of a bulk species at a particular node.

Use the LINK format to set an initial concentration of a wall species within a particular pipe.

The initial concentration of a bulk species within a pipe is assumed equal to the initial concentration at the downstream node of the pipe.

All initial concentrations are assumed to be zero unless otherwise specified in this section.

Models with equilibrium equations will require that reasonable initial conditions be set so that the equations are solvable. For example, if they contain a ratio of species concentrations then a divide by zero condition will occur if all initial concentrations are set to zero.

Examples:

[QUALITY]

;Set concentration of bulk species Cb to 1.0 at all nodes

GLOBAL Cb 1.0

;Override above condition for node N100

NODE N100 Cb 0.5

[PARAMETERS]

Purpose: Defines values for specific reaction rate parameters on a pipe by pipe or tank by tank basis.

Formats:

PIPE: pipeID paramID value

TANK: tankID paramID value

Definitions:

pipeID: the ID label of a pipe link in the network

tankID: the ID label of a tank node in the network

paramID: the name of one of the reaction rate parameters listed in the[COEFFICIENTS] section

value: the parameter's value used for the specified pipe or tank.

Remarks:

Use one line for each pipe or tank whose parameter value is different than the global value

Criticality Analysis

Bentley WaterCAD V8*i* provides the user with a unique and flexible tool to evaluate a water distribution system and identify the most critical elements. The user is allowed to shut down individual segments of the system and the results on system performance are determined. Rather than having to do this through the scenario manager, the user will be able to simulate a set of outages in a single run. This set can vary from a single element to each possible segment in a large system.

Bentley WaterCAD V8*i* reports a variety of indicators for each outage during a criticality analysis. Depending on the type of run, criticality analysis can report the flow shortfall, volume shortfall or pressure shortfall in the distribution system for each segment outage.

Before being able to conduct a criticality analysis, Bentley WaterCAD V8*i* must identify the segments to be removed from service. Once the options have been set in a Criticality Studies level of the Segmentation and Criticality manager, the user decides which scenario is to be used for the analysis and sets the rules for use of valving in the options tab.

In order to use criticality analysis, the user must make several decisions on the way that Bentley WaterCAD V8*i* performs the analysis. Each of those is described below.

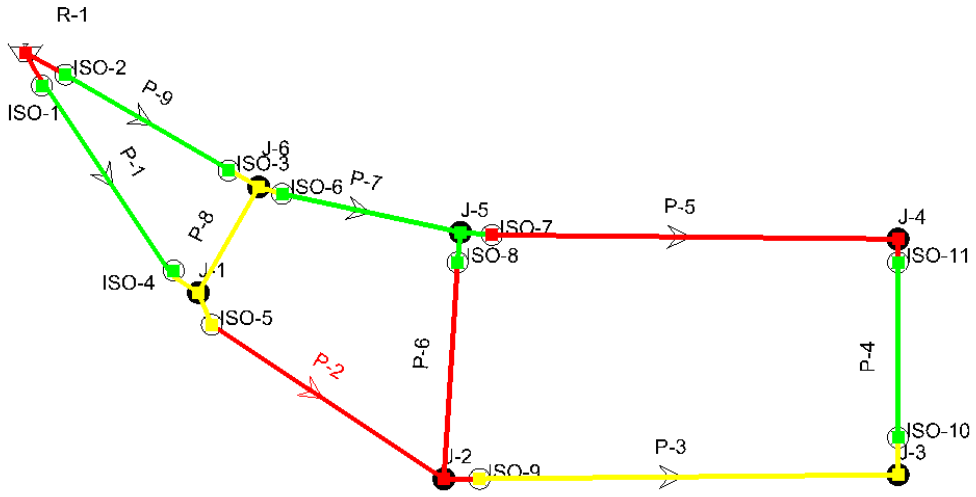
Segments vs. Individual Pipes

When a distribution system outage occurs, the portion of the system that is taken out of service is referred to as a “segment”. A “segment” or “Network segment” is the smallest portion of a distribution system that can be isolated by valving.

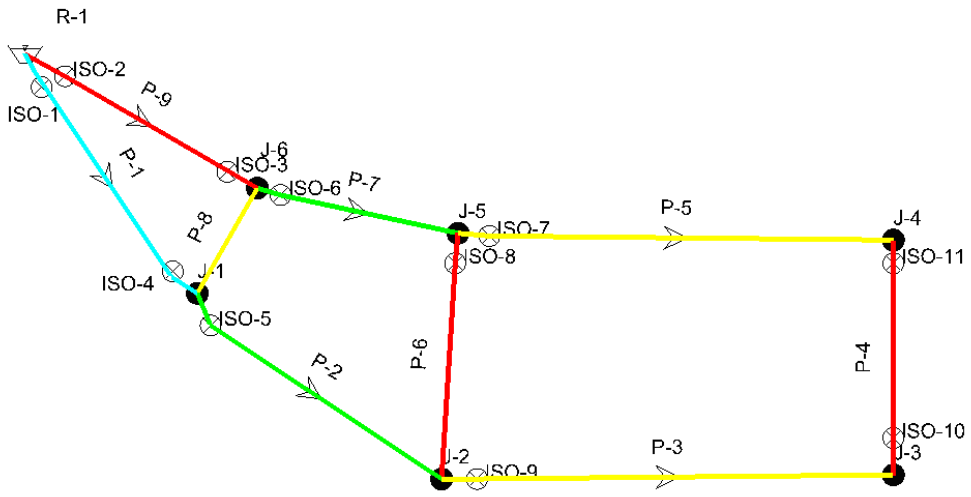
The user must decide which elements will be used to identify segments. This is done under the options tab under criticality studies. See the Segmentation section in the documentation for procedural details.

There are two general approaches to isolating portions of the system. The more correct way is to place all the isolating valves on pipe elements. In this way Bentley WaterCAD V8*i* can accurately identify which system elements are out of service during an outage. In some cases however, the user does not have sufficient data on the location of isolating valves. In this case, Bentley WaterCAD V8*i* assumes that each pipe element can be isolated and each distribution segment consists of a single pipe (not including the nodes at each end). The user identifies if isolating valves are to be used in the analysis by checking the box next to “Consider Valves?” on the Options tab of the Criticality Studies level. (Related to this is the ability of the user to identify if a valve is to be considered the boundary of a segment all of the time, only when it is closed in the selected scenario, or never.)

The figure below shows the segments that are identified if “Consider valves?” is checked. Note that the various colors assigned to elements by the program are not representative of any network attribute but are only used to differentiate adjacent segments.



The figure below shows the segments that are identified when the “Consider valves?” box is unchecked.



The user then picks the scenario to be used in the analysis by clicking New and picking the scenario from the list of available scenarios. Depending on the scenario selected, the criticality analysis will be either a steady state or extended period simulation and will use or not use pressure dependent demands (PDD). (If a fire flow analysis scenario is selected, it is treated as a steady state and if a water quality scenario is selected, it is treated as an EPS.)

Once the scenario has been selected for segmentation, the user can then decide if segments should be identified for the entire network or a subset of the network in the tab called “Segmentation scope”. If the scope of the segmentation analysis is a Subset of the system, an ellipse (…) button becomes available. By clicking this button, the user can decide on the elements to include using boxes, queries, polygons, or picking individual elements. Including any element in the Segmentation Scope means that the segment containing that element will be included in the segmentation and subsequent criticality analysis. Boundary elements between segments are not used if they are included in the Segmentation Scope. When done, the user right clicks and returns to segmentation scope. With the name of the scenario highlighted, clicking the GO arrow will start the segmentation. To delete the list of elements from the Segmentation Scope selection, pick the ellipse button and then pick the Clear button (last one on right).

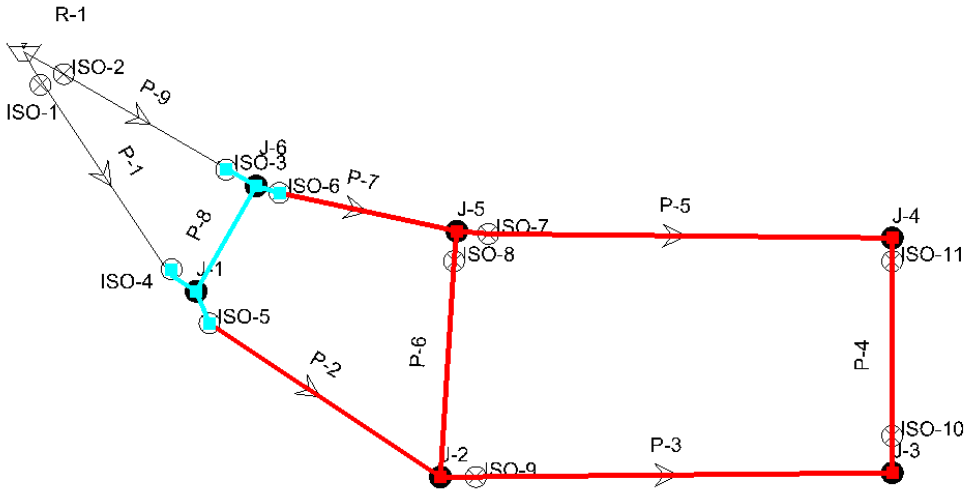
See the Segmentation topic for the details in running segmentation and Criticality Results regarding viewing the results.

Outage Segments

When a segment is taken out of service in a looped or multi-source system, virtually all of the other segments remain in service. However, in tree shaped systems, removing one segment from service also takes downstream segments out of service. These downstream segments are referred to as “Outage Segments”. To determine outage segments, highlight the Outage Segments level of the left pane and click the Go arrow. This will identify all outage segments.

Viewing and zooming to outage segments is similar to these operations in regular network segments. Segments must be identified before outage segments can be identified. In most cases in looped systems, the isolating segments usually contain no elements. However, there may be some surprises which can provide some insights into the adequacy of valving in a system.

The figure below shows the network segment that is being isolated in blue and the corresponding outage segment in red. Note that the various colors assigned to elements by the program are not representative of any network attribute but are only used to differentiate adjacent segments.



This system which at first looks as if it has adequate valving and parallel piping has a serious problem because of valving in the blue segment results in a large outage segment.

Running Criticality Analysis

After segments have been identified (not necessary to run outage segments), Bentley WaterCAD V8*i* can calculate the performance of the system when each segment is taken out of service. This is done by clicking on the Criticality button and hitting the Go arrow.

An important consideration in running criticality is whether the criticality is based on a full hydraulic analysis or simply the connectivity of the system. If the user checks the box labeled “Run hydraulic engine”, Bentley WaterCAD V8*i* will calculate the shortfall in the system based on a full hydraulic analysis. The type of run (steady vs. EPS; PDD vs. non-PDD) is determined by the calculation options of the selected scenario.

If the box is unchecked, Bentley WaterCAD V8*i* calculates shortfall based on connectivity. In that case, if a node is connected back to a source, it is assumed the demand is met. If the node is isolated for the source, it is assumed that it is not.

Understanding shortfalls

The criticality analysis works by identifying the shortfalls that occur when a segment is taken out of service. Depending on the type of analysis, different indicators of shortfall (i.e. drop in system performance) are used. The types of indicators of shortfall for each type of analysis are summarized in the table below.

Run with Hydraulic Engine	PDD?	Steady State/EPS	Flow Results	Pressure Results
No	N/A	N/A	No flow if not connected	N/A

Run with Hydraulic Engine	PDD?	Steady State/EPS	Flow Results	Pressure Results
Yes	No	EPS	No flow if not connected	Max Pressure Drop
Yes	No	Steady State	No flow if not connected	Max Pressure Drop
Yes	Yes	EPS	Volume reduction	Max Pressure Drop
Yes	Yes	Steady State	Flow Reduction	Max Pressure Drop

Criticality Results

Criticality results give an indication of the importance of the shutdown of a segment in terms of the amount of demand met. There are several different indicators depending on the type of analysis selected.

In some cases, especially when EPS runs are being made, the system that results during a segment shutdown may be one that can't be solved hydraulically because large numbers of nodes are disconnected from the system. In that case, the Is Balanced check box will not be checked. Users should look carefully at those segments to determine the importance of such an outage.

The key indicator of the importance of shutting down a segment is the System Demand Shortfall (%). When it is large (and the system is balanced), outage of the segment will have serious impacts. The results will be different depending on the type of analysis and:

- Whether the scenario uses Pressure Dependent Demand (PDD) or non-PDD calculation options.
- Whether the results are based on connectivity only (Run hydraulic engine not checked), a steady state scenario or an EPS scenario.

It is generally advisable to use PDD-based scenarios for criticality. Otherwise demands will be met regardless of the pressure as long as the pressure exceeds Minimum Pressure Required to Meet Demand (displayed at the top of the right pane). With PDD, a continuous relationship between demand met and pressure is used.

While actual water users are located along pipes, the model represents them as being located at nodes. Segments which are located entirely within a single pipe element in a looped system will have no shortfall even though there may be water users along the pipe.

The user-defined Maximum Allowable Demand Shortfall field is used to indicate whether the System Demand Shortfall criteria are satisfied. When Maximum Allowable Demand Shortfall is larger than the System Demand Shortfall, and Minimum Pressure to Supply Demand is smaller than Pressure Supplied at Worst Node, the "Are all demands met?" property will be checked (True).

Interpretation of results also depends on the type of run:

- **Connectivity only -** In this case, demand will not be met only when the nodes are isolated from the source. Otherwise it is assumed that demand is met when a node is connected.
- **Steady-State run -** With steady-state runs, the shortfall is based on calculated pressure and is useful for identifying the results of outages which are not particularly long (such that the tanks drain). The shortfall includes demands that are not met because the nodes are isolated plus demands that are not fully met because pressure drops.
- **EPS runs -** With EPS runs, the effects of tanks draining are also determined. With EPS runs it is much more likely to have nodes that become disconnected such that the hydraulic calculations will not balance. While the connectivity only and steady state runs are snapshots which give shortfall in flow units (e.g. gpm), the EPS runs give results in volume units (e.g. gallons).

To compare between scenarios, the user should pick the Criticality Studies level of the left pane and view the bottom half of the right pane. The Average System Shortfall is a good indicator for comparisons but is based only on segments for which the hydraulic calculations are balanced.

Individual values in the criticality results are described below (in general, results from a steady run will be given as Flow while results from an EPS run will be given as Volume; hence Flow/Volume is listed below):

- **Are all demands met?** - This is checked (True) only if the percent demand shortfall for this segment is less than the Maximum Allowable Demand Shortfall in %. This will generally be unchecked because most segments will have a node with a demand and the node is isolated from the system. When the default value for Maximum Allowable Demand Shortfall is 0, then any segment that sees any drop in supply when closed will fail to meet demands (and hence this box will be unchecked). This property may be checked if the demand inside the segment is 0 or if the Maximum Allowable Demand Shortfall is set greater than 0. If the pressure at the node with the lowest pressure is below the Minimum Pressure to Supply Demand, then "Are All Demands Met" will be unchecked.

- **Is balanced?** - This is checked if the hydraulic calculations are solved. For some segments, removing the segment may affect the network so severely (e.g. disconnecting all the sources) that the calculations cannot be run. These are usually segments that seriously affect the reliability of the network and the user should inspect these manually. If "Is balanced?" is not checked, many of the results fields are N/A (not applicable).
- **Maximum allowable demand shortfall (%)** - This value defaults to 0%. However, for non-PDD runs, the user can override this value by entering a value in the "Maximum allowable demand shortfall" field.
- **System Demand (Full)/System Demanded Volume** - This is the total of system demands when there are no segment outages. It is given in flow units for steady runs and volume units for EPS runs.
- **System Demand (Met)/System Supplied Volume** - This is the total water supplied when the segment is out of service in flow units for steady runs and volume units for EPS runs.
- **System Demand Shortfall (%)** - This value is calculated as 100%*[1-(Supplied/Demanded)]
- **Node with Largest Percent Demand/Volume Shortfall** - This is the node label for the node with the maximum percent demand shortfall defined below. If there are no nodes with a shortfall, then this value and the next field are set to (N/A).
- **Demand Shortfall at Worst Node (%)** - 100% * [1 - (Supplied/Demanded)] at node in previous field. If the pressure is less than the Minimum Pressure to Supply Demand, this value is set to 100%.
- **Node with Largest Demand/Volume Shortfall** - This is the node label for the node with the maximum demand shortfall (i.e. Demand - Supplied)).
- **Flow/Volume Demanded at Worst Node** - Demand - Supplied at node from previous field.
- **Flow Supplied at Worst Node** - Flow supplied at node identified in the previous field.
- **Node with Largest Pressure Shortfall** - Node with largest value of ("Min Pressure to Supply Demand" - Pressure). This field is only used for non-PDD runs because pressure is handled differently in PDD. When the scenario calls for PDD, the "Minimum Pressure to Supply Demand" property is ignored. If the value of Min Pressure to Supply Demand is 0, then this value is not calculated and is set to (N/A).
- **Pressure Demanded at Worst Node** - Minimum pressure to supply demand at the worst node.
- **Pressure Supplied at Worst Node** - Actual pressure at Node with Largest Shortfall at the worst node.

In the case of non-PDD demands for steady runs, there are two situations for a given node that fails to meet demands.

1. Nodes that are disconnected by the segment outage in which case the demands are not included in the simulation
2. Nodes that fail to meet minimum pressure in which case the demands are included in the simulation

For the case of an EPS with Non-PDD demands, when choosing to "run hydraulic engine", the program checks the pressure at each node at each time step, and identifies nodes that fall below the desired minimum pressure at any given time. For criticality purposes, the program then assumes these nodes supply zero demand. Without PDD, the program cannot determine the exact shortfall. However, the criticality results in this case will still be useful, as they will identify nodes that have insufficient pressure.

In the criticality results, the "Node with largest percent demand shortfall" and "Node with largest volume shortfall" will show the node that had the highest demand during the time when the pressure was below the desired minimum pressure.

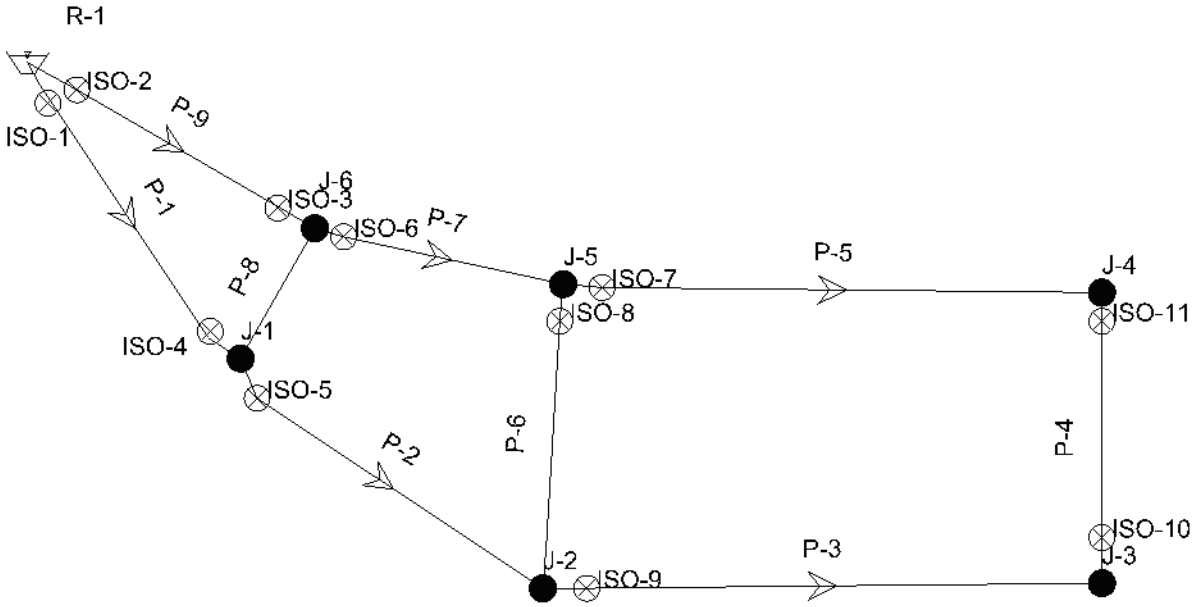
Segmentation

A distribution network segment is defined as the smallest portion of a distribution system that can be isolated. Segments are used in the Bentley WaterCAD V8*i* criticality analysis as the basic element of a system that can be isolated so that the effects of an outage can be evaluated.

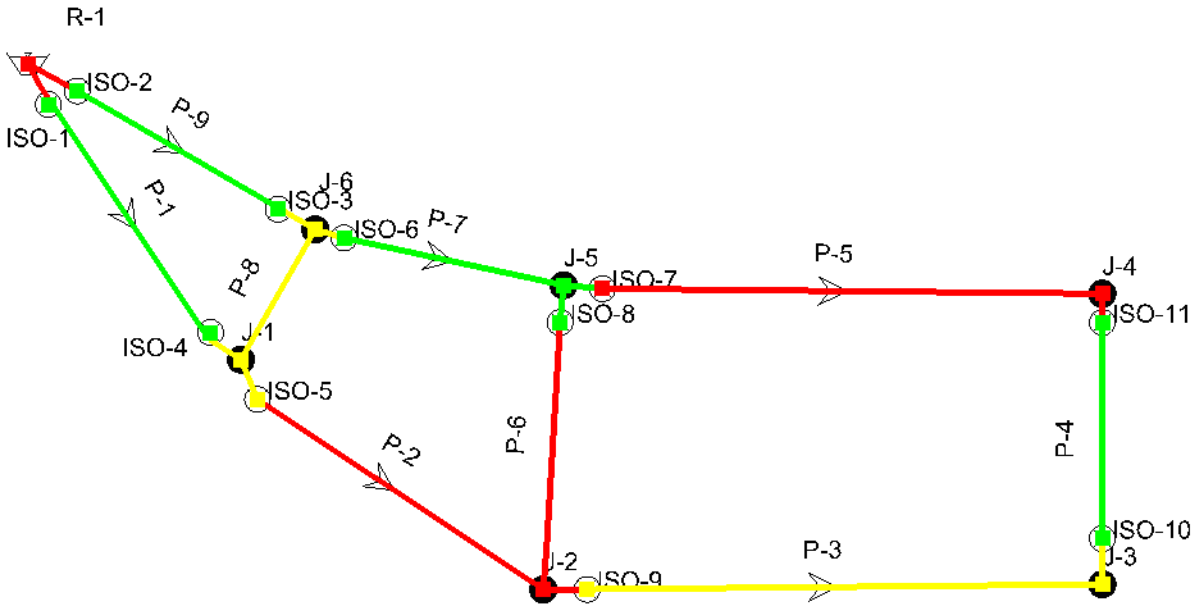
Bentley WaterCAD V8*i* allows a user to set up two types of segments:

1. **Using valves** - A segment is created when valves are closed to isolate a portion of a distribution system. If the user has entered isolating valves and these valves are assigned to pipes, then Bentley WaterCAD V8*i* automatically identifies segments. These segments can consist of a portion of a single pipe or several pipes and their interconnecting node elements. The user selects this type of segment by checking the “Consider valves?” box in the Options tab of the Criticality Studies manager.
2. **Pipe-by-pipe** - In some cases a user wants to conduct a criticality analysis but does not have information on the location of isolating valves. In this case, Bentley WaterCAD V8*i* will create segments such that there is one pipe link in each segment. The nodes at the end of the pipe links are not part of the segment when this method is used. The user selects this type of segment by unchecking the “Consider valves?” box in the Options tab of the Criticality Studies manager.

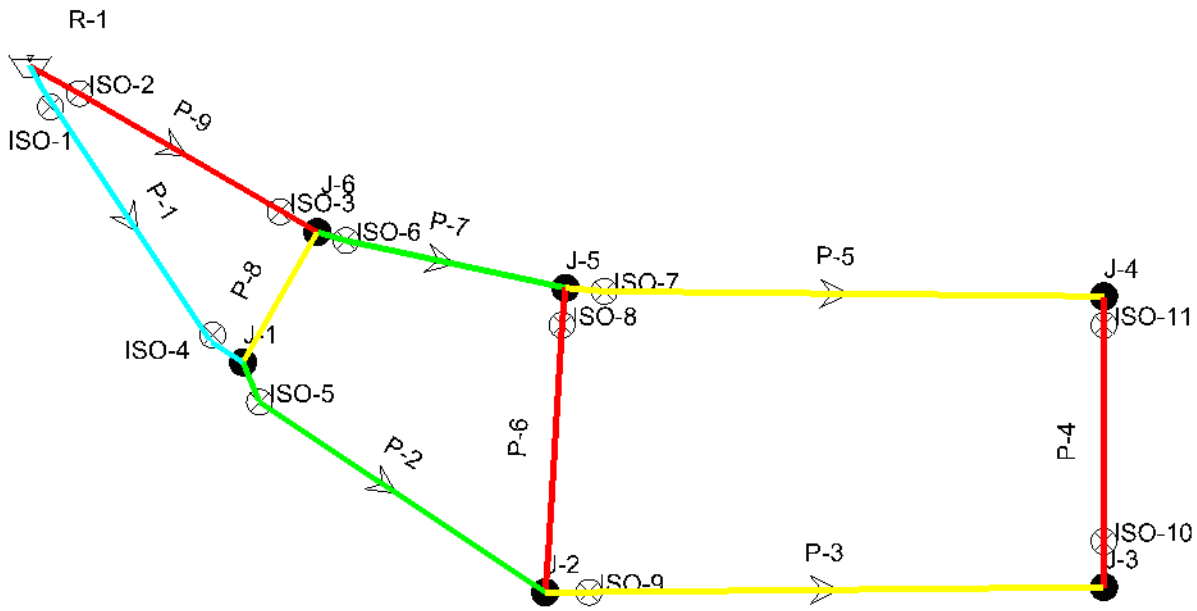
The first figure below shows a simple pipe network with valves.



If the “Consider valves?” Option is selected, then the segments (identified by color) are created based on valves that can be closed. The segments are identified by color in the figure below. Note that the various colors assigned to elements by the program are not representative of any network attribute, but are only used to differentiate adjacent segments.



If on the other hand, “Consider valves?” is unchecked, then each segment consists of one and only one pipe as shown below.



The option where valving is considered is a much more accurate reflection of the portion of the system that is out of service during a shutdown. Using the pipe-by-pipe segments can be misleading in come cases. For example if pipe P-8 is removed from the system, then by considering valving, the user can see that all downstream customers are out of service. However, in the pipe-by-pipe case, J-1 and J-6 are still in service and it looks as if downstream customers can be served.

Of course, to consider valves in the system, the isolating valves must be part of the pipe network. Adding isolating valves is explained in topic “Valves - Isolating.”

Depending on the approach used by the modeler, elements such as PRVs and General Purpose Valves may also be used to isolate segments. For each of these types of elements, the user can indicate whether they should be used to isolate the system. For each type of element, the user has three options:

- **Always use (default)** - valve is treated as an isolating valve for segmentation
- **Use when closed** - status of closed if assigned in initial conditions for that scenario
- **Do not use** - does not use valve as boundary to segment.

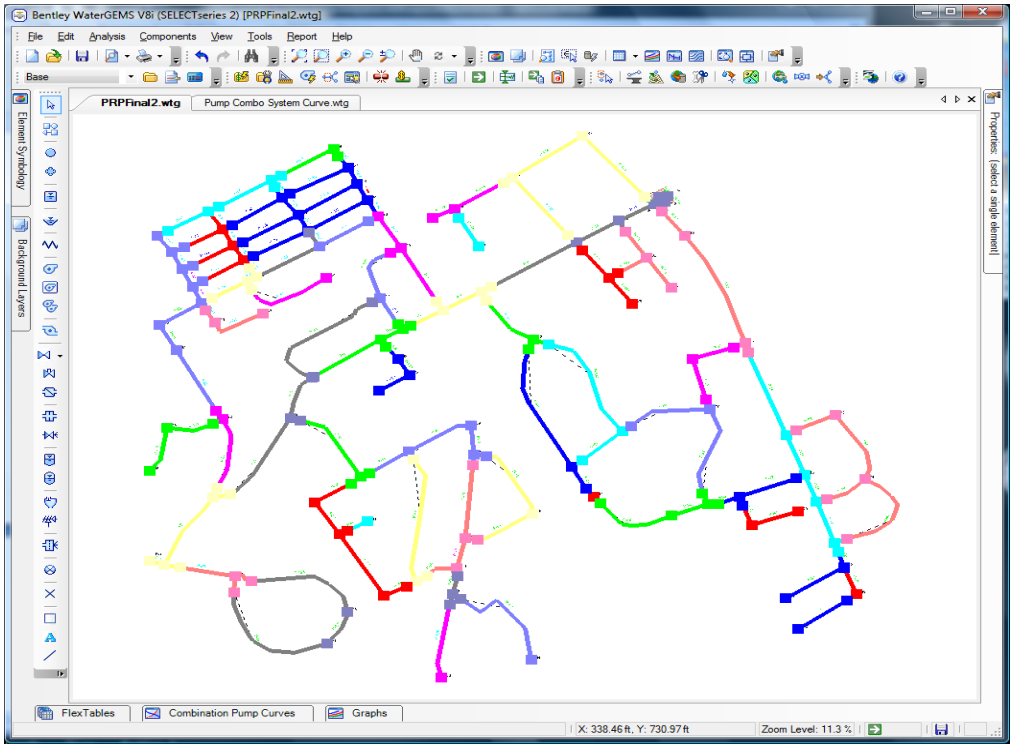
There are several buttons on top of the middle pane in the segmentation manager that are used to control the display of segments in the drawing and use of segmentation results.

The first button enables the user to Create a Selection Set including all of the elements from a specific segment. When the user picks this button, the user is given an opportunity to name the selection set. Hitting OK creates the selection set. The set includes pipes that are only partly in that segment. The user can also add the elements in the segment to a selection set or remove them from a selection set.

The second button Zooms to the selected segment and highlights the elements in that segment. If a pipe is only partly in that segment, the entire pipe is highlighted.

The Find in Drawing button is used to pick an element from the drawing and determine which segment it lies in. When the user picks Find, he is given a "Select from Drawing" prompt and must pick an element. The segment that the user picks is then highlighted in the middle pane list of segments and the details are given in the right pane. If a segment boundary valve is picked, then the segments on both sides of the valve are highlighted.

The Highlight Segments button color codes the drawing such that each segment has a different color. If All Segments is selected in the middle pane, then all segments are color coded and if one is selected, only that segment is color coded. Repeating this selection toggles off this color coding. This color coding is not a property of the element and as such is not handled by the Element Symbology tool and if an element is moved after this color coding, the color coding is not moved. It is usually advisable to minimize the segmentation dialog when viewing color coding.



The next button is the standard Refresh button which refreshes the drawing if needed.

The next button is the Report button which generates a report for printing.

Segmentation Results

The results of a segmentation analysis are shown in the right panes of the Criticality manager. The top half contains one line for each segment.

The segmentation results can be used to find segments which will become maintenance problems during a shutdown. To find troublesome segments, it is best to sort the segmentation results by right clicking on the appropriate column and choosing Sort Descending.

To find segments that require a large number of valves to be shut in order to isolate the segment, sort the Isolation Elements column. Then pick the segments that have the highest number of isolation elements and zoom to them to see where problem segments might exist.

To find the segments that are most likely to put a large number of customers out of service or are most likely to break, sort based on the length of pipe in the segment. If segments have a relatively even break rate, then the longest ones will have the most breaks and the longest ones are most likely to have the most customers out of service.

Sorting by Fluid Volume in the segment will give an indication of the amount of water that must be drained from the segment in order to de-water the pipe for repair.

The bottom half of the right pane gives details about the nodes included in each segment, the pipes involved in each segment and the isolating nodes needed to shut down each segment. In this portion of the results, there is one line for each element as opposed to the top half where there is one line for each segment. Usually this is best used by picking an individual segment from the middle pane and viewing the details of that segment.

To compare segmentation results between scenarios, the user should pick the Criticality Studies level at the top of the left pane. The top of the associated summary right pane (Segmentation Results Summary) gives overall statistics for each scenario. Usually the results are similar between scenarios unless they use different topologies in terms of valves.

Outage Segment Results

The outage segment results give an indication of which segments will be placed out of service when an upstream segment is shut down. In highly looped systems with multiple sources, there will be very few non-zero length outage segments, while in tree shaped segments with a single source, there will be numerous large outage segments.

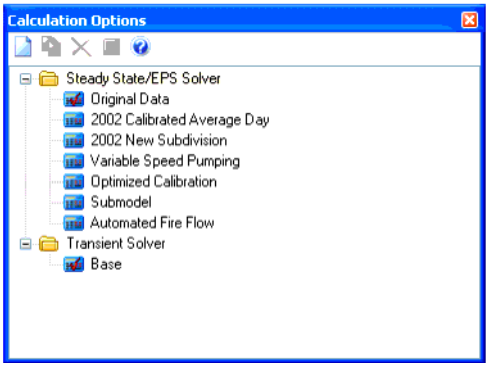
The outages segment list may be sorted based on Outage Set Length. Large outage segments usually indicate portions of the system where a single break or shutdown can place large numbers of customers out of service.

Use the zoom button on top of the middle pane to view the details of the individual outage segment sets and evaluate approaches to improve the system.

Calculation Options

Calculations depend on a variety of parameters that may be configured by you.

Choose Analysis > Calculation Options, Alt+3, or click the  button to open the Calculations Options dialog box.



The following controls are available from the Calculation Options dialog box.

New



Creates a new calculation option.

Duplicate



Makes a copy of the selected calculation option.

Delete



Deletes the selected calculation option. The base calculation option cannot be deleted.

Rename



Renames the selected calculation option.

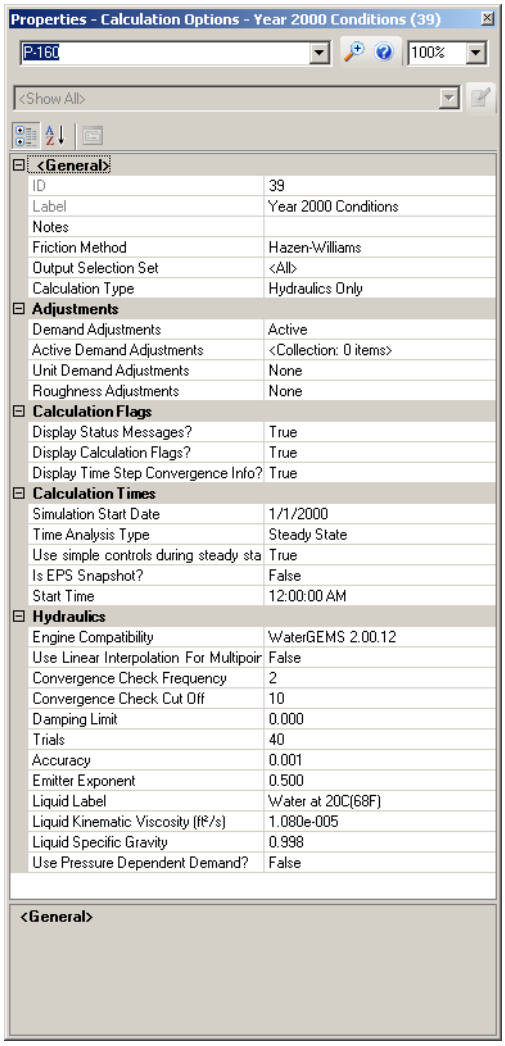
Help



Displays online help for the Calculation Options.

To view the Steady State/EPS Solver properties of the Base Calculation Options

Select Base Calculation Options under Steady State/EPS Solver and double click to open the Properties dialog box.



The following calculation option parameters are available for user configuration:

- **Friction Method**—Set the global friction method.
- **Output Selection Set**—Select whether to generate output for All Elements (the default setting) or only the elements contained within the chosen selection set.
- **Calculation Type**—Select the type of analysis to perform with this calculation options set.
- **Consider Pumps and Valves in Min. System Pressure Constraints?**— If True the pressures at pumps and valves will be considered.

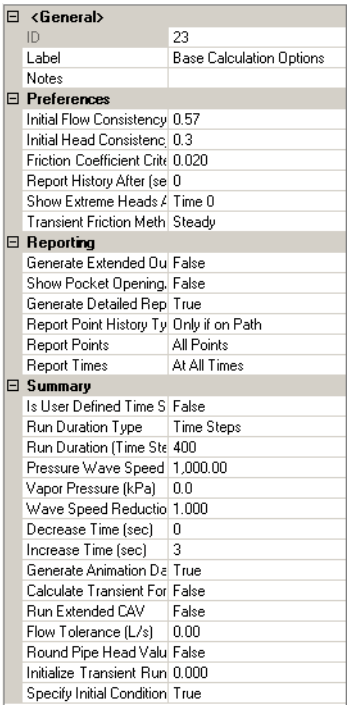
- **Demand Adjustments**—Specify whether or not to apply adjustment factors to standard demands.
- **Active Demand Adjustments**—The collection of demand adjustments that are applied during the analysis.
- **Unit Demand Adjustments**—Specify whether or not to apply adjustment factors to unit demands.
- **Active Unit Demand Adjustments**—The collection of unit demand adjustments that are applied during the analysis.
- **Roughness Adjustments**—Specify whether or not to apply adjustment factors to roughnesses.
- **Active Roughness Adjustments**—The collection of roughness adjustments that are applied during the analysis.
- **Display Status Messages?**—If set to true, element status messages will be stored in the output and reported.
- **Display Calculation Flags?**—If set to true, calculation flags will be stored in the output and reported.
- **Display Time Step Convergence Info?**—If set to true, convergence/iteration data for each time step will be stored in the output file and displayed in the calculation summary.
- **Simulation Start Date**—Select the calendar date on which the simulation begins.
- **Time Analysis Type**—Select whether the analysis is extended period or steady-state.
- **Use simple controls during steady state?**—When True, simple controls will be active during steady state analyses, else they will not be used. Note that logical controls are never used during steady state analysis.
- **Is EPS Snapshot?**—If True then an EPS snapshot is run instead of a regular steady state run. An EPS snapshot is a steady state run, but it considers the starting date and time of analysis and applies the appropriate pattern multipliers for that time. Note that since an EPS is not run, attributes such as tank levels are derived from the same initial conditions as a steady state run.
- **Equivalent Hydraulic Time Step**—In order that the pattern multipliers used in an EPS snapshot run exactly match those in an equivalent EPS run, specify the hydraulic time step of the EPS run that you wish to match.
- **Start Time**—Select the clock time at which the simulation begins.
- **Duration**—Specify the total duration of an extended period simulation.
- **Hydraulic Time Step**—Select the length of the calculation time step.
- **Override Reporting Time Step?**—Specify if you want the Reporting Time Step to differ from the Hydraulic Time Step.

- **Reporting Time Step**—Data will be presented at every reporting time step. The reporting time step should be a multiple of the hydraulic time step.
- **Set Water Quality Time Step?**—If set to True the Water Quality Time Step can be adjusted, otherwise it is computed by the calcuation engine. It is not recommended that you set this to True.
- **Water Quality Time Step**—Time interval used to track water quality changes throughout the network. By default, this value is computed by the numerical engine and is equivalent to the smallest travel time through any pipe in the system.
- **Engine Compatibility**—This field allows you to choose which engine compatibility mode you want to run in. Choose WaterGEMS 2.00.12 to get all of the latest engine improvements and fixes made by Bentley and an engine mode that is based upon EPANET 2.00.12. This is the default setting for new models. Choose WaterGEMS 2.00.10 to maintain compatibility with previous version of WaterGEMS (V8i SELECTseries 1 and earlier), where the computational engine is based on EPANET 2.00.10. This is the default for upgraded models. If you select one of the EPANET modes, any enhancements, calculation corrections, and bug fixes made by Bentley will be disabled in order to match EPANET version results. Imported EPANET models will default to the appropriate EPANET version.
- **Use Linear Interpolation for Multipoint Pumps?**—If set to true the engine will use linear interpolation to interpret the pump curve as opposed to quadratic interpolation.
- **Convergence Check Frequency**—This option sets the number of solution trials that pass during hydraulic balancing before the status of pumps, check valves, flow control valves, and pipes connected to tanks are updated. The default value is 2, meaning that status checks are made every other trial. A value equal to the maximum number of trials would mean that status checks are made only after the system has converged (whenever a status change occurs the trials must continue since the current solution may not be balanced). The frequency of status checks on pressure reducing and pressure sustaining valves is determined by the Damping Factor calculation option.
- **Convergence Check Cut Off**—This option is the number of solution trials after which periodic status checks on pumps, check valves, flow control valves, and pipes connected to tanks are discontinued. Instead, a status check is made only after convergence is achieved. The default value is 10, meaning that after 10 trials, instead of checking status at every trial indicated by the Convergence Check Frequency setting, status is checked only at convergence.
- **Damping Limit**—This is the accuracy value at which solution damping and status checks on PRVs and PSVs should begin. Damping limits all flow changes to 60 percent of what the would otherwise be as future trials unfold. The default of 0 indicates that no damping should be used and that status checks on control valves are made at every iteration. Damping might be needed on networks that have trouble converging, in which case a limit of 0.01 is suggested (relative to the default calculation hydraulic accuracy of 0.001).

- **Trials**—Unitless number that defines the maximum number of iterations to be performed for each hydraulic solution. The default value is 40.
- **Accuracy**—Unitless number that defines the convergence criteria for the iterative solution of the network hydraulic equations. When the sum of the absolute flow changes between successive iterations in all links is divided by the sum of the absolute flows in all links and is less than the Accuracy, the solution is said to have converged. The default value is 0.001 and the minimum allowed value for Accuracy is 1.0e-5.
- **Emitter Exponent**—Emitters are devices associated with junctions that model the flow through a nozzle or orifice. In these situations, the demand (i.e., the flow rate through the emitter) varies in proportion to the pressure at the junction raised to some power. The constant of proportionality is termed the discharge coefficient. For nozzles and sprinkler heads the exponent on pressure is 0.5 and the manufacturer usually states the value of the discharge coefficient as the flow rate in gpm through the device at a 1 psi pressure drop.
- **Liquid Label**—Label that describes the type of liquid used in the simulation.
- **Liquid Kinematic Viscosity**—Ratio of the liquid's dynamic, or absolute viscosity to its mass density.
- **Liquid Specific Gravity**—Ratio of the specific weight of the liquid to the specific weight of water at 4 degrees C or 39 degrees F.
- **Use Pressure Dependent Demand?**—If set to true the flows at junctions and hydrants will be based on pressure constraints.
- **Age Tolerance**—If the difference between two parcels of water is equal to or less than the value specified in this field, the parcels are considered to be of equal age.
- **Consituent Tolerance**—If the difference between two parcels of water is equal to or less than the value specified in this field, the parcels are considered to possess an equal concentration of the associated constituent.
- **Trace Tolerance**—If the difference between two parcels of water is equal to or less than the value specified in this field, the parcels are considered to be within the same percentile.

To view the Base properties of the Transient Solver Calculation Options

Select Transient Solver Base Calculation Options and double click to open the Properties dialog box.



<General>	
ID	23
Label	Base Calculation Options
Notes	
Preferences	
Initial Flow Consistency	0.57
Initial Head Consistenc	0.3
Friction Coefficient Crite	0.020
Report History After (se	0
Show Extreme Heads A	Time 0
Transient Friction Meth	Steady
Reporting	
Generate Extended Ou	False
Show Pocket Opening,	False
Generate Detailed Rep	True
Report Point History Ty	Only if on Path
Report Points	All Points
Report Times	At All Times
Summary	
Is User Defined Time S	False
Run Duration Type	Time Steps
Run Duration (Time Ste	400
Pressure Wave Speed	1,000.00
Vapor Pressure (kPa)	0.0
Wave Speed Reductio	1.000
Decrease Time (sec)	0
Increase Time (sec)	3
Generate Animation Da	True
Calculate Transient For	False
Run Extended CAV	False
Flow Tolerance (L/s)	0.00
Round Pipe Head Valu	False
Initialize Transient Run	0.000
Specify Initial Condition	True

The following calculation option parameters are available for user configuration:

- **Initial Flow Consistency**—Flow changes that exceed the specified value are listed in the output log as a location at which water hammer occurs as soon as simulation begins. The default value is 0.02 cfs.
- **Initial Head Consistency**—Head changes that exceed the specified value are listed in the output log as a location at which water hammer occurs as soon as simulation begins. The default value is 0.1 ft.
- **Friction Coefficient Criterion**—For pipes whose Darcy-Weisbach friction coefficient exceeds this criterion, an asterisk appears beside the coefficient in the pipe information table in the output log. The default value is 0.02.
- **Report History After**—Set the time at which reporting begins. The default value is 0.02.
- **Show Extreme Heads After**—Sets the time to start output of the maximum and minimum heads for a run. You can set these to show beginning at time = 0 (right away), after the first maximum or minimum, or after a specified time delay.

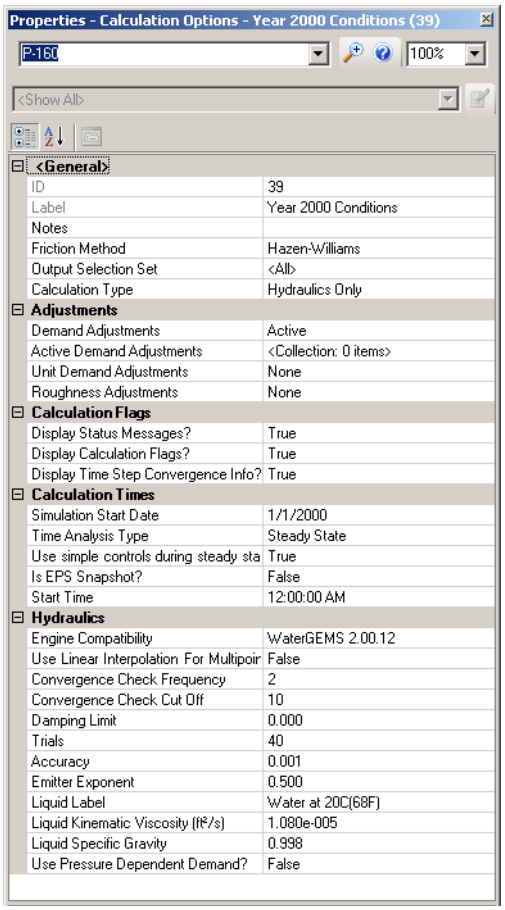
- **Transient Friction Method**—Select Steady, Quasi-Steady, or Unsteady friction method to be used for transient calculations.
- **Generate Standard Output Log?**—Toggles the standard output file.
- **Show Pocket Opening/Closing**—Toggles whether the list of vapor pockets open and close times will be appended to the output text file.
- **Generate Detailed Reports**?—Toggles the generation of ASCII output text files on or off. These can become voluminous for simulations with many time steps and they are not required for the operation of the FlexTables or graphics. Some users prefer to set this setting to False.
- **Report Point History Type**—Select All to generate point histories for all points in the text reports, or Only if On Path to generate report Histories only for those points that lie on a path.
- **Report Points**—Choose the report points type from the following:
  - No Points—No report points are defined.
  - All Points—All nodes in the model are report points.
  - Selected Points—Selecting this option makes the Report Points Collection field active, allowing you to define the report points.
- **Report Points Collection**—Clicking the ellipsis button in this field opens the Report Points Collection dialog, allowing you to choose the report points from the list of available points, or select them in the drawing.
- **Report Times**—Choose whether to report Periodically, At Specific Times, At No Times, or At All Times.
- **Report Period**—Specify the equal intervals of time (default) at which reports are generated. This option is only available when the Report Times property is set to Periodically.
- **Report Times Collection**—Opens the Report Times Collection dialog, allowing you to specify the times step to be reported. This option is only available when the Report Period property is set to At Specific Times.
- **Is User Defined Time Step?**—Selcts whether the time step is user-defined or automatically estimated.
- **Time Step Interval**— This option is only available when the Is User Defined Time Step? property is set to True.
- **Run Duration Type**—Selects whether the run duration is measured in time or time steps.
- **Run Duration**—Period of time simulated by the model.
- **Pressure Wave Speed**—Speed for the liquid being conveyed, the pipe material selected and its dimension ratio (DR), bedding, and other factors.

- **Vapor Pressure**—Pressure below which a liquid changes phase and become a gas (steam for water), at a given temperature and elevation.
- **Wave Speed Reduction Factor**—The low pressure wave speed reduction factor.
- **Decrease Time**—The time for the wave speed to decrease from its normal value to the reduced value at vapor pressure.
- **Increase Time**—The time for the wave speed to increase from its reduced value to the normal value at vapor pressure.
- **Generate Animation Data**—Set this property to True to generate animation data for selected report paths and points.
- **Calculate Transient Force**—Set this property to True to calculate transient forces.
- **Run Extended CAV**—Toggles the standard or extended Combination Air Valve (CAV) sub-model. The vacuum breaker component of CAV admit air into the pipeline during low transient pressures that is subsequently expelled at the outlet orifice(s). The extended model tracks momentum more accurately.
- **Flow Tolerance**—Flows below this value are assumed to be zero when running the transient calculations. This option is generally used to filter out insignificant flows that could otherwise cause numerical problems during the calculation. See Flow Tolerance for more details.
- **Round Pipe Head Values?**—Specifies whether pipe head values should be rounded or not. This option is generally used to filer out insignificant differences that could otherwise cause numerical probelms during the calculation.
- **Initialize Transient Run at Time**—If the “Specify Initial Condition” field is set to True, the transient simulation is initialized using results from a steady-state or extended period simulation. Enter a time here to initialize the transient simulation using results from the corresponding EPS time step.
- **Specify Initial Conditions?**—If set to True, you can manually specify the initial conditions for a transient simulation.

To create a new calculation option

1. Choose Analysis > Calculation Options and the Calculation Options dialog box opens.
2. Choose New.
3. Double-click on the newly created calculation option to open the Calculation Options Properties dialog box.

4. Set the fields for this calculation.



5. Close the properties box.
6. Close the Calculations Options box.

Controlling Results Output

There are two ways that you can limit the output data that is written to the result file from the water engine: by time step and by element. Limiting the reported results in this way will produce a smaller result file, thereby improving performance when copying results files during open and save operations. It also conserves hard disk space.

One way is to limit the reported time steps:

By default, the Overide Reporting Time Step calculation option is set to <All>. Under this setting, all results for all time steps are written to the results file.

To limit the output results to a specific interval (such as every 2 hours, every 4 hours, etc) set the Overide Reporting Time Step calculation option to Constant. The Reporting Time Step calculation option will become available. Enter the constant interval at which output results should be written to the results file in this field.

To limit the output results to specific time steps, set the Overide Reporting Time Step calculation option to Variable. The Reporting Time Steps calculation option will become available. Click the elipsis (...) button in this field to open the Reporting Time Steps dialog.

The other way is to limit the reported elements:

By default, the Output Selection Set calculation option is set to <All>. Under this setting, all results for all elements are written to the results file.

By choosing a previously created selection set in this field, you can limit the output data written to the results file to only include data for the elements that are contained within the specified selection set.

Reporting Time Steps Dialog Box

This dialog allows you to specify whether the output results for different time steps during an extended period simulaton will or will not be written to the results file.

You do this by specifying ranges of time during which:

- All of the time steps are reported on and written to the results file.
- None of the time steps are reported on and written to the results file.
- Time steps that fall within the specificed constant interval are reported on and written to the results file.

The first row in this dialog will always be 0.00 hours, which is the beginning of the first time range. To specify the first range of time, enter the end time step in the second row, for example 24 hours. Specify the type in the first row, for example <All>. In this example, all time steps between hour 0 (the start of the simulation) and hour 24 will be written to the results file. To specify further ranges of time, add new rows with the New button. Remove rows with the Delete button. The last range in the dialog will start at the time specified in the last row and end at the end of the simulation.

Note: If you have a large number of steps you can use an external program like Excel to create the table, then copy and paste it into the dialog.

Report Points Collection Dialog Box

This dialog allows you to specify which of the available points in the model will be report points.

Click the [>] button to add a highlighted point from the Available Items list to the Selected Items list.

Click the [>>] button to add all Available Items to the Selected Items list.

Click the [<] button to remove a highlighted point from the Selected Items list, returning it to the Available Items list.

Click the [<<] button to remove all report points from the Selected Items list, returning them to the Available Items list.

Click the Select From Drawing button to choose points from the drawing pane.

Report Times Collection

This dialog allows you to specify which of the available time steps in the model will be report times.

Click the [>] button to add a highlighted time step from the Available Items list to the Selected Items list.

Click the [>>] button to add all Available time steps to the Selected Items list.

Click the [<] button to remove a highlighted time step from the Selected Items list, returning it to the Available Items list.

Click the [<<] button to remove all time steps from the Selected Items list, returning them to the Available Items list.

Flow Tolerance

The transient calculation requires that there is not excessive friction in the pipelines. In some cases when the initial flow and headloss along a pipe are both very small, HAMMER will compute large friction factors for these pipes (generally because very low velocities result in small Reynolds number values, which results in high friction factors under laminar flow). This prompts an error message which prevents the model from running. To prevent this, it is possible to specify a Flow Tolerance value below which any flow is rounded down to zero. This prevents the friction factor error, because the friction factor for pipes with zero initial flow is based solely on the roughness parameter entered for the pipe. However, if the Flow Tolerance is adjusted, it is suggested that the 'Round Pipe Head Values?' parameter is set to 'True' and the pipe heads are rounded to a similar level of accuracy as the flows. This helps ensure that the head at either end of a pipe with zero initial flow is the same.

Note however, that in the majority of cases it is suggested that the default value is used for these parameters.

Determining the Transient Run Duration

Run duration is measured either in seconds or as a number of time steps. HAMMER determines the length of each time step automatically. Time steps typically range from a few hundredths of a second to a few seconds, depending on the system and the pressure wave speeds. The run duration has a direct effect on the modeling computation time, along with the time step selected for the simulation.

For simple systems or if the time required to compute the HAMMER model is not a concern, it is ideal (but not always necessary) to set run durations long enough to allow a final steady state to be achieved once all transient energy attenuates. This is quite manageable in many cases, such as for the sample file sample02.wtg, which requires about 30 to 40 seconds to reach a final steady state. Each system requires a different amount of time to reach a final steady state.

Transient Tip: Every pipe system has a characteristic time period, T = 2 L/a, where L is the longest possible path through the system and a is the pressure wave speed. This period is the time it takes for a pressure wave to travel the pipe system's greatest length two times. It is recommended that the run duration equal or exceed T. Another factor to consider when determining run duration is to allow enough time for friction to significantly dampen the transient energy. If in doubt, run HAMMER for a longer duration and examine the resulting graphs and time histories.

For larger systems, you can use the following guidelines to decide on the most appropriate run duration:

First run HAMMER for only a few time steps to identify the sources of transients (remember to output every time step using the Report Times attribute of the Calculation Options). You can also check for input errors by clicking the Validate button. Finally, click Compute to run the model, and then look for errors in the steady-state model or other initial transients in the comments at the end of the output file (.out).

Run HAMMER again for a duration of T=4 L/a (or greater) to verify that your simulation includes the maximum and minimum transient heads (Change the duration in the Calculation Options). These normally occur within this time frame. A longer run duration may be required if air pockets form or if a gas vessel or surge tank is installed, due to the persistence of oscillations in the system.

Run HAMMER again for a duration of T=20 L/a or greater, whatever is enough to allow friction to attenuate the transient energy and, consequently, to let the system approach or achieve a final steady state. See Selecting the Transient Friction Method.

The preceding procedure increases the likelihood that you will correctly simulate the key aspects of the hydraulic transient event for your system. However, remember that L is only a characteristic length which may not be directly applicable to branched or looped networks or plants. Always use sound engineering judgment in reviewing HAMMER results and interpreting the output.

Convergence Improvements for Control Valves

With WaterCAD V8*i* version SS2, some improvements have been made to the numerical solver to increase the stability of the solutions when control valves exist in the model. If the control valves are allowed to change status with each iteration, then it is possible to have oscillating solutions that may not converge. With this version, users have a much greater control over convergence in some situations with complicated control logic, especially those where control valves can interact with one another.

An easy first step when a convergence problem exists is to increase the number of "Trials" (iterations) for each solution. A default value of 40 is provided, but for systems with many complicated control valves, it may be necessary to increase this value to permit the other valves described above a chance to achieve a solution. A value of 200 or more is not unreasonable for problematic models. Since this is a global value it needs to be set high enough to cover the most tricky time steps in the model. There is no adverse effect of having this value too high.

To prevent oscillations in solutions, the numerical solver can be allowed to make several iterations before it changes the status of pumps, check valves, flow control valves and pipes connected to tanks. While this may mean that it takes more iterations to reach a final solution, it tends to make the solution process more stable. The default value of this parameter (called "Convergence Check Frequency" - see Calculation Options) is set to 2 but if there are multiple control valves in the system and convergence problems are being encountered, this number can be increased. It should be substantially less than the total number of allowable trials or else controls will not be allowed to find their correct status. A representative increase in value from the default of 2, might be 10. Note that when "Convergence Check Frequency" is increased it is likely necessary to make a corresponding increase in Convergence Check Cut Off. This particular option is discussed below.

Once the status of valves has stabilized, the numerical solver can quickly converge to the solution. After a certain point, it is not productive to check the status of pumps and valves. This number of iterations is called the "Convergence Check Cut Off" and by default is set to 10. In models with complicated controls, it may be necessary to increase this value to enable the controls to reach a stable set of values before their status becomes fixed. To do this, the convergence check cut off should be increased to a number that is still less than or equal to the number of trials. An increase in this value may be, but is not necessarily, accompanied by an increase in the previously described "Convergence Check Frequency" value.

In order for the solution to not overshoot the correct values, the changes made in each iteration are controlled by damping the size of changes. Usually the parameter "Damping Limit" is set to 0 by default which indicates that no damping is needed. However, when numerical solutions have difficulty converging, this limit can be increased to something roughly an order of magnitude larger than the flow "Accuracy" which is set to 0.001 by default. With that default, a value for Damping limit of 0.01 should help to dampen out oscillations. With a value of 0.01 set it means that when

convergence of the solution comes to within an accuracy value of 0.01 (as opposed to the tighter 0.001 value) damping will start by relaxing flow adjustments to 60% of the value they would be otherwise. Increasing the damping limit even higher than 0.01 may help in particularly difficult cases since damping will be initiated earlier. In all cases the damping limit needs to be relative to and higher than the calculation "Accuracy" value or 0 (damping off).

Another setting that can be modified to improve convergence that existed in the previous version of WaterCAD V8*i* is the "Accuracy" value. This value defines the measure by which the solution method determines whether the hydraulic calculations are balanced. The default value is 0.001 which means the sum of the flow changes in all the links from the previous trial to the current one is less than 0.1% of the sum of the flows in all the links in the system. i.e., the numerical solution has converged to within a tight tolerance. This is a very conservative value. In some cases for models that have many pipes with small or no flow, it may be necessary to increase the hydraulic accuracy value (make the model slightly less accurate) to account for this relative measure of convergence.

There is a tradeoff between speed and stability in these numerical solutions. The default values are set with an emphasis on performance and are good for typical systems. As these above options to dampen solutions are implemented, they tend to slow the convergence. However, when working with systems with multiple interacting control valves, it may be necessary to sacrifice performance for stability and change the numerical values described above.

Vapor Pressure

A liquid's vapor pressure limit is defined as the absolute pressure below which it flashes into its gas phase (vapor or steam for water) for the fluid temperature at which the system is operating. Vapor pressure is a fundamental parameter for any hydraulic transient analysis. Low transient pressures can cause a liquid to vaporize and, once one or more of these vapor pockets collapse later on, result in very large transient pressures, which may break pipes or other system components.

Tip: **For drinking-water systems at typical temperatures and pressures, HAMMER uses an approximate vapor pressure of -10.0 m or -14.2 psi (gauge) or -32.8 ft. by default, depending on the unit system in use. Typically, a liquid's vapor pressure can be obtained from tables (steam tables for water) given its temperature and absolute (not gauge) pressure. You might consider adjusting the vapor pressure if the elevation of your system is significantly different from mean sea level.**

The vapor pocket collapse process is analogous to the well-known tip-cavitation phenomenon, which causes pitting damage at pump impellers; however, vapor pockets can be orders of magnitude larger than cavitation bubbles and can result in system-wide transients.

Tip: To determine the impact of collapsing vapor pockets on your system, set the vapor pressure to a large negative value which you do not expect to occur, such as -1000 m, and run HAMMER with a different file name. Then reset the vapor pressure to its true value and run HAMMER again. The difference between these results is due to the effect of vapor pressure.

Heating or pressurizing a fluid increases its vapor pressure-an important consideration in industrial applications. Consider both operating temperature and pressure when determining a liquid's vapor pressure limit. (For example, water boils at a lower temperature at high altitudes due to the lower atmospheric pressure and lower absolute vapor pressure. Similarly, water boils at a higher temperature in a pressure cooker and this increased steam temperature accelerates the cooking process.) This is why the parameter library provided with HAMMER often provides values for liquids at different temperatures.

Selecting the Transient Friction Method

The Transient Friction Method option enables you to select the methodology for determining flow resistance and friction losses during calculations. This can be accessed from the Transient Solver calculation options (Analysis > Calculation Options). Available methodologies include:

- Steady Friction
- Quasi-steady Friction
- Unsteady Friction, also known as transient friction

For more information on the theory for each of these friction models, see Friction and Minor Losses.

Steady State Friction Method

In HAMMER, a hydraulic transient analysis usually begins with an Initial Conditions (steady state) calculation, which computes the heads and flows for every pipe in the system. Prior to beginning the transient calculations, HAMMER automatically determines the friction factor based on this information:

If a pipe has zero flow at the initial steady-state, HAMMER use the Friction Coefficient specified in the Pipe Physical properties.

If a pipe has a nonzero flow at the initial steady-state, HAMMER automatically calculates a Darcy-Weisbach friction factor, f, based on the heads at each end of the pipe, the pipe length and diameter, and the flow in the pipe. It uses this calculated value in the transient simulation.

Note: HAMMER always uses the Darcy-Weisbach friction method in performing the hydraulic transient calculations, regardless of which method is specified in the Steady State/EPS Solver Calculation Options. If required, HAMMER will automatically convert the friction factors to the appropriate format.

Quasi-Steady Friction Method

The quasi-steady friction method uses variable Darcy-Weisbach friction factors, f, at each point along the system, so that friction losses for an instantaneous velocity match the friction losses which would occur for fully developed steady flows with the same cross-sectional average velocity. For more information, see Quasi-Steady Friction.

Tip: Quasi-steady and unsteady friction models are the result of current research by others. Results should be compared with those obtained with a steady friction model. Always use engineering judgement when interpreting transient simulation results.

Transient or Unsteady Friction

Compared to a steady state, fluid friction increases during hydraulic transient events because rapid changes in transient pressure increase turbulent shear. HAMMER can track the effect of fluid accelerations to estimate the attenuation of transient energy more closely than would be possible with quasi-steady friction.

Computational effort increases significantly if transient friction must be calculated for each time step. This can result in long model calculation times for large systems with hundreds of pipes or more. Typically, transient friction has little or no impact on the initial low and high pressures, and these are usually the largest ever reached in the system (provided the simulation does not involve a vapor pocket collapse).

Tip: The steady-state friction method yields conservative estimates of the extreme high and low pressures which usually govern the selection of pipe class and surge-protection equipment. However, if cyclic loading is an important design consideration, the unsteady friction method can yield less-conservative but rigorous estimates of recurring and decaying extremes.

For more information on the implementation of the transient friction method in HAMMER, see Unsteady or Transient Friction.

Engine Compatibility Calculation Option

Previous versions of the software had a calculation option called "Use EPANET Compatible Results?" which was used to turn off computational changes Bentley had made to the core engine calculation that would change results compared to the results for an equivalent model run in the US EPA's EPANET software. In the current version of the software, however, the "Use EPANET Compatible Results?" has been replaced by a new "Engine Compatibility" calculation option that offers 4 choices as follows:

1. WaterGEMS 2.00.12
2. WaterGEMS 2.00.10
3. EPANET 2.00.12
4. EPANET 2.00.10

Previously, the "Use EPANET Compatible Results?" option was functionally the same as having choices 2 and 4 only. When the previous property was set to false, you were using option 2. When the previous property was set to true, you were using option 4. For this release of the software we have extended the engine support to include compatibility modes that include the revised engine convergence algorithms in EPANET 2.00.12, in addition to keeping the old behavior that was based on EPANET 2.00.10.

The intent of each of the compatibility modes is as follows:

1. WaterGEMS 2.00.12 - Computation engine based on EPANET 2.00.12 with Bentley's own enhancements and features.
2. WaterGEMS 2.00.10- Computation engine based on EPANET 2.00.10 with Bentley's own enhancements and features.
3. EPANET 2.00.12 - Computational engine based on EPANET 2.00.12 including any Bentley enhancements and features that do not change hydraulic results compared to EPANET, for models that are able to be completely represented in EPANET.
4. EPANET 2.00.10 - Computational engine based on EPANET 2.00.12 including any Bentley enhancements and features that do not change hydraulic results compared to EPANET, for models that are able to be completely represented in EPANET.

For those interested in what each engine compatibility mode means in more detail we provide the following compatibility matrix.

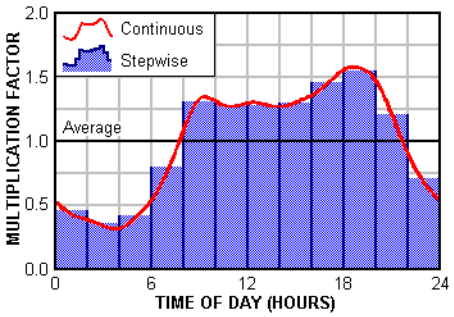
Item	WaterGEMS 2.00.12 Compatibility Mode	WaterGEMS 2.00.10 Compatibility Mode	EPANET 2.00.12 Compatibility Mode	EPANET 2.00.10 Compatibility Mode
Contains EPANET 2.00.12 computational enhancements for control valve convergence and water quality computational changes.	Yes. This is the *only* difference between WaterGEMS 2.00.12 mode and WaterGEMS 2.00.10 mode.	No	Yes. This is the biggest (but not only) difference between EPANET 2.00.12 mode and EPANET 2.00.10 mode.	No
Contains a number of computational improvements and enhancements over and above EPANET 2.0	Yes	Yes	Yes. Included if the improvement or enhancement did not change results compared to EPANET 2.00.12. e.g., results output control.	Yes. Included if the improvement or enhancement did not change results compared to EPANET 2.00.10. e.g., results output control.
Performance improvements for the calculation of models.	Yes	Yes	Yes. Performance changes do not affect results so they are always included.	Yes. Performance changes do not affect results so they are always included
Stability enhancements and potential crash bug fixes.	Yes	Yes	Yes. Stability changes do not affect results (of models that calculate in the presence of the bug) so they are always included.	Yes. Stability changes do not affect results (of models that calculate in the presence of the bug) so they are always included.
Supports ability to treat multi-point pump curves as piecewise linear curves (or power curves) as the user sees fit.	Yes	Yes	No. EPANET 2.00.12 chooses the representation of the curve based on the number of points in the curve.	No. EPANET 2.00.10 chooses the representation of the curve based on the number of points in the curve
Avoidance of reporting very large negative HGL/pressure for disconnected elements. (Avoids non sense results and problems with graph scales).	Yes	Yes	No	No
Improved pre-calculation data validation and checking.	Yes	Yes	Yes. Improved data validation do not affect results so it is always included.	Yes. Improved data validation do not affect results so it is always included.
Supports binary output files larger than 2Gb. Important for large models or long EPS runs.	Yes	Yes	Yes. Increases the size of output than can be produced compared to the official EPANET 2.00.12 release, but does not change results.	Yes. Increases the size of output than can be produced compared to the official EPANET 2.00.12 release, but does not change results.
Supports demands (inflow and outflow) on tank elements.	Yes	Yes	Yes. A warning is generated to advise that results may be different to the official EPANET 2.00.12 release because EPANET won't use the tank demand in the analysis.	Yes. A warning is generated to advise that results may be different to the official EPANET 2.00.12 release because EPANET won't use the tank demand in the analysis.

Patterns

The extended period analysis is actually a series of Steady State analyses run against time-variable loads such as sewer inflows, demands, or chemical constituents. Patterns allow you to apply automatic time-variable changes within the system. The most common application of patterns is for residential or industrial loads. Diurnal curves are patterns that relate to the changes in loads over the course of the day, reflecting times when people are using more or less water than average. Most patterns are based on a multiplication factor versus time relationship, whereby a multiplication factor of one represents the base value (which is often the average value).

Using a representative diurnal curve for a residence as illustrated below, we see that there is a peak in the diurnal curve in the morning as people take showers and prepare breakfast, another slight peak around noon, and a third peak in the evening as people arrive home from work and prepare dinner. Throughout the night, the pattern reflects the relative inactivity of the system, with very low flows compared to the average.

Typical Diurnal Curve



Note: This curve is conceptual and should not be construed as representative of any particular network.

There are two basic forms for representing a pattern: stepwise and continuous. A stepwise pattern is one that assumes a constant level of usage over a period of time, and then jumps instantaneously to another level where it remains steady until the next jump. A continuous pattern is one for which several points in the pattern are known and sections in between are transitional, resulting in a smoother pattern. For the continuous pattern in the figure above, the multiplication factor and slope at the start time and end times are the same. This is a continuity that is recommended for patterns that repeat.

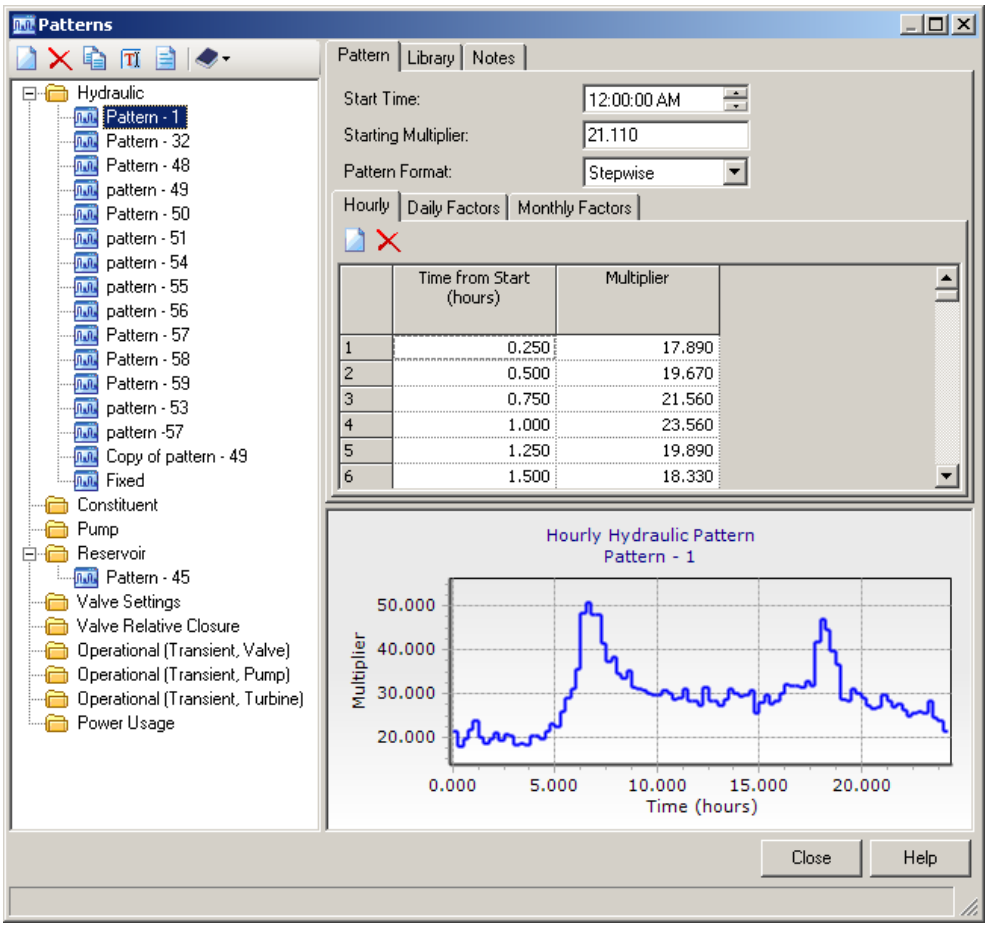
Because of the finite time steps used for calculations, this software converts continuous patterns into stepwise patterns for use by the algorithms. In other words for a time step a multiplier is interpolated from the pattern curve. That multiplier is then used for the duration of the time step, until a new multiplier is selected for the next time step.

Patterns provide a convenient way to define the time variable aspects of system loads. Patterns include:

- Pattern Manager

Pattern Manager

A pattern is a series of time step values, each having an associated multiplier value. During an extended period analysis, each time step of the simulation uses the multiplier from the pattern corresponding to that time. If the duration of the simulation is longer than the pattern, the pattern is repeated. The selected multiplier is applied to any baseline load that is associated with the pattern. You can also define daily and monthly multipliers for any pattern.



Patterns provide an effective means of applying time-variable system demands to the distribution model. The Pattern Manager allows you to create the following types of patterns:

- **Hydraulic**—This type of pattern can be applied to Junctions or Tanks. Use this pattern type to describe demand or inflow patterns over time.
- **Constituent**—This type of pattern can be applied to Reservoirs, Tanks, or Junctions. Use this pattern type to describe changes in Constituent Baseline Loads over time.
- **Pump**—This type of pattern can be applied to Variable Speed Pumps only. Use this pattern type to describe changes in the pump's Relative Speed Factor. In the Property dialog box for the pump, Is Variable Speed Pump needs to be set to True and the VSP type needs to be Pattern Based.
- **Reservoir**—This type of pattern can be applied to Reservoirs. Use this pattern type to describe changes in HGL over time, such as that caused by tidal activity or when the reservoir represents a connection to another system where the pressure changes over time.
- **Valve Settings**—This type of pattern can be applied to valves. Use this pattern type to describe changes to valve settings over time.
- **Valve Relative Closure**—This type of pattern can be applied to valves. Use this pattern type to describe changes to the relative closure of a valve over time.
- **Operational (Transient, Valve)**—This type of pattern can be applied to valves. Use this pattern to describe changes in a valve's status over time during a transient analysis.
- **Operational (Transient, Pump)**—This type of pattern can be applied to pumps. Use this pattern to describe changes in a pump's status over time during a transient analysis.
- **Operational (Transient, Turbine)**—This type of pattern can be applied to turbines.Uuse this pattern to describe changes in a turbine's status over time during a transient analysis.

  Note: In this program, an individual demand node can support multiple demands. Furthermore, each demand can be assigned any hydraulic pattern. This powerful functionality makes it possible to model any type of extended period simulation.

- **Power Usage**—This type of pattern can be applied to Power Meters in for use in energy management analysis.

The following management controls are located above the pattern list pane:

New 	Creates a new pattern of the highlighted type.
Delete 	Deletes the pattern that is currently highlighted in the list pane.
Rename 	Renames the pattern that is currently highlighted in the list pane.
Report 	Opens a report of the data associated with the pattern that is currently highlighted in the list pane.
Synchronization Options 	Browse the Engineering Library, synchronize to or from the library, import from the library or export to the library.

Tip: **Use the Report button to view or print a graph or detailed report of your pattern.**

The right half of the dialog consists of controls that allow you to define the settings for the pattern that is currently selected in the list of patterns on the left side of the dialog.

- **Start Time**—The first time step in the pattern. The start time format is a standard 24-hour clock. The format is Hour:Minute:Second AM or PM (e.g., 12:45:30 PM).
- **Starting Multiplier**—The multiplier value of the first time step point in your pattern. Any real number can be used for this multiplier (it does not have to be 1.0).
- **Pattern Format**—The following pattern formats are available:

  - **Stepwise**—The multiplier values are considered to be the average value for the interval between the specified time and the next time. Patterns using this format will have a staircase appearance. Multipliers are set at the specified time and held constant until the next point in the pattern.
  - **Continuous**—The multipliers are considered to be the instantaneous values at a particular time. Patterns using this format will have a curvilinear appearance. Multipliers are set at the specified time, and are linearly increased or decreased to the next point in the pattern.

Hourly patterns consist of a number of time step points, defined in the table below the Pattern Format control on the Hourly tab.

Note: The minimum time step for hourly patterns is 1 second. WaterCAD/WaterGEMS are not intended to be used for demand changes at such a short interval. In those cases, transient phenomena may dominate and those changes can be better modeled using HAMMER.

- **Time From Start**—The amount of time from the Start Time of the pattern to the time step point being defined.
- **Multiplier**—The multiplier value associated with the time step point.
- **Relative Closure**—The percentage of full flow that the valve allows at the associated time step point. This attribute is only available for **Operational (Transient, Valve)** pattern types.
- **Relative Speed Multiplier**—The percentage of full speed that the pump is running at during the associated time step point. This attribute is only available for **Operational (Transient, Pump)** pattern types.
- **Gate Opening Percent** —The percentage compared to fully open for the turbine gate opening at the associated time step point. This attribute is only available for **Operational (Transient, Turbine)** pattern types.

Daily and Monthly factors are defined in the same way as hourly ones, the difference being that rather than defining time steps you enter multipliers for each day of the week (for Daily patterns) or for each month of the year (for monthly patterns).

A graph of the currently selected pattern is displayed in the lower right corner of the dialog.

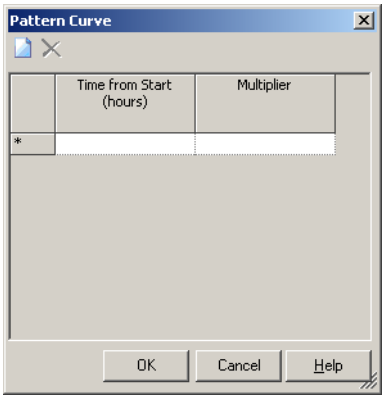
Note: **Patterns must begin and end with the same multiplier value. This is because patterns will be repeated if the duration of the Extended Period Analysis is longer than the pattern duration. In other words, the last point in the pattern is really the start point of the pattern's next cycle.**

An Extended Period Analysis is actually a series of Steady State analyses for which the boundary conditions of the current time step are calculated from the conditions at the previous time step. This software will automatically convert a continuous pattern format to a stepwise format so that the demands and source concentrations remain constant during a time step.

An individual node can support multiple hydraulic demands. Furthermore, each load can be assigned any hydraulic demand pattern. This powerful functionality makes it easy to combine two or more types of demand patterns (such as residential and institutional) at a single loading node.

Pattern Curve Editor

This dialog allows you to define pattern curves for the Patterns Engineering Library.



The following buttons are located above the time step points table on the left:

- **New**—Creates a new row in the time step points table.
- **Delete**—Deletes the currently highlighted row from the time step points table.

The time step points table contains the following columns:

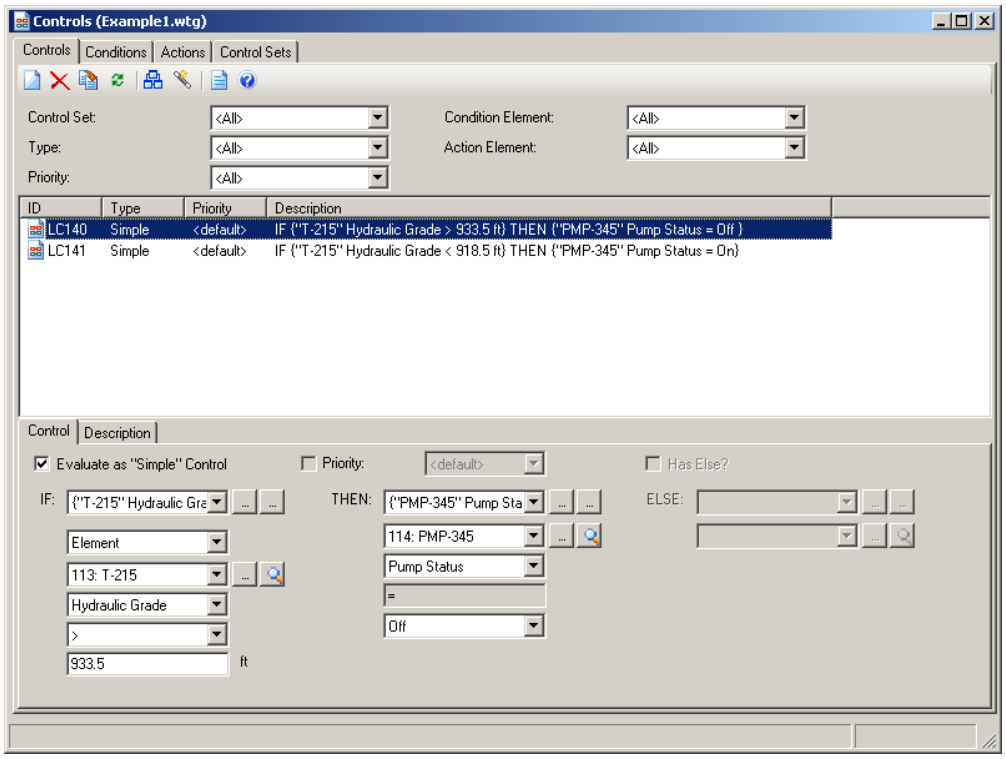
- **Time from Start**—Lets you specify the amount of time from the Start Time of the pattern to the time step point being defined.
- **Multiplier**—Lets you specify the multiplier value associated with the time step point.

Controls

Controls give you a way to specify for virtually any element based on almost any property of the system. Controls are included in a scenario when they are specified in the Operational Alternative. The controls become part of an Operational Alternative when you specify the name of a Control Set to use in a given Operational Alternative.

The Control Manager is the main work center for controls. The Control Manager manages all controls, conditions, actions, and control sets in the system. The Control manager allows you to define controls using advanced IF, AND, and OR condition logic, which can trigger any number of THEN or optional ELSE actions.

Choose Components > Controls to open the Control Manager.



The Control Manager consists of the following tabs:

- **Controls**—Manage all controls defined in the system.
- **Conditions**—Define the condition that must be met prior to taking an action.

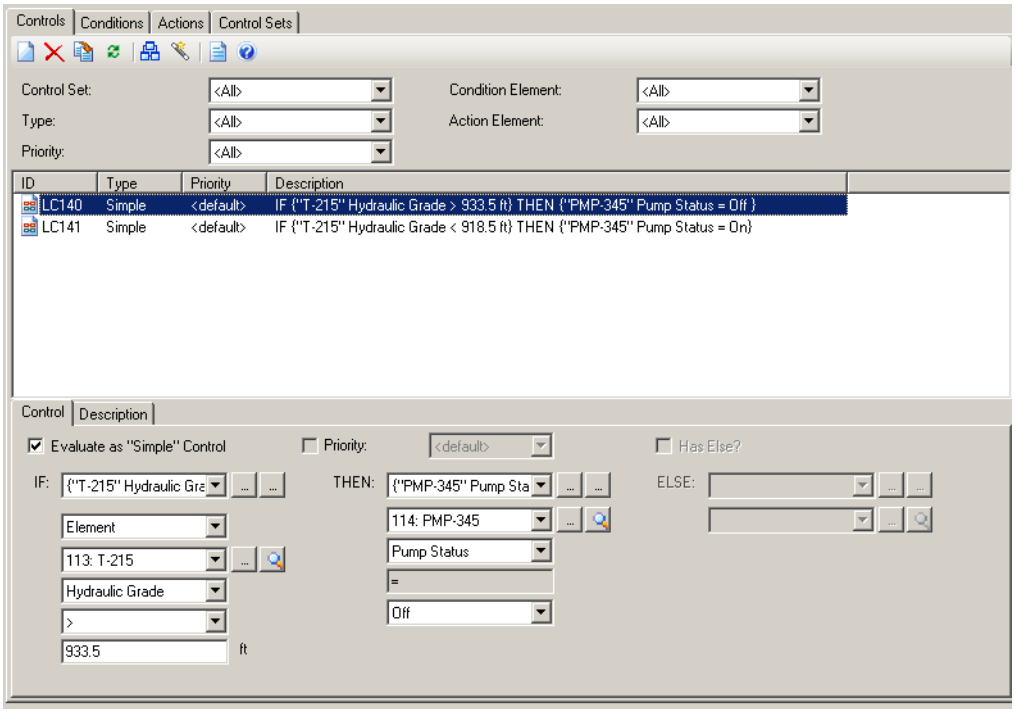
- **Actions**—Define what should be done to an element in the system in response to an associated control condition.
- **Control Sets**—Assign groups of controls to Control Sets.

Controls Tab

The Controls tab allows you to manage all controls defined in the system. Controls can be one of two types: simple or logical. Simple controls are made up of an IF condition and a THEN action statement. Logical controls are made up of an IF condition, a THEN action, and an optional ELSE action, and can be assigned a priority for resolving potential conflicts between logical controls.

Controls, Conditions, and Actions are assigned a non-editable application-provided ID (e.g., LC01).

The Controls tab is divided into sections:



•The pane in the center of the dialog box is the Controls List. This list displays a list of all Logical Controls defined in the system.

- Located above the Controls List is a toolbar with the following buttons:
  - **New**—Creates a new control.
  - **Delete**—Deletes the highlighted control. You can hold down the Ctrl key while clicking on items in the list to select multiple entries at once.

  - **Duplicate**—Creates a copy of the currently highlighted control.
  - **Refresh**—Refreshes the highlighted control
  - **Control Sets**—Opens the Control Set dialog.
  - **Control Wizard**—Opens the Control Wizard.
  - **Report**—Generates a summary of the selected control, listing the ID, conditions, actions, and elements incorporated into the control.
- Below the toolbar is a set of filters that allow you to only display controls that meet criteria defined by the filter settings. The following filters are available:
  - **Type**—When a Type filter other than <All> is specified, only controls of that type will be displayed in the Controls list.
  - **Priority**—When a Priority filter other than <All> is specified, only controls of that priority will be displayed in the Controls list.
  - **Condition Element**—When a Condition filter other than <All> is specified, only controls containing the selected Condition element will be displayed in the Controls list.
  - **Action Element**—When an Action filter other than <All> is specified, only controls containing the selected Action element will be displayed in the Controls list.

You can edit or create controls consisting of an IF condition, a THEN action, and an optional ELSE action. The lower pane is split into sections:

- **Evaluate as Simple Control**—Turn on in order to evaluate the condition as a simple control.
  - **IF Condition**—The drop-down list allows you to choose from a list of conditions that have already been created in the Conditions tab.
  - **THEN Action**—The drop-down list allows you to choose from a list of actions that have already been created in the Actions tab.
  - **ELSE Action (optional)**—The ELSE action is used when the conditions for the control are not met. To specify an ELSE action, click the check box to activate the drop-down list. The drop-down list allows you to choose from a list of actions that have already been created in the Actions tab.
- **Priority**—This area of the dialog box is optional. To set a priority for the control being created, turn on to activate the priority drop-down list. You can set a priority of 1-5, 5 being the highest priority. If multiple controls meet a certain condition and they have conflicting actions, the control with the highest priority will be used.

Note: **At calculation time, the priority is used to determine the logical control to apply when multiple controls require that conflicting actions be taken. Logical controls with identical priorities will be prioritized based on the order they appear in the Logical Control Set alternative. A rule without a priority value always has a lower priority than one with a value. For two rules with the same priority value, the rule that appears first is given the higher priority.**

Relative speed pump patterns take precedence over any controls (simple or logical) that are associated with the pump.

Hovering the mouse cursor over a control in the list will open a tooltip which displays the conditions and actions that make up that control.

When creating a new condition or action for a new control, the condition and action input fields will be initialized with the data used in the last condition or action that was created.

Once created, the Logical Control will be assigned an application generated ID (e.g., LC04).

- **Description**—This area is preset with a default description. There is an option to change the default description. To do so, turn on to activate the description field, and enter your description in the text box.
- **Summary**—This area of the dialog box displays a description of the control.
- **Status Pane**—When one or more filters are active, the lower left corner of the dialog will show the number of controls currently displayed out of the number of total controls. Additionally, a FILTERED flag is displayed in the lower right corner.

Logical, or rule-based controls allow far more flexibility and control over the behavior of your network elements than is possible with simple controls. This is accomplished by allowing you to specify one or more conditions and then link these to one or more Actions by using logical IF, AND, THEN, OR, and ELSE statements.

Note: **Logical Controls are not executed during Steady State analyses.**

Logical controls consist of any combination of simple conditions and simple actions. Controls are defined as:

IF:	Condition 1 AND condition 2 OR condition 3 AND condition 4, etc., where condition X is a a condition clause.
THEN:	Action 1 AND action 2, etc. where action X is an action clause.

ELSE (Optional):	Action 3 AND action 4, etc. where action X is an action clause.
Priority (Optional):	Priority where priority is a priority value (1 to 5, 5 being the highest priority).

In addition to the high level of flexibility provided by allowing multiple conditions and actions, the functionality of Logical controls is also enhanced by the range of Condition types that are available. You can activate the stated actions based on element demands, element hydraulic grade or pressure, system demand, clock time, time from start, tank level, or time to fill or drain a tank.

You can also create composite conditions and actions. You can cause actions to be performed when multiple conditions are met simultaneously, or when one or the other conditions are met. You can also activate multiple actions when a single condition is met.

EXAMPLE:

To create a logical control in which a pump (PMP-1) is turned on when the level in a tank (T-1) falls below a specified value (5 ft.) or when the system demands exceed a certain level (5000 gpm):

- **Conditions**—Because this control needs to be triggered by multiple conditions, a Composite Condition is chosen. In this instance, the operator OR is chosen to link the conditions, because the pump should be turned on if either condition is true.

  IF condition—{T-1 Level < 5 ft.}

  OR condition—{System Demand > 5000 gpm}

- **Actions**—Because this control has a single desired outcome if one of the conditions is met, a simple action is chosen. The first action in a logical control is always linked to the conditions by a logical THEN statement. In this instance, an ELSE action will also be used, to keep the pump off if neither of the conditions is true.

  THEN action—{PMP-1 Status = On}

  ELSE action—{PMP-1 Status = Off}

The finished logical control looks like this:

IF {T-1 Level < 5 ft.} OR {System Demand > 5000 gpm} THEN {PMP-1 Status = On} ELSE {PMP-1 Status = Off}

This example illustrates the power of using logical controls. To achieve the same functionality using simple controls, you would need to create four separate controls—one to turn the pump on if the tank level is below the specified value, one to turn the pump off if the tank level is above a specified value, one to turn the pump on if the system demand is greater than the specified value, and one to turn the pump off if the system demand is less than the specified value.

Tip: **Use the optional ELSE field to cause actions to be performed when the conditions in the control are not being met. For example, if you are creating a control that states, "If the level in Tank 1 is less than 5 ft., Then turn Pump 1 On," use an ELSE action to turn the pump off if the tank level is above 5 ft.**

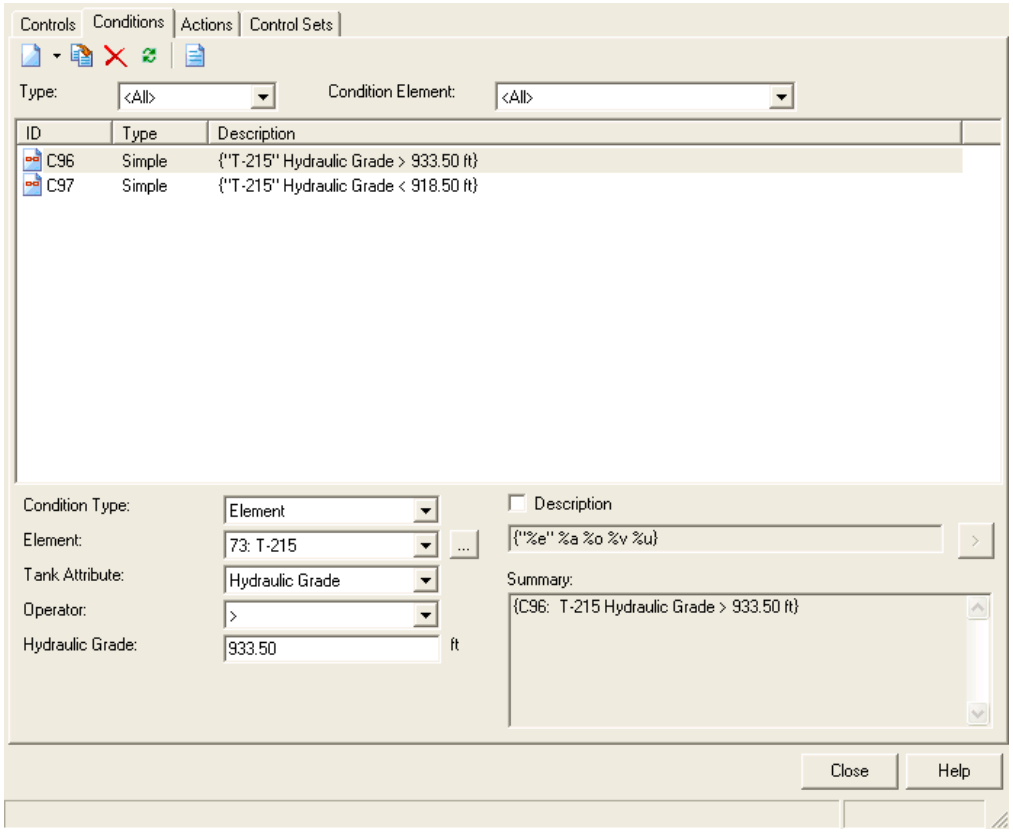
Note: **Logical Controls are not executed during Steady State analyses.**

When defining a logical control, you have the option to share conditions and/or actions. In other words, more than one control can reference the same condition or action. Keep in mind that when you change an underlying condition or action, it will affect all controls that reference that condition or action.

Conditions Tab

Conditions allow you to define the condition that must be met prior to taking an action. The Conditions tab provides a list of all conditions defined in the system. There are two types of conditions: simple conditions and composite conditions.

The Conditions tab is divided into sections:



- The pane in the middle of the dialog box is the Conditions List. The Conditions List displays a list of all logical conditions defined in the system. The list contains four columns: ID (the application defined id, e.g., C01 for simple, CC01 for composite), Type (simple or composite), description, and references (logical control references).
- Located above the Conditions List is a toolbar with the following buttons:
  - **New**—Create a simple or composite condition.
  - **Duplicate**—Copy the selected condition.
  - **Delete**—Deletes the selected condition. You can hold down the Ctrl key while clicking on items in the list to select multiple entries at once.
  - **Refresh**—Refreshes the selected condition.
  - **Report**—Generates a summary of the selected condition.
- Below the toolbar is a set of filters that allow you to only display controls that meet criteria defined by the filter settings. The following filters are available:
  - **Control Set**—When a control set is specifed, only conditions that are a component of that control set are displayed in the Conditions list.

  - **Type**—When a Type filter other than <All> is specified, only conditions of that type will be displayed in the Conditions list.
  - **Condition Element**—When a Condition filter other than <All> is specified, only conditions containing the selected Condition element will be displayed in the Conditions list.

- The controls used to create or edit a condition vary depending on whether the condition is simple or composite:

Simple Conditions

The input fields for a simple condition change depending on the condition type that is selected in the condition Type field. The Simple Condition Types and the corresponding input data are as follows:

Element—This will create a condition based on specified attributes at a selected element. The fields available when this condition type is selected are as follows:

- **Element**—The Element field allows you to specify which element the condition will be based upon, and provides three methods of choosing this element. The drop-down list displays elements that have been used in other logical controls, the Ellipsis (…) button, which opens the Single Element Selection dialog box, and the Select From Drawing button, which allows you to select the element using the graphical Drawing view.

Attribute—This field displays the available attributes for the element type currently specified in the Element field.

- **Pressure Junctions**—The following attributes are available for use when a Junction is chosen in the Element field:
  - **Demand**—This attribute is used to create a condition based on a specified demand at the corresponding junction (e.g., If J-1 has a demand…).
  - **Hydraulic Grade**—This attribute is used to create a condition based on a specified hydraulic grade at the corresponding junction (e.g., If J-1 has a hydraulic grade of…).
  - **Pressure**—This attribute is used to create a condition based on a specified pressure at the corresponding junction (e.g., If J-1 has a pressure of…).

- **Pumps**—The following attributes are available for use when a Pump is chosen in the Element field:
  - **Discharge**—This attribute is used to create a condition based on a specified rate of discharge at the corresponding pump (e.g., If PMP-1 has a discharge of…).

  - **Setting**—This attribute is used to create a condition based on the Relative Speed Factor of the corresponding pump (e.g., If PMP-1 has a relative speed factor of 1.5…).
  - **Status**—This attribute is used to create a condition based on the status (On or Off) of the corresponding pump (e.g., If PMP-1 is On…).

Note: Relative Speed Pump patterns take precedence over any controls (Simple or Logical) that are associated with the pump.

- **Tanks**—The following attributes are available for use when a Tank is chosen in the Element field:
  - **Demand**—This attribute is used to create a condition based on a specified demand at the corresponding tank. For tanks, this demand can represent an inflow or outflow (e.g., If T-1 has a demand…).
  - **Hydraulic Grade**—This attribute is used to create a condition based on a specified hydraulic grade at the corresponding tank (e.g., If T-1 has a hydraulic grade of…).
  - **Pressure**—This attribute is used to create a condition based on a specified pressure at the corresponding tank (e.g., If T-1 has a pressure of…). Note that tank pressure is calculated referenced from the tank *base elevation* and that the generic elevation field for tanks is not considered. This is done to allow the modeling of elevated tanks. For non-elevated tanks elevation is the base elevation.
  - **Level**—This attribute is used to create a condition based on a specified water level at the corresponding tank (e.g., If the water in T-1 is at a level of…).
  - **Time to Drain**—This attribute is to create a condition based on the amount of time required for the tank to drain (e.g., If T-1 drains in X hours…).
  - **Time to Fill**—This attribute is to create a condition based on the amount of time required for the tank to fill (e.g., If T-1 fills in X hours…).

- **Reservoirs**—The following attributes are available for use when a Reservoir is chosen in the Element field:
  - **Demand**—This attribute is used to create a condition based on a specified demand at the corresponding reservoir. For reservoirs, this demand can represent an inflow or outflow (e.g., If R-1 has a demand…).
  - **Hydraulic Grade**—This attribute is used to create a condition based on a specified hydraulic grade at the corresponding reservoir (e.g., If R-1 has a hydraulic grade of…).
  - **Pressure**—This attribute is used to create a condition based on a specified pressure at the corresponding reservoir (e.g., If R-1 has a pressure of…).

- **Pipes**—The following attributes are available for use when a Pipe is chosen in the Element field:
  - **Discharge**—This attribute is used to create a condition based on a specified rate of discharge at the corresponding pipe (e.g., If P-1 has a discharge of…).
  - **Status**—This attribute is used to create a condition based on the status (Open or Closed) of the corresponding pipe (e.g., If P-1 is Open…).
- **Valves**—The following attributes are available for use when a valve is chosen in the Element field:
  - **Discharge**—This attribute is used to create a condition based on a specified rate of discharge at the corresponding valve (e.g., If PRV-1 has a discharge of…).

  Note: The Setting attribute is not available when a GPV is selected in the Element field.
- **Setting**—This attribute is used to create a condition based on the setting of the corresponding valve. The type of setting will change depending on the type of valve that is chosen. The valves and their associated setting types are as follows:
  - **PRV**—Choosing the Setting attribute in conjunction with a PRV will create a condition based on a specified pressure at the PRV (e.g., If PRV-1 has a pressure of…).
  - **PSV**—Choosing the Setting attribute in conjunction with a PRV will create a condition based on a specified pressure at the PRV (e.g., If PSV-1 has a pressure of…).
  - **PBV**—Choosing the Setting attribute in conjunction with a PRV will create a condition based on a specified pressure at the PRV (e.g., If PBV-1 has a pressure of…).
  - **FCV**—Choosing the Setting attribute in conjunction with a PRV will create a condition based on a specified rate of discharge at the PRV (e.g., If FCV-1 has a discharge of…).
  - **TCV**—Choosing the Setting attribute in conjunction with a PRV will create a condition based on a specified headloss coefficient at the PRV (e.g., If TCV-1 has a headloss of…).
- **Status**—This attribute is used to create a condition based on the status (Closed or Inactive) of the corresponding valve (e.g., If PRV-1 is Inactive…).

System Demand—This will create a condition based on the demands for the entire system. The fields available when this condition type is selected are:

- **Operator**—This field allows you to specify the relationship between the Attribute and the target value for that attribute. The choices include Greater Than (>), Greater Than Or Equal To (>=), Less Than (<), Less Than Or Equal To (<=), Equal To (=), or Not Equal To (<>).
- **System Demand**—This field lets you set a system-wide demand.

Clock Time—This will create a condition based on the clock time during an extended period simulation. If the extended period simulation is for a period longer than 24 hours, this condition will be triggered every day at the specified time.

- **Operator**—This field allows you to specify the relationship between the Attribute and the target value for that attribute. The choices include Greater Than (>), Greater Than Or Equal To (>=), Less Than (<), Less Than Or Equal To (<=), Equal To (=), or Not Equal To (<>).

Time From Start—This will create a condition based on the amount of time that has passed since the beginning of an extended period simulation. The following fields are available when this condition type is selected:

- **Operator**—This field allows you to specify the relationship between the Attribute and the target value for that attribute. The choices include Greater Than (>), Greater Than Or Equal To (>=), Less Than (<), Less Than Or Equal To (<=), Equal To (=), or Not Equal To (<>).

Target Value—This field's label will change depending on the attribute that is chosen. The value entered here is used in conjunction with the operator that is chosen to determine if the condition has been met.

Description—This area of the dialog box is preset with a default description. There is an option to change the default description. To do so, click the check box to activate the description field, and enter your description in the text box. Additionally, the description field supports the following expandable masks:

%#	ID
%e	Element
%a	Attribute
%o	Operator
%v	Value
%u	Unit

Note: Click the description list box to select one of the predefined masks.

Aside from reducing the amount of data input, using these masks provides the additional benefit of automatically updating the corresponding information when changes are made to the various condition components.

Summary— This area of the dialog box displays an automatically updated preview of the expanded description.

Composite Conditions

When a Composite Condition is being defined or edited, the lower part of the dialog box is comprised of a two column table and two buttons. The buttons are as follows:

- **Insert**—Adds a new row to the Condition list.
- **Delete**—Deletes the highlighted row from the Condition list.
- **Refresh**—Updates the referenced conditions.

The table contains two columns, as follows:

- **Operator**—This column allows you to choose the way in which the related Condition logic will be evaluated. The available choices are **If**, **And**, and **Or**.

  Note: The first condition in the list will use the If operator. Any additional conditions will allow you to choose between AND and OR.

  Any combination of AND and OR clauses can be used in a rule. When mixing AND and OR clauses, the OR operator has higher precedence than AND. Therefore, "IF A or B and C" is equivalent to "IF (A or B) and C". If the interpretation was meant to be IF A or (B and C), this can be expressed using two Logical Controls: Logical Control 1: "IF A THEN..." and Logical Control 2: "IF B AND C THEN..."

- **Condition**—The drop-down list allows you to choose a condition that was already created beforehand.

Description—This area of the dialog box is preset with a default description. There is an option to change the default description. To do so, click the check box to activate the description field, and enter your description in the text box. Additionally, the description field supports the following expandable masks:

%#	ID
%v	Value

Aside from reducing the amount of data input, using these masks provides the additional benefit of automatically updating the corresponding information when changes are made to the various condition components.

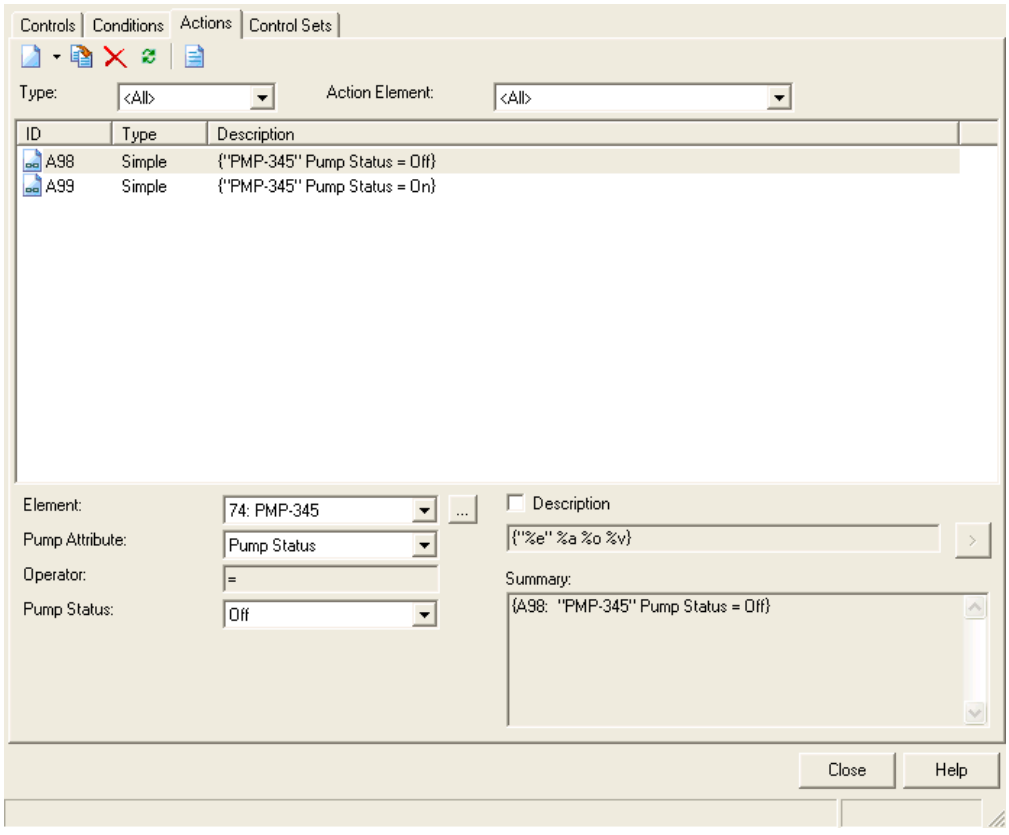
Note: Click the description list box to select one of the predefined masks.

Summary—This area of the dialog box displays an automatically updated preview of the expanded description.

Actions Tab

Actions allow you to define what should be done to an element in the system in response to an associated control condition. The Actions tab provides a list of all actions defined in the system. There are two types of actions: simple actions and composite actions. Actions have an application-provided non-editable ID (e.g., A01 for simple, AA01 for composite).

The Actions tab is divided into sections:



- The Actions List displays a list of all logical actions defined in the system. The list contains four columns: ID (the application defined ID, e.g., A01 for simple, AA01 for composite), Type (simple or composite), description, and references (logical control references).
- Located above the Conditions List is a toolbar with the following buttons:
  - **New**—Opens the New Logical Action dialog box, where you can create a new logical action.
  - **Edit**—Depending on whether a simple or composite action is highlighted, this button opens the Simple Logical Action or Composite Logical Action dialog box, which allows you to edit the highlighted action.
  - **Delete**—Deletes the highlighted action. You will be prompted to confirm this action. You can hold down the Ctrl key while clicking on items in the list to select multiple entries at once.
  - **Find**—Opens the Find Logical Action dialog box, which allows you to find a particular action based on a variety of criteria.
  - **Report**—Generates a summary of the highlighted action.

  – Below the toolbar is a set of filters that allow you to only display controls that meet criteria defined by the filter settings. The following filters are available:
    - **Control Set**—When a control set is specifed, only actions that are a component of that control set are displayed in the Actions list.
    - **Type**—When a Type filter other than <All> is specified, only actions of that type will be displayed in the Actions list.
    - **Action Element**—When an Action Element filter other than <All> is specified, only actions containing the selected Element will be displayed in the Actions list.
- The controls used to create or edit an action vary depending on whether the action is simple or composite:

Simple Actions

The following controls are used to define or edit Simple Actions:

- **Element**—The Element field allows you to specify which element the action will be based upon and provides three methods of choosing this element. The drop-down list displays elements that have been used in other logical controls, the Ellipsis (…) button, which opens the Single Element Selection box, and the Select From Drawing button, which allows you to select the element using the graphical Drawing view.
- **Attribute**—This field displays the available attributes for the element type specified in the Element field. Not all attributes are available for all element types. The available attributes include:

– **Status** – This attribute is used to change the status of a pipe, pump, or valve when the related conditions are met. The available choices are dependant on the element type.

– **Setting**—This attribute is used to change the settings of a pump or valve when the related conditions are met. The setting type varies depending on the type of element.

Note: Pipes can only utilize the Status Attribute, Pumps and all Valves except for the GPV can utilize either the Status or Setting Attribute. GPVs can only use the Status Attribute.

For all valves except for the GPV, there is no explicit Active status with which to base a control upon—the status choices are Inactive or Closed. After a control sets a valve to Inactive or Closed, to reactivate the valve another control must be created with a Setting attribute. This is because a valve cannot be set to Active, but must have specific input data to work with.

For GPVs, there is no Inactive setting. GPVs can only be set to Active or Closed. If the GPV is not closed, the valve will always produce the headlosses associated with it through the Head-Discharge Points table.

- **Operator**—The operator for logical actions is always EQUAL TO (=).
- **Attribute Value**—This field's label will change depending on the attribute that is chosen. Depending on the element type and the attribute that was chosen, the input field may also change to a drop-down list, which contains the possible settings for that element. Not all settings are available for all element types.

Note: Pipes can be set to Open or Closed, Pumps can be set to On, Off, or have their relative speed factors increase or decrease. GPVs can be set to Active or Closed. All other valves can be set to Inactive, Closed, or have their respective settings changed, depending on the Valve type.

Description—This area of the dialog box is preset with a default description. There is an option to change the default description. To do so, click the check box to activate the description field, and enter your description in the text box. Additionally, the description field supports the following expandable masks:

%#	ID

%e	Element
%a	Attribute
%o	Operator
%v	Value (and Unit, if applicable)

Aside from reducing the amount of data input, using these masks provides the additional benefit of automatically updating the corresponding information when changes are made to the various control components.

Note: Click the description list box to select one of the predefined masks.

Summary—This area of the dialog displays an automatically updated preview of the expanded description.

Composite Actions

When a Composite Action is being defined or edited, the lower section of the dialog box is comprised of a single column table and two buttons. The Table contains a list of the Actions to be used. Each row is a drop-down list that allows you to choose an action that was already created beforehand.

- **Insert**—Adds a new row to the Action list
- **Delete**—Deletes the highlighted row from the Action list.

Description—This area of the dialog box is preset with a default description. There is an option to change the default description. To do so, click the check box to activate the description field, and enter your description in the text box. Additionally, the description field supports the following expandable masks:

%#	ID
%v	Value

Aside from reducing the amount of data input, using these masks provides the additional benefit of automatically updating the corresponding information when changes are made to the various control components.

Note: Click the description list box to select one of the predefined masks.

Composite logical actions consist of multiple simple logical actions. These actions are linked with an AND statement.

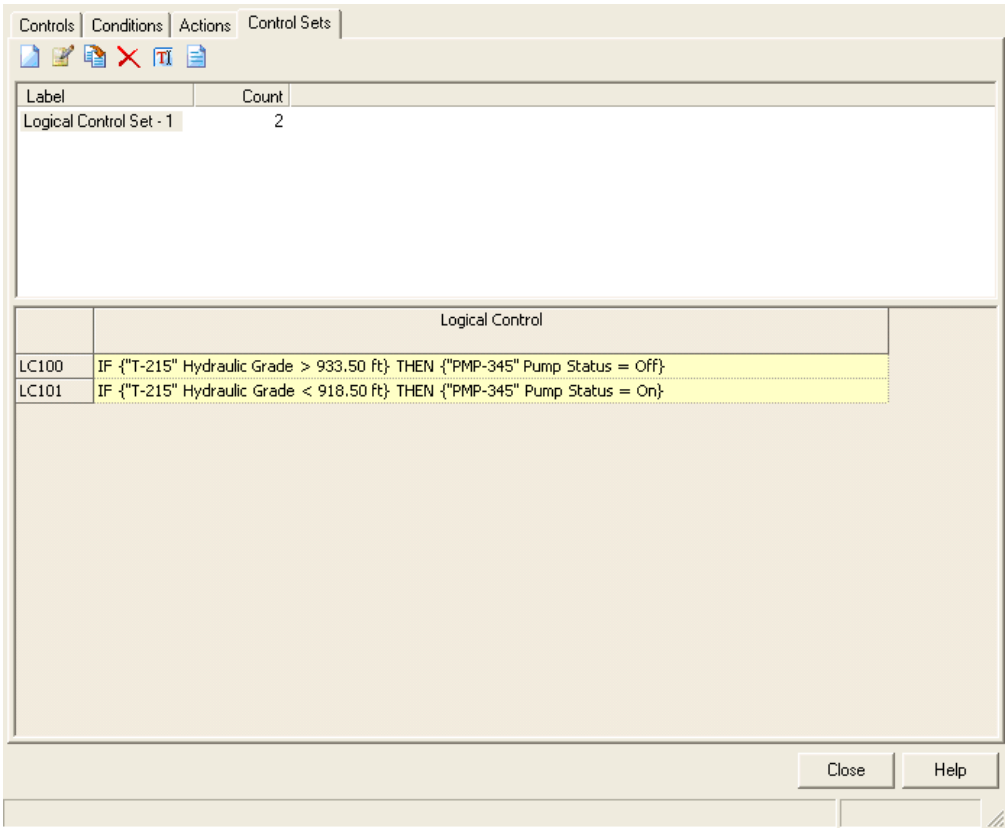
Summary—This area of the dialog box displays an automatically updated preview of the expanded description.

Control Sets Tab

The Control Sets tab allows you to create, modify and manage control sets. Control sets are a way to organize your controls, and also provide the means to use different controls in different scenarios.

A Control Set is made up of one or more control statements (called Controls) of the form: **If (condition) then (action) else (action)**. The actions and conditions are defined under the Conditions or Actions tab under control.

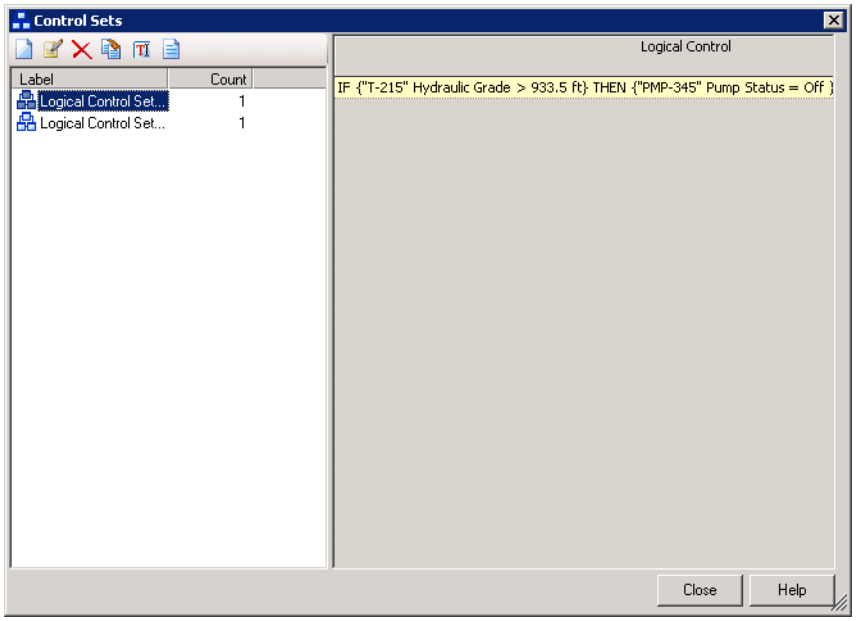
The following options are available in this dialog box:



- **New**—Opens the Logical Control Set editor dialog box. From this window, you can add previously created logical controls to the new control set.
- **Edit**—Opens the Logical Control Set editor dialog box, which allows you to edit the highlighted control set.
- **Duplicate**—Prompts for a name, then opens the Logical Control Set editor to allow you to add or remove controls from the control set.
- **Delete**—Deletes the highlighted control set. You will be prompted to confirm this action. You can hold down the Ctrl key while clicking on items in the list to select multiple entries at once.
- **Rename**—Allows you to rename the highlighted control set.
- **Report**—Generates a summary of the highlighted control set, listing the ID, conditions, actions, and elements for all of the logical controls contained within the control set.

Control Sets Dialog Box

This dialog allows you to create and manage control sets.



The dialog consists of a list pane that displays all of the control sets defined for the current project, a series of buttons that allow you to create, edit, manage, and report on the control sets, and a display pane that shows the details of the currently selected control set.

The following controls are located above the control set list pane:

New



Opens the Logical Control Set Editor, allowing you to define a new control set.

Edit



Opens the Logical Control Set Editor, allowing you to edit the currently highlighted control set.

Delete



Deletes the control set that is currently highlighted in the list pane.

Duplicate



Creates a copy of the control set that is currently highlighted in the list pane.

Rename



Renames the control set that is currently highlighted in the list pane.

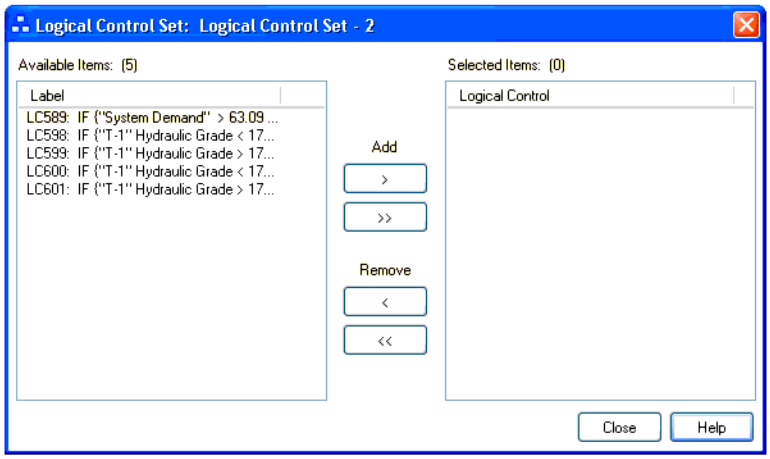
Report



Opens a report of the data associated with the control set that is currently highlighted in the list pane.

Logical Control Sets Editor Dialog Box

The Logical Control Set Editor is divided into two panes.

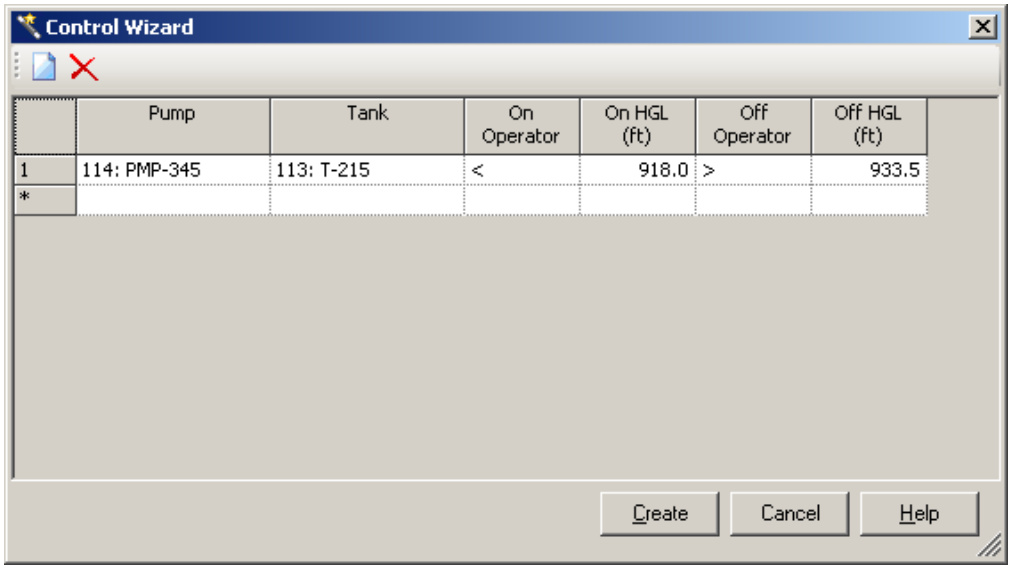


The left pane, labeled Available Items, contains a list of all of the logical controls that have been created in the current project. To add controls to the Selected Items pane on the right, highlight the desired controls and click the [>] button under Add. To add all of the controls to your Logical Control set, click the [>>] button under Add. To remove a control from the Selected Items pane, highlight it and click the [<] button under Remove. To remove all controls from the Selected Items pane, click the [<<] button under Remove.

Note: Priority is based upon the order that the controls appear in this dialog box. The first control in the control set has the highest priority, and so on. Any control with a set priority will overrule any control with no set priority.

Control Wizard

The Control Wizard lets you quickly create pump controls based on tank HGL.



The dialog consists of a table containing the following columns:

- **Pump**: Choose the pump to be controlled. You can select it from the menu, click the ellipsis (...) button to use the Find function, or click the cursor button to select the pump from the drawing.
- **Tank**: Choose the controlling tank. You can select it from the menu, click the ellipsis (...) button to use the Find function, or click the cursor button to select the tank from the drawing.
- **On Operator**: This field allows you to specify the relationship between the HGL and the target On value for the HGL. The choices include Greater Than (>), Greater Than Or Equal To (>=), Less Than (<), Less Than Or Equal To (<=), Equal To (=), or Not Equal To (<>).
- **On HGL**: The HGL value at which the pump turns on.
- **Off Operator**: This field allows you to specify the relationship between the HGL and the target Off value for the HGL. The choices include Greater Than (>), Greater Than Or Equal To (>=), Less Than (<), Less Than Or Equal To (<=), Equal To (=), or Not Equal To (<>).
- **Off HGL**: The HGL at which the pump turn off.

Click the New button above the table to add a new row; click the Delete button to remove the currently selected row.

Active Topology

The Bentley WaterCAD V8*i* Active Topology feature lets you create alternatives in which selected elements are displayed differently in the drawing view.

While these elements are in the inactive state, they are not evaluated in network calculations. This ability allows you to easily create before and after scenarios for proposed construction projects and test the redundancy of existing networks.

While elements are inactive, they are not included in any hydraulic equations. Inactive elements are also not evaluated when generating contour plots, and are not available for inclusion while generating profiles. Inactive elements are differentiated visually from Active ones in the main drawing pane, in the Aerial View window, and in either of the plan view types. When generating project inventory reports, element details reports, or element results reports, inactive elements are not included.

Inactive elements will not appear in the corresponding tabular reports, unless the Include Inactive Topology option is turned on. The default setting does not include inactive elements. Inactive elements are still available for inclusion in selection sets.

Any changes made to the Active Topology are applied to the Active Topology Alternative associated with the current scenario, and an unlimited number of active topology alternatives can be created.

Active Topology Selection Dialog Box

While it is possible to make elements active or inactive by:

1.checking or unchecking the "Is active?" box in the alternative manager under the Active Topology Manager,

2. unchecking the "Is active?" box in a FlexTable, or

3. picking True of False in property grid next to "Is active?" for individual elements,

another way of making elements active or inactive is the Active Topology Selection Tool, which is accessed under Tools > Active Topology Selection.

When you select the Active Topology Selection command, a Select tool opens. Selecting elements at this time can make them active or inactive according to the commands below.

Making an element "inactive" means that the element remains in the data file but it is not included in any hydraulic analysis calculations. Inactive elements will appear in FlexTables but calculated values will be set to NA.

Changing the active status using this tool only affects the Active Topology Alternative of the current scenario.



The Select tool consists of the following controls:

Done 	Select **Done** when you are finished selecting elements to bring you back to the Active Topology Selection dialog box.
Add 	This option is the default mode when you click the **Select From Drawing** button. Clicking elements while in this mode selects (highlights) elements, making them Inactive. Clicking on an element that is already inactive causes the tool to give a beep and the element remains inactive.
Remove 	While in this mode, clicking elements deselects them, making them Active. Clicking on active elements has no effect.
Query 	Allows you to select elements using a query.
Find 	Opens the Find dialog, allowing you to find the desired element(s).
Clear 	Removes all elements from the inactive elements pane, thereby causing all elements to become active in the current scenario.

The Done, Add, and Remove commands are also available from the right-click context menu while the Select tool is active.

Note: Selecting a node element to become Inactive will also select all adjacent pipes to become Inactive. This is because all pipes must end at a node.

In AutoCAD mode, you cannot use the right-click context menu command Repeat to re-open the Active Topology Selection dialog box.

External Tools

External Tools allows you to manage commands which you can run directly from within WaterCAD V8*i*. A command can be anything you can double-click to run from within Windows Explorer, such as a program (e.g. Notepad.exe), a folder (e.g. C:\Windows\), a document (e.g. a .txt or a .doc file), or an internet resource (e.g. www.bentley.com).

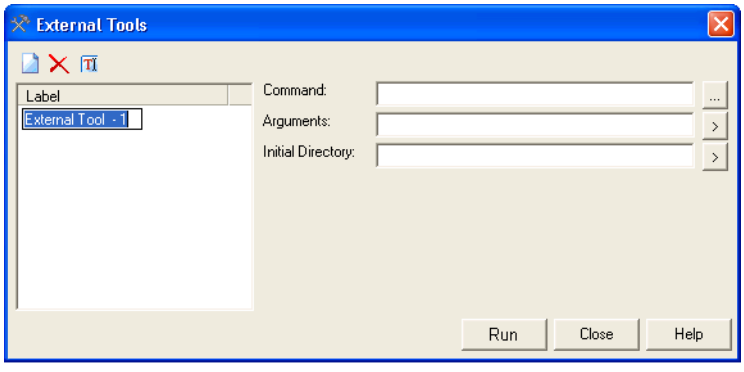
The External Tool Manager consists of the following elements:

- **External Tool List Pane**—This pane lists the external tools that have been created.
- **New**—Creates a new external tool in the list pane.
- **Delete**—Deletes the currently highlighted external tool.
- **Rename**—Allows you to rename the currently highlighted external tool.
- **Command**—This field allows you to enter the command you would like to run. Click the ellipsis button to open the standard Windows Open dialog to select an item interactively.
- **Arguments**—This optional field allows you to enter command line variables to be passed to your command when run. Arguments containing spaces must be enclosed in quotes.
- **[>] Predefined Arguments**—The button located to the right of the Arguments field provides a submenu of predefined arguments. You can concatenate directory and filename arguments to specify a full path name. The available predefined arguments are:
  - **Project Directory**—This predefined argument expands to the directory where your project was last saved. The argument string is %(ProjDir). e.g. C:\Users\MyUser\Documents\Bentley\WaterGEMS\ .
  - **Project File Name**—This predefined argument expands to the file name of your project when it was last saved. The argument string is %(ProjFileName). E.g. Project.wtg.

  - **Project Store File Name**—This predefined argument expands to the datastore filename of your project when it was last saved. The argument string is %(ProjStoreFileName). E.g. Project.wtg.sqlite.
  - **Project Working Directory**—This predefined argument expands to the directory where your project is located when it is being edited. The argument string is %(ProjWorkingDir). E.g. C:\Users\MyUser\AppData\Local\Temp\Bentley\WaterGEMS\.
  - **Project Working File Name**—This predefined argument expands to the filename of your project when it is being edited. The argument string is %(ProjWorkingFileName). E.g. Project.wtg.$$$.
  - **Project Store Working File Name**—This predefined argument expands to the datastore filename of your project when it is being edited. The argument string is %(ProjWorkingStoreFileName). E.g. Project.wtg.sqlite.$$$.
  - **Working Directory**—This predefined argument expands to the top-level temporary working directory. The argument string is %(ProjWorkDir).

- **Initial Directory**—Specifies the initial or working directory of the tool or command.
- **[>] Predefined Initial Directories**—The button located to the right of the Initial Directory field provides a submenu of predefined directory variables. The available predefined directory variables are:
  - **Project Directory**—The variable string is %(ProjDir). Refer to the Predefined Arguments above for more information.
  - **Project Working Directory**—The variable string is %(ProjWorkingDir). Refer to the Predefined Arguments above for more information.
  - **Working Directory**—The variable string is %(ProjWorkDir). Refer to the Predefined Arguments above for more information.

- **Run**—This button executes the external tool using the specified settings.



SCADAConnect Overview

SCADAConnect is the name given to several types of features aimed at better integrating hydraulic models with operational data. This is sometimes referred to as "Live Modeling" or "Real Time Modeling" but since it often involves a SCADA (Supervisory Control and Data Acquisition) system, the name SCADAConnect is used in Bentley products. Several different groups of features are covered by the SCADAConnect name including:

1. Ability to import field data from SCADA systems, data loggers and other external data sources for use in modeling.
2. Ability to run hydraulic analyses from a simplified user interface developed for operations personnel who are not full-time modelers.
3. Ability to display model results in a SCADA Human Machine Interface (HMI).
4. Ability to establish alarms and alerts to help review model runs.

Each of these groups of features is described further below with links to additional details.

Importing Data to Hydraulic Models

The users can connect the model to external data using a SCADA element which the user places in the model and connects to a model element and a value in an external data source (see SCADA Element on page 10-887) Each of these elements represents an individual signal. In the element, the user defines the model element and property (field) associated with the SCADA signal. At the same time, the user identifies the external signal under Components > SCADA signals (see SCADA Signals Setup on page 10-889). This involves identifying the data source and whether it is some type of data file or a direct connection to an OPC server. The user then identifies which signals (tags) from the data source are to be made available to the SCADA element in the model.

Once the link between the model elements and the external data sources have been established, the user can use external data for a variety of purposes including:

i. Viewing external data in the model in tabular or graphical form.

ii. Comparing model results with external data for model calibration using tabular and graphical views.

iii. Importing initial conditions for use in a model run.

iv. Importing field data for use with Darwin Calibrator.

Running Model using SCADAConnect Simulator

The user can run a hydraulic model from a simplified user interface such that someone can get started using models without detailed knowledge of all WaterGEMS/CAD features. This is intended for use by operators or engineers who are not regular model users. The user can access this SCADAConnect Simulator by picking Tools > SCADAConnect Simulator . To use this feature, a SCADA Baseline scenario must already have been set up. The user can then make changes to demands and override controls to make a run which can be based on initial conditions from the baseline scenario, some historical point in time or current (Live) conditions (see SCADAConnect Simulator on page 10-917).

When the user hits the Compute button [show button], the model runs and the user can view the results through a wide range of features (tabular, color coding, annotation, profiles) and perform additional calculations such as energy use. If the user has set up the features to publish results to a SCADA HMI, then the model results can be viewed in the HMI (see below).

Viewing Model using SCADA HMI

SCADAConnect enables users to view model results using a SCADA HMI. This is oriented toward users who are operators and are accustomed to working with the HMI. To use this feature, the user must have set up an OPC server to receive data from the model and HMI screens to display the values from the OPC server. Usually the user will not need to set up the server and the HMI from scratch but use the existing server and HMI as a starting point with modifications to reflect additional data than can be made available from the model which is not available from the normal SCADA system.

The user sets up the mappings from the model to the OPC server using a simple SCADA Results Publishing table which identifies the connection between the model properties and tags in the OPC server (see Displaying Model Results in SCADA Human Machine Interface (HMI)-Overview on page 10-922). To view values in the HMI, the user need only change the time in the Time Browser.

Alarms and Alerts

The user can also create alarms and alerts within SCADAConnect. Alarms are settings assigned to SCADA elements which, when triggered, produce messages. (Alarms can also be set on tank elements based on level. Alarms can also be set on tank elements based on level.) Alerts are similar but are set up in an Alerts manager and are associated with model hydraulic elements, not SCADA elements (see Alarms on page 10-912 and Alerts on page 10-915).

SCADA Element

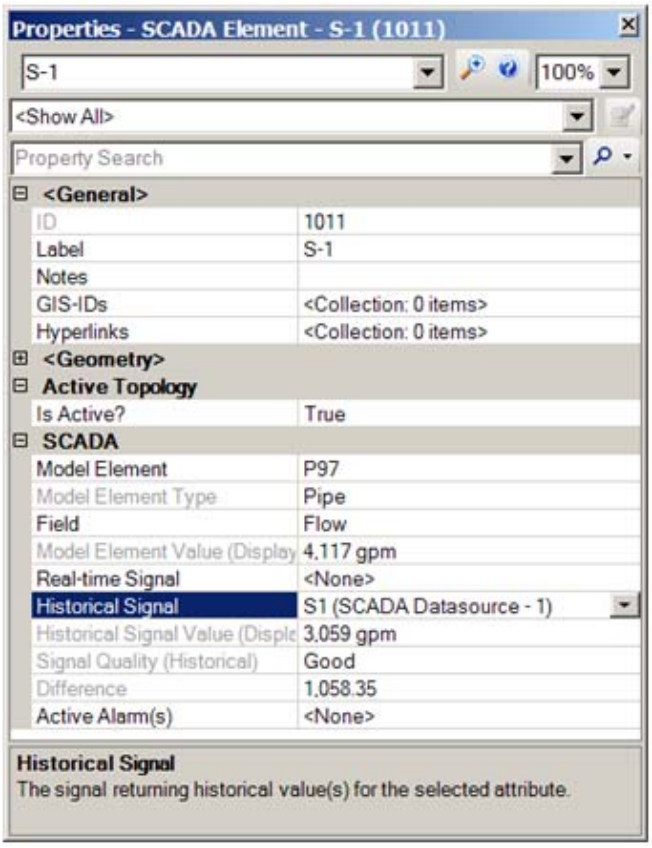
A SCADA element is an element created in order to link model elements with external time series data usually from a SCADA system (although it could be a data logger or database/spreadsheet file). These elements can be used to display external data in a model or set up alarms for model results.

A SCADA element can be placed as any other element although it is not used in hydraulic calculations.

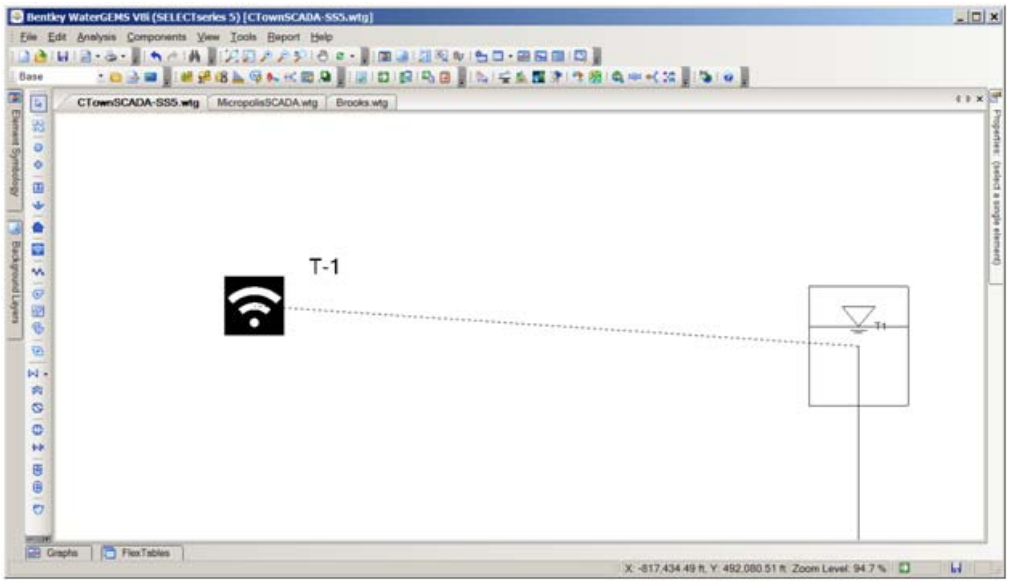


The SCADA element must be linked to a model element and can also be linked to some type of external signal. Each SCADA element has only one property so that an element with multiple properties must have one element per property (e.g. a pump with suction and discharge pressure and flow would have three SCADA elements).

The user enters the required data in the SCADA element property grid or flex table.



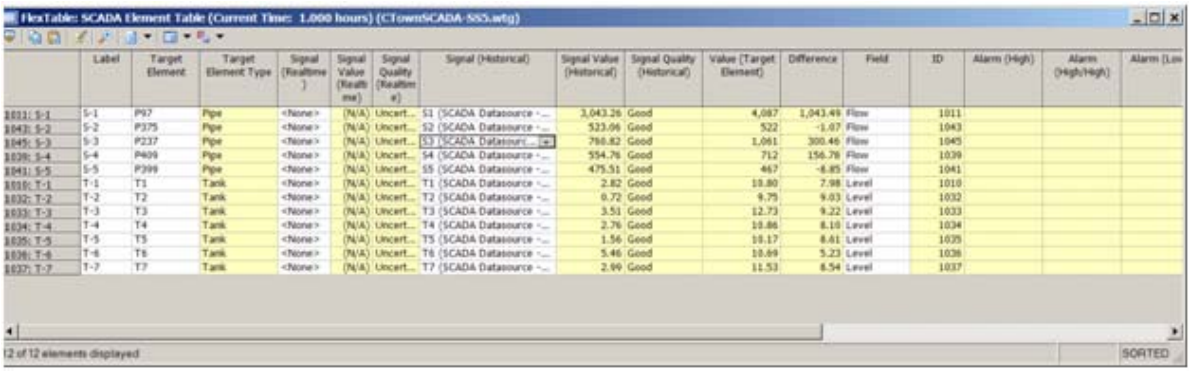
The most important property of the SCADA element is the model element with which it is associated. To set this, the user picks Select Model Element in the Model element property. The user then picks the model element associated with the SCADA element. The two elements should be located close to one another and are connected by a dashed line.



Once the model and SCADA element have been connected, the user selects the field/property that will be shared. If there is an external signal that will be passed, the user identifies the type of signal (real time or historical) and selects the signal from the drop down list of available signals that have been established in the SCADAConnect Signals Manager (see SCADA Signals Setup on page 10-889). The signal value, quality and difference between the signal and the model result are then displayed if available. The quality field is sound on some data sources indicating whether the value appears to be good.

The user can also set up alarms for that model element which will be displayed at the end of a model run (see Alarms on page 10-912).

The full list of SCADA elements can be viewed in the SCADA flex table as shown below (View > Flex Table > SCADA element).



The mappings between the model and external data are also used in importing initial conditions and loading Darwin Calibrator.

Once SCADA elements have been created, they can be detected in Network Navigator queries such as

- All SCADA elements
- Orphaned SCADA elements
- Find associated SCADA elements
- Elements with SCADA data

SCADA Signals Setup

In order to work with external data sources (in particular SCADA systems), it is necessary to set up the link between WaterGEMS/CAD and the data source. There are several type of data sources that can be used including database sources (which can include a wide range of file types), OPC Historical data source, OPC Real Time data source and a Citech data source.

The SCADA Signals dialog can be reached using either Component > SCADA Signals or Tools > SCADAConnect Simulator > SCADA Signals (first button on top of dialog). When opening the dialog for the first time, the user should pick the New button and then identify the type of data source from the four listed above. Depending on the type of data source selected, dialog will open enabling the user to define the data to be shared between the model and datasource. See the help topic for the individual datasource types (see SCADA Signals - Database on page 10-891, SCADA Signals - OPC on page 10-897, and SCADA Signals - Citect on page 10-900).

The other buttons on the top of the dialog include

Edit which enables the user to change the datasource properties.

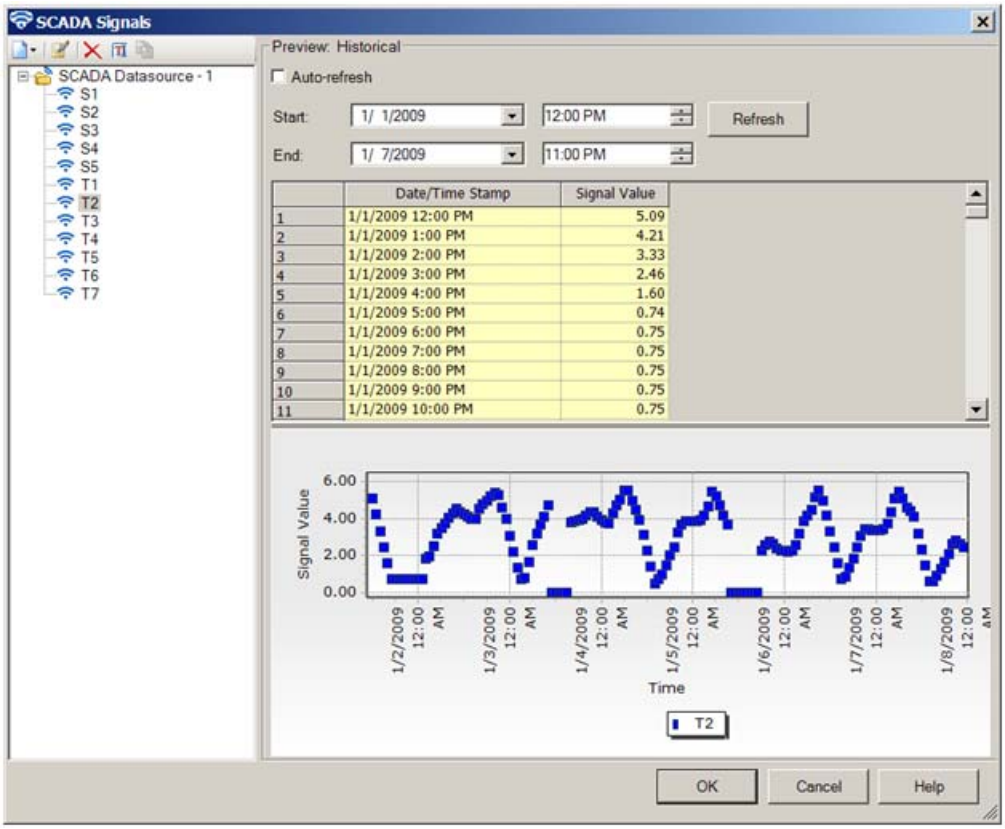
Delete which deletes the selected existing data source or SCADA signal. This does not delete any data from the datasource.

Rename which enables the user to change the name of the selected datasource or SCADA signal.

Duplicate which enables the user to copy the highlighted existing datasource.

The left pane of the dialog lists the datasources that have been defined. When the data-source list is expanded, each signal within that datasources is listed.

The right pane lists the signals when a datasource is highlighted in the left pane. When one of the individual signals is highlighted in the left pane, the right pane displays the range of times covered by the signal, a tabular view of the data and a graph of the data. If values are not shown, the user should pick the refresh button.



At the bottom of the SCADA Signals dialog the user can choose OK to save the signals or cancel to leave without saving.

The check box for Auto-refresh is not persisted. Therefore, it must be checked each time this dialog is opened.

SCADA Signals - Database

When the user selects a database source, the user is indicating that the SCADA data (whether it is from a SCADA system, logger or some other source) will be in the form of a file or database as opposed to retrieving data directly from a SCADA OPC server. Datasources can be managed with ProjectWise. The file types and format are described below.

As of SELECTseries 5, WaterGEMS/CAD currently supports the following although more could be added In the future:

- Excel 3.0
- Excel 4.0
- Excel 5.0
- Excel 2003/XP/2000/97 (8.0)
- Excel 2007 (12.0)
- Access 2.0
- Access 97/7.0 (3.0)
- Access 2003/2002/2000 (4.0)
- Access 2007 (12.0)
- ODBC Source
- OLEDB Source
- SQL Source
- Oracle connection

There are essentially two formats for the signals to be presented to the model: One value per row or multiple values per row.

One value per row: In this format, the signals should be stored in a way that each row/record contains a signal name, a time stamp and value columns. It is also possible to indicate the quality of the data (e.g. good, bad, questionable). The order of the columns does not matter and there may be columns that are not used. An example of such data is shown below.

Table 10-1: Database Source Example

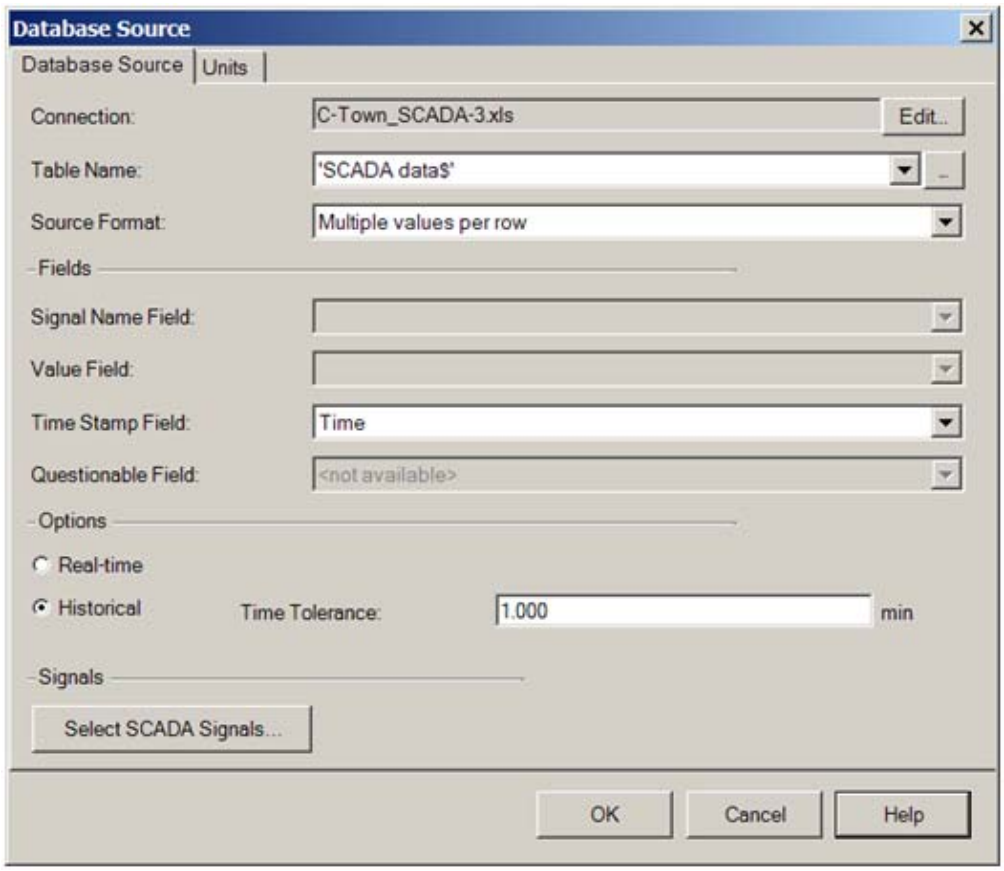
3:45:14 15 June, 2016	Flow Pump #7	234.156	Good
3:45:16 15 June, 2016	South Tank	18.187	Good
3:45:16 15 June, 2016	West Pressure Point	Nan	Bad

Multiple values per row. In this format, usually found when SCADA data have been processed, there can be multiple values for each record or row. The signal name needs to be the first row of the table. The signal names will usually correspond to the tags from the SCADA system. Not all of the columns/fields in the data source need to be used. An example of such data is shown below.

Table 10-2: Database Source Example 2

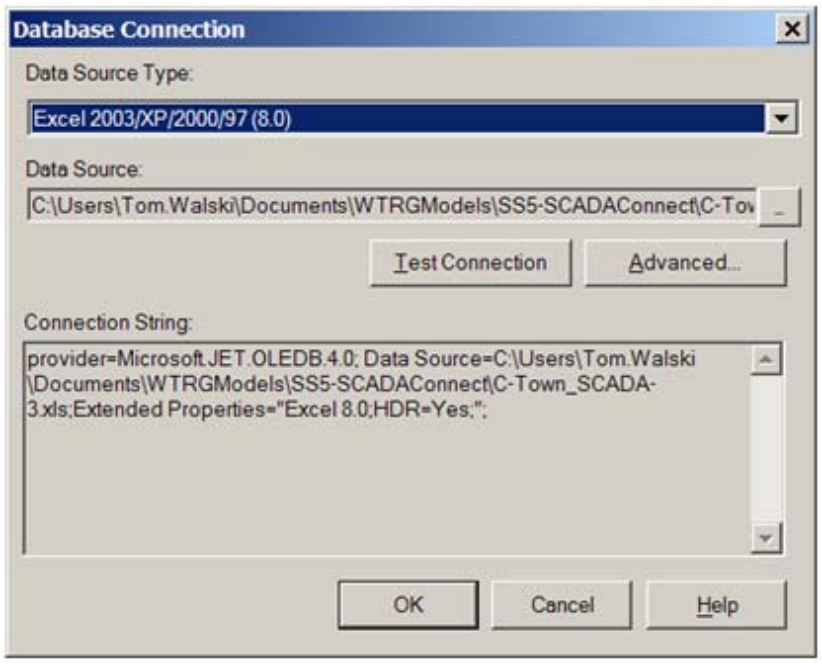
Time Stamp	Pump B Flow	Level South Tank	West Pressure Gauge	Flow Plant TM
3:45:14 15 June, 2016	375.788	34.44	87.5	12.356
3:45:15 15 June, 2016	376.114	34.41	87.1	12.319
3:45:16 15 June, 2016	0.015	34.38	85.6	12.189

Once the user has selected a Database source when defining a Signal connection or choosing to Edit a database source, the user will see the following dialog, except ODBC, OLEDB and SQL sources which have some special requirements shown later.

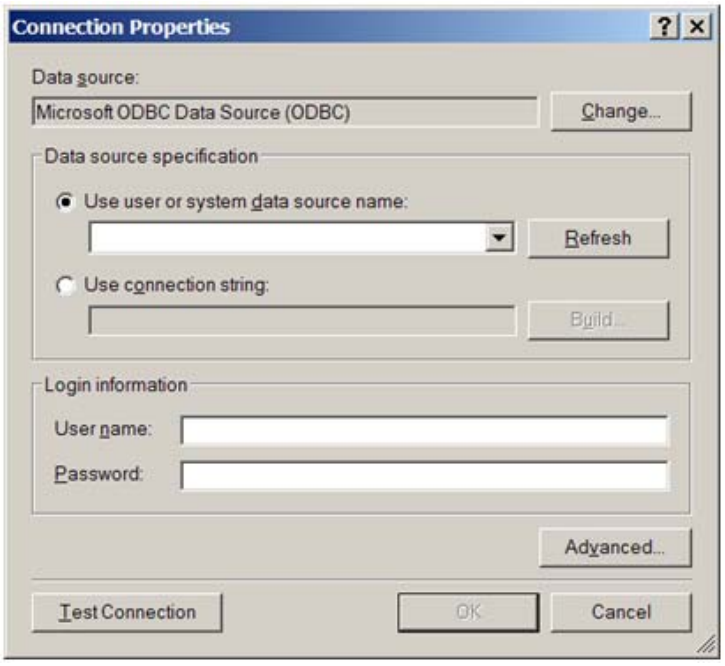


The fields are described below:

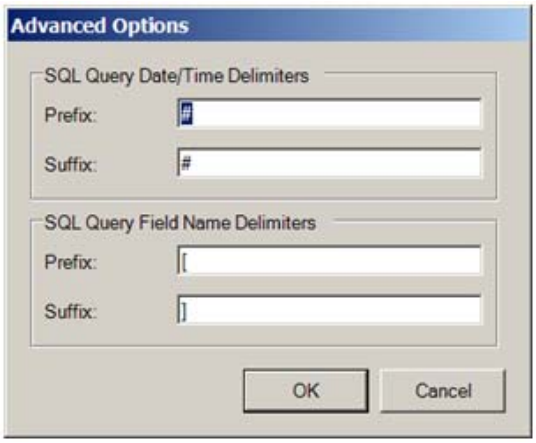
Connection actually establishes the connection with the data file. Picking Edit will open the dialog below:



The Data Source Type field will open a drop-down list of the available types as listed earlier. The Data Source is the actual file with a full path. Once the path is provided, it is suggested to pick Test Connection to ensure that the source is set up correctly. For ODBC, OLDDB, SQL and Oracle connections, additional information, which includes such items as login information, is required in a dialog as shown below:



The Advanced button open a dialog that is useful for SQL queries as shown below:



The Connection String is automatically established by the program. Advanced users can edit this string.

The Table Name field opens a drop down which enables the user to pick the table from the table corresponding to this datasource. If multiple datasources are used, each must have its own connection.

The Source Format identifies which of the formats: one value per row or multiple values per row is to be used.

The Signal Value Field identifies which field in the file is to be used as the signal name when the one value per row format is used. It is not used for multiple values per row.

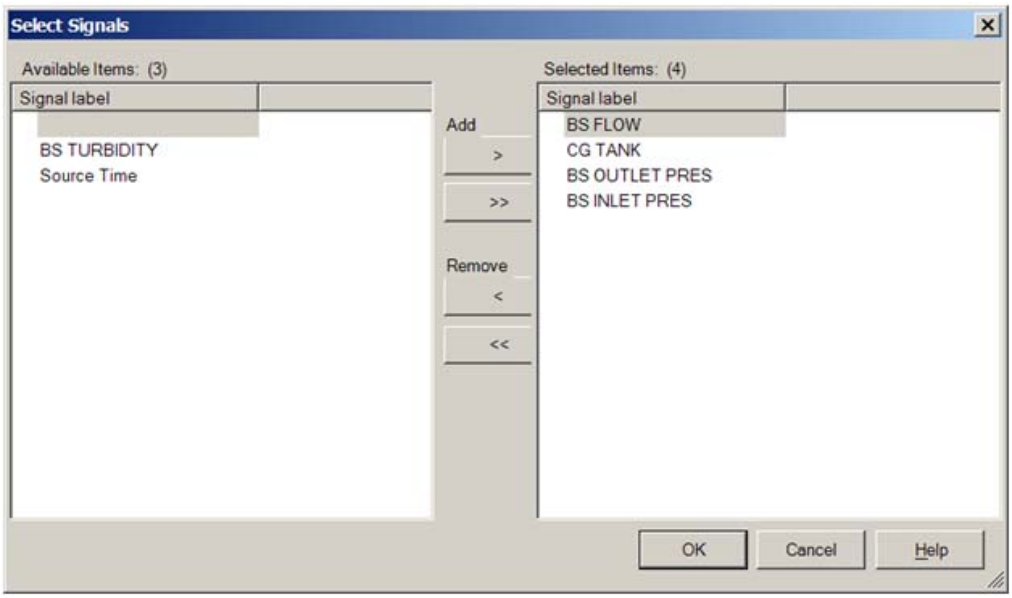
The Value Field identifies which field in the file is to be used as the Value when the one value per row format is used. It is not used for multiple values per row.

The Time Stamp Field is used to identify which field contains the time stamp. Any Windows-compliant Date/Time format can be used.

The Questionable Field identifies which field in the file is to be used as the data quality field when the one value per row format is used. It is not used for multiple values per row.

The Options portion of the dialog identifies which type of data this is to be considered. If the Real Time button is picked, then only the most recent value is imported while if Historical is picked, all values in the time band are used except for cases when a single value is needed such as Initial Conditions or Darwin Calibrator, in which case the Time Tolerance is used to pick the correct value from the historical datasource.

Once the user has identified the Data Source, the user can pick the Select SCADA Signals button which opens the dialog below which enables the user to select the signals that will be available in the model. These should correspond to the properties that are available for model elements plus any user defined properties. This is done by highlighting the signals in the left pane and picking Add to move them to the right pane.



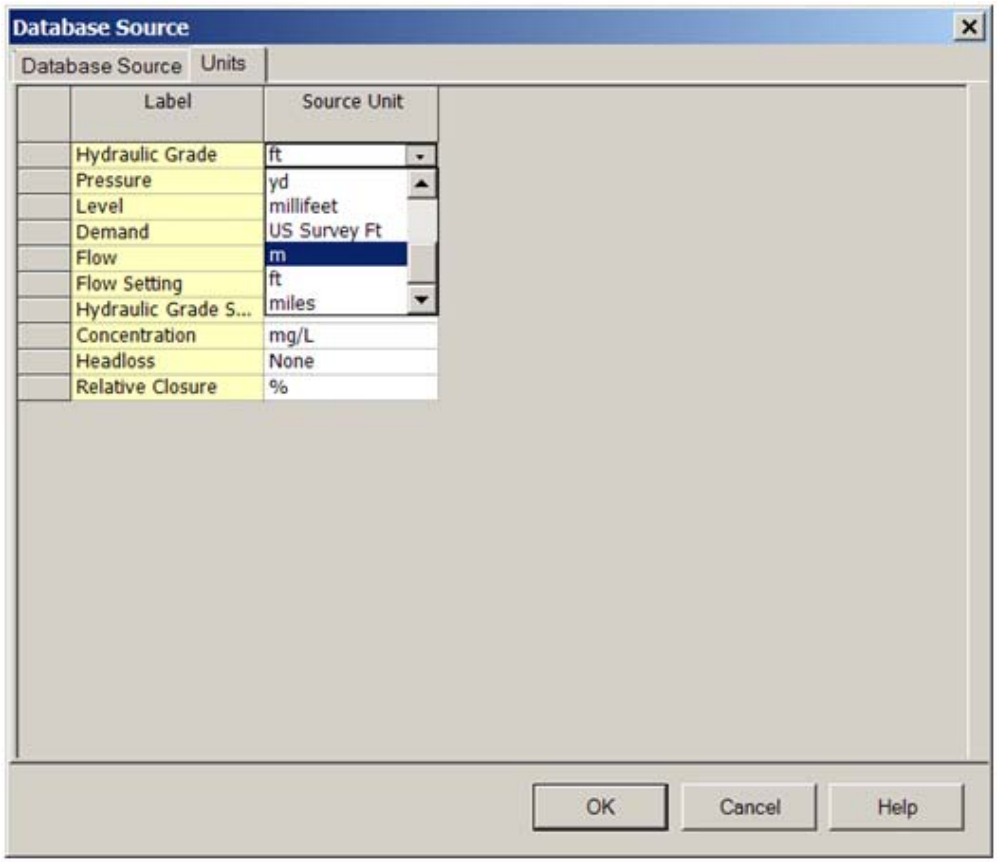
These signals are added to the list of available signals by highlighting them in the left pane and picking the Add button to move them to the right pane.

Clicking OK on the Database Source dialog performs a number of validations including verifying that:

- Datasource is available.
- The selected table is valid.
- Signal names are valid.
- Questionable field exists (if it was selected)
- Time/date field is available (if Historical is selected)

The second tab on the Datasource editor dialog is the Units tab which enables the user to specify the units for the SCADA signals. The default values are the values specified for the parameters in the model. However, if the units in the SCADA system data-source are different, this is where the user can indicate what those units are so that they can be adjusted when being imported.

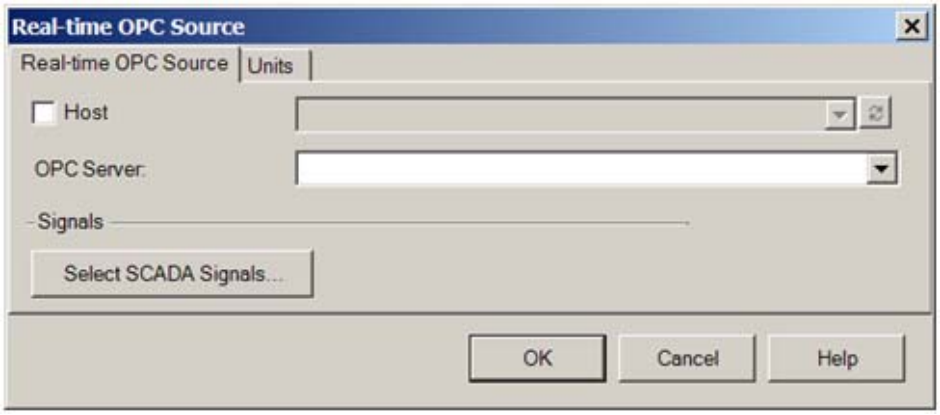
The user picks the field in the right column and then selects the correct units from the drop down list.



SCADA Signals - OPC

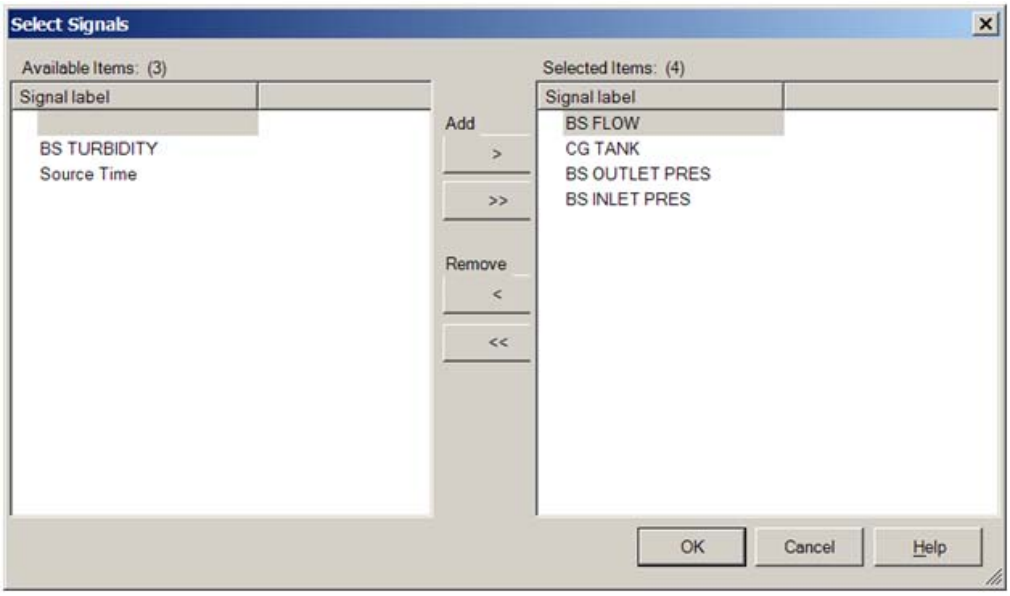
When setting up a SCADA connection, the user can import data from an OPC server (in addition to database datasources). The OPC connection can be used to import values that are either real time values or historical values.

Upon selecting an OPC source, the following dialog open for the user to select the location of the server. If it is the same computer as the model, the user should leave the Host box unchecked. If it is a different computer, the user should check the host box and the program will search for a list of connected computer from which the user can choose.



The user then picks the OPC server from the drop down list provided. The program looks for all OPC servers that are connected to the computer or if Host is selected, all the servers that the computer can reach.

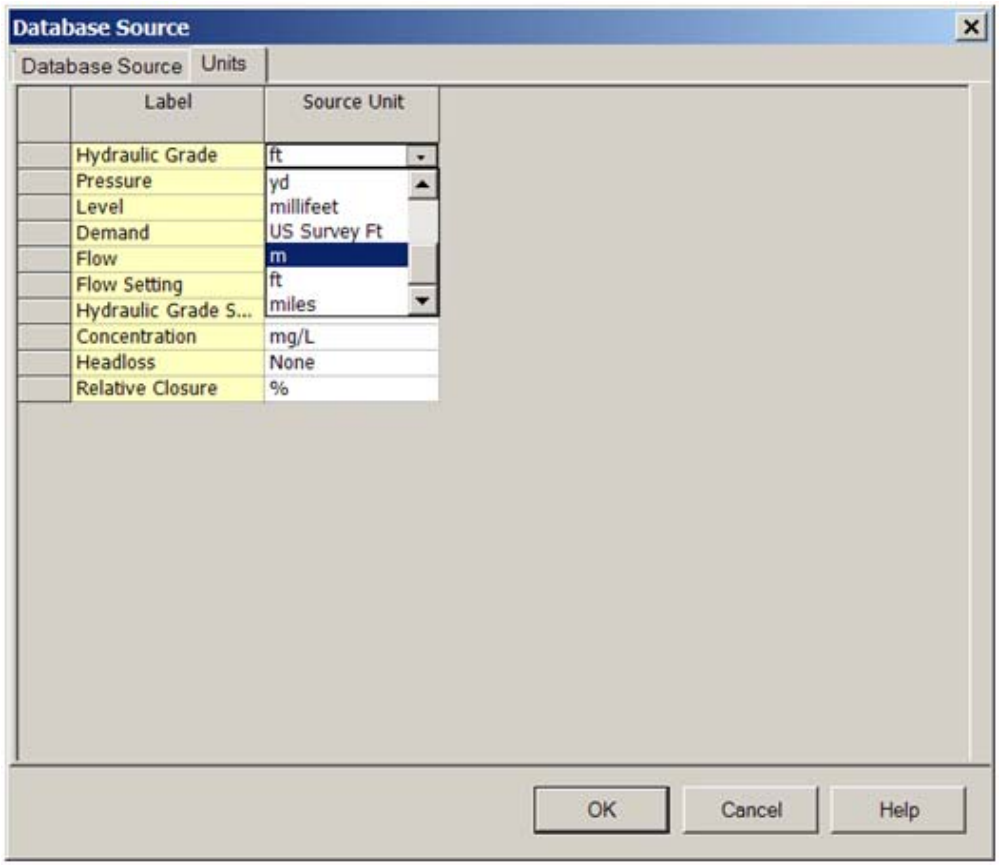
Once the server connection is available, the user picks the Select SCADA Signals box and a dialog opens which enables the user to pick which signals are to be made available for the model.



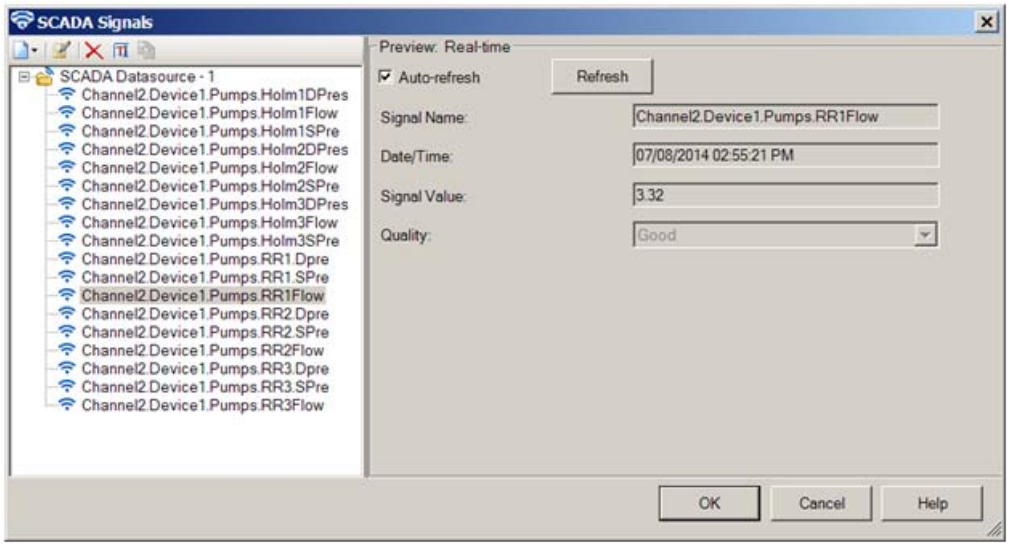
These should correspond to the properties that are available for model elements plus any user defined properties. This is done by highlighting the signals in the left pane and picking Add to move them to the right pane.

The second tab on the Datasource editor dialog is the Units tab which enables the user to specify the units for the SCADA signals. The default values are the values specified for the parameters in the model. However, if the units in the SCADA system data-source are different, this is where the user can indicate what those units are so that they can be adjusted when being imported.

The user picks the field in the right column and then selects the correct units from the drop down list.

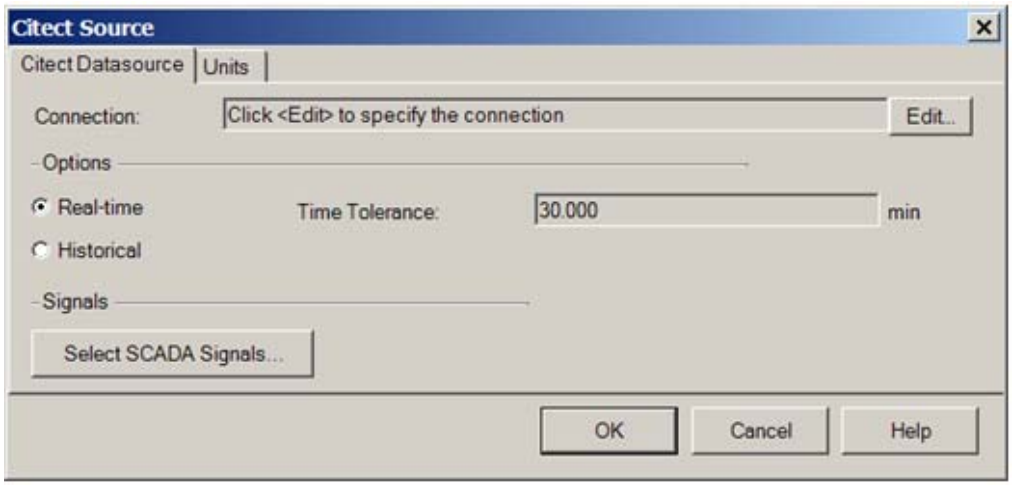


Once the connections have been set up, both Historical and Real Time data can be viewed and imported. When real time data are imported from an OPC server, the latest value is displayed as shown below (see Refresh and Auto Refresh on page 10-905).

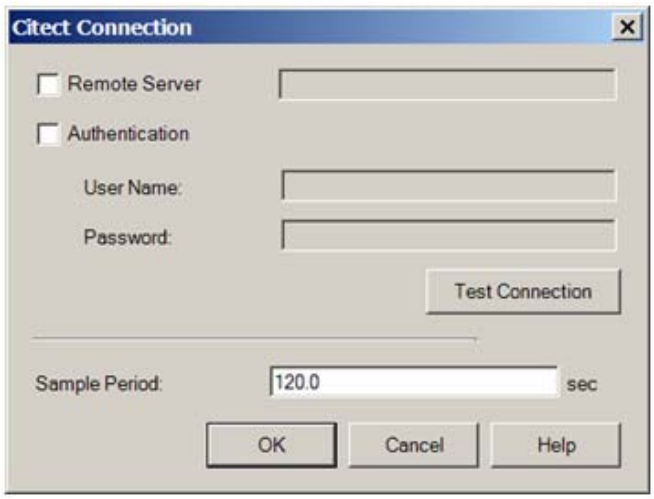


SCADA Signals - Citect

The user can also set up customized SCADA connections directly with a Citect SCADA system by choosing Citect source when creating a new connection. This will open the following dialog which enables the user to establish a connection with a Citect server.



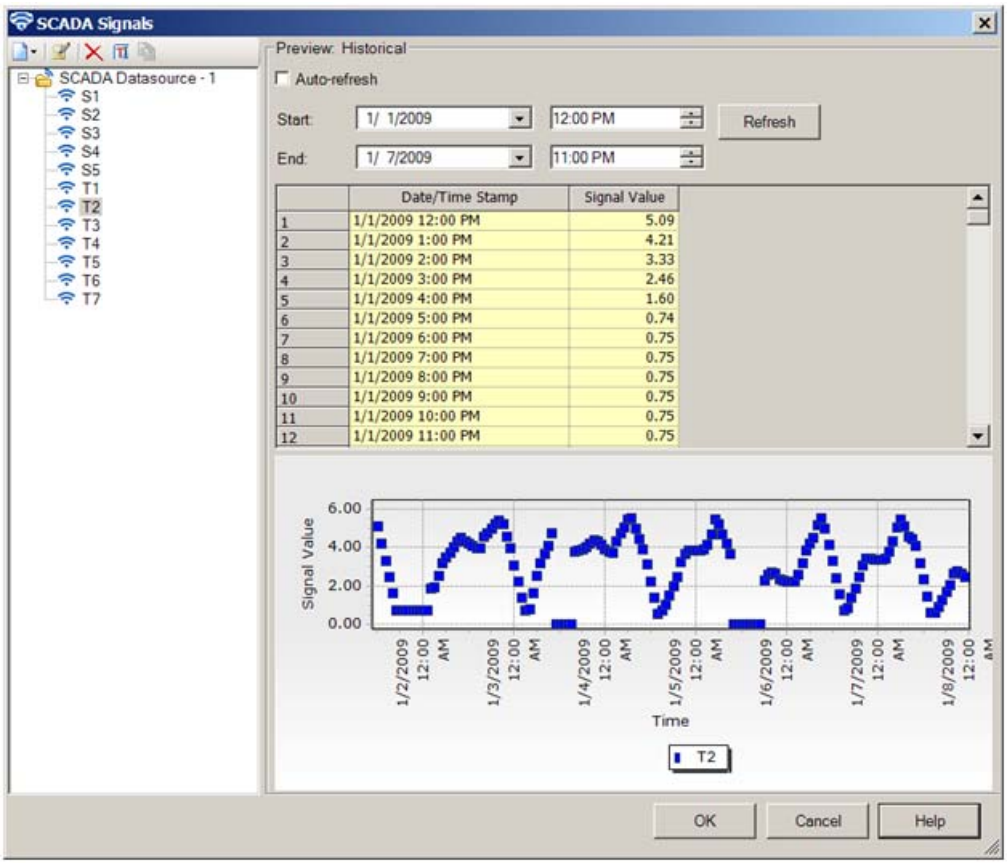
By picking the Edit button, the user can identify the server and enter any authentication credentials. Citect server credentials are required for a remote connection.



Most of the other behaviors are similar to connections with other OPC/hitorical servers. The user an specify whether the data are to be Historical or Real-time and can use the Select SCADA Signals to identify which signals are to be connected.

Viewing SCADA Data in Model

A user can view external data, either from a file or an OPC server, directly in WaterGEMS/CAD. Once the connections have been established, the user can enter the SCADA Signals manager and pick any signal. (The signal does not necessarily need to be linked to a SCADA element.) The user picks one of the signals from the left pane and will be able to view the time series as shown below.



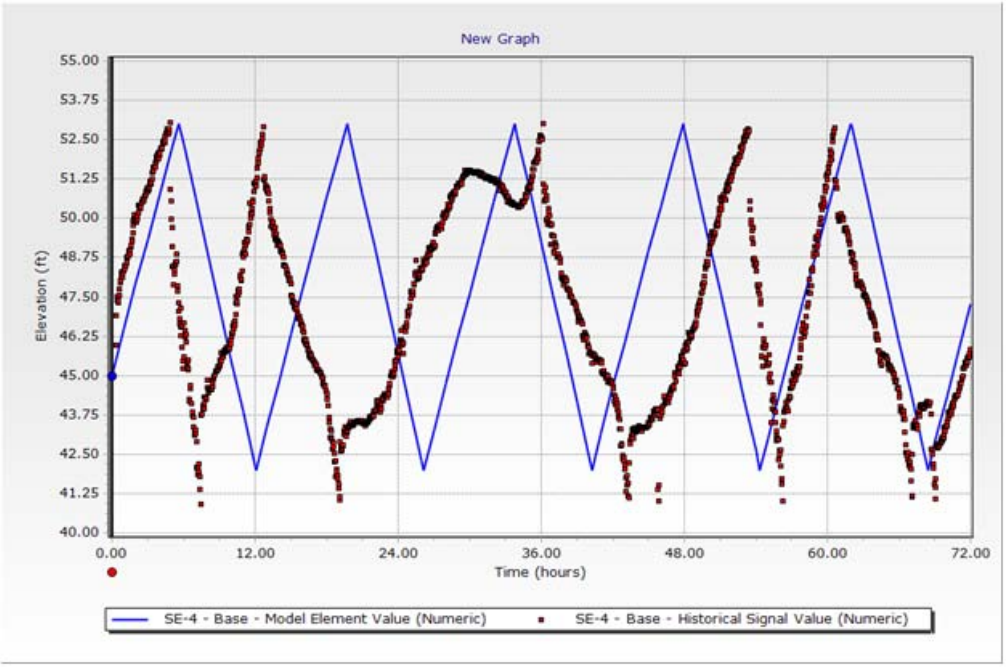
If the display is empty, the user should pick the Refresh button.

Checking the Auto-refresh button means that the tabular and graphical views of the data are automatically displayed when a new signal is selected.

Right clicking on the top of the Signal Value column enable the user to change Units and Formatting.

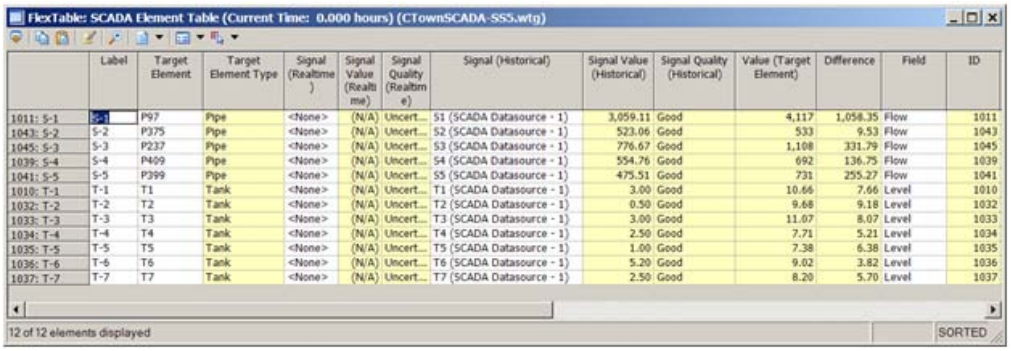
Right clicking on the bottom border of the graph area enables the user to set chart options in the graphical view.

Once SCADA data are available, they can be viewed graphically by right clicking on the SCADA element and picking graphs. If the SCADA signal has been set up correctly and the SCADA element is associated with a model element, then the SCADA signal and model results will be plotted on the same graph.



All of the graph manager features are available for use with these graphs.

For a given time step, the values for SCADA data can be viewed in the SCADA flex table.



FlexTable: SCADA Element Table (Current Time: 0.000 hours) (CTownSCADA-SSS.wtg)

	Label	Target Element	Target Element Type	Signal (Realtime)	Signal Value (Realtime)	Signal Quality (Realtime)	Signal (Historical)	Signal Value (Historical)	Signal Quality (Historical)	Value (Target Element)	Difference	Field	ID
1011: S-1	S-1	P97	Pipe	<None>	(N/A)	Uncert...	S1 (SCADA Datasource - 1)	3,059.11	Good	4,117	1,058.35	Flow	1011
1043: S-2	S-2	P375	Pipe	<None>	(N/A)	Uncert...	S2 (SCADA Datasource - 1)	523.06	Good	533	9.53	Flow	1043
1045: S-3	S-3	P237	Pipe	<None>	(N/A)	Uncert...	S3 (SCADA Datasource - 1)	776.67	Good	1,108	331.79	Flow	1045
1039: S-4	S-4	P409	Pipe	<None>	(N/A)	Uncert...	S4 (SCADA Datasource - 1)	554.76	Good	692	136.75	Flow	1039
1041: S-5	S-5	P399	Pipe	<None>	(N/A)	Uncert...	S5 (SCADA Datasource - 1)	475.51	Good	731	255.27	Flow	1041
1010: T-1	T-1	T1	Tank	<None>	(N/A)	Uncert...	T1 (SCADA Datasource - 1)	3.00	Good	10.66	7.66	Level	1010
1032: T-2	T-2	T2	Tank	<None>	(N/A)	Uncert...	T2 (SCADA Datasource - 1)	0.50	Good	9.68	9.18	Level	1032
1033: T-3	T-3	T3	Tank	<None>	(N/A)	Uncert...	T3 (SCADA Datasource - 1)	3.00	Good	11.07	8.07	Level	1033
1034: T-4	T-4	T4	Tank	<None>	(N/A)	Uncert...	T4 (SCADA Datasource - 1)	2.50	Good	7.71	5.21	Level	1034
1035: T-5	T-5	T5	Tank	<None>	(N/A)	Uncert...	T5 (SCADA Datasource - 1)	1.00	Good	7.38	6.38	Level	1035
1036: T-6	T-6	T6	Tank	<None>	(N/A)	Uncert...	T6 (SCADA Datasource - 1)	5.20	Good	9.02	3.82	Level	1036
1037: T-7	T-7	T7	Tank	<None>	(N/A)	Uncert...	T7 (SCADA Datasource - 1)	2.50	Good	8.20	5.70	Level	1037

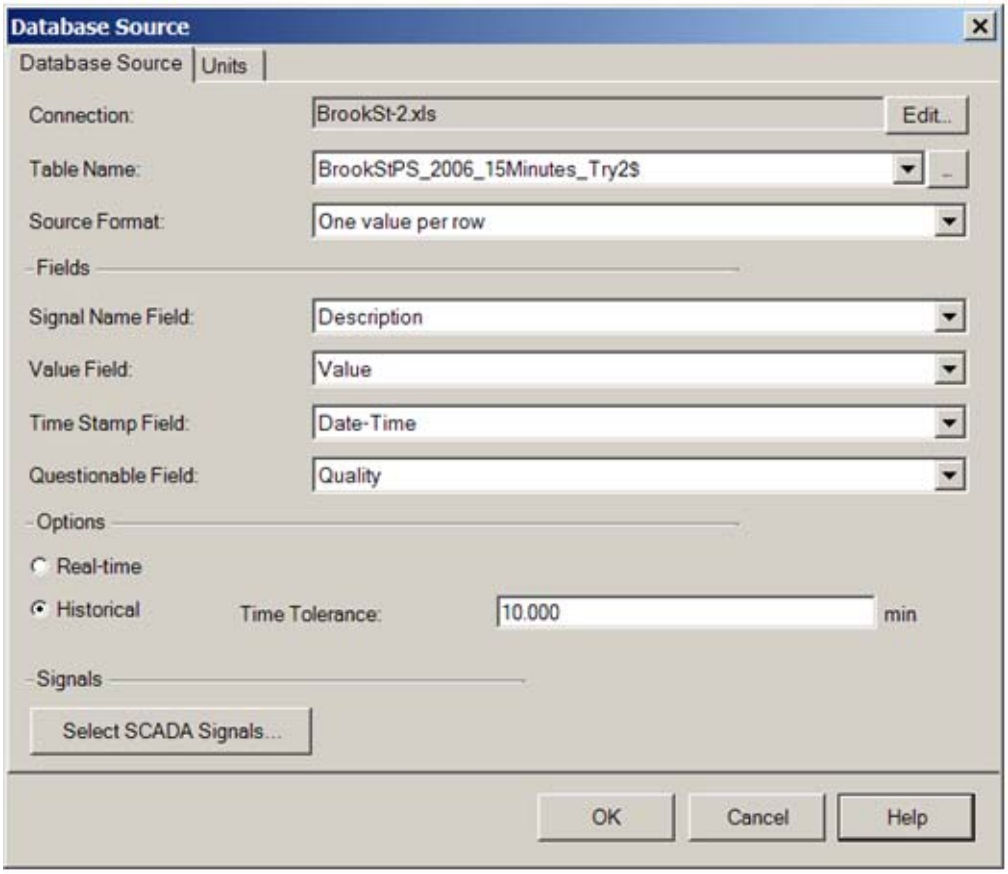
12 of 12 elements displayed SORTED

The user can also annotate or color code by values associated with SCADA elements.

Time Tolerance

When using historical data, there are cases where the user must select a single value from a time series of values, such as for loading initial conditions or Darwin Calibrator. The user usually needs data for a single point in time but a SCADA value may not be available at exactly that time. For example, the user may need a value for 8:00:00 but values may only be available for 7:58:14 and 8:02:11. SCADAConnect will use the value from the time closest to the time required.

However, there may be instances where the nearest value available is far from the required time. The user sets the band of time SCADAConnect will look for values by entering a Time Tolerance.

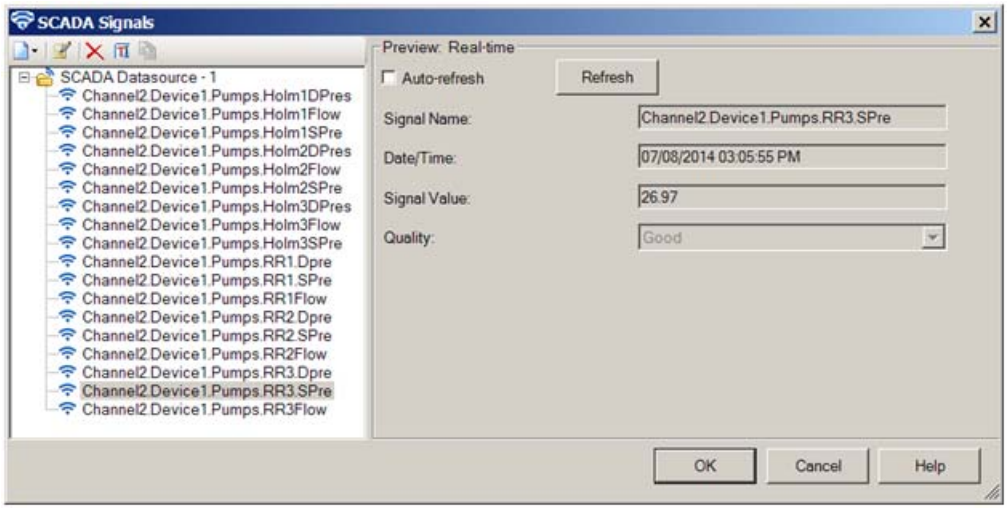


If there is no value within the time tolerance, no value is imported and an error message is listed in the SCADA Log which can be reached from Tools > SCADAConnect Simulator >SCADA Log (eighth button in SCADAConnect Simulator dialog). The log message is written on attempting to read the signal value.

Refresh and Auto Refresh

When viewing signals in the SCADA Signals dialog, WaterGEMS/CAD does not instantly read and display the values. Instead, the user must pick the refresh button to display the result for the selected signal when the first signal is selected or the user switches signals.

To force an automatic refresh of the display when switching signals, the user should check the auto-refresh check box.

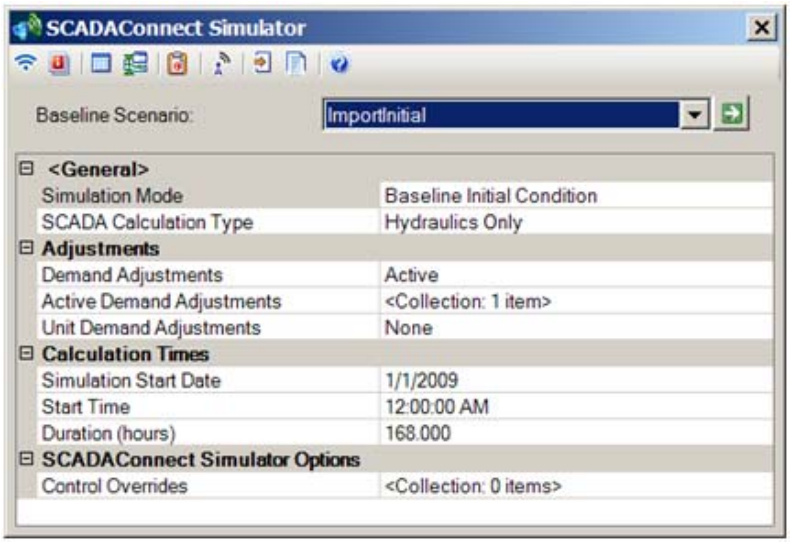


Importing Initial Conditions with SCADAConnect

A user can override the initial condition associated with a scenario with new initial conditions from an external data source such as a SCADA system. This tool enables user to import tank levels and other data to start a model run with exact conditions from an external data source at a specific time. For example, the user may want to set tank levels based on the current tank levels in the distribution system. There must be a SCADA signal element for each value to be imported and the SCADA connections must be set up before attempting to import.

Note: **There are actually two ways to import initial conditions. The first is described here, the second consists of setting up a Historical or Live run in the SCADAConnect simulator. With the method described here, the initial condition actually modifies the values in the initial setting alternative of the current scenario being run.**

The user picks Tools > SCADA Connect Simulator. Initial conditions are imported to the Current Scenario, not the baseline scenario. If the user does not want to overwrite the an existing scenario, the user should set up a new scenario to receive the imported values.



The user then picks the Import Initial Conditions button (seventh from left).



This opens the following dialog:



The user then identifies if the import is from a historical data source or real time data. If the data source is a historical time series, then the user must provide a date and time which can be typed directly or indicated from the drop down calendar. Data must be available within the Time Tolerance specified when the signal was created for historical data. If not, then the initial condition from the baseline scenario will be used and an entry will be made in the SCADA log.

If the user checks the Create new Selection Set box, the created Selection Set contains all model elements with updated initial settings.

If the user selects Real Time, then the model uses the current value from the OPC server or the latest value from the database source. No time tolerance is applied for real time data.

Values of initial conditions are imported for all model elements that have SCADA elements assigned and are attempting to import data from an external source which must be identified prior to the import.

The Ignore Inactive Elements check box indicates that initial settings should not be imported for inactive SCADA elements.

The properties that are set can be stored in the Initial Settings alternative with the exception of water quality values which are stored in the Constituent alternative. The values that can be imported for any element depend on the initial settings that can be set for that element as summarized below:

Pumps and Variable speed pump batteries can have on/off status changes and variable speed pumps can directly set the pump relative speed factor. The import field should contain a 0 or 1 (On=0, Off=1). If status is indicated by some other value (e.g. text value of On or Off=0) the user needs to adjust the data source.

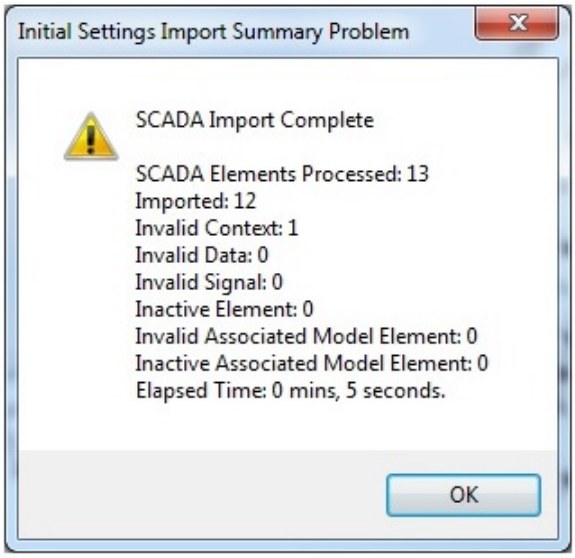
Pipes can be open (0) or closed (1).

Control valves such as Pressure reducing valve (PRV), Pressure Sustaining Valve (PSV), Pressure Breaker Valve (PBV), Throttling Control Valve (TCV) or Flow Control Valve (FCV) can have their initial status (0=active, 1=inactive, 2=closed) or their setting (numerical value) set from an external source. A TCV setting can be specified as either relative closure or headloss coefficient. A General Purpsoe Valve (GPV) can only be Active or Closed.

The initial condition for a tank must be specified as the Elevation as a Hydraulic Grade Line relative to the model data, not a level relative to tank bottom. A tank with 10 ft of water and a bottom elevation of 230 ft would need to have a value of 240 ft, not 10 ft.

Constituent values can be imported for any junction node but not HAMMER specific elements.

At the end of the import, a summary of the import is provided.



The sections in this dialog are as follows:

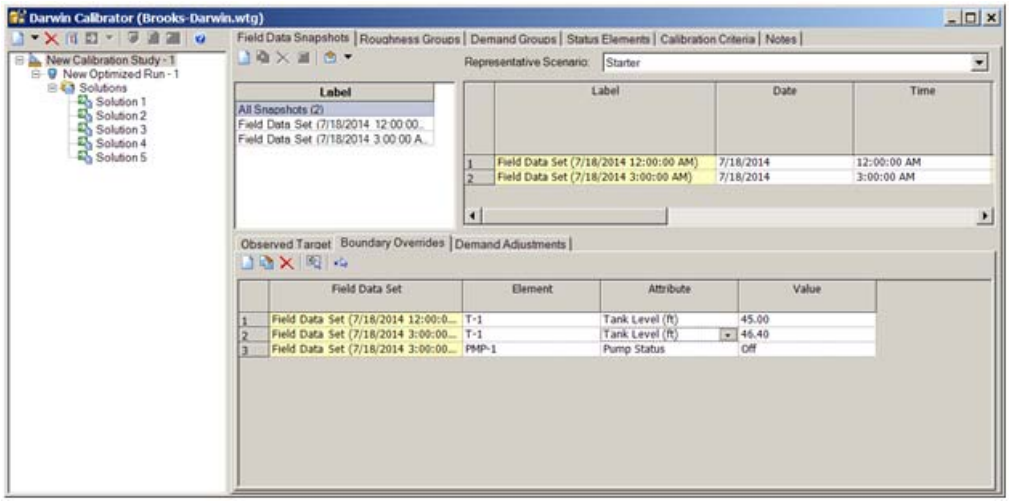
- **Imported** - The number of SCADA elements successfully processed and imported.
- **Invalid context** - The SCADA element mapping is not appropriate for use in the import (doesn't have an equivalent initial settings field) or the SCADA field has not been assigned.
- **Invalid Data** - No data was available. For real-time no value is available. For historical no value is available for the requested time within the time tolerance of the data source. The data source may also be invalid.
- **Invalid Signal** - The signal is not properly defined. The signal may not be defined or may have been deleted.
- **Inactive Element** - The SCADA Element is inactive and the "Ignore inactive elements" option was selected.
- **Invalid Target Element** - The SCADA Element target element is undefined or deleted.
- **Inactive Target Element** - The SCADA Element target element is inactive and the "Ignore inactive elements" option was selected.
- **Elapsed Time** - The total time taken to execute the import.

Loading Darwin Calibrator from SCADA Data

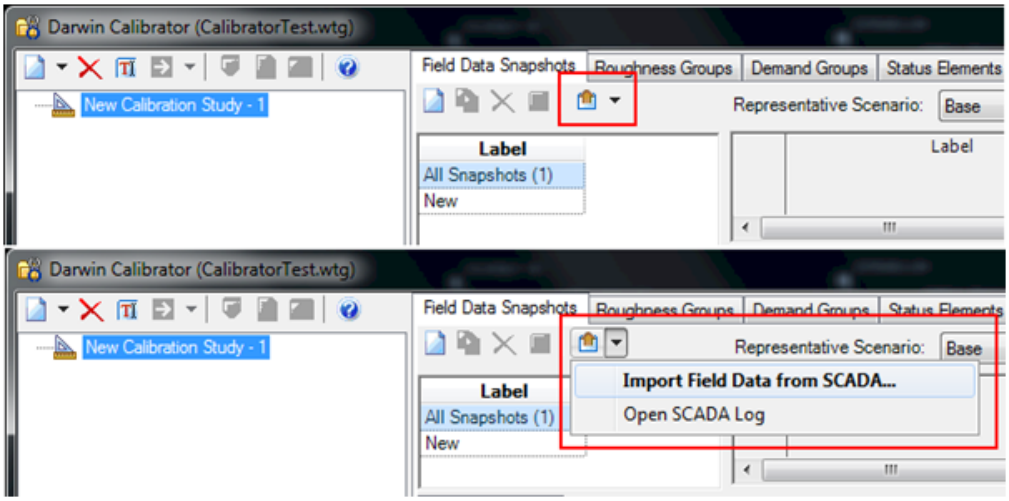
SCADAConnect enables a user to automatically load data from an external data source, such as a SCADA system, into Darwin Calibrator to assist in calibrating a model. See the Darwin Calibrator help for instructions for using Calibrator (Calibrating Your Model with Darwin Calibrator on page 11-1045).

Before using Darwin, the user must first set up a SCADA element for each signal, set up the connection to the data source (see SCADA Signals Setup on page 10-889), assign the SCADA signal to the SCADA elements and assign the SCADA element to the SCADA element to a hydraulic model element (see SCADA Element on page 10-887).

The SCADA data are imported to as many Field Data sets in Darwin as there are time steps in the field data set. Open Darwin Calibrator, pick or create a Calibration study in the left pane and pick the Field Data Snapshot tab in the right pane.



Pick the Import Field Data from SCADA button at the top of the right pane.



Select whether historical or real time data are to be imported and if it is historical data, the time setting for the import should be indicated.



The **Ignore inactive elements** options allows the user to exclude certain SCADA signals from the import by making either the SCADA Element inactive or the target element of the SCADA Element inactive (or both). The default Field Data Set Label is taken from the time assigned to the import data or the current time for Real Time data. The user can overwrite this time.

Hit OK to import the data. On completion of the import, the following message will appear.



The definition of each entry is shown below.

- **Imported** - The number of SCADA elements successfully processed and imported.
- **Invalid context** - The SCADA element mapping is not appropriate for use in the import (doesn't have an equivalent initial settings field) or the SCADA field has not been assigned.
- **Invalid Data** - No data was available. For real-time no value is available. For historical no value is available for the requested time within the time tolerance of the data source. The data source may also be invalid.
- **Invalid Signal** - The signal is not properly defined. The signal may not be defined or may have been deleted.
- **Inactive Element** - The SCADA Element is inactive and the "Ignore inactive elements" option was selected.
- **Invalid Associated Model Element** - The SCADA Element target element is undefined or deleted.
- **Inactive Associated Model Element** - The SCADA Element target element is inactive and the "Ignore inactive elements" option was selected.
- **Elapsed Time** - The total time taken to execute the import.

Note that if any of the line items that contain "Invalid" are non zero, the message will show a warning icon as pictured above, else an information icon is shown. Details of the issues when a warning is present are able to be obtained from the SCADA Log, which is the eight button from the left in the SCADAConnect Simulator dialog (see SCADAConnect Simulator on page 10-917).

If the user wants to import data from multiple historical times, the steps from Import Field Data from SCADA should be repeated for each time. The imported values are displayed in the lower right pane.

Darwin determines if the imported values are to be Observed Target values (pressures, pipe flows), Boundary Overrides (tank level, pump status) or Demand Adjustment (junction demands).

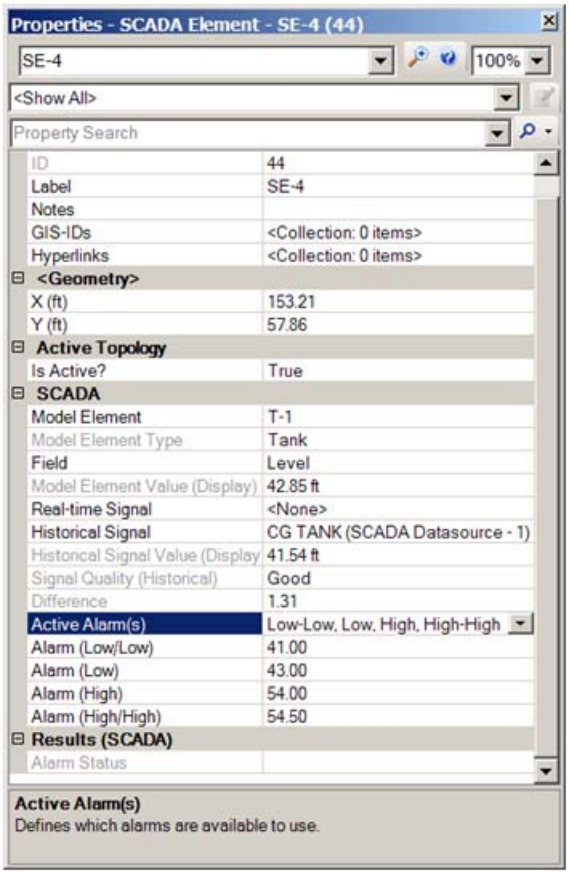
Alarms

Alarms refer to messages that are generated by WaterGEMS/CAD when specific values are exceeded in model results. Alarms can be created at any SCADA element or for high and low values of tank elements. They differ from alerts in that alerts can be triggered at any type of element and can include multiple elements in one alert. As a property of a SCADA element, alarms are intended to mimic the response of alarms in a SCADA system.

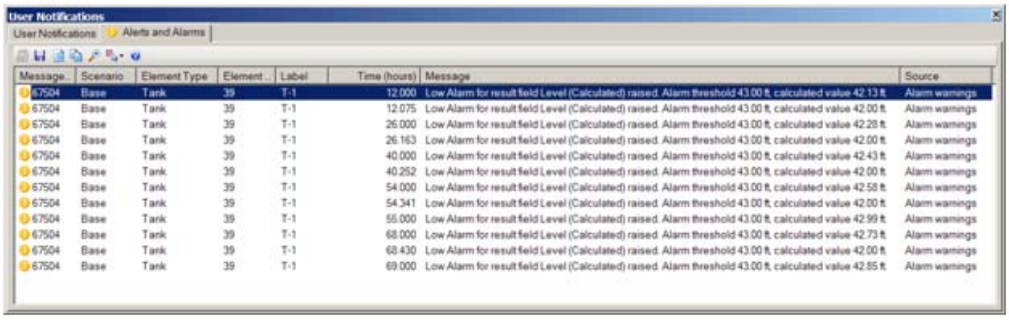
The alarms can be established by setting up a SCADA element with a property on which an alarm can be based. Under the Active Alarms property, the user can set up 4 different combinations of alarms

- Low
- High
- Low and High
- Low-low, Low, High, and High-High

Once the user picks the combination of alarm setting, the user fills in the numerical value.



When the model is run, the user can view the alarms by selecting Analysis > User Notifications and picking the Alarms and Alerts tab.



The buttons on the top of the display include:

The first button is not active for alarms.

The second button enables the user to save the alarms and alerts in a csv file.

The third button generates a report that can be printed.

Thefourth button copies the highlighted alarm.

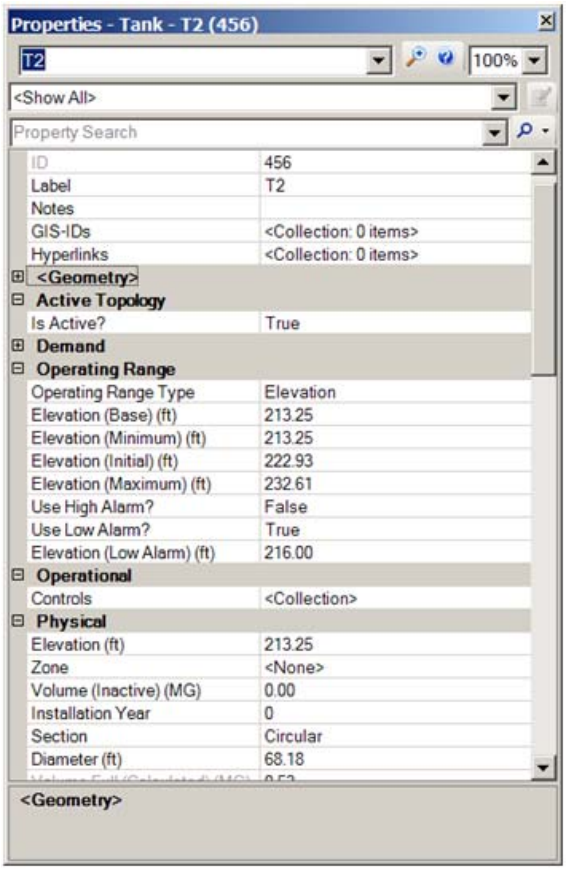
40020 “Base" "Tank" "456” “T2” 16.598 “Tank T2 low alarm level is violated” Calculation Warnings

The fifth button zooms to the element associated with the highlighted alarm.

The sixth button selects the highlighted element in the model. When multiple elements trigger an alert, it is to do this in the Details version of alerts.

The final button is Help.

Alarms can also be set up for Tank elements in the element property grid. The user selects where High and/or alarms are desired by setting Use High/Low Alarm to True and setting the numerical value as either an Elevation or a Level.



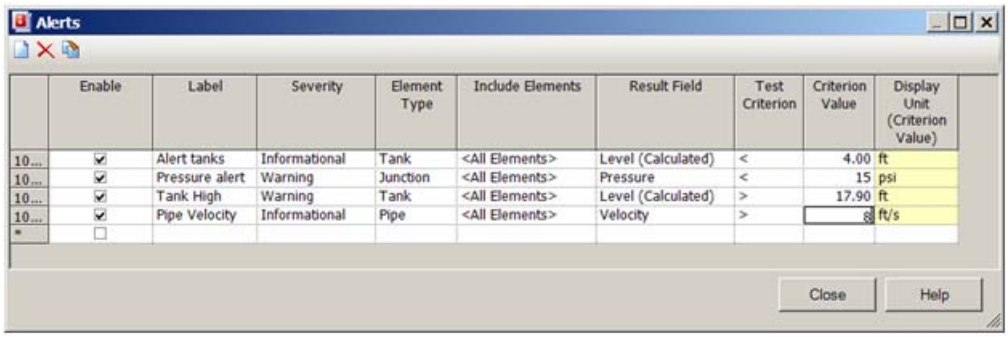
Alerts

A user can establish Alerts which are settings that will trigger messages when an alert Criterion Value has been exceeded. Alerts differ from Alarms in that they can be associated with any type of hydraulic model element. Alerts are set up in the Alerts manager which can be accessed from Components > Alerts or as the second button from the SCADAConnect Simulator dialog.

Alerts can be used for example, to find all junction elements that fall out of a given pressure range during a model run.

Alerts Manager

The alerts manager is shown below:



From the buttons at the top of the dialog, the user can create new alerts, delete existing alerts or duplicate an alert.

Checking a box in the column labelled when the alert is to be used in the next model run.

The Label is a user supplied name for the alert.

Severity is used to differentiate between different levels of alerts and can be Informational, Warning or Critical. They affect the color of the symbol at the beginning of an alert.

Element type refers to the type of element covered by the alert. Each element type has its own alerts such that junctions and hydrants would need their own alerts.

The Include Elements column enables the user to apply an alert to all elements of the selected type or to a selection set of elements.

The Results Field column enables the user to specify which result property for the elements will be used in the comparison.

The Test Criterion column identifies which type of relationship is to be used for the comparison.

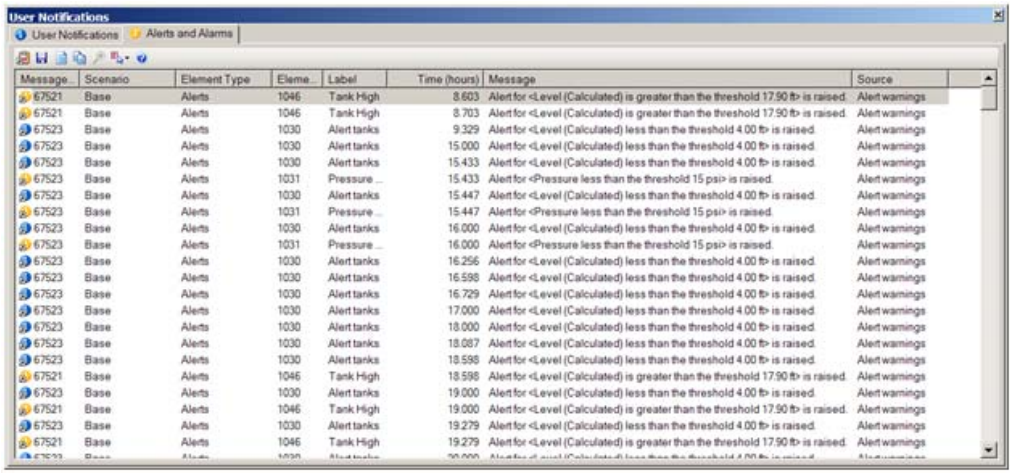
The Criterion Value column enables the user to set the threshold numerical value used for the comparison. It is best to not set the threshold too tightly or it will result in a very large number of alerts.

The Units column is a read-only column showing the user the units for the criterion value.

Alerts Results

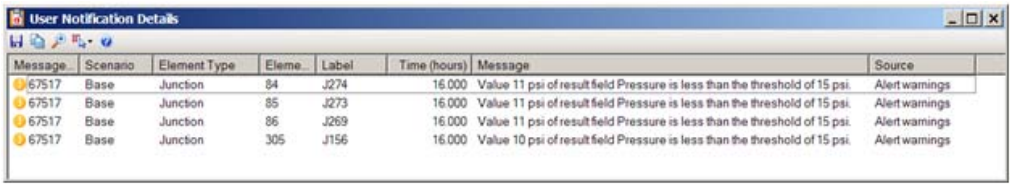
Alerts are calculated at the time the scenario is run, not when the alerts are entered.

The alert messages can be found with user notifications under Analysis > User Notifications and picking the Alarms and Alerts tab.



Most of the columns are self-explanatory. The color of the message is an indication of its severity.

When multiple elements trigger alerts, these can be viewed by picking the Details (first button on top of table) to view multiple alerts at a given time.



The second button enables the user to save the alarms and alerts in a csv file.

The third button generates a report that can be printed.

The fourth button copies the highlighted alert.

40020 “Base” “Tank” “456” “T2” 16.59 “Tank T2 low alarm level is violated.” Calculation Warnings

The fifth button zooms to the element associated with the highlighted alert.

The sixth button selects the highlighted element in the model. When multiple elements trigger an alert, it is to do this in the Details version of alerts.

The final button is Help.

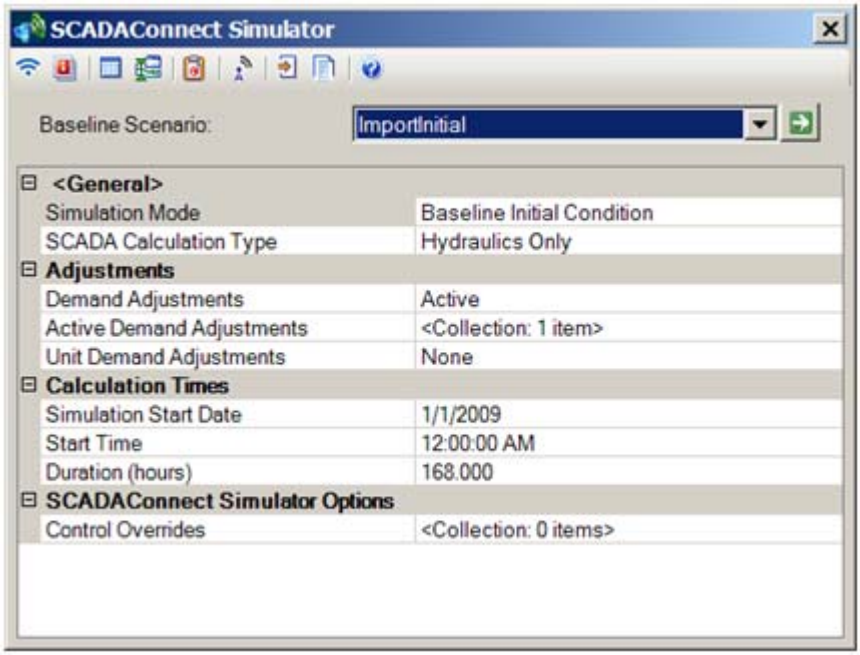
SCADAConnect Simulator

SCADAConnect Simulator provides a way for users to modify and run a model scenario from a very simple user interface without the need to interact with some of th e more sophisticated features of WaterGEMS/CAD. In order to run the simulator, it is necessary to have a model with an existing Extended Period Simulation (EPS) scenario already created (and labeled as SCADA).

When using a baseline scenario as a starting point for some SCADA Simulator runs, the user may not want to modify an existing scenario on which the later SCADA runs are based. Runs from SCADA Simulator can modify the Calculation options (e.g. start time, demand adjustments, control overrides) or Initial settings (e.g. tank levels, pump status). It is advisable to create a Child scenario of the original scenario and make that the new Baseline Scenario. It would have a different set of Calculation options and a different Initial Settings alternative. For example, if the starting scenario is "MaxDay" with Calc option "EPS24" and initial alternative "Base-initial", the user may create a new scenario called "MaxDayfromSCADA" with Calc option "SCADACalcOptions" and Initial settings "Initiasettings from SCADA". Setting up the SCADA Simulator this way would enable the user to try a wide variety of adjustments, and initial settings without modifying the starting scenario.

To run a simulation, the user picks Tools > SCADAConnect Simulator.

The Simulator opens to enable the user to run the simulation.



The buttons on the top row include:

SCADA Signals which enables the user to identify the SCADA signals with are to be used (see SCADA Signals Setup on page 10-889).

Alerts which opens the Alerts Manager which opens the Alerts manager (see Alerts on page 10-915).

Flex tables which opens the SCADA element flex table (see SCADA Element on page 10-887).

Time browser which adjust time to view results.

User notifications which opens the display of User Notifications, Alarms and Alerts.

SCADA Results publishing which enables the user to publish the results of the model run to an OPC server for use in the SCADA HMI (see Displaying Model Results in SCADA Human Machine Interface (HMI)-Overview on page 10-922).

Initial conditions which enables the user to import initial conditions from an external data source and use them instead of the conditions from the Baseline scenario.

SCADA Log which opens the log of messages that were generated by SCADA Connect.

Help which opens this help topic.

In the grid portion of the Simulator dialog, the user can provide the following information.

Baseline scenario is the existing EPS scenario that the user will run with any modifications made in this dialog. Default value is the current scenario.

Simulation mode indicates which of four different ways the scenario can be run (default value is baseline):

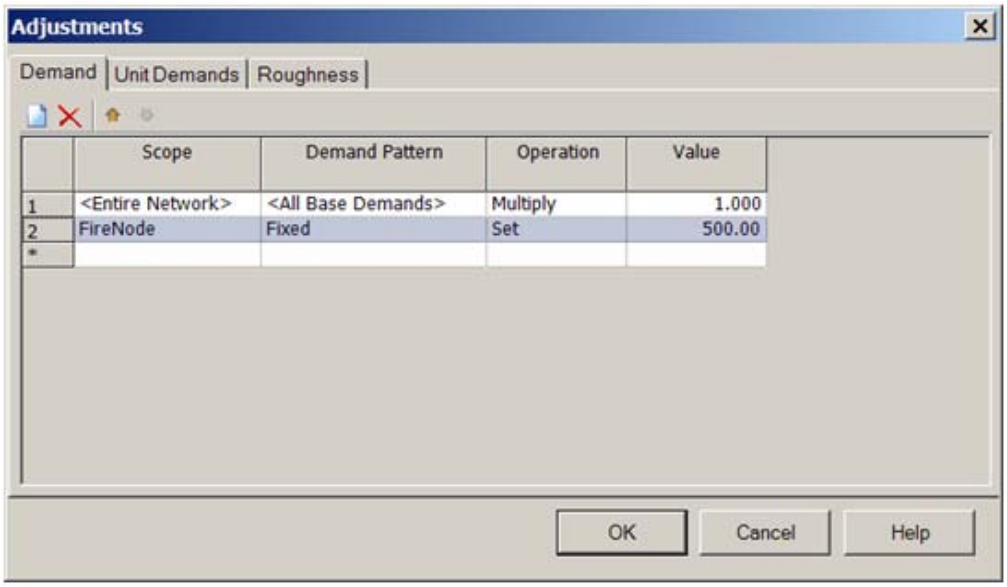
1. Baseline initial which runs the baseline scenario with no modifications
2. Historical which loads the model with initial conditions from a data and time for data found in an historical source
3. Live which loads the model with initial conditions from current data provided from an OPC server or the latest value from a database source.
4. Live (Auto Compute) which loads the data as from a Live run but will reload and rerun the model at a user specified time interval.

SCADA Calculation Type which identifies if the model run is to be hydraulics only, age, constituent or trace type runs. Default value is Hydraulics only.

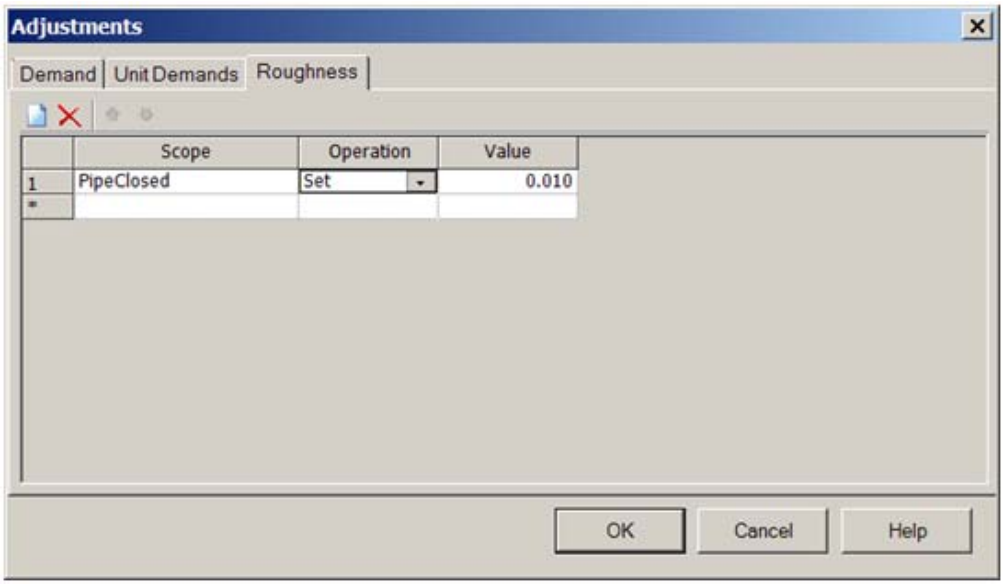
Demand adjustment enables the user to alter demands and pipe roughness when Active is selected using values from the entries below (Active Demand Adjustment or Unit Demand Adjustments).

When the ellipse button is selected next to Active Demand Adjustments or Unit Demand Adjustments, the Adjustment Dialog opens. This can be used to simulate a demand on a day that differs from the baseline or a fire or leak at a specific location or throttling or closing a pipe or pipes. Any adjustments apply to the duration of the EPS run. Within the dialog there are tabs to edit the Demands, Unit Demands and Roughness. Scope refers to a selection set of elements to which the adjustments are to be

applied while Demand pattern is a filter so that demand adjustments are only applied to elements that have a particular demand pattern. Changes made with adjustments are not passed into the physical or demand alternatives and will not be detected using Scenario Comparison.



Values for roughness depend on the Friction Method specified in the Calculation options and Set can refer to Hazen-Williams, Darcy-Weisbach or Manning coefficients.



Any adjustments are saved in the Calculation Options for the current scenario and are therefore shared with any other scenarios that use the same calculation option. For this reason it is desirable to set up a separate Calculation Option for scenarios that will use adjustments so that they don't affect other scenarios.

The Calculation Time is used to set the times in the model. This property is not used in the model because in a Live run, the model uses the current time as the starting time and the Starting Time property is not used.

The duration refers to the length of the simulation.

If Live (Auto Compute) is selected as the Simulation Mode, then the user sets the interval between auto runs entering a time in Auto Compute Interval.

The user can override controls for elements by picking the ellipse button in the Control Overrides property (see Control Overrides in SCADAConnect Simulator on page 10-921).

Control Overrides in SCADAConnect Simulator

The simulator user may want to control elements in the model in a different way that specified in the control statements in the Baseline scenario. This can be done in the Simulator dialog by picking the ellipse button in the Control Overrides field. This opens the Control Overrides dialog. Pick New to create a new override.

The user is then prompted to pick the element to override. This includes pumps, pipes and most control valves.



The columns in the Control Override are described below:

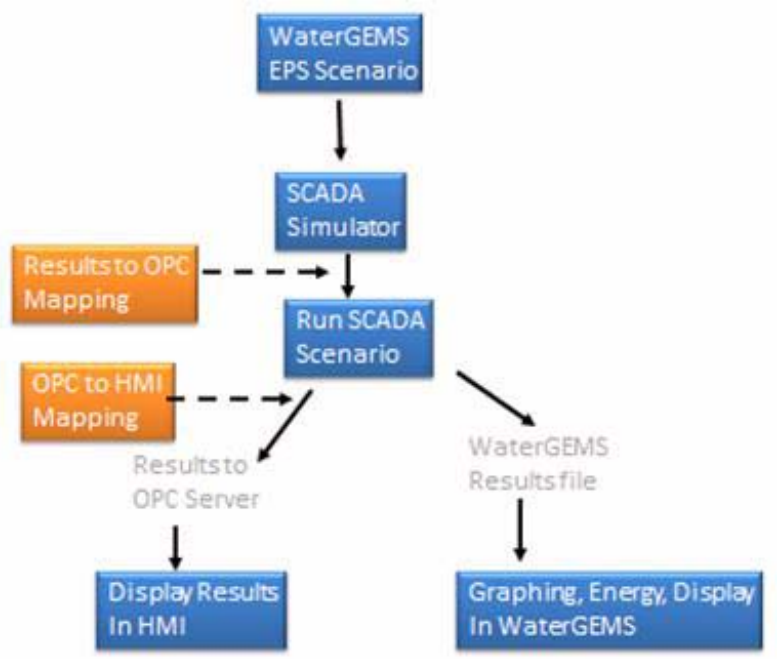
- Enabled? Should be checked if the user wants to use the override in the current simulation.
- Controlled element is the label of the element to be controlled.
- Element type is a read-only column identifying the type of element.
- Attribute is used to specify which setting or status is to be overridden. These are only to be used for control properties. For example, pipes can be open or closed but their diameter cannot be changed here.

- Value is the new status or setting. Status refers to on/off, open/closed properties while setting refers to continuous properties like pump speed.
- Start time is the time when the override begins.
- Duration is the length of time the override is in force.
- Priority is used to determine which control takes preference when there are conflicts between overrides.
- Notes can be used as reminders of why the override is being used.

Control overrides are saved with the Calculation options for a scenario. If the user wants to experiment with overrides but does not want to affect an existing scenario, then the user should create a new scenario with a new set of calculation options.

Displaying Model Results in SCADA Human Machine Interface (HMI)-Overview

WaterGEMS/CAD enables a user to run an EPS analysis and display the results in a SCADA HMI. The process is summarized in the figure below. The blue boxes are the work flow for a user. The orange boxes are configuration steps that are set up once (unless changes are needed).

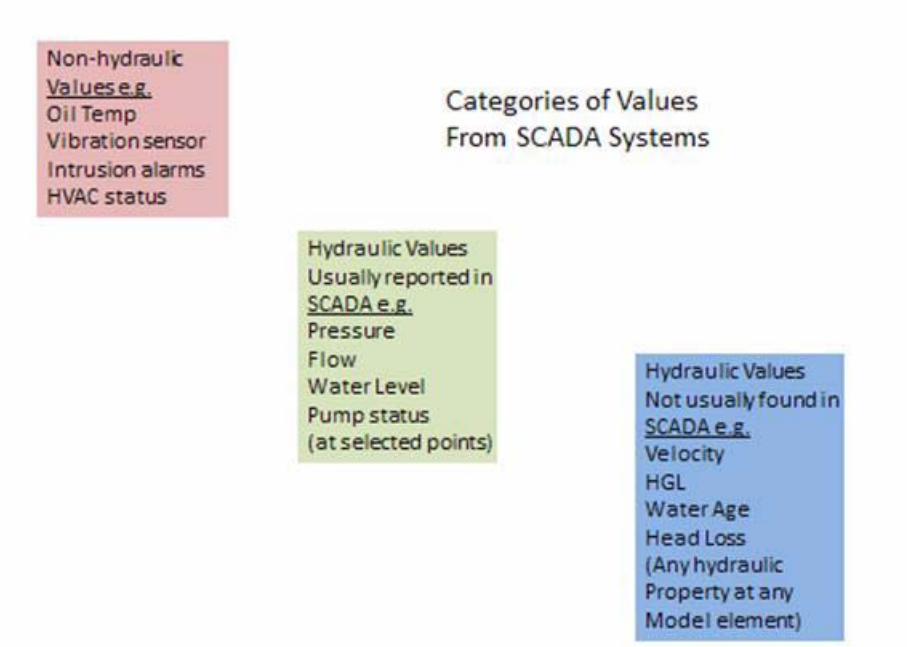


Model results are not directly displayed in the HMI. As is typical of SCADA systems, values are placed in an OPC server and these values are then displayed from the OPC server to the HMI. Therefore it is necessary to define mappings from the model to the OPC server tags and then identify which tags are displayed in which locations in the HMI. ("Tags" refer to the name of a property in the server.)

Configuration Steps

Because there are numerous brands of OPC servers and HMI software, it is not possible to give detailed steps on setting up the server and configuring the HMI. Users are referred to the documentation of those individual products for instructions. The user needs to install the OPC server and the HMI software before running WaterGEMS/CAD SCADA simulations. The server can be set up on the computer running the model or hosted on another computer networked to the model.

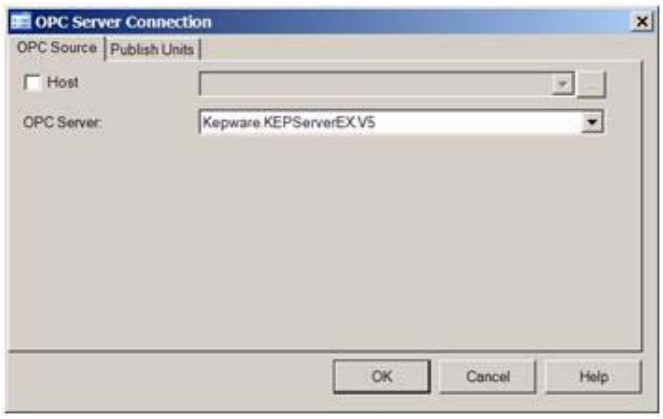
It is best to use the same versions of the server and software that the system operators are using so that users will be familiar with the software. It may be possible to make a copy of the server and display files for use with the model. However there is not a one-to-one relationship between model parameters and values that are displayed in the operator's HMI. The model can calculate far more properties at more location than the SCADA system but the model is not concerned with non-hydraulic properties. This can be visualized as shown below. The properties in the blue box are an example of information that can be provided to the HMI from the model that is not available from the SCADA system.



Once the OPC server is set up, the user can define the tags that are to be published from each model run. These would normally include the hydraulic properties that are displayed in the SCADA system but can include a wide variety of values that the model can generate. (These are essentially "software sensors" as opposed to the physical sensors in the field.) Before associating model element with SCADA tags, the user must set up the tags in the SCADA OPC server in accordance with the procedures for that server.

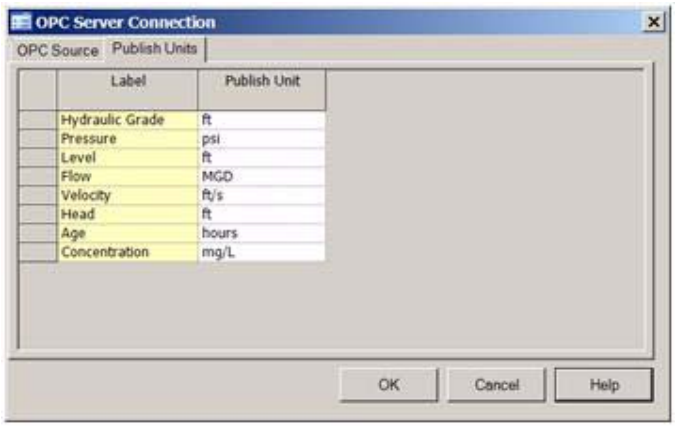
The mappings from the model to the server are set up by selecting Tools > SCADAConnect Simulator > SCADA Results Publishing (sixth button from left in SCADAConnect Simulator dialog).

This opens the SCADA Results Publishing table where the user defines which properties are associated with each tag on the server. The first time this is done, the user must first pick the OPC server by selecting the fourth button.

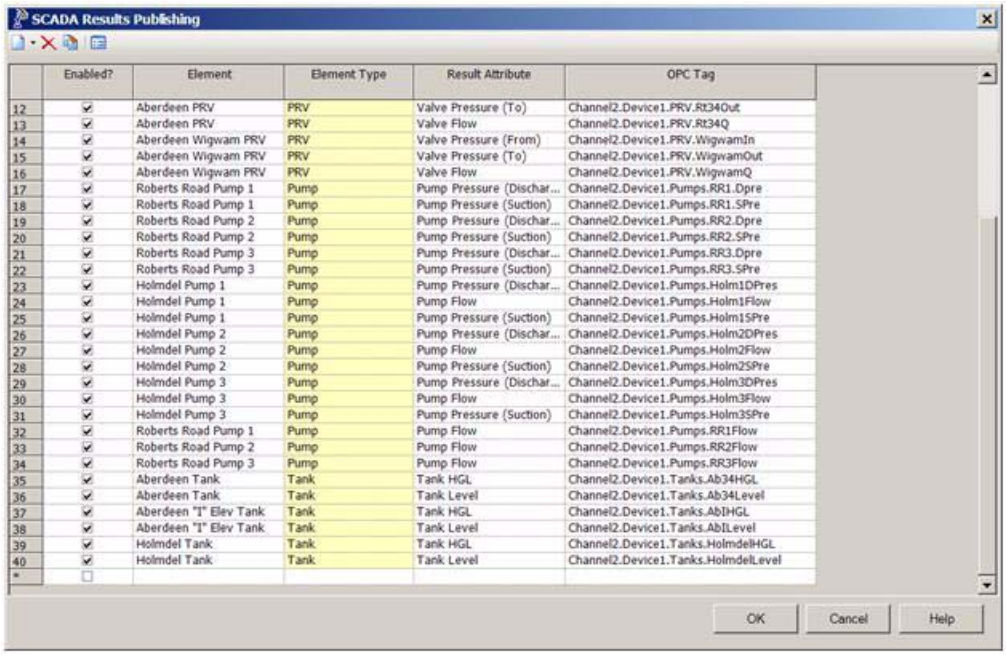


If the server is on a different computer than the model, the user identifies this by checking the Host box and navigating to the server location using the ellipse button.

The user can also specify the units associated with the values in the OPC server which may be different from the values in the model.



The user then fills in the SCADA Results Publishing Table by specifying which model element and property are associated with each tag. If the user does not wish to use to publish the result for a given property in a given run, the Enable box should be unchecked.



	Enabled?	Element	Element Type	Result Attribute	OPC Tag
12	☑	Aberdeen PRV	PRV	Valve Pressure (To)	Channel2.Device1.PRV.Rt34Out
13	☑	Aberdeen PRV	PRV	Valve Flow	Channel2.Device1.PRV.Rt34Q
14	☑	Aberdeen Wigwam PRV	PRV	Valve Pressure (From)	Channel2.Device1.PRV.WigwamIn
15	☑	Aberdeen Wigwam PRV	PRV	Valve Pressure (To)	Channel2.Device1.PRV.WigwamOut
16	☑	Aberdeen Wigwam PRV	PRV	Valve Flow	Channel2.Device1.PRV.WigwamQ
17	☑	Roberts Road Pump 1	Pump	Pump Pressure (Dischar...	Channel2.Device1.Pumps.RR1.Dpre
18	☑	Roberts Road Pump 1	Pump	Pump Pressure (Suction)	Channel2.Device1.Pumps.RR1.SPre
19	☑	Roberts Road Pump 2	Pump	Pump Pressure (Dischar...	Channel2.Device1.Pumps.RR2.Dpre
20	☑	Roberts Road Pump 2	Pump	Pump Pressure (Suction)	Channel2.Device1.Pumps.RR2.SPre
21	☑	Roberts Road Pump 3	Pump	Pump Pressure (Dischar...	Channel2.Device1.Pumps.RR3.Dpre
22	☑	Roberts Road Pump 3	Pump	Pump Pressure (Suction)	Channel2.Device1.Pumps.RR3.SPre
23	☑	Holmdel Pump 1	Pump	Pump Pressure (Dischar...	Channel2.Device1.Pumps.Holm1DPres
24	☑	Holmdel Pump 1	Pump	Pump Flow	Channel2.Device1.Pumps.Holm1Flow
25	☑	Holmdel Pump 1	Pump	Pump Pressure (Suction)	Channel2.Device1.Pumps.Holm1SPre
26	☑	Holmdel Pump 2	Pump	Pump Pressure (Dischar...	Channel2.Device1.Pumps.Holm2DPres
27	☑	Holmdel Pump 2	Pump	Pump Flow	Channel2.Device1.Pumps.Holm2Flow
28	☑	Holmdel Pump 2	Pump	Pump Pressure (Suction)	Channel2.Device1.Pumps.Holm2SPre
29	☑	Holmdel Pump 3	Pump	Pump Pressure (Dischar...	Channel2.Device1.Pumps.Holm3DPres
30	☑	Holmdel Pump 3	Pump	Pump Flow	Channel2.Device1.Pumps.Holm3Flow
31	☑	Holmdel Pump 3	Pump	Pump Pressure (Suction)	Channel2.Device1.Pumps.Holm3SPre
32	☑	Roberts Road Pump 1	Pump	Pump Flow	Channel2.Device1.Pumps.RR1Flow
33	☑	Roberts Road Pump 2	Pump	Pump Flow	Channel2.Device1.Pumps.RR2Flow
34	☑	Roberts Road Pump 3	Pump	Pump Flow	Channel2.Device1.Pumps.RR3Flow
35	☑	Aberdeen Tank	Tank	Tank HGL	Channel2.Device1.Tanks.Ab34HGL
36	☑	Aberdeen Tank	Tank	Tank Level	Channel2.Device1.Tanks.Ab34Level
37	☑	Aberdeen "1" Elev Tank	Tank	Tank HGL	Channel2.Device1.Tanks.AbIHGL
38	☑	Aberdeen "1" Elev Tank	Tank	Tank Level	Channel2.Device1.Tanks.AbILevel
39	☑	Holmdel Tank	Tank	Tank HGL	Channel2.Device1.Tanks.HolmdelHGL
40	☑	Holmdel Tank	Tank	Tank Level	Channel2.Device1.Tanks.HolmdelLevel
*	☐				

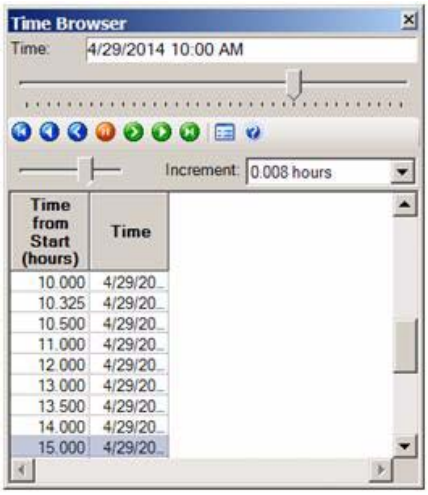
Once the mappings are complete, when a model run is completed, the results can be published to the OPC server for use in the HMI.

When model results are associated with a tag in the OPC server, the user must define how the result value is displayed in the HMI. It is best to model this behavior after the actual HMI although, as stated above, not all the values in the HMI are hydraulic values and the model can populate more tags than the actual HMI. The user can create a new set of HMI screens or can modify the existing screen to accept additional values.

Model to HMI Work Flow

Once the mappings from the model to the OPC server and on to the HMI display have been completed, the results of any model run, for which the Calculation Type is designated as SCADAConnect Simulator, are available for display. At the start of a SCADA runs, a series of checks are made to determine such things as whether a tag is available in the OPC server for each mapped property or if a model element is active for each tag. Any errors or warnings can be found in the SCADA Log which is the eight button at the top of the SCADAConnect Simulator dialog.

When the HMI is started, the results will be those corresponding to time 0. To advance to different times or run continuously, the user must open the Time Browser (Analysis > Time Browser). The user can advanced time using the buttons at the top of the dialog or pick a specific time to view from the list of times in the lower portion of the dialog.



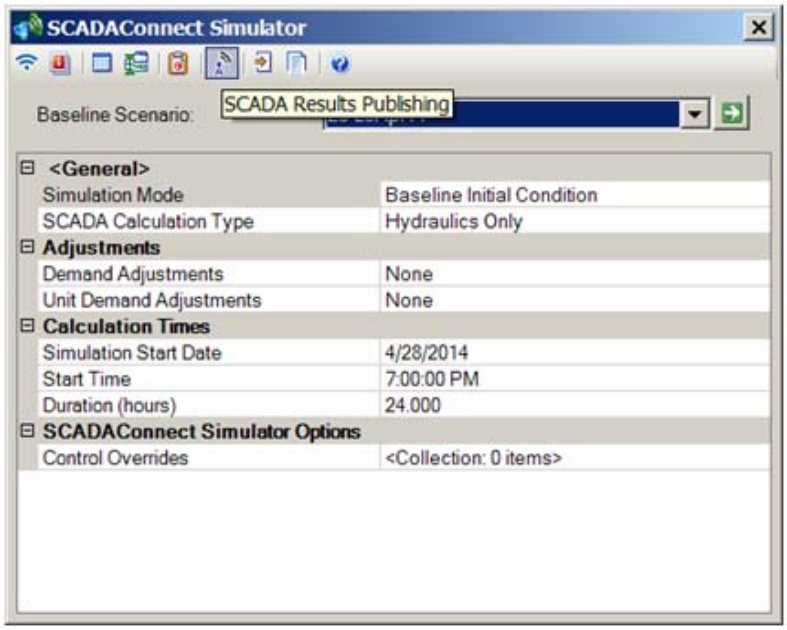
The controls for the HMI such as panning, switching screens etc. depend on the brand and version of the HMI and instructions can be found with that software.

SCADAConnect Results Publishing Dialog

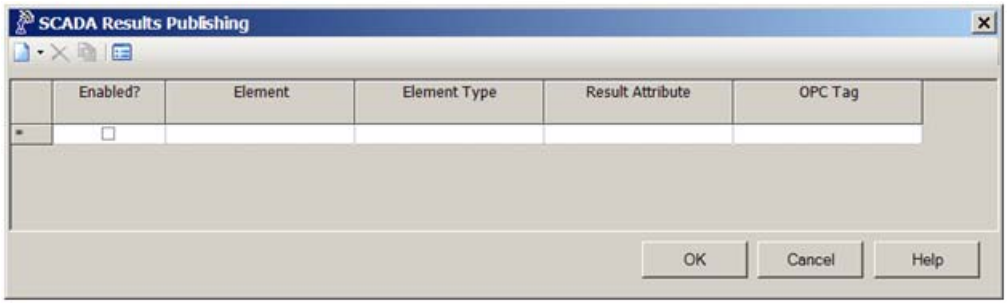
Model results can be published to an OPC server for display in a Human Machine Interface (HMI). The user must identify the mapping between the model element and the tag/signal in the OPC server. The hydraulic models do not directly display their results in the HMI but instead publish the results to any OPC compatible server. The HMI then reads from the server and display the results. When the model Time Browser advances to a new time, the published results are updated. The signal mappings to the server are explained below.

In most cases, the OPC server to HMI programming already exists for the real SCADA system and can be used as an example when setting up model to HMI mappings. The difference is that the model is only interested in hydraulic properties and can display far more hydraulic properties at more locations than the actual SCADA system.

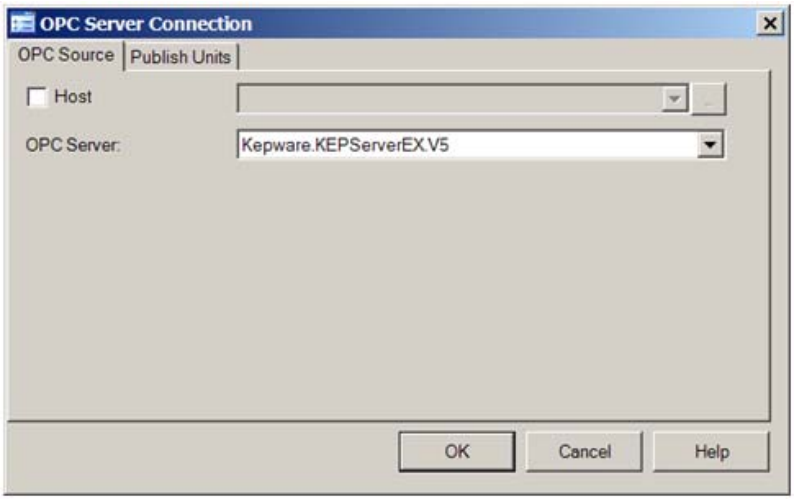
The SCADA results publishing dialog can be accessed using Tools > SCADAConnect Simulator and picking the sixth button (SCADA Results Publishing) in that dialog:



This opens up the Results Publishing Dialog. The first step is to identify the OPC server to be used. This is done by picking the fourth button on top of the dialog, Define OPC Server Connection.



This opens the dialog below where the user identifies the OPC server to be used. If the server is hosted on a different computer, the user should check the box labelled Host and search for the computer on the network. If the server is located on the user's computer, the user need only search from the drop down list of servers.



The other buttons at the top if the dialog include, Open an addition signal to model mapping, Delete a mapping and Duplicate a mapping.

Before opening this table, the signals/tags need to be identified in the OPC server. This varies between OPC servers and the user needs to consult the documentation for the specific brand and version of the server.

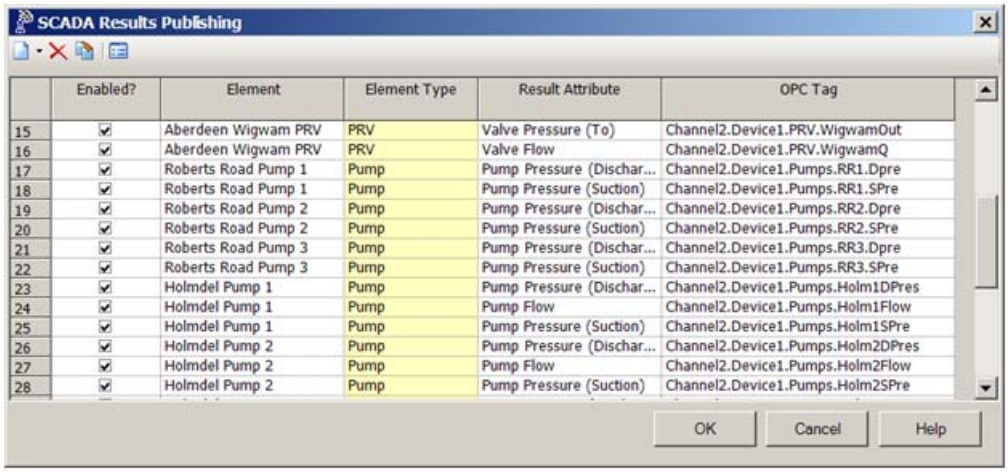
The columns in the table include

Enable which when checked indicates that the row in this table is to be used for publishing. Unchecked, the results are ignored.

Element indicate which model element is to be used for the mapping. The Element Type is automatically populated as a read-only field.

The Result Attribute identifies which property from the model element is to be mapped to the OPC tag.

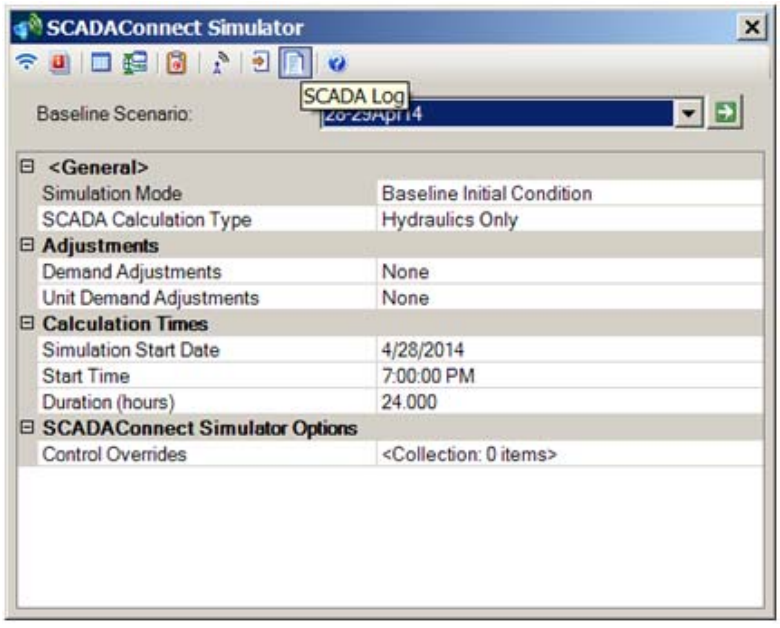
The OPC Tag is the tag/signal name of that property in the OPC server. These tags should be established before opening this table.



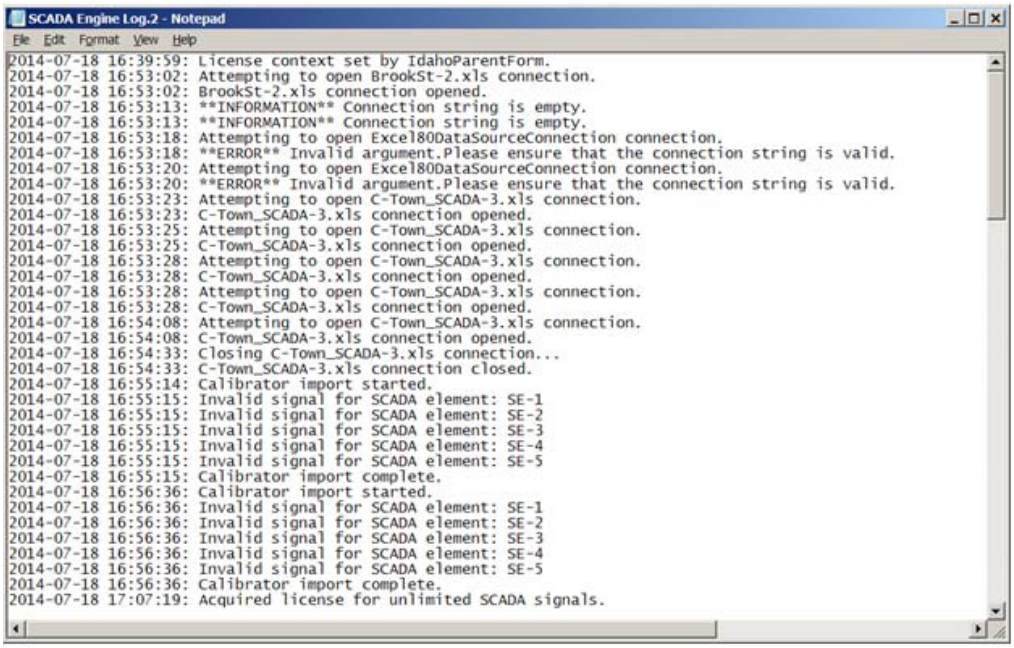
Picking OK sets up the mappings in the model such that as the Time Browser is advanced, the correct value is published to the OPC server.

SCADA Log

On occasion, the SCADA to model connections generate error/information messages. These messages are displayed in the SCADA Log which can be accessed by Tools > SCADAConnect Simulator and selecting the eight button.



This opens the SCADA Log as shown below. The log is a Notebook file which can be helpful in diagnosing problems and communicating with Bentley Technical Support.



SCADAConnect Overview

SCADAConnect is a tool in Bentley hydraulic models that enables a user to interact with SCADA systems or other external data sources (e.g. data loggers) that provide temporal data. These data can be used to initialize model runs, calibrate models, and help modeling and operation personnel share information with minimal effort.

To use SCADAConnect, the user establishes a connection between a data source and the model. The data source can be a data file with time series data or an OPC server. The user then associates individual signals from the data source with signal elements in the model (see SCADA Element on page 10-887). These signal elements are in turn associated with hydraulic model elements.

SCADAConnect can be accessed by picking Tools > SCADAConnect (xref SCADAConnect toolbar) or Components > SCADA Signals (see SCADA Signals Dialog on page 10-931).

While SCADAConnect works with both 32 and 64 bit systems, it is best to run the 32 bit version of the Bentley modeling product with a 32 bit SCADA system and the 64 bit version of the model with a 64 bit SCADA system.

SCADAConnect Toolbar

When the user selects Tools > SCADAConnect, the following toolbar becomes available:



The first button opens the SCADA signals dialog which can also be reached from Components > SCADA Signals. This is where connections and signal mappings are created (see SCADA Signals Dialog on page 10-931).

The second button opens the SCADA flex tables which can also be opened from View > Flex Tables. It enables the user to view SCADA and model values in the same table.

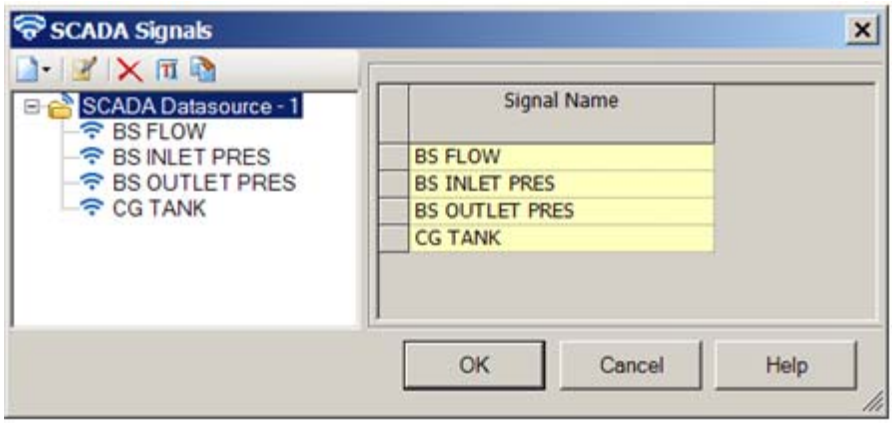
The third button opens the initial setting dialog which enables the user to select a time and import values of certain properties into the initial conditions alternative such as wet well and pump status (seeInitial Setting Import Dialog on page 10-939).

The fourth button creates a SCADAConnect log which enables the user to view what SCADAConnect did and is helpful in debugging problems.

The fifth button opens this Help topic.

SCADA Signals Dialog

The SCADA Signals dialog enables the user to associate a SCADA data source with a model and then map the individual signals to signal elements in the model.



The buttons on top of the left pane are described below:

New enables uses to create a new Database or OPC data source. See help topics on Database Source Dialog and Real Time or Historical OPC Source Dialog for details. [xref to those help topics]

Edit enables the users to view and modify previously defined database or OPC sources by opening the detailed dialogs.

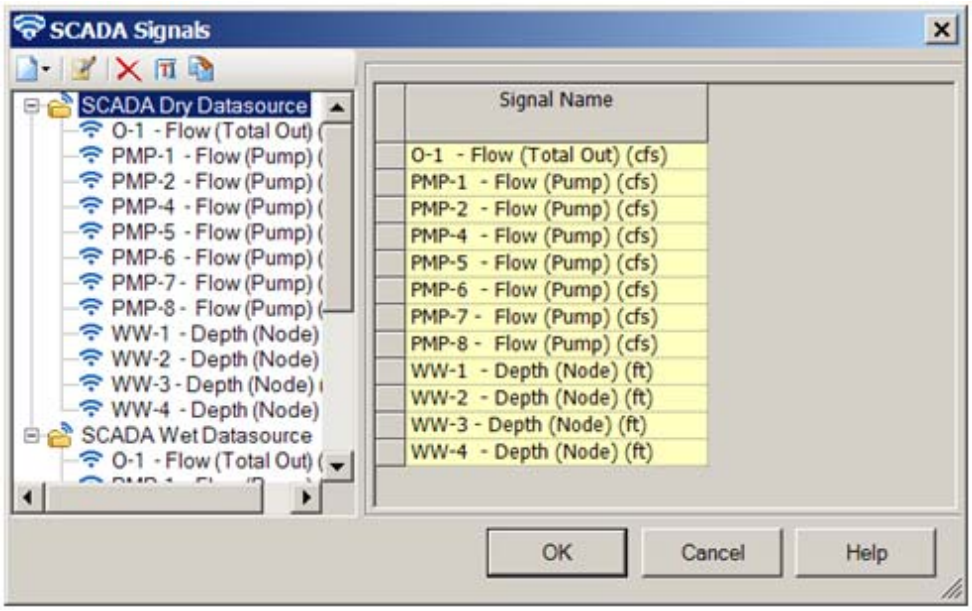
Delete deletes the existing highlighted datasource or signal.

Rename allows the user to rename the selected data source or signal.

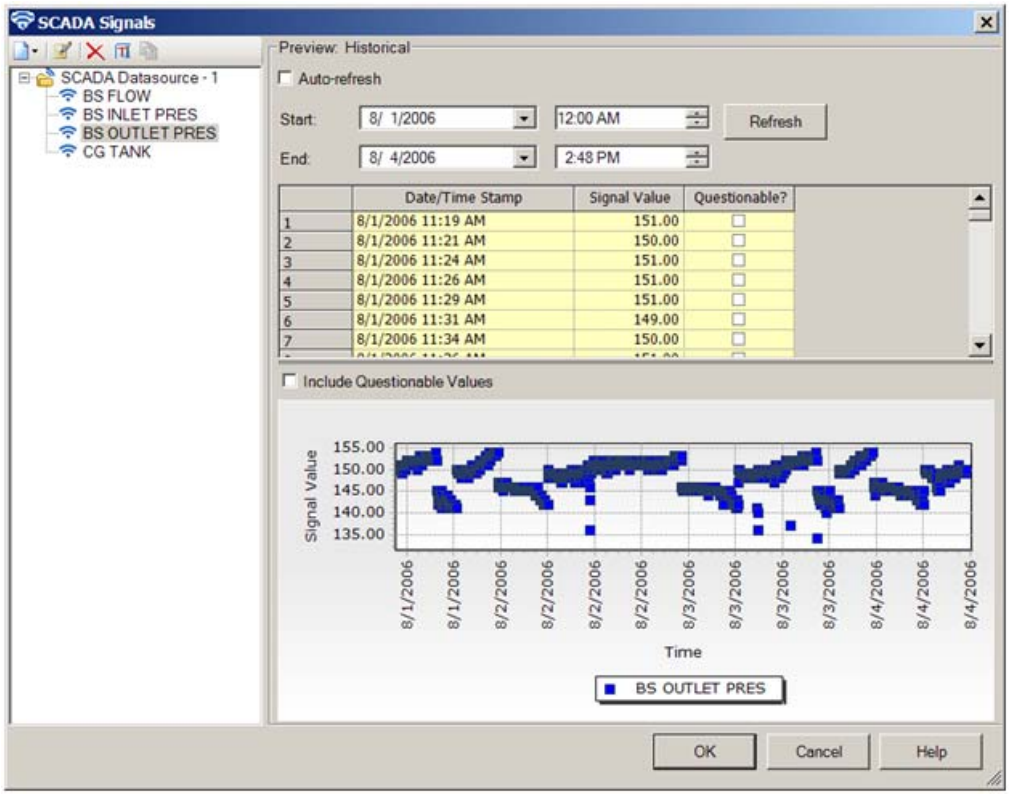
Duplicate enables the user to duplicate an existing datasource.

The behavior of the right pane depends on whether the user has selected a data source or a signal in the left pane.

If a datasource has been selected, the right pane will display a list of signal names.



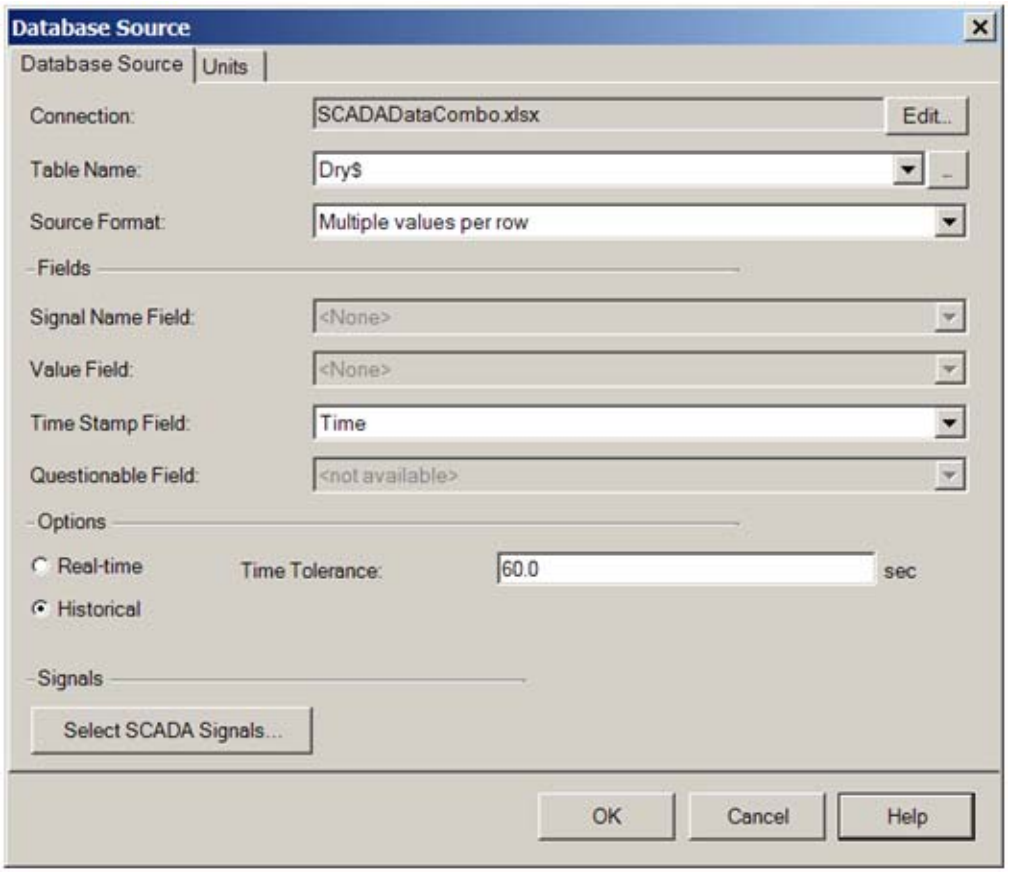
If a signal has been selected, the right pane will display a preview of the data for that signal. To ensure that the data are current, the user can pick Refresh button or Auto Refresh check box.



SCADA Database Source Dialog

Picking the New > Database Source or Edit a Database source in the SCADA signals dialog opens the Database Source editor.

The first step is to establish a database connection which initiated by pick the Edit button, in the Connection field (see Database Connection Dialog on page 10-936)..



Once the Connection to the file has been established, the user can further refine the connection. For example in an Excel file or an Access database, the user can specify which table is to be used for the connection. If multiple tables are used, each one must have its own connection.

The user must pick from one of two data source formats: One value per row or Multiple values per row.

In the "One value per row" format, each row/record must contain the signal name field (e.g. Pressure Main St.), the value field (e.g. 60) and a time stamp filed (e.g. 2/4/2015 14:30). Optionally, the user may also specify if there is a field indicating the quality of the data, Questionable Field (e.g. Missing). There may be other fields on each record but these are ignored. The order of the fields doesn't matter. The Time Stamp Field must include a recognized Windows data/time value. A typical row might include:

3:40 am 3/5/2016 Flow Pump 3 34 Good

In Multiple values per row, there must be a field/column header with the signal name and there may be multiple fields in a single row. A typical first row and subsequent row may include:

Date/time	Pressure Main St	Flow Main St.	Temperature	Tank Level South
2 Oct 14 13:08.15	48.34	834.2	35	12

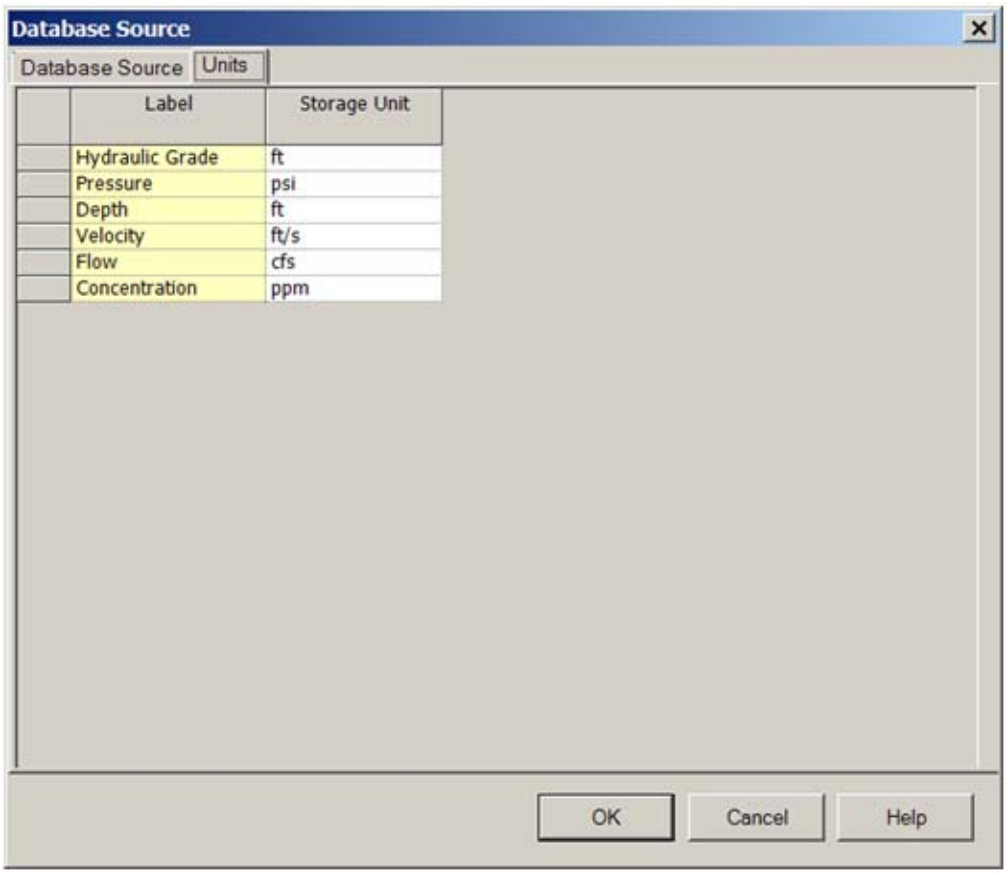
Within a Database source, there are two ways that time is used. The user selects these under Options. In "Historical", the values can be read for any time for which data are available. For "real time", only the most recent value is read and then, only if it is within the time tolerance specified by the user. This is used for importing initial conditions for starting a run or viewing the latest value that has been placed in the datasource file. The time is based on the clock in the computer running the model.

The "Time Tolerance" field is used because the time stamp on a SCADA signal may not correspond exactly to the time for which data are called for from the model. For example, the model may require a value at 5/16/2015 3:15:00 am but the closest value in the SCADA system may be 5/16/2015 3:13:25 am. If the difference is greater than the tolerance, no value is obtained. It is best to set a wide tolerance initially which can be reduced as higher quality information is required. If multiple values are available within the tolerance, the value closest to the required time is used.

To select which signals from the SCADA system are to be mapped to the SCADA signal element, the user picks "Select SCADA Signals" (see OPC Select Signals Dialog on page 10-940).

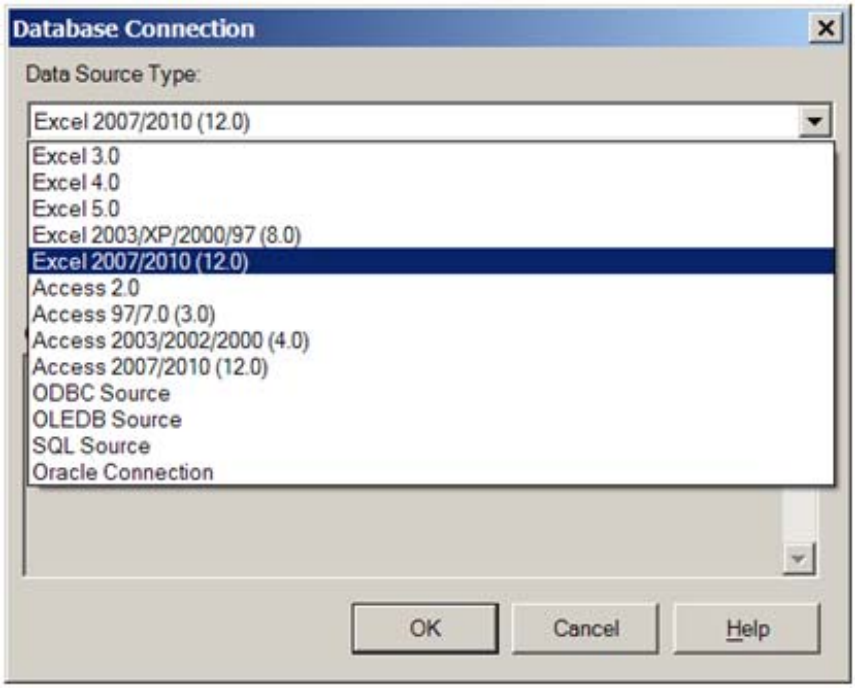
The units for an imported value are assumed to be the display units in the model. If there is a chance that the units may be different in the SCADA file than in the model,

the user can override the units by selecting the units tab and entering the "Storage units".

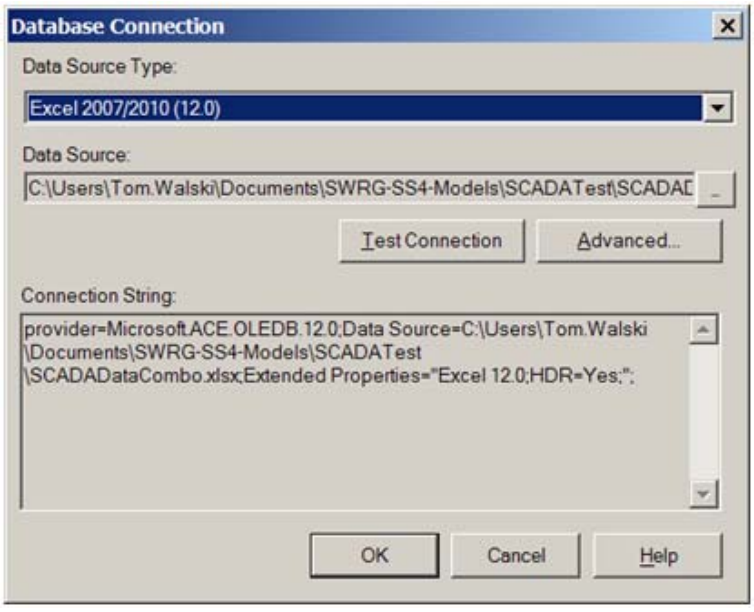


Database Connection Dialog

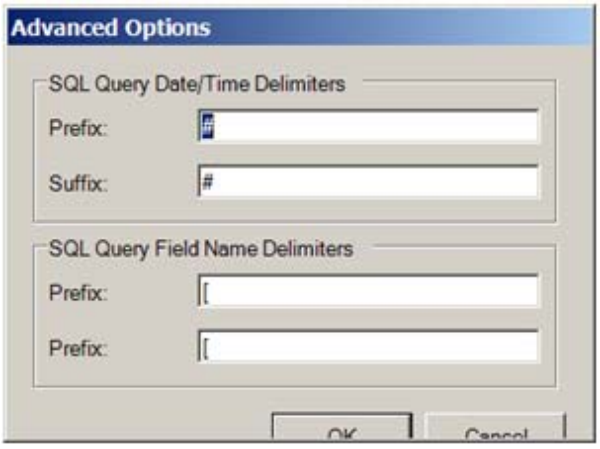
Picking the Edit button next to the Connection field allows the user to pick the type of data source from a list which includes the following file types.



Once the Data Source type has been selected, the user can pick the ellipse button in the Database Connection dialot to navigate to the data source file. Once the file has been selected, the user should pick Test Connection to check that the file matches the Data Source Type and is in a valid format.

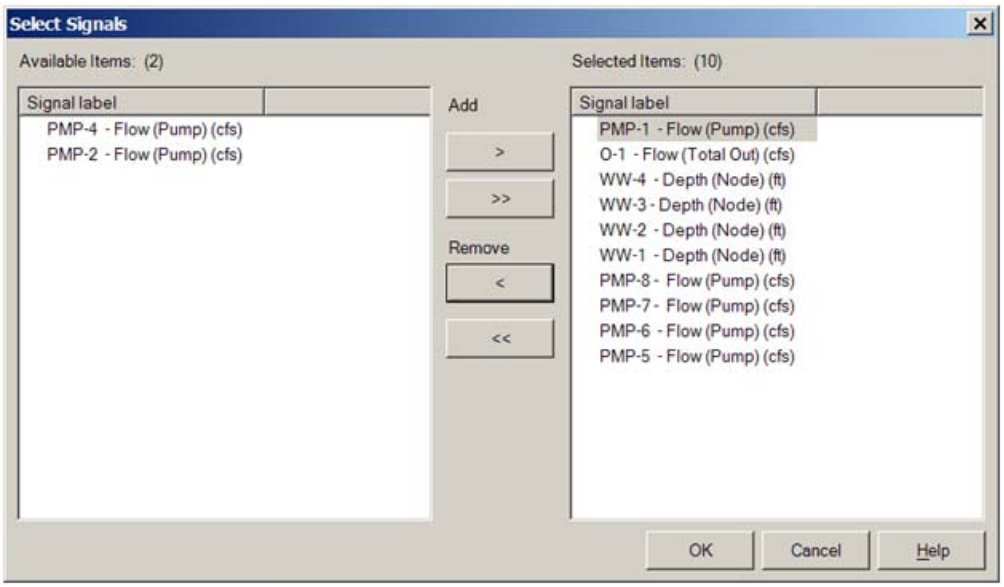


If the Advance button is selected, the user can set up the prefix and suffix for SQL type queries of the data source.



Database Select Signals Dialog

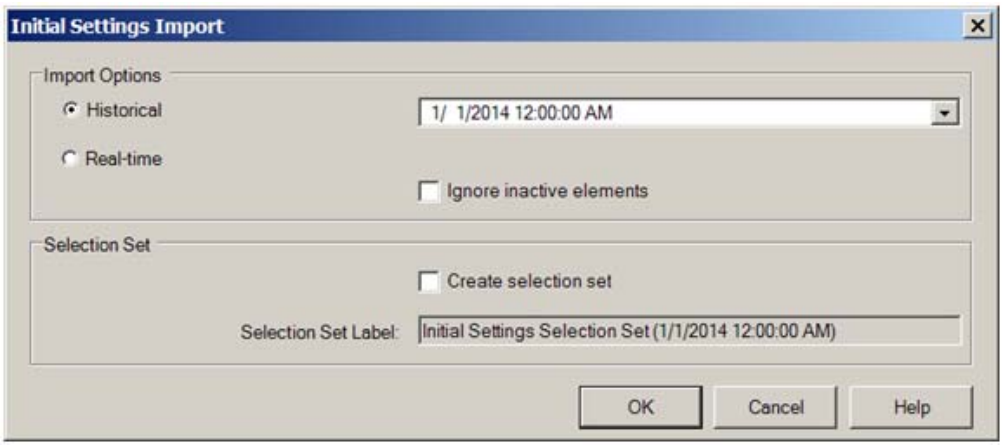
Not all of the signals captured by the SCADA system are of interest to hydraulic modeling and can be assigned to SCADA signal elements. Some examples of signals that aren't needed may include amperage, temperature, intrusion, loss of phase, etc. However, these signals may be in the SCADA datasource. To select which signals are to be associated with the model, the user selects "Select SCADA Signals" from the Datasource dialog (see SCADA Database Source Dialog on page 10-933).



To select signals, pick the signal from the available signals in the left pane and pick the Add button (>) to move it to the right pane. Alternatively, a user can Add all the signals (>>) and Remove (<) the few that are not needed.

Initial Setting Import Dialog

When the Initial settings button is selected in the SCADAConnect toolbar, the following dialog opens:

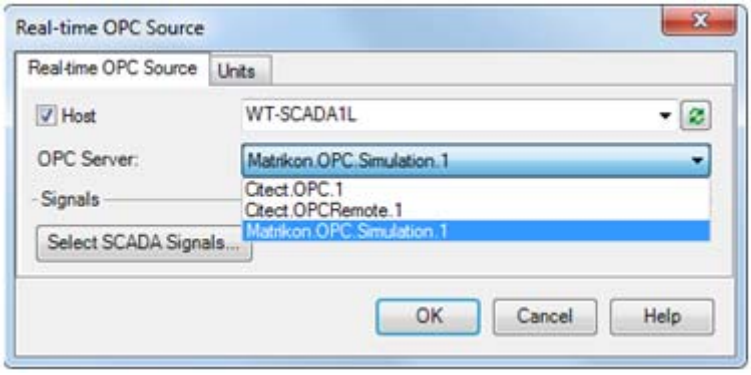


The user selects whether to use a Historical or Real Time datasource. The mapping between the SCADA system and the hydraulic model must be created before the initial condition can be imported. If a Historical data set is selected, the user must pick a time for which values are imported. If a real time datasource is selected, the most recent values are used.

If "Ignore inactive elements" is checked, only values for active elements are imported. This can help speed up imports.

Real-time OPC Source Dialog

When the user selects Real-time OPC source as the signal source, the program can pull current data directly from an OPC server. The following dialog opens:

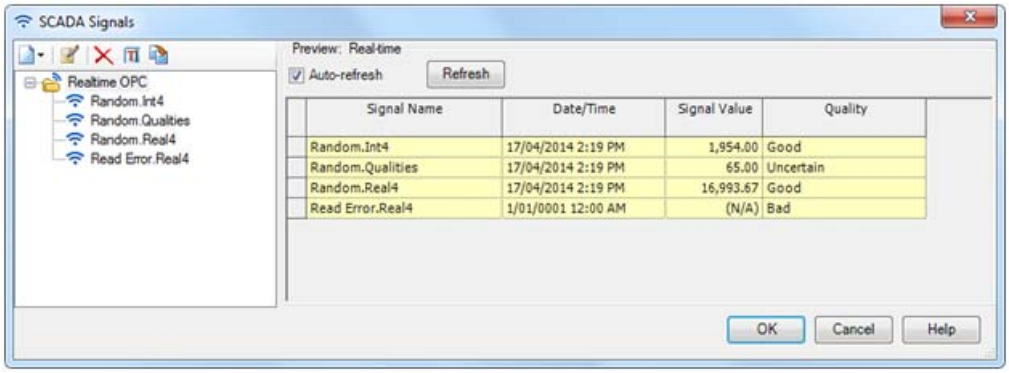


The Host field is used to identify the computer hosting the OPC server on the network. The refresh button to the right of the field searches the network for the server. If unchecked, it is assumed that the server is on the computer running the hydraulic model.

The OPC Server is the name of the server since there can be several servers on a computer.

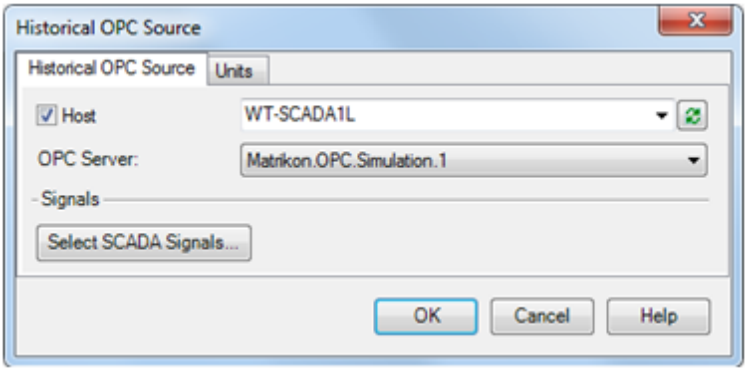
The Select SCADA Signals button opens the dialog to select signals as described in the Select Signal (OPC) help topic. [xref Select Signals (OPC)]

The real time OPC signal values can be viewed in the SCADA Signals dialog as shown below:



Historical OPC Source Dialog

When the user selects Historical OPC source as the signal source, the program can pull current data directly from and OPC server. The following dialog opens:



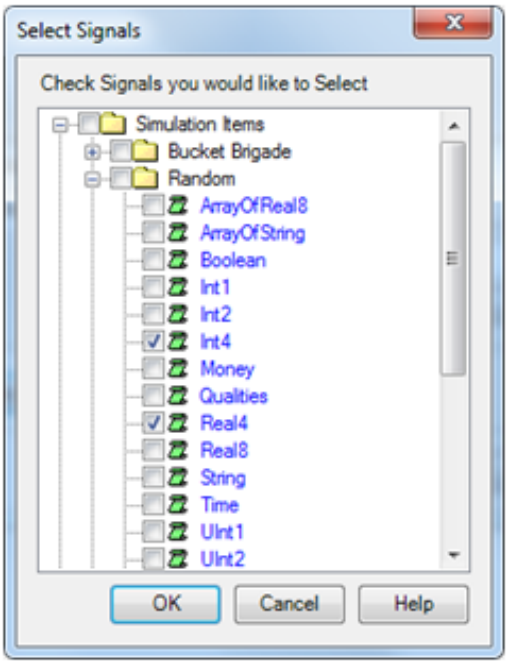
The Host field is used to identify the computer hosting the OPC server on the network. The refresh button to the right of the field searches the network for the server.

The OPC Server is the name of the server since there can be several servers on a computer.

The Select SCADA Signals button opens the dialog to select signals as described in the Select Signal (OPC) help topic (see OPC Select Signals Dialog on page 10-940).

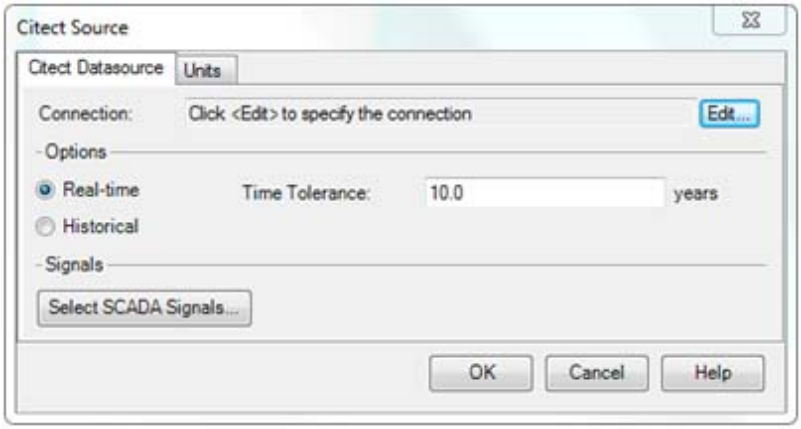
OPC Select Signals Dialog

When Select Signals button is picked in either the Real-time or Historical OPC setup, the following dialog is opened. The user expands the tree view and picks the desired signals.



Citech Source Dialog

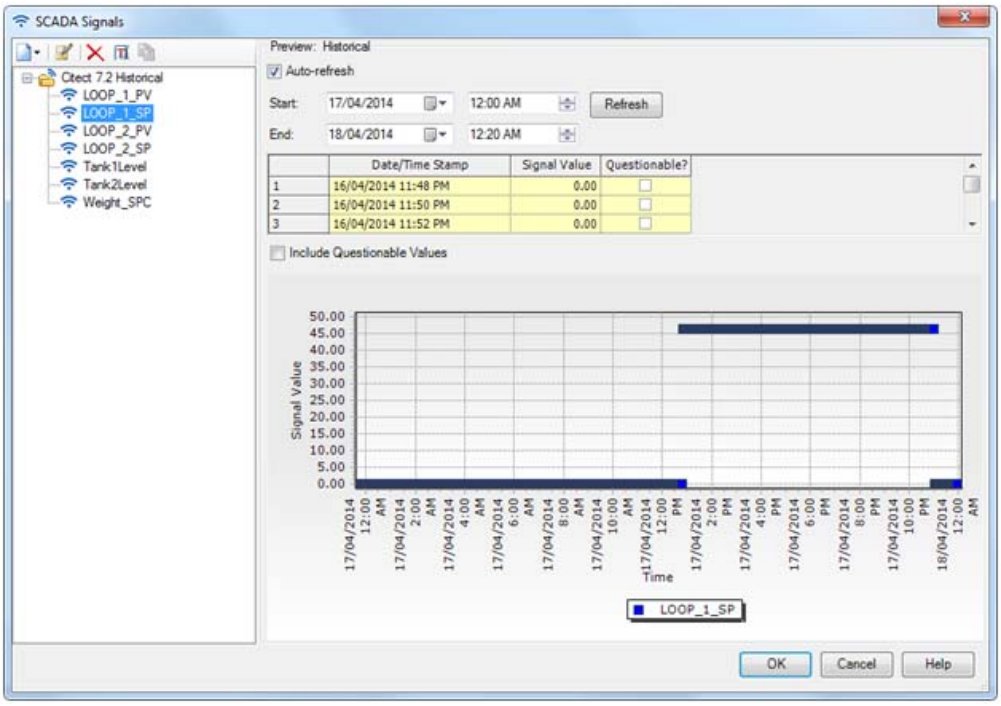
When the user selects a Citech data source when setting up SCADA signals, the following dialog opens.



The edit button can be used to pick the Citech connection as shown in Citech Connection Dialog help (see Citech Connection Dialog on page 10-942).

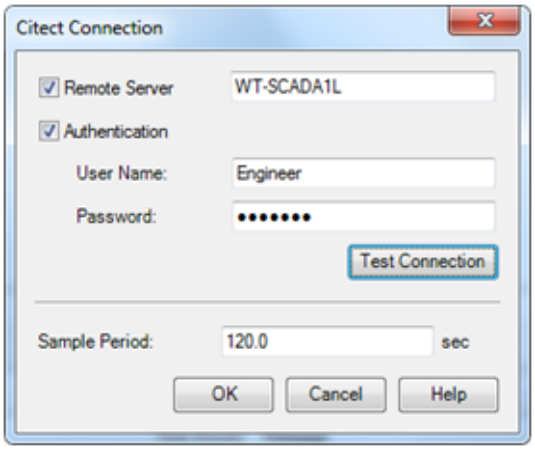
The user can pick whether historical or real time values are required and can set a tolerance which limits the range of times over which values will be checked. The default value is a very large number.

The signal values from a Citech server will appear as shown below:



Citech Connection Dialog

When the user picks a Citech connection, it is established in the dialog below:



When Remote server is checked, the use must enter the computer name. If unchecked, the server is assume to be on the same computer as WaterGEMS. Authentication can be specified for those servers requiring it.

SCADAConnect Manager

The SCADAConnect manager contains the following menus:

File

Exit - Exit SCADAConnect.

Tools

Connection Manager - Specify several different databases or data servers. Multiple SCADA Connection links can be established for multiple files/data/server locations. This is only needed for database data sources and Citect connections.

Data Source Manager - Specify tables or data sources from within each data server specified in the Connection Manager.

Load to Calibrator Field Dataset - Populates a new calibration field data set in Darwin Calibrator with SCADA data which may be historical or real-time.

Load to Initial Settings - Populates the initial settings alternative with real-time SCADA data.

Load to Extended Data - Load the SCADA data into a Custom Field (User Data Extension).

Load Demand Base Average Values -Calculates an average demand value over a full day and writes it to the demand alternative of the model.

Demand Inversing -Opens the Demand Inversing dialog box to calculate daily zone demands based on SCADA data.

View SCADA Data - Displays the SCADA data directly from the SCADA database source without importing to the hydraulic model in a tabular grid for a specified time period.

Options - Provides access to customizable options.

SCADAConnect Workflow

To use the SCADAConnect, the usual steps consist of

Open SCADAConnect. (**Tools** > **SCADAConnect**)

Setup SCADA connection in Connection Manager (if needed).

Provide database information in Data Source Manager.

Establish Signal Mapping.

View or Load data from SCADAConnect to the model (Load SCADA Data and Viewing SCADA Data).

Demand Inversing if necessary.

Use data in hydraulic model.

See also: Options and Miscellaneous SCADAconnect Operations.

Connection Manager

The Connection Manager is used to create a new SCADA connection and to edit the existing SCADA connection. SCADA connection is basically an item of the Connection Manager in which information like, the data source, type of connection used, method of connection are provided. You can also specify the server location, user name, password etc. depending upon the selected connection method. Two generic types of connections can be established.

Database Connection

Citect Connection

Database Connection

SCADAConnect can read data from a variety of sources. Upon creating a new connection, the user will need to select whether the connection is to a database or a Citect server. The Database Connection method helps to establish a communication to a SCADA file. A file can be accessed (or opened) using different methods (or drivers) such as ODBC, OLEDB or SQL. To simplify this to a general hydraulic modeler, SCADAConnect offers two additional methods where modeler can simply specify a file such as Excel or These direct file methods also uses OLEDB method. Following are the methods you can select to provide information about your SCADA data.

Access File

Access 2.0

Access 2003/2002/2000 (4.0)

Access 2007 (12.0)

Access 97/7.0(3.0)

Excel File

Excel 2003/ XP /2000/97 (8.0)

Excel 2007 (12.0)

Excel 3.0

Excel 4.0

Excel 95/5.0 (5.0)

ODBC Source

More options are available under Connection Properties

OLEDB Source

More options are available under Connection Properties

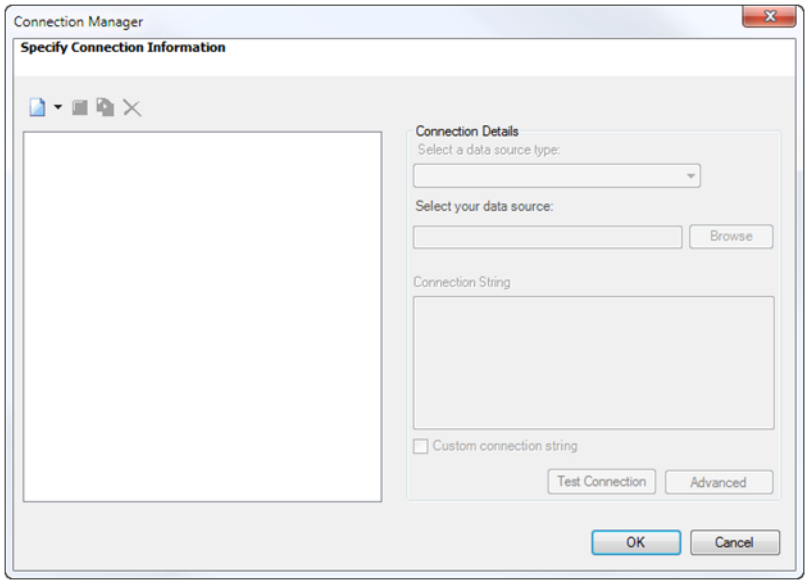
SQL Connection

More options are available under Connection Properties

For connection to Citect, go to Citect Connection. To create a new connection,

Within SCADAConnect, go to Tools > Connection Manager.

The Connection Manager opens.



Click the New button to create a new connection. You can select databse or Citect connection. Go to Database Connection for creating a connection using database option. Or, go to Citect Connection for creating a connection using Citect option (see Citect Connection help).

Rename the newly created Database Connection to a suitable name such as Access DB Connection. It's always a good idea to name the Database Connection similar to the data source as this Database Connection will be used at other location such as Data Source Manager.

Select a data source type from Database: Connection Details group.

For Example: Select Access 2003/2002/2000 (4.0) from the drop down.

Click on the Browse button and provide your Access file or Excel file. Connection String box will automatically be filled in with the appropriate connection string.

Connection String is a special string which contains the information that the provider needs to know to be able to establish a connection to the database or the data file. It is a passed in code to an underlying driver or provider in order to initiate the connection.

The Browse button will show a Connection Properties window when selected data source type is, ODBC Source or, OLEDB Source or, SQL Connection.

Click on Test Connection button. This button normally validates the path, file format, and existence of the file. You should see a "Connection Succeeded" message box.

The Advanced button allows changing the prefixes and suffixes for the Name and Date/Time field. Under advanced circumstances only, these delimiters need changes; otherwise default should work.

If you receive any message other than "Connection Succeeded" like "Could not open connection" then make sure you made the right selection or provided the right options. If you are still not getting the "Connection Succeeded" message, check the log file. Go to Enable Advanced Logging for more details.

Connection Properties

Connection Properties will only be available when the selected data source type of Connection Manager is, ODBC Source, OLEDB Source, or SQL Connection. Connection Properties facilitates some advanced features related to establishing communication with database file/server. There are four generic connection properties, which run as:

Connection Properties for Microsoft ODBC Data Source

Connection Properties for Microsoft Access Database File

Connection Properties for OLEDB Source type

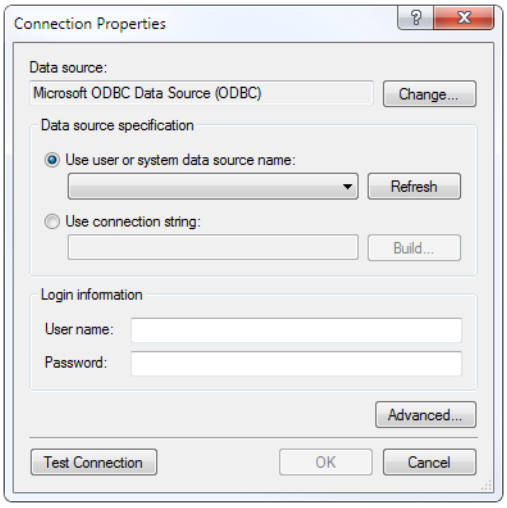
Connection Properties for SQL Source type

For creating connections to Citect, go to Citect Connection.

There is Advanced option available under each aforementioned categories.

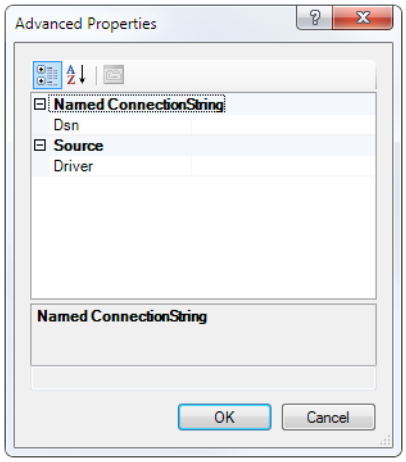
Connection Properties for Microsoft ODBC Data Source

For the ODBC data source, you can either select the **Use user or system data source name** or **Use connection string**. When the first option is selected, you can chose from the provided item for example, MS Access Database from the dropdown or you can create your own connection string by selecting the second option, **Use connection string**.



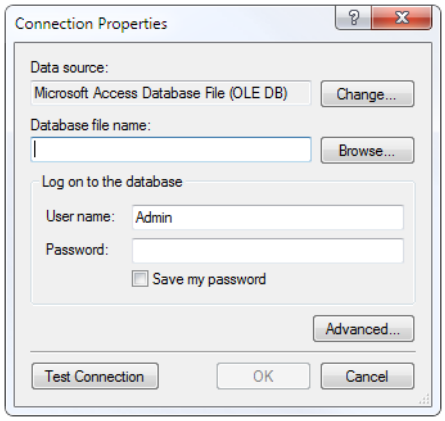
Advanced Connection Properties for ODBC

Dsn and the Driver value can be manually typed into the provided grid.



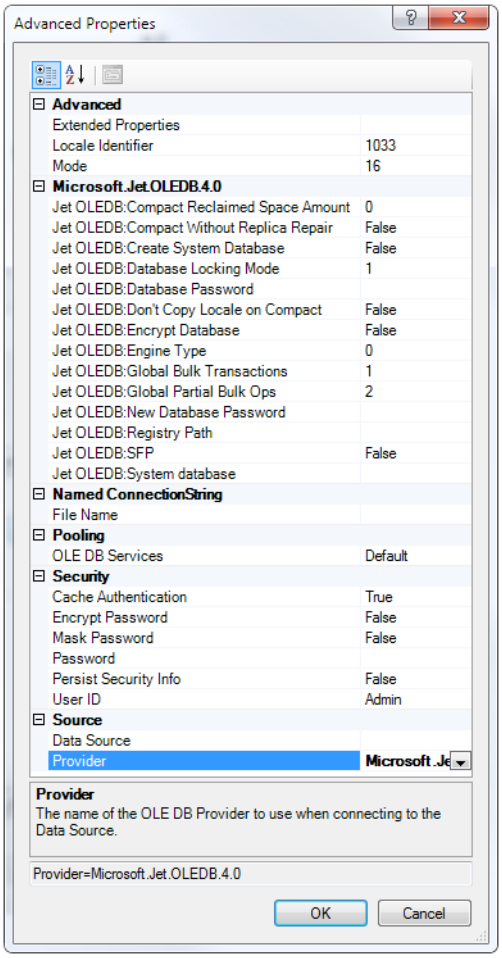
Connection Properties for Microsoft Access Database File

Click on the Browse button to select the Access file and then provide the User Name and Password if necessary. Advanced configuration settings are available under Advanced button.



Advanced Connection Properties for Access

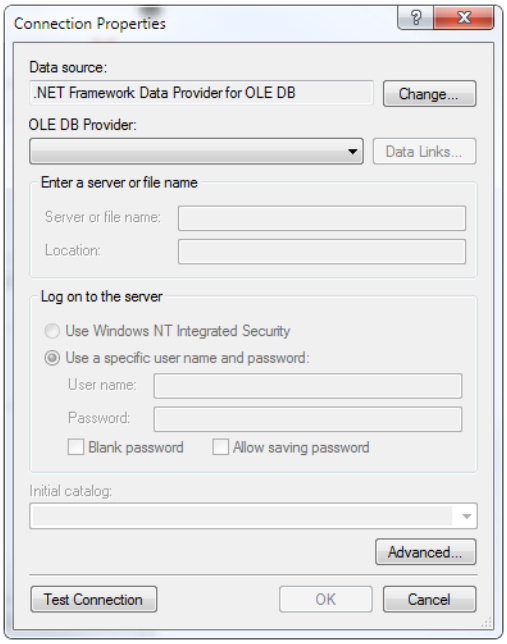
Select the appropriate option and/or provide the necessary text or value.



Connection Properties for OLEDB Source type

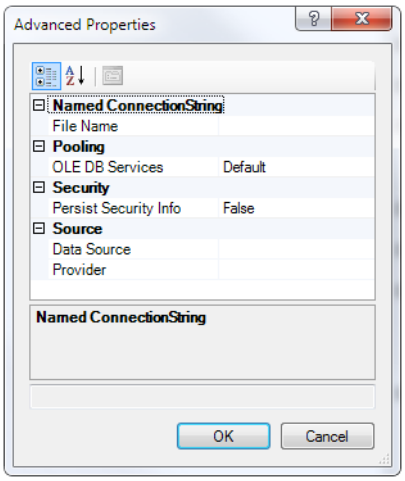
Based on the data source, select the **OLEDB Provider**. For example, **Microsoft Jet 4.0 OLE DB Provider**. The **Data Links** button will provide further settings options. Depending on the OLE DB Provider selected, you may or may not have to provide

some/all information. When **Microsoft Jet 4.0 OLE DB Provider** is selected, only **Server or file name** and **User name/Password** are required.



Advanced Connection Properties for OLEDB

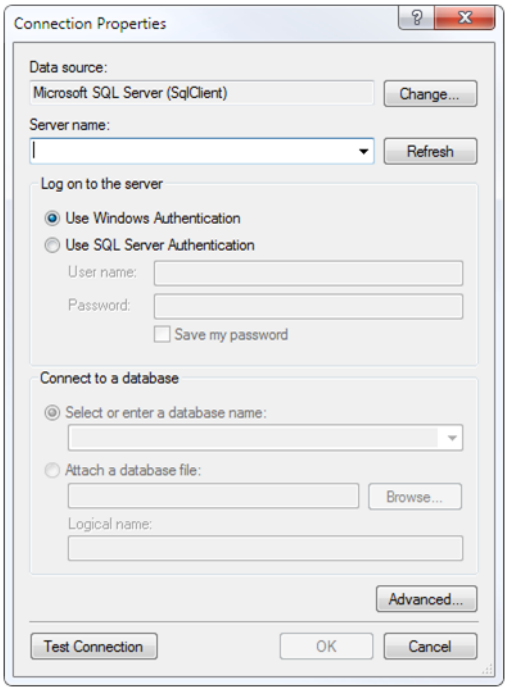
Advanced Properties are dependent on the **OLE DB** provider selection. The following screen capture is without selecting any OLE DB provider.



Connection Properties for SQL Source type

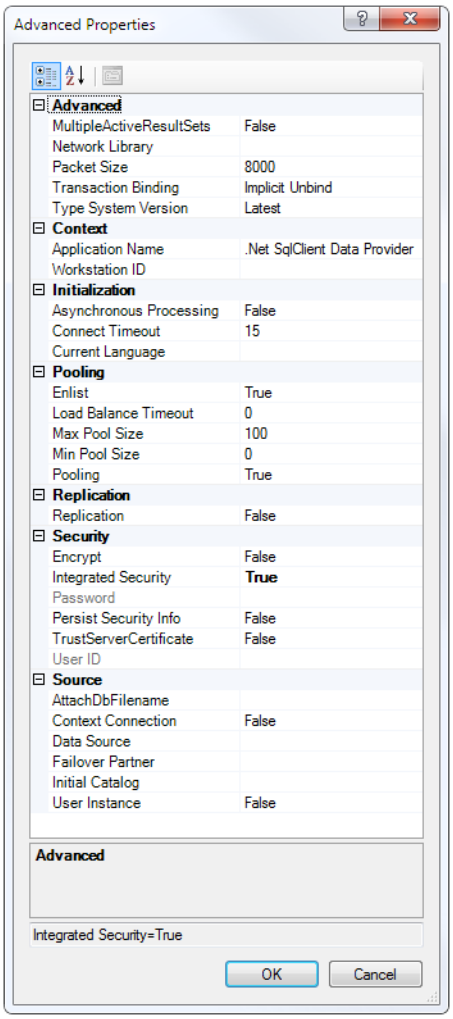
Server name on SQL Properties is dependent on your computer. The drop down will be populated if SQL server is installed on your computer. The example of server name

would be MYCOMPUTERNAME\SQLEXPRESS. Depending on how the SQL server is setup, you need to select appropriate options on the screen below.



Advanced Connection Properties for SQL

Select or provide the appropriate values on the available field.



Citect Connection

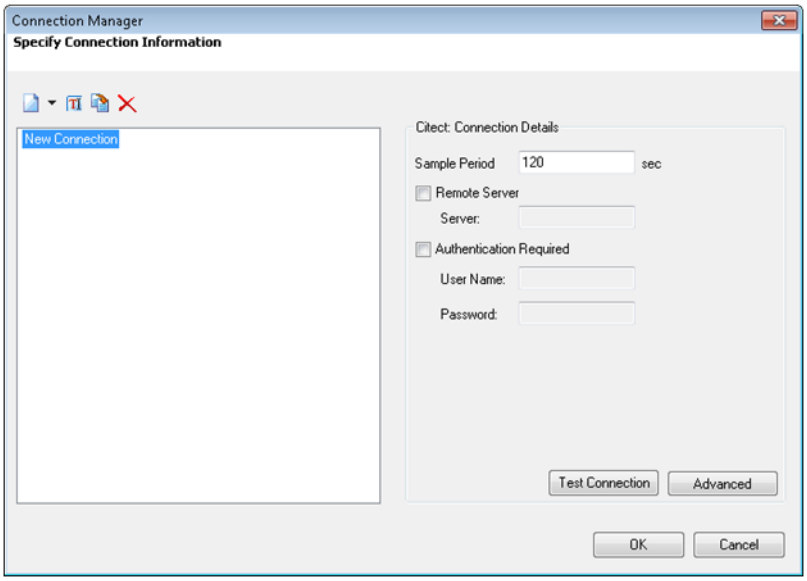
SCADAConnect can establish a connection with Citect SCADA and communicate to fetch data from its data source. In order to setup a Citect connection, the connection manager of SCADAConnect needs to be configured.

Within SCADAConnect, go to **Tools** > **Connection Manager**.

The Connection Manager opens.

Click the **New** button and select **Citect**.

Connection Manager screen will change as below.



Sample Period: Provide the data sampling interval. This number will be the frequency (or polling interval) used by SCADAConnect to pull the data from the Citect data source.

Remote Server: Provide the server name where the Citect is installed. If the Citect is running on the same computer where the SCADAConnect is, you can uncheck the Remote Server (no server name) or leave the server value blank.

Authentication Required: If your Citect requires User Name and Password (typically they do), enter the User Name and Password. Click on Test Connection button to verify the connection.

Data Source Manager

The Data Source Manager helps to specify either the SCADA data table and table related attributes that are going to be used to fetch the data to the SCADAConnect or the computer name and server which hosts the OPC server software. The OPC server can reside on the computer running the model or any other reachable computer. The OPC server must be set up beforehand. Depending on the SCADA connection created on Connection Manager, following type of database sources can be selected. So, it is necessary to create a SCADAConnection on Connection Manager first.

Database Source

Citect Data Source

Database Source

The **Database Source** option will allow defining the Table Name and related fields for **SCADA connection** created on Connection Manager. If there are more than one table where the SCADA information are stored then multiple database sources need to be created.

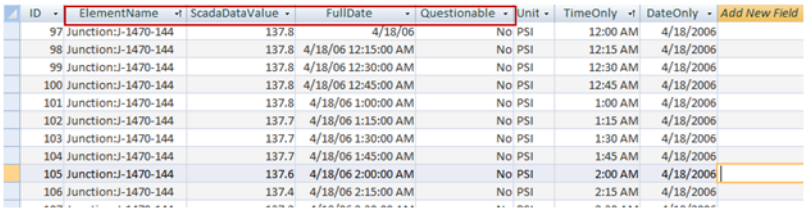
For example: If SCADA stores instantaneous data on a table called "LiveData" and the historical data are stored or backed up on a table called "HistoricalData" then two database source connections are required to communicate with each table.

It's always a good idea to label the database source connection with the Table Name as this database source connection will be referred at other locations.

There isn't any restriction on the order of any column or any limitation on number of rows in any data table. There are two formats for database files:

Each row contains one value and a column should store one type of information, such as Date should only store date vales. Each row must contain the Data/Time, tag (label) and value for the signal. It may also contain information as to whether the values are questionable. The following screen is just an example where the highlighted columns can be mapped to SCADAConnect. For more details about signal mapping go to Signal Mapping.

2. Each row can contain multiple values. Each row must contain a data/time identified and multiple signal values associated with that time. The tags associated with each column must be placed in the first row of the table.

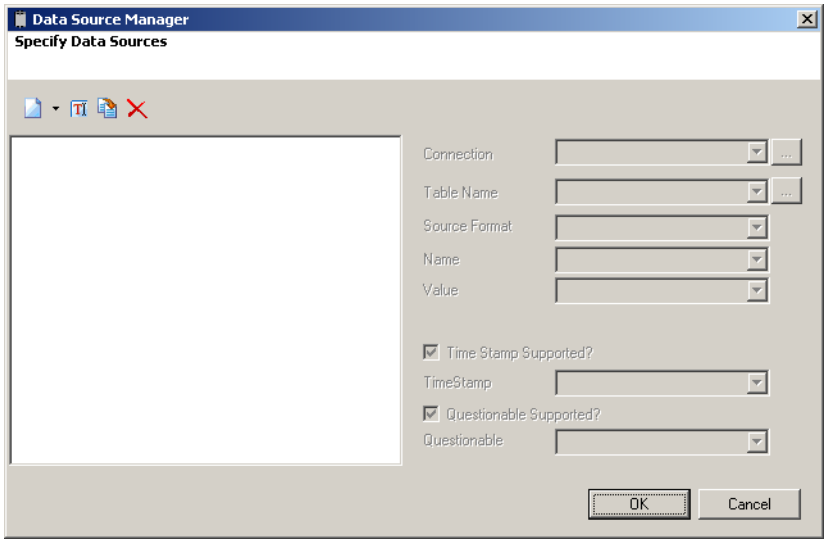


ID	ElementName	ScadaDataValue	FullDate	Questionable	Unit	TimeOnly	DateOnly	Add New Field
97	Junction:J-1470-144	137.8	4/18/06	No	PSI	12:00 AM	4/18/2006	
98	Junction:J-1470-144	137.8	4/18/06 12:15:00 AM	No	PSI	12:15 AM	4/18/2006	
99	Junction:J-1470-144	137.8	4/18/06 12:30:00 AM	No	PSI	12:30 AM	4/18/2006	
100	Junction:J-1470-144	137.8	4/18/06 12:45:00 AM	No	PSI	12:45 AM	4/18/2006	
101	Junction:J-1470-144	137.8	4/18/06 1:00:00 AM	No	PSI	1:00 AM	4/18/2006	
102	Junction:J-1470-144	137.7	4/18/06 1:15:00 AM	No	PSI	1:15 AM	4/18/2006	
103	Junction:J-1470-144	137.7	4/18/06 1:30:00 AM	No	PSI	1:30 AM	4/18/2006	
104	Junction:J-1470-144	137.7	4/18/06 1:45:00 AM	No	PSI	1:45 AM	4/18/2006	
105	Junction:J-1470-144	137.6	4/18/06 2:00:00 AM	No	PSI	2:00 AM	4/18/2006	
106	Junction:J-1470-144	137.4	4/18/06 2:15:00 AM	No	PSI	2:15 AM	4/18/2006	

To create a new database source,

Within the SCADAConnect go to **Tools** > **Data Source Manager**.

The Data Source Manager opens.



Click on the **New** button and select **New Database Source**.

If the source is a dtabase, from the Connection drop down, select your connection that has been created on Connection Manager. If the drop down is not listing any items, go to the Connection Manager and create a **Database Connection** or **Citect Connection** as needed.

The Connection Ellipsis button (..) will open the Connection Manager directly.

If the source is an OPC server, the user must identify the computer on which the OPC server is located. This may be the computer running the model in which case, the user would select "Local Machine". If it is a networked computer, it must contain the network location in the form //computer name. Because a computer may have several OPC servers on it, once the computer is selected, the user must indicate which OPC server is to be used.

From the **Table Name** drop down, select the table which contains the SCADA data. In other words, you need to select the table where the SCADA is storing the data. It could be historical data table or a real-time data table.

If **Table Name** drop down is not listing any items then make sure you have right connection is selected in connection drop down.

The Table Name Ellipsis Button will allow advanced user to provide custom queries. This is often used when the table provided in Table Name needs filtering or additional manipulation. Custom queries can be applied to any database sources. For more details visit custom queries.

Example of a custom Table Name query:

SELECT [ElementName], [ScadaDataValue], [FullDate] FROM ScadaDataTable;

Source format. The user must select from the two formats:

- One value per row

- Multiple values per row.

If the user selects One value per row the following steps apply:

From the **Name** drop down, select the field (or column) where the SCADA signal name that corresponds to hydraulic element are stored.

For example: The column which stores the "LakewoodTank" or "MainStreetBooster". These "LakewoodTank" or "MainStreetBooster" are like a tag name used by SCADA to store the data. These "Lakewood Tank", "MainStreetBooster" tags will later be mapped to the Lakewood Tank and Main Street Pump element of the hydraulic model. The name of the SCADA tag does not need to be the same as the name of the hydraulic model element. For example, Lakewood Tank may be mapped to T-7.

If the **Name** drop down is not listing any items then make sure you have right **Table Name** and/or right connection is selected in connection drop down.

From the **Value** drop down, select the field (or column) where the data read by SCADA are stored. These values (or numbers) will be imported to the hydraulic model. These valves could be imported to different location in hydraulic model, such as Initial Settings, Darwin Calibrator etc.

If the **Value** drop down is not listing any items then make sure you have right Table Name and/or right connection is selected in connection drop down.

Check the **Time Stamp Supported**. If you SCADA data contains a field that stores date then check box. Without checking this box, the Time Stamp drop down will not be editable and you will not be able to provide the date or time field and if Time Stamp is not provided, later, when importing the data from SCADA to the hydraulic model, SCADAConnect will not be able to filter your SCADA data based on any 'From Date Time' and 'To Date Time' In other words, you must provide this to take full advantage of SCADAConnect.

From the **Time Stamp** drop down, select the field (or column) where the time is stored. This Time Stamp field in SCADA data must have the full time such as 4/28/80 12:15:00 AM. If time and date are stored in separate field (or column), use custom queries or edit the SCADA time and date field.

If SCADA stores any attribute and flags the validity of the recorded data then it's a good idea to check the **Questionable Supported** box. If you SCADA does not flag a read yet you want to create some logic and filter those reads then that is doable in

SCADAConnect. For example, any flow value below 30 units can be disregarded. For more details see Custom Queries.

From the **Questionable** drop down, select the field (or column) where the data are stored. This field must be a Boolean type . If the Questionable data equals **True**, we can ignore such data when viewing or importing any data to the hydraulic model. In other words you need to select a field (or column) where SCADAConnect stores the flagged information.

If your SCADA does not flag a read yet you want to create some logic and filter those reads then that is doable in SCADAConnect. For example, any flow value below 30 units can be disregarded. For more details go to custom query.

Click the **OK** button in Data Source Manager.

If the user selects the Multiple values per row format, the following steps apply. (If the user selects Multiple values per row, it is not possible to use the Questionable field.)

Citect Data Source

The Citect Data Source option only works with Citect Connection.

For the Citect data source the user only has to choose, if the Citect source should be used for real-time or historical scada data access.

OPC Real-time source

First the Computer name combo box shows the accessible computer names. If the OPC server is installed on the current machine, the host name will be shown as "Local machine" (Default for a new OPC source).

To choose the associated OPC-server a list of accessible OPC Data Access server available on the selected computer is shown in the combo box.

OPC Historical source

First the Computer name combo box shows the accessible computer names. If the OPC server is installed on the current machine, the host name will be shown as "Local machine" (Default for a new OPC source).

To choose the associated OPC-server a list of accessible OPC Historical Data Access server available on the selected computer is shown in the combo box.

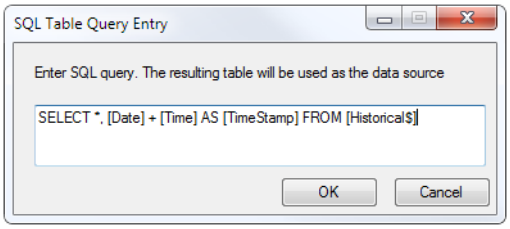
Citect Data Source

The Citect Data Source option only works with Citect Connection.Remaining options for any Citect or database source connection remains the same. So, for details about the rest of the fields, go to Database Sources.

Custom Queries

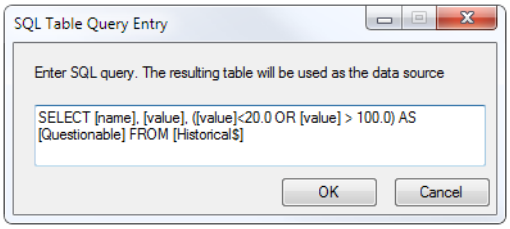
Use Custom Queries to create a customized, intermediate data table that SCADAConnect can read. The query can add new fields based on available field values in the data source, allowing data to be translated from a specific user format to the SCADAConnect format. It can also be used to add validation of the SCADA data.

For example, if the signal data supports a timestamp field, SCADAConnect expects the data to be presented in a single Date/Time field. However, if the timestamp in the data source is stored in two separate fields, a custom query can be written to present the two fields to SCADAConnect as a single DateTime field.



This will generate an intermediate data table with all the fields from the table plus a new calculated field called TimeStamp that contains the Date/Time values. This TimeStamp field is the field name that should be entered in the Time Stamp of Data Source Manager dialog.

Another example would be to use a query that will add extra data validation to remove errors. If signal values are known to always be within a certain range, the following query could be written to mark those signals as **Questionable** and then allow SCADAConnect to skip those values.



This will generate a field called **Questionable** that can be used in the Questionable of Data Source Manager dialog. When the data is read by SCADAConnect, data records with values outside this range will have the **Questionable** field set to **TRUE**, and SCADAConnect will discard the value.

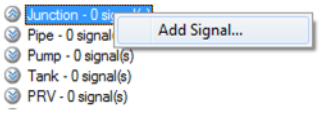
When custom queries are entered, they should have valid SQL syntax for the data source being used. Custom queries are sent to the database provider and therefore the Advanced Options of the Connection Manager do not apply to these queries.

Signal Mapping

Signal mapping is a process of particularizing assigning a SCADA signal (or SCADA tag or name) to a hydraulic element. Options like selecting a Data source, specifying the target of imported data such as Calibration field datasets, Element Initial Settings, Element User Data Extensions, etc. are available in Signal Mapping Editor.

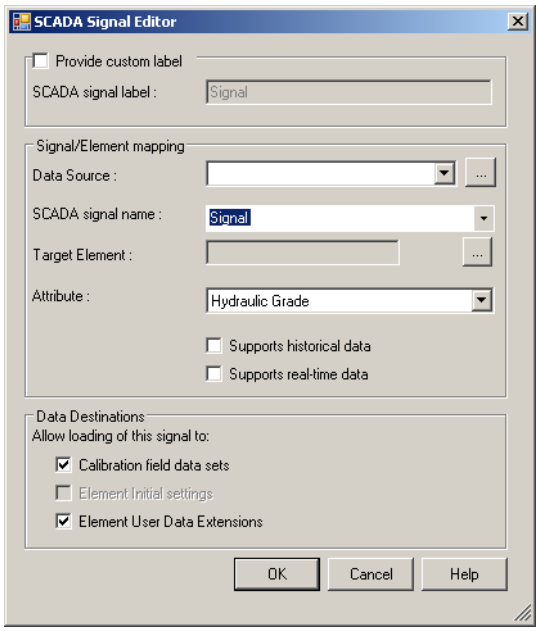
To start mapping a signal or creating a new mapping signal:

Within SCADAConnect, right click on any element type and select **Add Signal**. For example, right click on Junction - 0 signal(s) and select Add Signal.



If a signal already exists then right clicking on that signal will provide options to **Delete Signal** or **Edit Signal**. Clicking on **Edit Signal** will launch the **SCADA Signal Editor**.

SCADA Signal Editor



The SCADA Signal Editor is comprised of three basic options group.

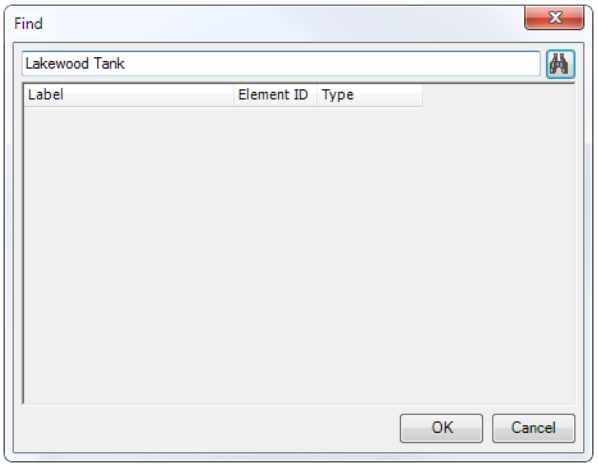
Provide custom label: In Provide custom label, you can specify a user supplied custom name to mapped signal. This option is particularly helpful when the signal name is cryptic and long, which make navigation less comfortable.

Check this Provide custom label box to provide a custom label which will be displayed in SCADAConnect main window. If this box is not checked then signal name in SCADAConnect window will automatically be generated based on the options selected under Signal/Element mapping group.

Signal/Element mapping: Main mapping of SCADA signal to a hydraulic model element goes in this group. Signal mapping is a process of saying, for example, my ABC attribute in the Name field of my SCADA data corresponds to XYZ of a hydraulic model. First select the data source from which the signal should be read. From the **SCADA signal name** drop down, select the signal (or SCADA tag name) that you intend to map with a hydraulic element, for example, "LakewoodTank Level" or "MainStreetBooster flow rate ". If the SCADA data contains two or more type of attributes corresponding to the same hydraulic element, two or more number of signals need to be mapped in SCADAConnect. For example: SCADA stores information about flow as well as pressure out of the "MainStreetBooster" then corresponding to each attribute, one for the flow and one for the pressure, needs to be mapped.

Click on the **Target Element Ellipsis** button. The **Find** window will open and type in the element ID or label that you wish to map to, for example, Lakewood Tank or Main Street Pump or 420. Click on Find [ICON] button. This will search and list all the available element(s) that matched the search criteria.

Select the interested element and click **OK** on the Find window.



In **Attribute** drop down, select the appropriate item.

For Example: If SCADA data is storing a pressure read, you need to select **Pressure** from the drop down. The items available under **Attribute** will change based on type of element selected under **Target Element**.

To specify, that the signal mapping can be used to read historical data the "Support historical data" needs to be checked. This option is only selectable, if the selected data source supports historical scada data.

With the Support real-time data check box the user can allow the usage of the signal mapping for reading real-time data. This option is only selectable if the selected data source supports real-time scada data.

Data Destinations: Specify where you want SCADAConnect to import your data.

Check **Calibration field data sets**, if you want the data imported to Darwin Calibrator

Check **Element Initial settings**, if you want the data to Initial Setting Alternative

Check **Element User Data Extensions** , if you want to import the SCADA data to a custom defined field. This will facilitates to create color-coding and annotations from Element Symbology .

To import the SCADA data to the **Time Series Field Data**, go to **Load to Extended Data** and to calculate the demand value based on the **SCADA data**, go to **Demand Inversing**.

Load SCADA Data

Loading SCADA data is a process of importing the SCADA data to a specific location in a hydraulic model. Specifying the database source of the SCADA data and where in the hydraulic model the importer should import the data are managed during signal mapping process.

If **Calibration field data sets** is checked under the **Data Destinations** group of **SCADA Signal Mapping** window, you will be able to import the data to the Darwin Calibrator. Specifying the correct SCADA Signal Data Sources are equally important as well, otherwise data may get imported from an unintended database source.

Before loading any data to the hydraulic model, make sure the units are correctly specified. For more information, go to XREF units.

There are four different ways to utilize/visualize the imported SCADA data.

Load to Calibrator Field Dataset

Load to Initial Settings

Load to Extended Data

Load Demand Base Average Values

When Real-time option is selected, SCADAConnect will import the latest data stored by the SCADA system. If a SCADA System is storing data at every 15 minute interval, say at 1:00 AM, 1:15 AM, 1:30 AM and so on, and SCADAConnect is used to load the real-time data at 1:35AM, it will import the data stored at 1:30AM.

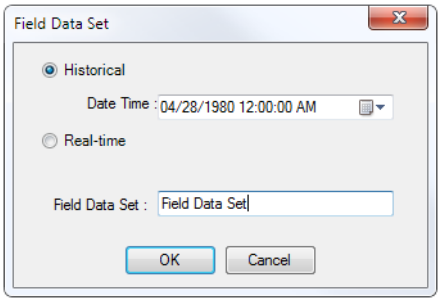
Load to Calibrator Field Dataset

One of the major benefits of using SCADAConnect is to calibrate the hydraulic model using the data read by SCADA system. Historical or real-time data can be imported to the Darwin Calibrator Field Dataset directly.

To import the SCADA data to Darwin Calibrator Field Dataset:

Within in SCADAConnect window, click on **Tools** > **Load to Calibrator Field Dataset**.

The Field Data Set dialog box opens.



Select either **Historical** or **Real-time** options.

When the **Historical** option is selected, the database source specified in the SCADA Signal Data Sources group of SCADA Signal Editor will be used (same is true for **Real-time** option).

Specify a desired name in the **Field Data Set** box which you established in Darwin Calibrator.

Click **OK**.

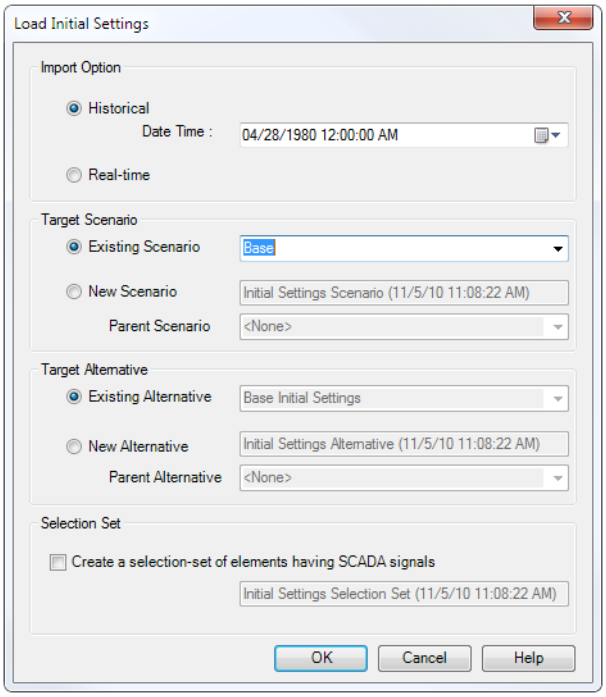
Load to Initial Settings

Loading the SCADA data to initial setting will update the initial setting/value field of hydraulic model. For example, if the real-time tank level read by SCADA is 28ft and executing this (Load to Initial Settings) will load 28ft of value into the tank initial level of the hydraulic model in the specified scenario and alternative.

To import the SCADA data to Initial Settings fields:

Within the SCADAConnect window, click on **Tools** > **Load to Initial Settings**.

The Load Initial Settings dialog opens.



Options under Load Initial Settings are categorized under following groups:

Import Option: Specify whether data from **Historical** database source or **Real-time** database source should be used while fetching the SCADA data.

Target Scenario: When an existing scenario is provided in the **Existing Scenario** drop down, initial settings data of the hydraulic model will be overwritten by the fetched SCADA data.

A new scenario can also be created by selecting **New Scenario** option. Specify the name of the newly created scenario and the Parent Scenario of this new scenario. If **<None>** is selected, a new base scenario will be created.

Target Alternative: Similar to Target Scenario, specify whether the imported SCADA data should overwrite the existing data or create a new alternative. To overwrite the existing hydraulic data, select **Existing Alternative** and select the desired alternative from the drop down. Accordingly, to create a new alternative, select **New Alternative**, provide a name and then select the Parent Alternative. If **<None>** is selected, a new base alternative will be created.

Selection Set: A selection set, containing elements whose data were imported will be created when Create a selection-set of elements having SCADA signals is checked. This method is particularly helpful to double check the updated elements attribute.

Load to Extended Data

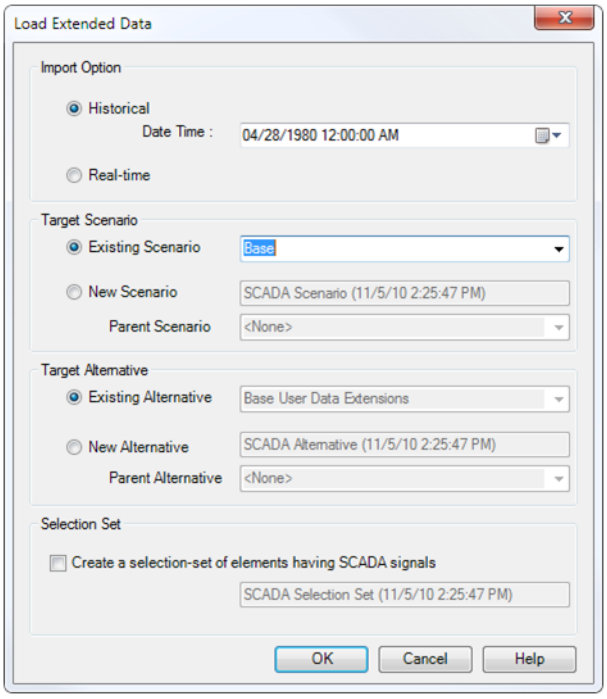
Creating color-coding and annotation is also possible for the imported SCADA data. To utilize this feature, SCADA Data needs to be loaded to a custom data field, normally called as User Data Extensions (UDX). These UDX will automatically be created by SCADAConnect once the steps described below are completed.

Once the data is in a field of a hydraulic model, you can access the data of that field in Flex Table, Graph, Color-coding, Annotations, etc.

To Import the SCADA data to a custom field or UDX:

Within in SCADAConnect window, click on **Tools** > **Load to Extended Data**.

The Load Extended Data dialog opens:



There are four major options group available under Load Extended Data.

Import Option: Specify whether data from **Historical** database source or **Real-time** database source should be used while fetching the SCADA data.

Target Scenario: When an existing scenario is provided in the **Existing Scenario** drop down, initial settings data of the hydraulic model will be overwritten by the fetched SCADA data.

A new scenario can also be created by selecting **New Scenario** option. Specify the name of the newly created scenario and the Parent Scenario of this new scenario. If **<None>** is selected, a new base scenario will be created.

Target Alternative: Similar to Target Scenario, specify whether the imported SCADA data should overwrite the existing data or create a new alternative. To overwrite the existing hydraulic data, select **Existing Alternative** and select the desired alternative from the drop down. Accordingly, to create a new alternative, select **New Alternative**, provide a name and then select the Parent Alternative. If **<None>** is selected, a new base alternative will be created.

Selection Set: A selection set, containing elements whose data were imported will be created when Create a selection-set of elements having SCADA signals is checked. This method is particularly helpful to double check the updated elements attribute.

Once the information are provided on above four options, Click **OK** on Load Extended Data dialog box to complete the import process.

Load Demand Base Average Values

Viewing SCADA Data

Before loading SCADA data to the hydraulic model, the SCADA data can be reviewed from View SCADA Data dialog box. This option not only builds the confidence in the process of fetching the data but also helps to find any problematic data because of viewing graphs capabilities. Other than viewing the SCADA data in tabular view, the following methods can be performed.

Graph

Graph with Questionable Values

Create Time Series

Create Time Series with Questionable Values

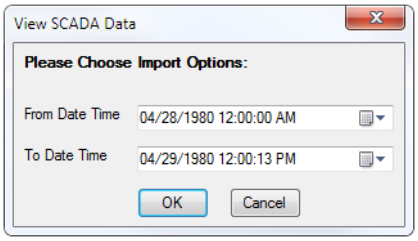
To view the SCADA data, signal mapping, providing database source and providing the connection gateway are necessary. During the entire process of viewing SCADA data, data are temporarily loaded to the SCADAConnect directly from the SCADA data source, none of the data displayed here are loaded or imported to the hydraulic model element.

Only SCADA signals that are associated with elements that are active in the current scenario are updated.

To view SCADA Data:

Within SCADAConnect, go to **Tools** > **View SCADA Data**.

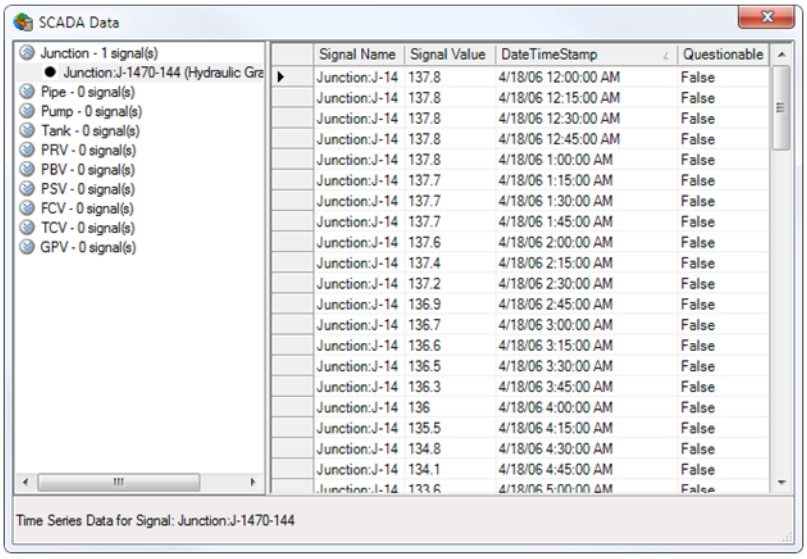
The View SCADA Data dialog opens, where from and to date time can be provided. The value of From Time Date inserted here is used as the starting time (time from start = 0) when the SCADA Data are listed or plotted regardless of the Time Date of the actual SCADA data. It is recommended to set the Time Date in this field to the same value as the SCADA data.



Populate the appropriate **Date Time**.

Click **OK**.

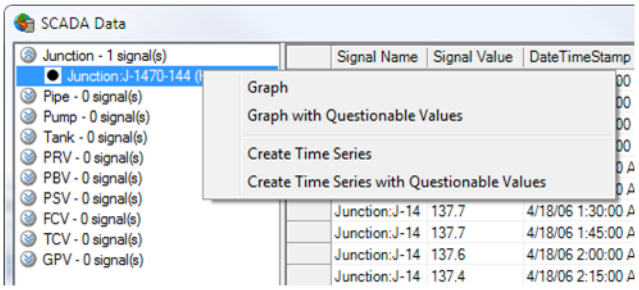
If you do not see SCADA Data dialog box then make sure you provided correct signal mapping, database source and the connection. If options are properly selected/ provided, the SCADA Data dialog opens.



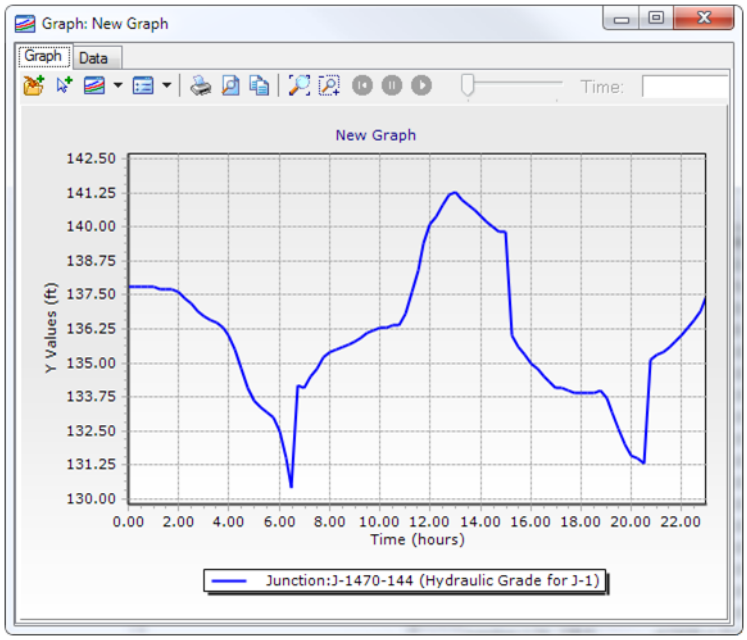
Graph

To view a graph of any mapped signal follow the steps described below. However, to learn more about graphing, see Graphs.

Right click on the desired signal and select **Graph**.



A New Graph dialog box opens.



Since the graph is displayed on Standard User Interface of the hydraulic model, native command of graph window can be utilized.

For example, if a pressure SCADA data are viewed in graph, then from Add to Graph button of the Graph you can add any desired element from a WaterCAD V8*i* EPS run and compare the data directly.

If SCADA data contains Questionable value = TRUE, then such data will be discarded and not be displayed in the Graph.

Graph with Questionable Values: When this option is selected, the SCADAConnect will not filter any data on Questionable field (or column) and will display all the attributes in the graph.

Create Time Series: To create Time Series Field Data using the SCADA Data, right click on any mapped signal in the SCADA Data window and select Create Time Series.

A message box confirming the creation of the Time Series Field Data will be displayed. To check the newly exported data go to Components > Time Series Field Data.

Create Time Series with Questionable Values: When this option is selected, the SCADAConnect will not filter any data on Questionable field (or column) and will export all the attributes in the Time Series Field Data.

Demand Inversing

Demand inversing is a method to adjust the assigned pressure junction demands in the water model to accurately match the real world demands. To calculate the real demands, Demand inversing requires the user to identify the boundaries of each zone, the inflow and outflow points, the tanks signals, and the SCADA tag associated with each value needed.

With this information, and SCADA data for a full day, the real world total daily demands of each zone can be calculated. The application can then find a multiplication factor that can be applied to each pressure junction's demand field in the model to make the simulated zone demand equal to the real world demand for that day.

To run this tool, you must have one or more Zones defined in your model.

Demand Inversing Workflow

To use the Demand Inversing tool, the usual steps consist of:

Open Demand Inversing tool. (Within SCADAConnect, **Tools** > **Demand Inversing**).

Update all the tabs (Calculations, Reference Consumption, Flow Signals and Tank Signals) as necessary. See Demand Inversing Dialog Box. Make sure that the zones for which demands are being calculated have been selected in the "Reference Consumption" tab.

Go the **Compute** menu and execute **Estimate Zone Flow** or **Demand Multiplier**. The Estimate Zone Flow command will compute the model over a 24-hour period and calculate the Estimated Daily Zone Demand Volume value in the "Reference Consumption" tab. This can be useful if the daily zone demand of the physical system is known and you want to manually compute a user-defined demand multiplier that can be applied to the demands. More often the Demand Multiplier command is used, which computes the Estimated Daily Zone Demand Volume value as described above, but also reads the SCADA data from a given day and calculates the Daily Zone Demand Volume from SCADA data value. It then does a simple calculation between

the estimated daily flow and the measured SCADA daily flow to come up with a Demand Multiplier.

4. To have SCADAConnect create or update a demand alternative, go to **Update** and click on **Write Demands**. This will update the demand alternative by applying the Demand Multiplier to every input demand value in the selected zone of the model, make the simulation's demand usage match the data gotten from the physcial system. If no errors are encountered in the process SCADAConnect will and issue a message "Demand Alternative <name> Successfully Created." If the demand alternative does not show up in the alternative manager, click to Expand All in the alternative docking manager to force a refresh of the tree and see it.

Demand Inversing Dialog Box

The Demand Inversing dialog consists of the following:

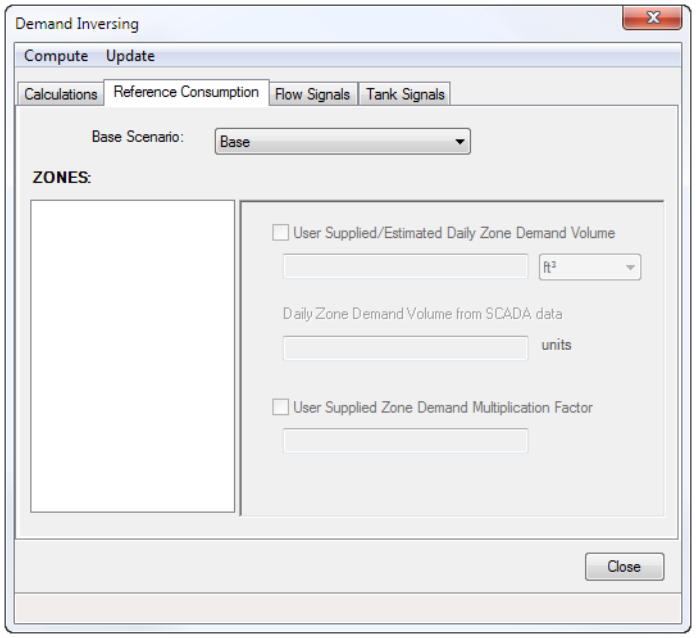
Tabs:

Calculations: In calculation tab, you can provide the destination alternative of the calculated demand. Demand can either be overwritten or a new demand alternative can be created.

To overwrite an existing demand alternative, select the desired alternative from the drop down. And, to create a new alternative, select the Create New Alternative and provide a name. If you want this alternative to be a child of an existing alternative, select the alternative from Parent Alternative.

Reference Consumption: In this section, information about the demand (or consumption), the zone and the scenario needs to be populated and therefore the

hydraulic model must have zone(s) specified otherwise Demand inversing tool will not run. The Reference Consumption tab looks like this:



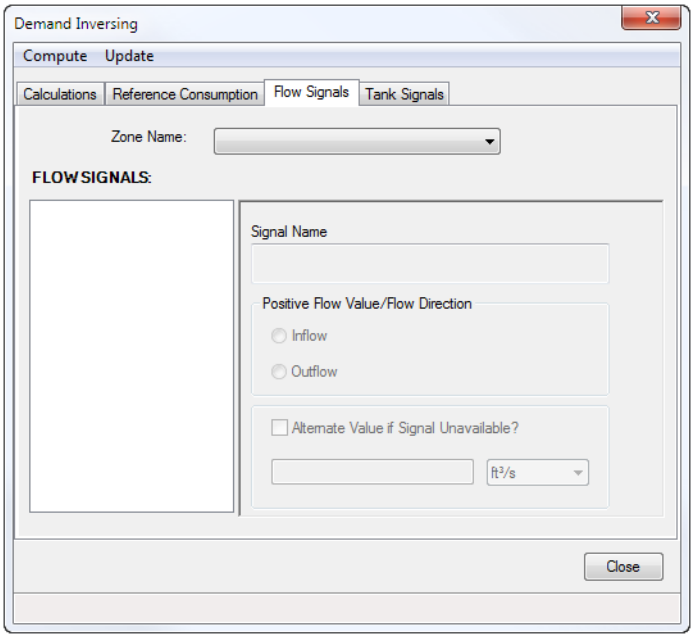
Base Scenario: The hydraulic model scenario to obtain the demand.

User Supplied/Estimated Daily Zone Demand Volume: Check this to manually supply a daily water demand volume for the selected zone.

Daily Zone Demand Volume from SCADA data: The real world zone demand volume calculated from the SCADA data, based on inflow/outflow to/from the zone, and tank level changes in the zone, is populated here when Estimate Zone Flow is executed from Compute menu.

User Supplied Demand Multiplication Factor: Check this to supply a custom multiplication factor to be applied to the junction demands of the model. The default behavior is to use a multiplication factor that, when applied to the estimated zone volume, will equal the average zone volume from the SCADA data. This will be populated when Demand Multiplier is executed from Compute menu. This is not required and has a default value of 1.

Flow Signals: In this flow signals tab, you will define whether a flow is coming in or going out from the selected zone and selected signal. The screen of flow signals looks like this:



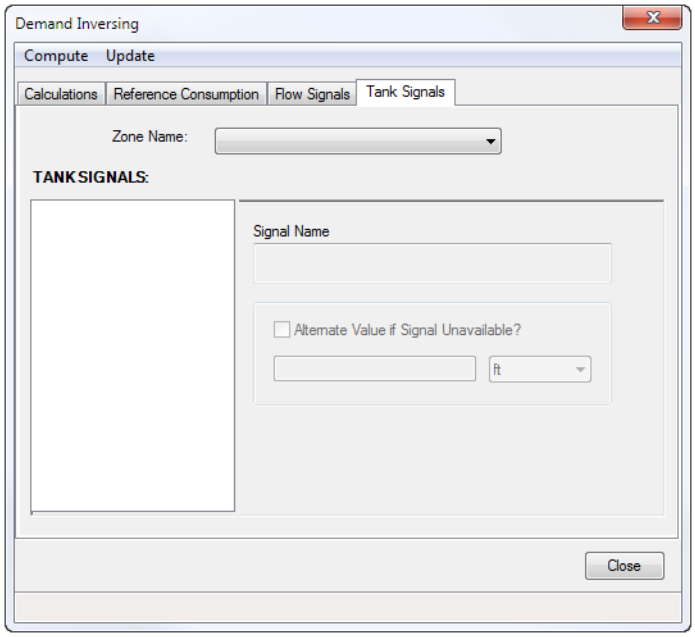
Zone Name: Select and configure each zone that was selected to calculate in Reference Consumption tab.

Flow Signals: Check each flow signal that represents an inflow/outflow to the selected zone.

Flow Direction: Select whether the positive SCADA flow value of this pipe equals flow into the selected zone or flow out of the selected zone.

Alternate Value if Signal Unavailable: Supply an average flow value through this signal that can be used when this SCADA data is not available.

Tank Signals: If there are any tanks signals mapped within the specified zone then those tanks need to be provided to Demand inversing tool. The tank signals tab looks like this:



Zone Name: Select and configure each zone that was selected to calculate in Reference Consumption tab.

Tank Signals: Check each tank signal that represents a tank in the selected zone.

Alternate Value if Signal Unavailable: Supply an average value related to the selected tank that can be used when this SCADA data is not available.

Menu Items:

Compute: Runs the calculation.

Estimate Zone Flow: This tool estimates the daily zone demand volume each zone selected, by calculating the selected Base Scenario then fills in the **Daily Zone Demand Volume** for each selected zone in the **Reference Consumption** tab.

Demand Multiplier: This menu item estimates the demand multipliers as well as the daily zone demand volume in the **Reference Consumption** tab.

Update: Updates the results.

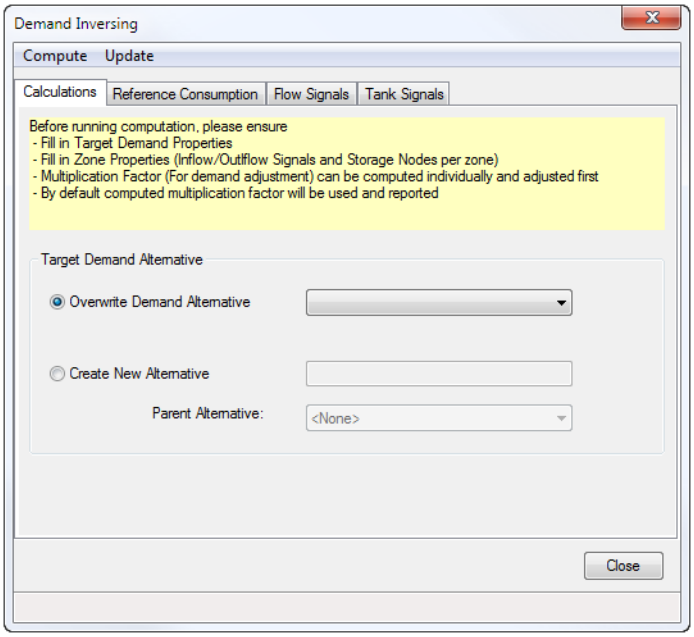
Write Demands: Write demand will take the multiplier value from the **Reference Consumption** tab and multiply the hydraulic model's pressure junction demand

values by this factor. This newly calculated demand value will go to the alternative as described in **Calculations** tab.

To launch the Demand Inversing tool:

Within SCADAConnect, go to **Tools** > **Demand Inversing.**

The Demand Inversing window opens.



Options

SCADAConnect includes customization Options, divided into the following tabs:

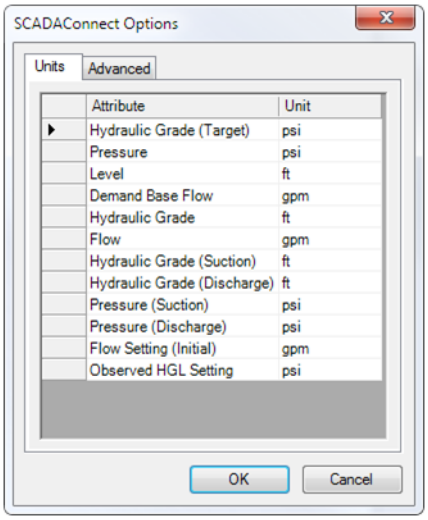
Units from SCADAConnect

Advanced

Units from SCADAConnect

Specify the units where each of the attribute types are stored within the SCADA system. In other words, if a SCADA data are stored in a unit called "A", then select the unit "A" from the available units.

For example, if flow is stored in L/s in SCADA system, then select L/s unit for Flow. These can be different from the units for that attribute in the hydraulic model.



Units must be set to the units of the SCADA data. Units that are set in the hydraulic model do not matter.

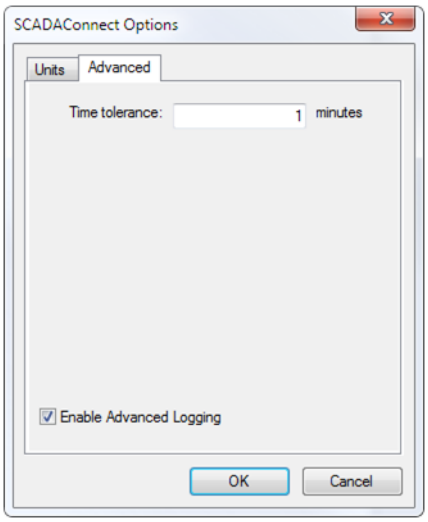
Advanced

Time tolerance: SCADA data may not be available at the time that the user requests it depending on the polling interval of the SCADA system. In order not to miss a valid data point because it does not fall exactly at the requested time, the user can specify a Tolerance. Specify the time tolerance for retrieval of historical data from the SCADA database. Time tolerance refers to the intervals centered about the specified time for the historical data query. The time tolerance should be large enough to cover the full range of signals to be retrieved. This is defined by the SCADA polling interval.

For example, if the time of a field data set for a historical data import is 12:00:00, then a time tolerance of three (3) minutes specifies a time span of six (6) minutes, from 11:57:00 to 12:03:00. This time span defines the query made against the SCADA system historical data by SCADAConnect and thus defines the range of valid time stamps for data loaded from the SCADA system into the model field data set.

The time tolerance should be set to the smallest value possible that captures a full snapshot of SCADA data. Avoid unnecessarily large settings. Only whole numbers can be entered.

The time tolerance only applies for a historical import in which historical data from the SCADA system are returned for the specified time span.



Enable Advanced Logging: When this check box is checked, the hydraulic model maintains a text log file. This log file will specially be helpful when you are not able to import the data and want to figure out what is occurring. Sometime this file may content technical terms which are beyond the normal hydraulic modeler. After looking at this log file, if you could not resolve the issue, contact Bentley Technical Support.

Generally the default location to this log file is at the following location:

Windows XP: C:\Documents and Settings\<User>\local Settings\Application Data\Bentley\WaterGEMS\8\SCADAConnect.log

Vista or higher:
C:\Users\<User>\AppData\Local\Bentley\WaterGEMS\8\SCADAConnect.log

Miscellaneous SCADAconnect Operations

Color-coding

Annotating

Color-coding

In order to color-code the elements in the drawing, the SCADA data must be imported to User Data Extension (UDX). To import the SCADA data into a UDX field follow the steps provided in Load Extended Data.

Steps to color-code:

Load data to User Data Extension or go to Load Extended Data.

Go to **View** > **Element Symbology**.

Right click on the desired element and select **New** > **Color Coding**. (Desired element must have at least one signal mapped).

Color Coding Properties opens.

Click on > button, next to the Field Name drop down and select **SCADA Data**.

Provide **Minimum**, **Maximum**, **Steps** and desired **Color** values.

Click **Apply** and **OK**.

Annotating

In order to annotate the elements in the drawing, the SCADA data must be imported to User Data Extension (UDX). To import the SCADA data into a UDX field follow the steps provided in Load Extended Data.

Steps for Annotation:

Load data to User Data Extension or go to Load Extended Data.

Go to **View** > **Element Symbology**.

Right click on the desired element and select **New** > **Annotation**. (Desired element must have at least one signal mapped.

Annotation Properties opens.

Click on > button, next to the Field Name drop down and select **SCADA Data**.

Provide **X Offset**, **Y Offset**, and **Height Multiplier** values.

Click **Apply** and **OK**.

SCADA Signals in Network Navigator

A user can display the elements that have SCADA signals created from SCADAConnect. If SCADA signal are not mapped then nothing will be selected on the drawing so mapping of SCADA signal should be done first, if not done already. For more details, go to mapping signals.

There are two ways to select the elements in the drawing.

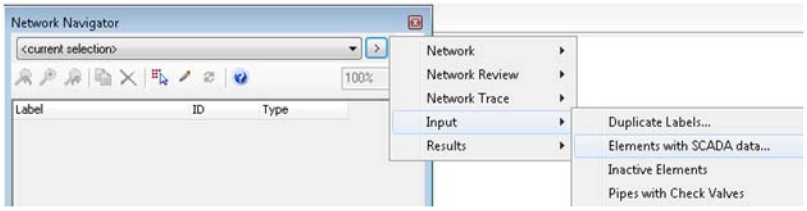
Using Network Navigator

Using Queries

Selecting Elements using Network Navigator:

Go to **View** > **Network Navigator**.

Click on > button, left to green arrow button and select **Input** then **Elements with SCADA Data**.

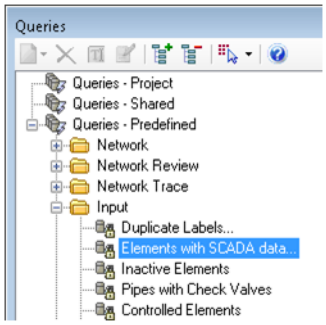


Make sure **Select in Drawing** button is toggled.

Selecting Elements using Query:

Go to **View** > **Queries**.

Expand, **Input** and double click, **Elements with SCADA data**.



SCADAConnect Simulator

The SCADAConnect Simulator for WaterCAD V8*i* application consists of some tools that someone, who is not necessarily a hydraulic modeler, can use to run a WaterGEMS model to simulate the performance of a water system and evaluate the response of the system to various operational changes. It is intended to enable a water distribution system operator to have access to much of the functionality of a sophisticated hydraulic model without the need to learn many of the work flows which are not needed by the operator. In particular, the operator would not be concerned with creating or calibrating the model as this should be done before the model is made available to the operator.

Before the SCADAConnect Simulator application can be run, some preliminary configuration is required (see SCADAConnect Simulator Configuration). If initial conditions are to be imported from a database/spreadsheet source or from a live SCADA feed, the mapping of signals from these sources to WaterGEMS initial

conditions must be created. If the results of the model run are to be viewed in the SCADA Human Machine Interface (HMI), results must be published to an OPC server for display in an HMI.

The SCADAConnect Simulator application can be started by clicking on the SCADAConnect Simulator shortcut icon from the Start menu under Bentley > WaterGEMS  . This will open the SCADAConnect Simulator dashboard which is the way that the operator will interact with the hydraulic model (see SCADAConnect Simulator Interface). At this point, the operator can set

Which file (water project) contains the WaterGEMS model

Which existing WaterGEMS scenario will be used as a starting point for creating a run

How the run will determine the initial condition for elements like tank and pumps

Starting time for a historically based run

Duration of the run

Override the demands and controls from the specified scenario

Once a run is complete, the operator can view the user notifications that were generated during the run. These would include:

Model messages which would include any problems with the model

SCADA messages which would include any warning from the SCADA system

Alarms messages which would include the kinds of messages that the normal HMI might display such as high or low tank levels

Once the run is complete, the operator can view results, modify some parameter, such as overriding a pump control, and restart the model run.

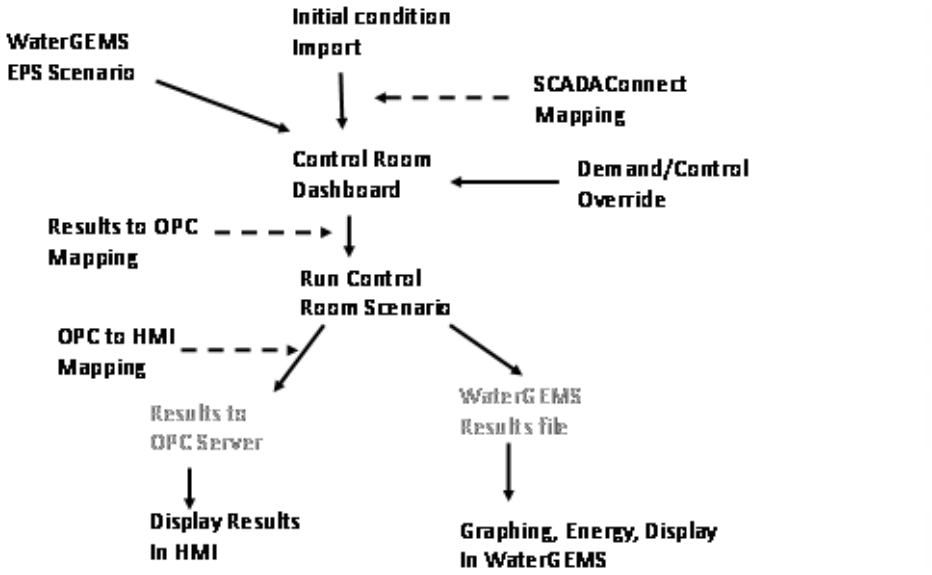
In general, there are two ways to view model results:

WaterGEMS: For those who are familiar with WaterGEMS, the results from a SCADAConnect Simulator dashboard run can be viewed in WaterGEMS using its visualization tools. This is especially useful for those wanting to view time series graphs or calculate energy costs as these features are not available from the SCADAConnect Simulator dashboard. Upon opening WaterGEMS after running in SCADAConnect Simulator, the user must access the results using File > Import > Results and select the .out file corresponding to the run as specified in Tools > Configuration > Output path. The results will be located in the scenario named "[Imported Results] Baseline Scenario Name".

SCADA HMI: For those who want to view the model results in the SCADA HMI interface (or don't know the WaterGEMS interface), the results from a model run can be viewed in the HMI. This requires that the link between the model and the HMI through an OPC server have been configured (see SCADAConnect Simulator Configuration) (OPC is the name for the specification standard used to communicate between different control devices and software). With this interface, the operator can look at the flows, pressures and other properties just as the operator can with real data in the HMI. The operator can move the time slider on the dashboard to view how values in the HMI can change over time.

The steps in running the SCADAConnect Simulator application are described in help topic SCADAConnect Simulator Interface.

An overview of the process to fully use the control room feature is show below.



Dashed lines are done during configuration. Gray boxes should be transparent to user.

SCADAConnect and SCADAConnect Simulator

The SCADAConnect feature and the SCADAConnect Simulator application are two different but related WaterGEMS features which enable the hydraulic model to work with data from water system operation and use the SCADA Human Machine Interface to display model results.

SCADAConnect is used to bring data from a SCADA system or other similar data sources, such as loggers or data files, for use in WaterGEMS for creating initial

conditions, loading data for Darwin Calibrator, importing data for graphing and tabular views, and performing simple demand inversing.

The SCADAConnect Simulator application can be used to take an existing WaterGEMS scenario, modify it (if desired) to reflect a different initial conditions, controls or demands and calculate the scenario to produce modeling results. These model results can then be displayed in the SCADA HMI so that operators can understand how their system would perform in an environment with which they are familiar. The immediate results of a run are notifications of alarms or warning that may have occurred during the run. Results from a model run are provided to the SCADA HMI through an OPC server which must be configured to take advantage of that functionality. Results from the runs from the SCADAConnect Simulator can also be viewed in WaterGEMS, and features such as graphing and energy costing can be used in WaterGEMS for those users who want to run the WaterGEMS interface.

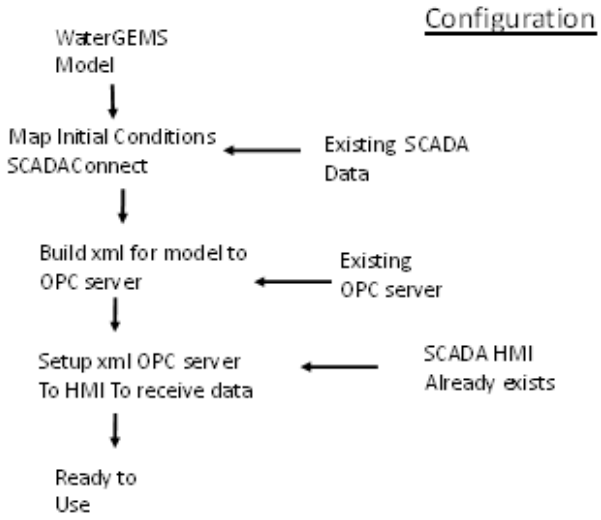
The SCADAConnect feature is started from Tools > SCADAConnect from within WaterGEMS. The SCADAConnect Simulator application is started from its own shortcut. SCADAConnect Simulator makes use of SCADA mappings defined in SCADAConnect for loading live or historical initial conditions.

When WaterGEMS is installed, the user can select whether or not the SCADAConnect Simulator will be installed. To run SCADAConnect Simulator, the user must have a license that enables SCADAConnect (SCADAConnect licensing is included with WaterGEMS and is an add-on for WaterCAD).

SCADAConnect Simulator Configuration

The SCADAConnect Simulator allows you to take a WaterGEMS model, make modifications to operational properties, run the model, view messages and display results in WaterGEMS with essentially no further configuration. However, to fully use the power of SCADAConnect Simulator, it is desirable to import initial conditions from external sources and display the results of a model run in the SCADA HMI (Human Machine Interface). To import initial conditions, the user must create mappings from the external data source to WaterGEMS using the SCADAConnect feature. To display results in the SCADA HMI, the user must configure SCADAConnect Simulator to publish results to an OPC server, for viewing an HMI.

An overview of the configuration steps to fully use SCADAConnect Simulator is shown below.



Importing initial conditions from external sources

The WaterGEMS model used in the SCADAConnect Simulator will need to contain an extended period simulation (EPS) scenario to serve as the baseline condition in the system. That scenario will contain initial settings with regard to tank levels and pump/valve status. However, the user may want to use some other initial settings which may be provided from a historical data source or live (real time) data from a SCADA Source. The Historical data is used to simulate past conditions while the Live data is used to view future operating conditions.

The initial settings must be mapped from the external data source using the SCADAConnect feature in WaterGEMS. SCADAConnect allows you to map how each signal from the external source is associated with a WaterGEMS property. For example, TNKSOUTH LEVEL may map to WaterGEMS element "TNK-5" and property "Level". The details of setting up these mappings are provided in the help topic Signal Mapping.

If an initial setting is not mapped from the data source, the value in the baseline scenario is used.

Publishing model results to OPC server

To view model results in your SCADA HMI, it is necessary to publish the model results to an OPC server. (OPC is the standard for sharing control data between devices.) The server could be the same computer that the model is running on, or it could be some other networked computer. For every calculation result to publish for viewing in a simulation HMI-display, a new signal tag in required. These signal tags need to be added to the OPC server; either a new OPC-server, or the existing real-time

OPC-server. The OPC-server needs to support classic OPC-DA requests for reading/ writing signal values. Setting up an OPC-server (if required) or adding new signal tags is Vendor specific. Please refer to the documentation of your SCADA software.

The mappings from WaterGEMS to the OPC server will generally parallel the signals used in the real-time HMI display to receive actual field data from the real OPC server. The user may want to use the list of available signals in the SCADA system as a starting point for setting up the mappings. Many of the actual SCADA signals will not correspond to a model result and can be eliminated (e.g. intrusion alarms, motor temperature).

Each result attribute in the model to be published corresponds to a row in the results publishing xml file. To specify a mapping between a result property of a model element and a simulation signal the user needs to specify:

ElementType of the model element (e.g. **Tank**, **Pump**, **Valve**, **Node** or **PressurePipe**),

ElementID of the model element (not the label),

ComputedAttribute (e.g. **PumpSpeed**, **PumpStatus**, **PumpFlow**, **TankLevel**, **TankHGL**, **PressurePipeFlow**, **PressurePipeVelocity**, **ValveSetting**, **ValveStatus**, **ValveFlow** and **NodePressure**)

OPCTag consists of the OPC-servername of the simulation OPC Data Access server and the full signal name separated by a backslash.

AttributeStorageUnit: optional - possibility to specify a different unit for the published value (e.g. AttributeStorageUnit="flow:mgd")

An example of result publishing configuratuion is shown below:

```
<?xml version="1.0" encoding="utf-8"?>

<ControlRoomOutputSetManager>

  <StorageElementHeader Id="0" Version="1"
ElementName="ControlRoomOutputSetManager" />

  <ControlRoomOutputSetManager Count="3">

    <ControlRoomOutputSetElement ElementType="Tank"
ElementId="34" OpcTag="WaterGEMSData.TankResults.Level
T1" ComputedAttribute="TankLevel" />

    <ControlRoomOutputSetElement ElementType="Pump"
ElementId="28"
```

```
OpcTag="WaterGEMSData.PumpResults.PumpState P-1"
ComputedAttribute="PumpStatus" />

    <ControlRoomOutputSetElement ElementType="Pump"
ElementId="28" OpcTag="WaterGEMSData.PumpResults.PumpFlow
P-1" ComputedAttribute="PumpFlow" />

  </ControlRoomOutputSetManager>

</ControlRoomOutputSetManager>
```

This file is identified in the SCADAConnect Simulator dialog using Tools > Configuration.

Configuring your HMI to Show the Simulation OPC

The user must also define HMI screens for showing values from the simulation OPC server. This step is specific to each HMI vendor. In general, it is best to start with a copy of your existing HMI as a starting point, and modify that to show values from the simulation OPC server.

SCADAConnect Simulator Interface

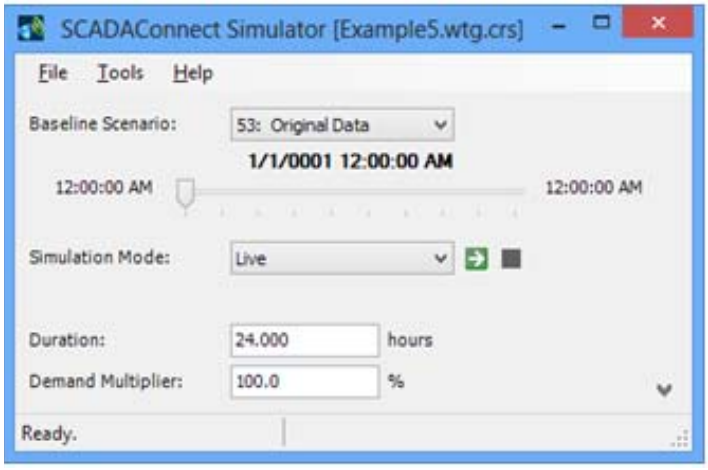
SCADAConnect Simulator provides a user with a way to make a run of a WaterGEMS model without the need to learn the details of building a model or using more advanced features. It is intended so that a control room operator can use WaterGEMS in a familiar environment and optionally, view results in a SCADA Human Machine Interface (HMI). Refer to the SCADAConnect Simulator Configuration section for getting things set up to perform a run.

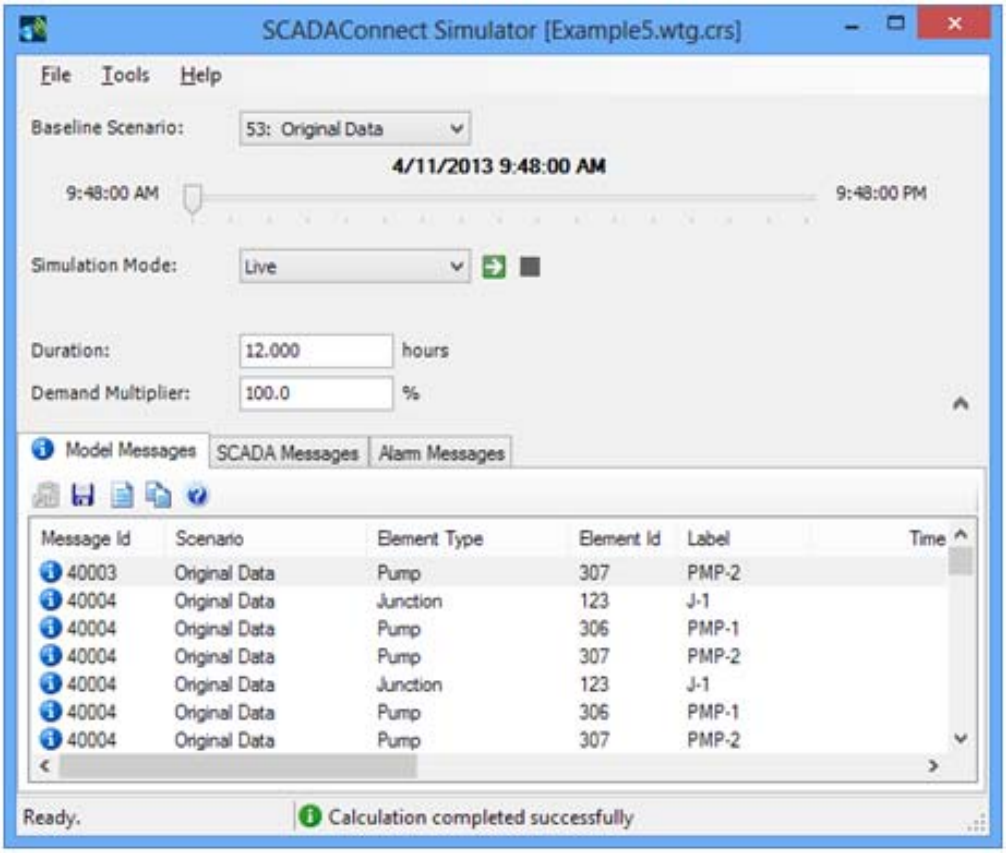
To start the SCADAConnect Simulator application, click on the SCADAConnect Simulator shortcut (or select "SCADAConnect Simulator for WaterGEMS" under the Windows Start menu.

SCADAConnect Simulator Window

The SCADAConnect Simulator window supports two view modes: Compact and Full.

Click the chevron icon to toggle between view modes.





File: Open an existing model project file using File > Chose Water Project. The user then browses to a file with a .wtg extension containing the model and picks Open.

Scenario: The WaterCAD V8*i* model must contain at least one Extended Period Simulation (EPS) scenario to be run. Pick one of the EPS scenarios from the Baseline Scenario drop down menu. This Baseline Scenario contains the scenario that the user can run (If no EPS scenarios exist, open the project in WaterGEMS to add one).

Simulation mode: The Simulation Mode allows you to specify how the initial conditions (e.g. tank levels, pump on/off or speed) are to be applied. The selections include

Baseline - the model uses the unmodified initial conditions that are defined in the Baseline scenario.

Historical - the model uses initial conditions taken from a historical SCADA data source (spreadsheet, database file, or from an historical OPC server containing initial conditions). When this option is selected, the user must identify the Start date and time to start the run (data for this date and time must exist in the configured SCADA data source). The historical SCADA Source and the signal mappings need to be configured using the WaterCAD V8*i* SCADAConnect feature. This option is used to model past conditions.

Live - the model uses initial conditions taken from the most recent values from the SCADA OPC server. The mapping to the OPC server must have been created using SCADAConnect. This option is used to model forward from the current time.

Live (Auto Compute) - the model behaves essentially like the Live option but the model will automatically load initial conditions and start a run at a time interval specified in the Auto Compute Interval box. This is used to continuously run the model to forecast future conditions.

If a value of an initial condition needed for a model run is not available from the historical or Live data source, the value from the initial conditions in the baseline scenario are used.

Start Time: For a historical run the user needs to specify the Start date and time for the historical run here (this field is only shown if the Historical Simulation Mode has been selected).

Duration: The user must then select the Duration for the run. The default value is the value from the Baseline Scenario, but it can be overridden.

Demand Multiplier: By default the model will use the water demands associated with the baseline scenario. The user can globally adjust the demands by changing the Demand Multiplier. A value of 120% would multiply all demands by a factor of 1.2. To make more advanced demand adjustments (e.g. if demands are to be changed at only a small subset of nodes), a new scenario should be set up in WaterCAD V8*i*.

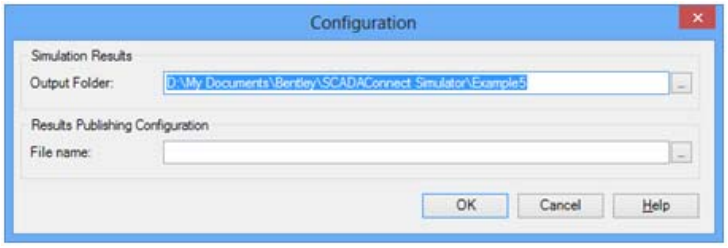
User Notifications Pane: The bottom portion of the window displays different types of run notifications. There are three types:

Model Messages contain notifications of problems and issues with the model. If they are red, it means that the run did not complete successfully; yellow means that there are warnings that the user should investigate and blue are information relating to the model run.

SCADA Messages contain error messages related to attempting to read initial conditions from the configured data source.

Alarm Messages contain calculated related alarms that were triggered during the model run, such as high tank level.

Configuration Settings: Before starting a run, the user must identify where the results will be saved on the computer for viewing by selecting Tools > Configuration. Click the elipsis button [...] to interactively specify the folder.



The output folder is a required field and is the folder where SCADAConnect Simulator places any output created from a run. Use the ellipses button to browse for the folder interactively.

The **Results Publishing Configuration** field is used to identify the xml file that describes results to be published to the OPC server so that the results can be displayed in the SCADA HMI. Click the ellipsis button [...] to interactively specify the file name (see SCADAConnect Simulator Configuration).

Overriding controls: Pumps and valves by default are controlled using control statements associated with the Baseline scenario. The user can override those controls by picking Tools > Control Overrides. See SCADAConnect Simulator Control Overrides.

Time Slider: After a successful run the user can use the time slider to choose the result time for displaying the calculation results in the HMI display (similar to the Time Brower in WaterGEMS).

SCADAConnect Simulator Control Overrides

The Baseline Scenario contains control statements that determine how and when pumps and valves are operated. These all you to control pump on/off status (or pump speed for variable speed pumps), and open/closed status for pipes and valves.

The user can override controls by picking Tools > Control Overrides. This opens the Control Overrides Window:



The tabs along the top of the window indicate the type of element to be controlled. The word "status" refers to digital properties that can be turned on/off or open/closed. The word "setting" refers to analog properties that vary continuously such as pump speed (for variable speed pumps) or valve setting.

The New button adds an entry to the table, while the Delete button removes the highlighted entry.

Using pump status as an example, each entry is describes below. Other tabs behave similarly.

Override Enabled: It is possible to create an override but not use it for a given run. This is controlled by the override enabled check box. Checking the box means that the override will be used in the next run.

Label: This field should contain a name for the override so that the user can remember the purpose of that override. It need not duplicate information in subsequent fields.

Controlled Element: This field contains the name of the model element that is being controlled. It is populated by picking an element from the drop down list.

Pump Status: This field is set to on or off for pumps. For elements where the value is "setting", this is a numerical value.

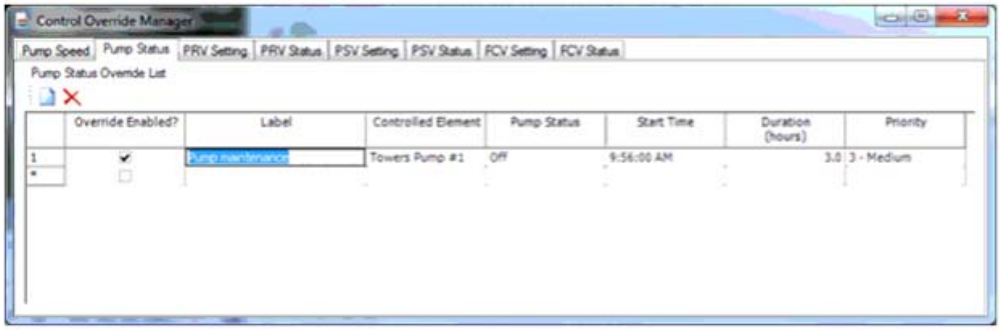
Start Time: The start time is the time at which the pump override becomes effective. The time can be adjusted by picking the hour, minute or second value and clicking the forward or backward button.

Duration: The Duration is the amount of time that the override will be in effect. After this time is exceeded, the controls associated with the baseline scenario will once again be in effect.

Priority: The priority determines which control statement is used when there is a conflict between control statements. By default this value is 0. Higher values (up to 5) take precedence.

Example: Pump Status Control Override

An example of a control override for pump status is shown below. In it, the pump, Tower Pump #1 will be turned off at 9:56 am for 3 hours for maintenance and that priority will override default priority. After 3 hours, the pump will become available for control by other control statements. If the user would want to force it on at the time, it would require adding another override.



Flushing Simulation

WaterCAD V8*i* flushing module can be used to simulate the effect of flushing water distribution systems.

There are several purposes for flushing distribution systems including increasing velocity to scour pipes, reducing water age, testing operation of hydrants, etc. The WaterCAD V8*i* implementation of flushing is oriented toward increasing velocity in mains to flush out solids and stale water. The primary indicator of the success of flushing is the maximum velocity achieved in any pipe during flushing operation.

Type of Flushing

The basic concept in flushing is an "Event". This corresponds to one snapshot during a flushing program. Flushing analysis consists of simulating many flushing events.

WaterCAD V8*i* can analyze two general types of flushing, Conventional and unidirectional:

- Conventional flushing consists of opening up hydrants or blowoffs one at a time without any isolation valve operation.
- unidirectional flushing (UDF) consists of one or more hydrants or blowoffs while isolation valves (or pipes) may be closed to control the direction of flow.

Depending on the target velocities and layout of the system, conventional flushing is often adequate. unidirectional flushing will improve velocity although it requires additional labor. A recommended workflow is to first simulate conventional flushing and then identify areas which are not adequately flushed and require unidirectional flushing. If a secondary goal is to test the operation of every hydrant, then conventional flushing is usually adequate while if valve exercising is also a goal, unidirectional flushing becomes more attractive.

Starting model

For flushing analysis, it is best to start from an all-pipe model. Small pipes without a means of flushing (e.g. 2 in. pipes) can be excluded. Ideally, the model will also contain every hydrant and isolating valve at its exact location. This is especially important for UDF because the location of a hydrant relative to the closed valves is very important.

If a model does not contain hydrant elements, junction nodes can be used as flushing points. The error should be small for conventional flushing although for UDF a valve may be closed valve between the hydrant and junction. If hydrant elements are used, it is not necessary in explicitly include the hydrant lateral in the model because the lateral length and its associated head losses can be accounted for within the hydrant element.

If isolating valves are not included in the model, the user can simulate valve closing by closing pipes, although it is up to the user to insure that a valve is actually available in the field to close the pipe.

Specifying hydrant flows

Hydrant flows may be specified directly in flow units or as an emitter coefficient. Because hydrant flow is a function of pressure and the user does not usually know the pressure at the hydrant beforehand, it is more accurate to specify the emitter coefficient. For standard North American hydrants that comply with AWWA Standard C502 or C503, the emitter coefficient would be 150-180 gpm/psi$^{0.5}$ (11-14 L/s/m$^{0.5}$) for the 2.5 in. (63 mm) outlet and 380-510 gpm/psi$^{0.5}$ (30-40 L/s/m$^{0.5}$) for the 4.5 in. (115 mm) outlet depending on the model of hydrant, size of barrel and length of barrel. See Advanced Water Distribution Modeling and Management (p 451-453) for more discussion on this. In terms of flow units, free discharge from a hydrant can vary from 500 to 1500 gpm (32-95 L/s) depending primarily on the strength of the distribution system at that point.

Flushing Manager

The Flushing Manager is used to set up flushing events, evaluate their effects and set up reports which can be given to operators to carry out flushing programs. The flushing manager can be opened by selecting Analysis > Flushing Manager or picking the flushing manager button from the Analysis toolbar.  .

Flushing in WaterGEMS/WaterCAD is designed to simulate the kinds of flushing performed to increase velocity or shear stress in pipes to remove any deposits and thus improve water quality. Velocity or shear stress can be compared with target values to determine if flushing was successful. This type of flushing is based on steady analysis. If flushing is being performed to decrease water age, it is best modeled by setting up an extended period simulation run to view the changes in water age or some other constituent.

Note: For users of WaterGEMS and WaterCAD SS3 and earlier, flushing was controlled in the flushing alternative. For SS4 and later, this functionality was moved to the Flushing Manager and a large number of additional features were added. Opening a file created in SS3 or earlier will result in the information from the flushing alternative being transferred to the Flushing Manager.

The following Help topics provide details on the steps involved with setting up flushing and viewing results.

- Flushing Terminology
- Flushing Work Flow
- Starting Flushing Manager
- Flushing Area Options
- Flushing Event Creation
- Flushing Manager Toolbar Buttons
- Flushing Results Browser
- Flushing Area Report (Flex Table)
- Flushing Options Dialog
- Flushing Notifications
- Flushing Operator's Report

Flushing Terminology

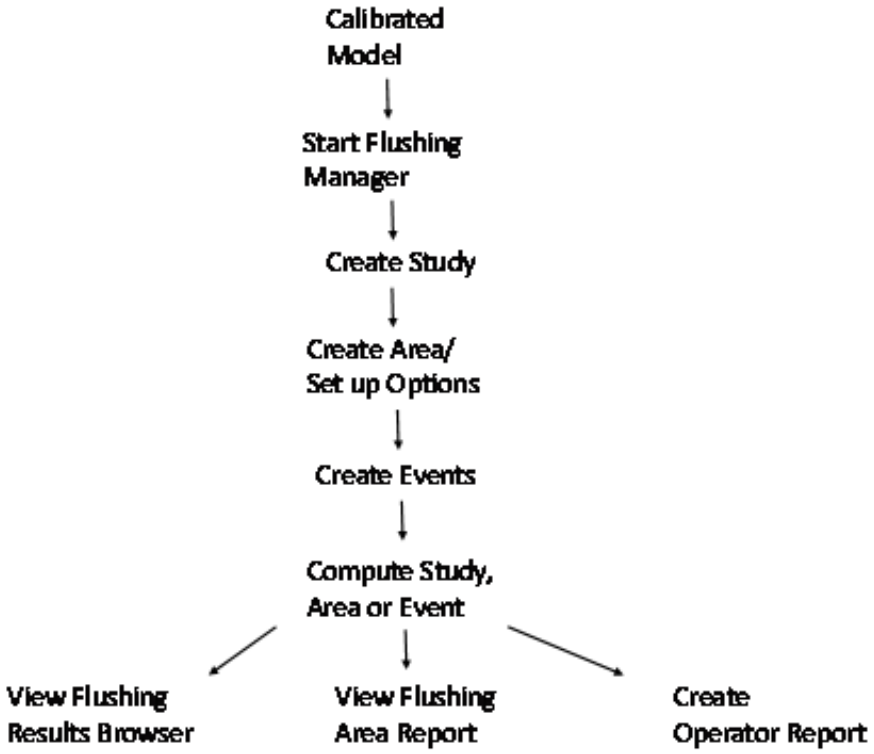
Some terms used in flushing are explained below:

- **Event** refers to a single operation of a flowed hydrant(s) with any associated valve operation. It corresponds to a single steady state simulation with a flowed hydrant(s). Events may be conventional or unidirectional.
- **Conventional event** refers to opening a single hydrant with no associated valve operation (valves are set according to the representative scenario). These are treated separately making it very easy to set up a large number of conventional events (as opposed to the more detailed steps needed for unidirectional flushing). The user may wish to quickly assess the performance of conventional flushing as a first step before moving to unidirectional flushing.
- **Unidirectional flushing (UDF)** refers to flushing where isolation values (or pipes) may be closed and more than one hydrant may be flowed. UDF can generate higher velocities and shear stress. The user can compare with conventional flushing to determine if the additional effort is justified.

- **Pipe run** refers to the collection of pipe links that a user wishes to flush in a UDF event. The volume of water in the pipe run is used as the minimum amount of water that must be flushed and the time to flush that volume is used as the minimum time of flushing. A pipe run should consist of pipes in series from the flowed hydrant. There is no pipe run for a conventional event since flow direction cannot be controlled.
- **Flushing Area (or Area)** refers to a set of flushing events that are usually focused on a given portion of the system. By computing an area, every event in that area is simulated. An area is associated with a single representative scenario which controls boundary conditions. An area might consist of a neighborhood to be flushed or a collection of events that can be run by a crew in a single shift. In general flushing areas should not significantly overlap.
- **Pipe set** refers to the user wants to flush in a given area. These are the pipes considered when determining properties like "Pipe length met target". The Pipe set should encompass all pipe runs in the area. A pipe set is a required input. It is created by picking the ellipse button next to pipe set.
- **Nodes of Interest** are nodes for which auxiliary results are saved. These are useful for monitoring nodes than may have low pressure during flushing. Nodes of interest are an optional input.
- **Flowed elements** can be either junction nodes or hydrants. For conventional flushing with no valve closure, hydrants are generally close enough to nodes that the results are virtually the same. However, in UDF where a valve may be closed between the hydrant and junction, it is important to represent the flowed hydrant explicitly in the model.
- **Controlled (Closed) elements** can be represented either by a closed isolation valve or a closed pipe element in UDF. (There are no closed elements in conventional flushing.) Closing an isolation valve is a more precise way of modeling UDF but some models do not contain isolation valves. When a pipe element is closed, it is assumed that an operable valve is present. A closed pipe cannot be part of a pipe run.
- **Flushing study** refers to a group of areas that possibly cover the entire system. Computing a study will run all of the events in all of the areas in the study. A set of studies may be used to compare different approaches to flushing a system. One study may rely heavily on conventional flushing while another may rely on UDF. There needs to be at least one study.
- **Representative scenario** refers to the existing scenario that established the boundary conditions and demand that relate to a flushing area. This determines which pumps are operating, what the demands are and what tank levels are set to during the flushing analysis.
- **Output scenario** is the name given to the scenario that contains the results of the flushing analysis. There is one output scenario per area and the current scenario should be set to the output scenario to view results in the flushing result browser once the user leaves the flushing manager.

Flushing Work Flow

The overall work flow for modeling flushing is shown below:



To perform an analysis of a set of flushing events (i.e. a flushing area), the user must create flushing events. Upon opening the flushing manager initially, there will be a default study "Flushing Study" which will have one area called "Base Flushing" in the left pane.

The user creates new studies or areas by right clicking on the study node in the left pane. Right clicking on the study node creates new events.

Within a flushing area, the user defines the representative scenario, target velocity and shear stress, pipe set, method to determine flow (emitter or flow) and auxiliary output if desired. It is a good idea to create a selection set corresponding to the pipe set before entering the flushing browser.

The user then creates events within an area. Conventional events are made up of the hydrant (or junction) to be flowed while UDF events are made up of flowed elements, controlled (closed) elements and pipe runs. The user can also identify the extent of the drawing that will appear in the optional reports.

Once the events have been defined, the user can compute the flushing events for either the study or the flushing area. The results can be reviewed with the Flushing Results Browser which presents results based on events or the Flushing Results Flex Table which presents results based on pipes.

The user can then optionally prepare a report for the operators who will conduct the flushing containing instructions and drawings for each event.

Starting Flushing Manager

The Flushing Manager can be started by selecting Analysis > Flushing Manager or picking the Flushing Manager button from the calculation toolbar. [show button]

The Flushing Manager opens and the user much create a study and a flushing area. This can be done by picking the New button from the top of the left pane and selecting New Study or New Area. An area is a subset of a study. A study or area can also be created by right clicking on a study node in the left pane. [show flushing manager with at least one study, area and event]

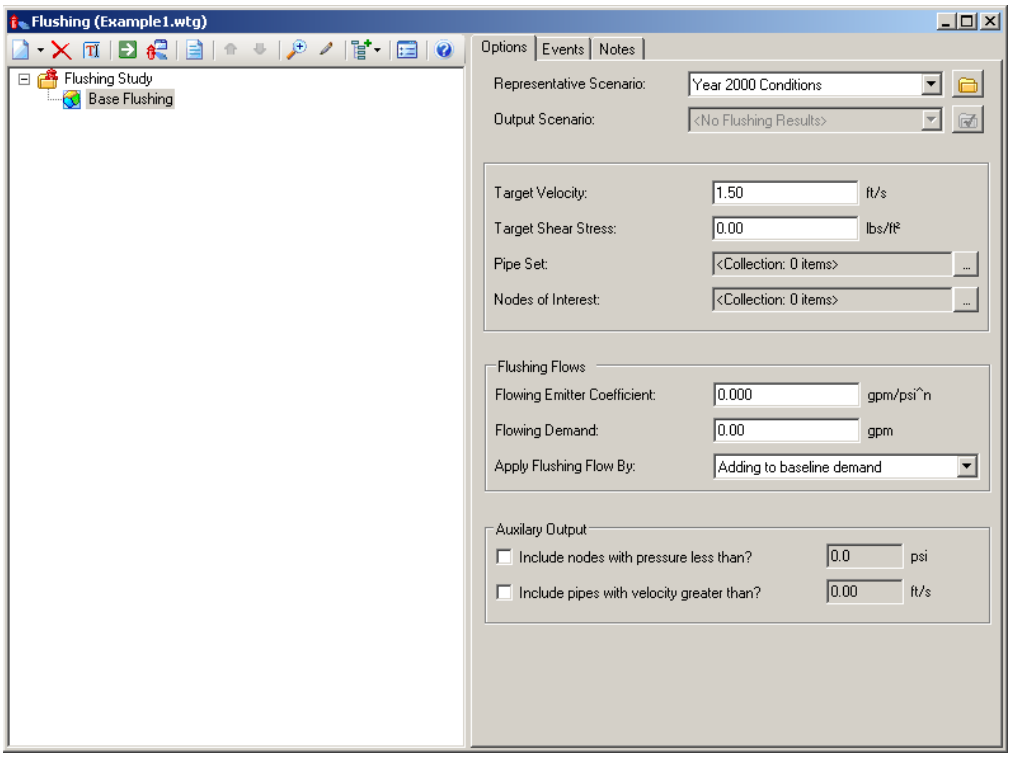
When the study node is highlighted in the left pane, the right pane lists the flushing areas that are associated with that study. The user can edit the Representative Scenario in the right pane.

Right clicking on the study node opens a list containing

- **Add** - create new study or area
- **Delete** - delete the study
- **Rename** - renames the study
- **Compute** - computes all of the active events in the study
- **Zoom To** - zooms to the extents of the elements in the study
- **Highlight** - highlights the elements in the study
- **Expand Children** - expands the tree view of areas in the study
- **Collapse Children** - collapses the tree view of areas in the study

Flushing Area Options

When the flushing area is selected in the left pane, the user can set up global options for the events within that area. Most of these are set in the options tab for the areas node. The most important is the Representative Scenario which establishes the boundary conditions (tank levels, pump status, demands) for the area.



The Output Scenario is the scenario where the results of the flushing analysis will be stored. The output scenario is created automatically the first time the area is computed.

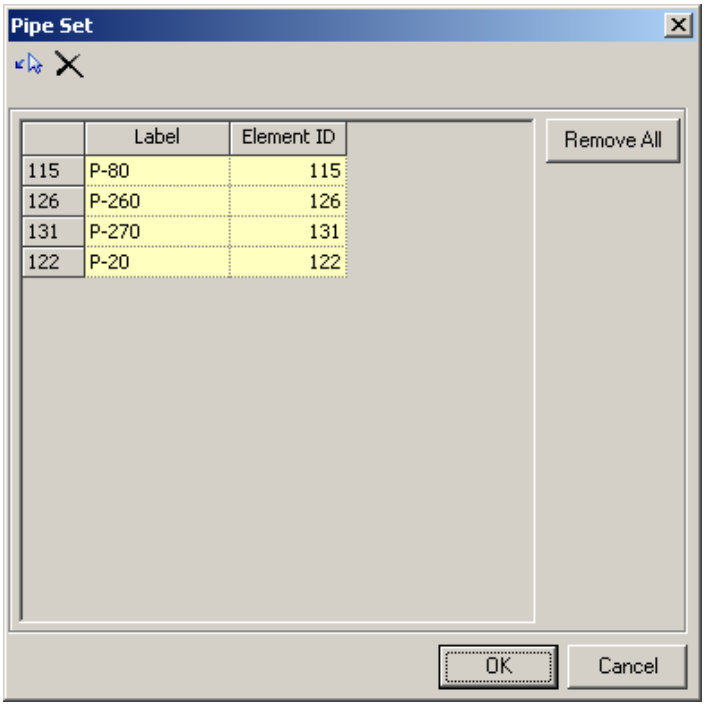
The Target Velocity is the velocity that should be exceeded for the flushing to be considered successful for that pipe. The user may specify a Target Shear Stress as well as a Target Velocity or in addition to a Target Velocity.

The Pipe Set is the collection of pipes for which the target velocity will be compared with the maximum velocity achieved by flushing. It is created by clicking the ellipse button and entering the pipe set dialog. Picking the Select from Drawing button enables the user to select the pipes to be included in the set using the standard element selection dialog.

The first toolbar button is used to select elements from the drawing. The standard select from drawing toolbar is displayed when in selection mode. Only pipes can be selected for this dialog.



(It may be advisable to create a selection set of pipes before entering the flushing manager.) The delete button can remove individual elements while the Remove All button removes all at once.



The Nodes of Interest ellipse operates similar to the Pipe Set except that it selects nodes that will always appear in the auxiliary results. Most nodes will not have data saved for each flushing event. Only those that meet the auxiliary results criteria or appear in the Nodes of Interest will be included.

Under flushing flows, the user can specify either the emitter coefficient for the hydrant or junction being flowed or the actual flow rate. Because flow rate depends on pressure and the user does not usually know the flow rate ahead of time, it is usually more accurate to specify and emitter coefficient. Typical values are 250 gpm/psi$^{0.5}$ (20 L/s/m$^{0.5}$). See page 453 of *Advanced Water distribution Modeling and Management* (Bentley). Do not specify both an emitter coefficient and a flow.

Depending on the selection from the drop down menu "Apply Flushing Flow By", the hydrant flow can be added to the node demand or used in place of the nodal demand.

Under Auxiliary Output, the user can save values for all elements for each event. However, in most cases the user is not interested in values for properties in elements far from the flushing. The user must therefore specify condition for which element data are saved and available for display for individual events. If the box, "Includes nodes with pressure less than?" is checked, properties for elements with pressure less than the specified value are saved for display/ If the box, "Include pipes with velocity greater than?" is checked, properties of pipes with high velocity are saved. This makes it possible to use color coding to display results of flushing without saving a great deal of unneeded values.

The Events tab enables the user to get a quick view of the events that are contained in the area and if desired, make events active or inactive for the next run. Click the Conventional Event Quick Edit button to open the Conventional Event Quick Edit dialog, allowing you to globally edit local flows and emitter coefficients across multiple events.

The Notes tab enables the user to enter a text description of the area.

Right clicking on an area in the left pane opens the following options

- **Add** - create new event
- **Delete** - delete the area
- **Rename** - renames the area
- **Compute** - computes all of the active events in the area
- **Shift Up** - moves the area up the list of areas
- **Shift Down** - moves the area down the list of areas
- **Zoom To** - zooms to the extents of the elements in the area
- **Highlight** - highlights the elements in the area
- **Expand Children** - expands the tree view of areas in the area
- **Collapse Children** - collapses the tree view of areas in the area

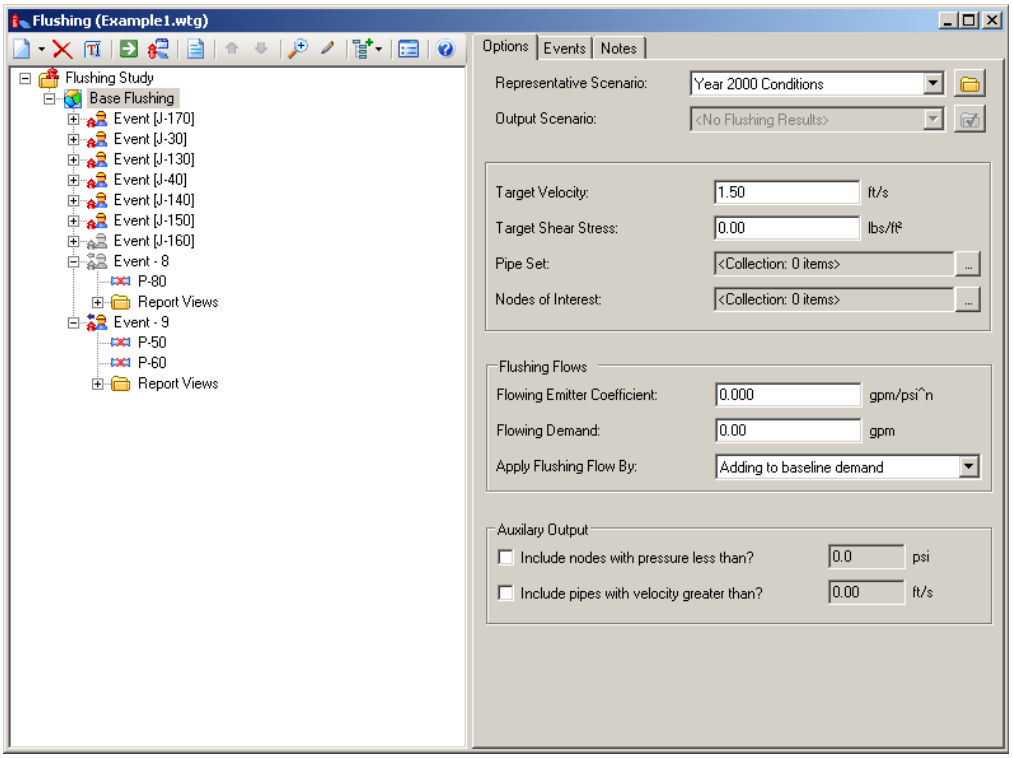
In the left pane, the type of event and its status is designated by the icon representing that event

 - conventional active

 - UDF active

 - conventional inactive

- UDF inactive



The event Active check box is on the top of the right pane when the event is highlighted in the left pane. Inactive events are not computed.

Flushing Event Creation

Once a study has been defined, the events that make up the study can be created, the user can create events. Events are created by picking the New button at the top of the left pane when the area is highlighted and selecting Conventional or UDF, or right clicking on an area and selecting Conventional or UDF. It is assumed that the flushing events are conducted in the order in which they are listed. The key to order is usually to flush from clean water into un-cleaned areas.

Conventional flushing events have the advantage of being very easy to set up a large number of conventional events in essentially one step. When conventional is selected as the type of event, the user sees the Selection dialog where individual junctions of hydrants are selected, junctions or hydrants can be selected by polygon or they can be selected based on a selection set that has been previously defined. Having a selection set already defined if not all the nodes in a polygon are to be flowed can be helpful.

UDF events can only be created one at a time because the user must select flowed elements, controlled elements and optionally the pipe run to be flushed. In this case a special form of the select dialog is opened.

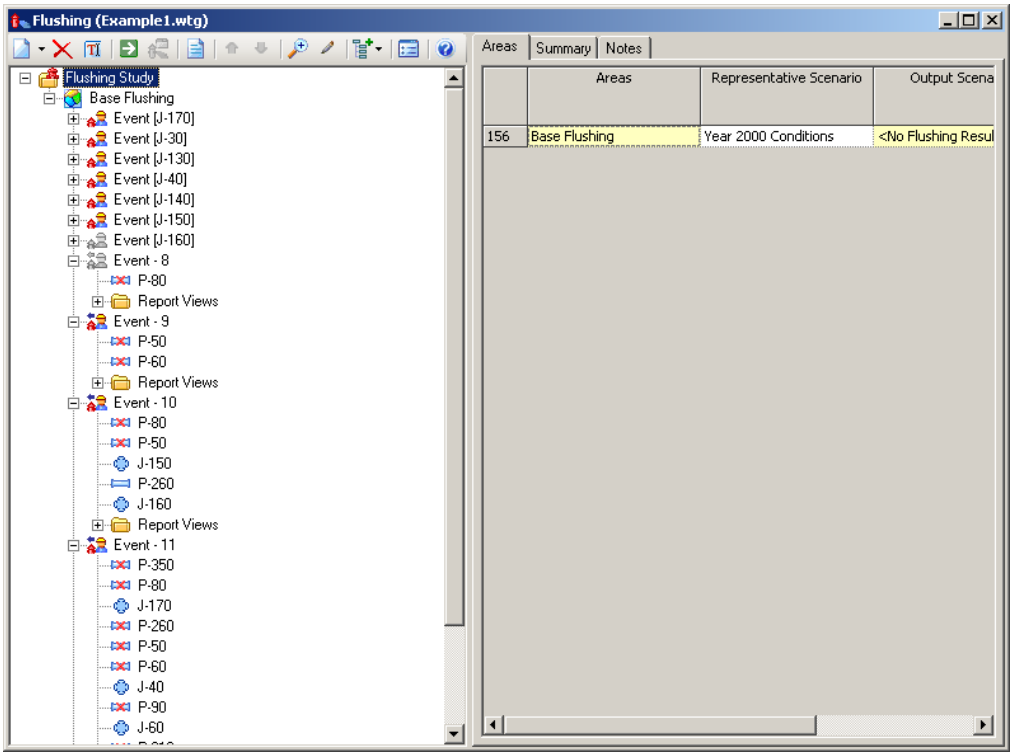


If a node element is selected, it is considered to be flowed if it is a hydrant or junction and is considered to be closed if it is an isolation valve or control valve. If a pipe element is selected, there are two options. The default is that the selected pipe is closed. If the user has selected the third button on the Select dialog, the pipes that are being selected are part of a pipe run. Picking the second button will switch back to closed.

The Report Views node contains a list of drawing views that will be included in the Operator Report. The report views show the coordinates of the corners of the view. The primary view is created initially automatically based on the extent of the elements involved in an UDF event and the flowed hydrant with a buffer around it (default = 300 ft) for a conventional event.

Once an event is created, if the event is expanded in the left pane, there is one row for each element that is flowed, closed or part of a pipe run. The following icons are displayed

When an event is highlighted, the right pane displays details of the elements included in that event.



The element label and type are properties of the element selected and status is an editable field indicating if the element is open/closed, flowed or part of a pipe run. The user can overwrite the flow emitter or flows specified in the area tab by checking the Specify Local Flows check box for that element and inserting a different flow or emitter for that element.

Notes fields are very important if the results of the flushing analysis are to be given to operators to locate elements to operate. This might include "Southwest side of Adams St. and 3rd Ave." as a hydrant description or "In front of 319 Penn Ave. - watch out for big dog" as the location of a valve that needs to be closed.

Flushing Manager Toolbar Buttons

The buttons at the top of the left pane in the flushing manager are described below:

- **New** - creates new study, area or event depending on which node is highlighted
- **Delete** - deletes the highlighted study, area or event

-  **Rename** - start editing of highlighted study, area or event
-  **Duplicate** - creates a copy of the highlighted area or event
-  **Compute** - starts analysis of highlighted study or area
-  **Flushing Browser** - opens up flushing browser for the selected scenario
-  **Report** - opens up preview of operator report
-  **Move Up** - moves selected area or event up the list
-  **Move Down** - moves selected area or event down the list
-  **Zoom** - zooms to extent of selected study, area or event
-  **Highlight** - highlights elements in selected study, area or event, as follows:
  - Circles with an X represent closed nodes.
  - Circles represent open nodes.
  - Dashed lines represent Pipes.
  - Solid lines represent Pipe-Runs.
-  **Expand/Collapse** - expands or collapses selected node
-  **Options** - enables user to set default colors and extent of view in conventional flushing
-  **Help** - opens flushing help

Flushing Results Browser

The flushing results browser contains the results of a flushing run listed by event. It is assumed that the flushing events were conducted in the order in which they are listed. The content of the browser corresponds to the current scenario. If it is not a flushing output scenario, no events would be displayed.

The display in the flushing browser corresponds to the current scenario which needs to be a flushing output scenario. The scenario can be switched to the flushing browser in the main drawing or by picking the button "Make Output Scenario Current" next to the output scenario selection in the right pane of the flushing manager.

Before opening the browser it is helpful to set up color coding and annotation that will highlight the flushing events. Usually color coding pipes by velocity or shear stress and junctions and hydrants by demand will be the most useful. For example, pipes with a velocity over 4 ft/s (1.2 m/s) might be red with thickness three times that of other pipes.

Toolbar buttons at the top of the browser enable the user to:

-  **Zoom** - zooms to extent of flushing event
-  **Highlight** - highlights elements in flushing event. In highlighting, the pipe run color will override element symbology color coding
-  **Reset** - cancels out the selected event and displays results for representative scenario
-  **Report** - opens preview of flushing browser report
-  **Help** - opens flushing help

The columns listed in the browser include

- **Label** - the event label
- **Flushing Type** - conventional or UDF
- **Pipe Length Met Target** - length of pipe in flushing area that met both velocity and shear stress target during event
- **Cumulative Pipe Length Met Target** - sum of length of pipe in flushing area that met target of all events up to current event
- **Incremental Pipe Length Met Target** - difference between cumulative length for this event and previous event. If pipe length that me target is large but incremental length is small, event may be duplicating effects of other prior events.
- **Minimum Pressure Node** - node with lowest pressure in the flushing area or nodes of interest
- **Minimum Pressure** - pressure at node in previous column
- **Travel Time (Pipe Run)** - minimum flush volume divided by hydrant flow
- **Volume** (**Minimum, Pipe Run)** - volume of water in pipe run that must be flushed (0 if no run specified).
- **Flow (Pipe Run)** - flow in the pipe run that must be flushed

When an event is highlighted, the property grid and flex tables will contain values corresponding to that event. If elements are not associated with the event, they will have NA in many fields.

To view flushing by pipes instead of by event, use the Flushing Area Report (Flex Table).

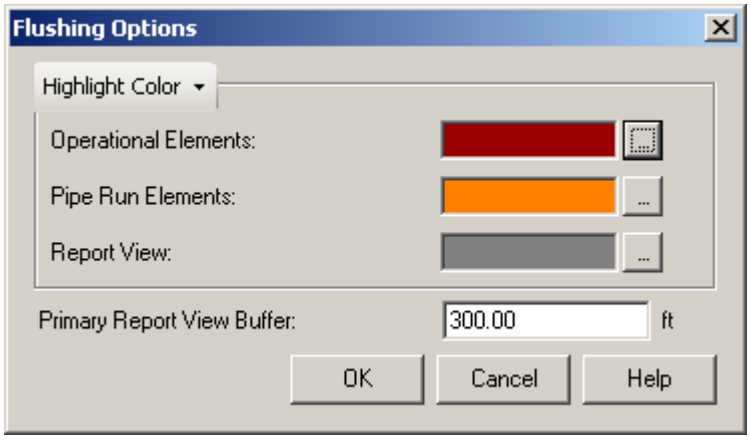
Flushing Area Report (Flex Table)

While the flushing results browser displays flushing results on an event basis, the flushing area results flex table presents the results on a pipe basis listing whether the pipe met the flushing target and which event was the most effective in flushing that pipe.

The flushing flex table can be opened as any other flex table by selecting View > Flex Table > Flushing report when the current scenario is a flushing output scenario. By default, the table will open with all pipes. It is usually helpful in large models to make a selection of elements and pick "Open on Selection" (right click on flushing area report button) when opening the flex table.

Flushing Options Dialog

The flushing options dialog enables the user to set the highlight color for operational or pipe run elements or bounding boxes for views. The user can also set the bounding box size for the operator report for conventional flushing.



Flushing Notifications

Several notifications can be generated during a flushing run. They are listed below:

Table 10-3: Flushing Notifications

Notification Text	Notification Category	Response
Pipe set not specified.	Error	Specify pipe set.
At least one flushing event element is not active during the flushing run.	Warning	Elements must be active to affect flushing results. Have you deleted any pipes since pipe set was created?
At least one run pipe is not included in flushing pipe set.	information	Ideally, pipes in a run should be included in pipe set.
At least one run pipe is closed during the flushing run.	Warning	Pipes in run should not be closed.
At least one event contains a pipe run that is not continuous.	Warning	Check for gaps in the pipe run.

Flushing Operator's Report

The output report is intended to be prepared by a modeler and given to field operations crews so that they have explicit direction on which elements to operate. Before opening the operator's report button, the modeler should

1. Set up the desired background layer.
2. Decide the extent of the view to display and if additional views are desired, set up those views.
3. Include detailed notes to help the operators locate the elements (e.g. an operator may not know where H-21 is located but will know "Hydrant in front of 31 Elm St.").

The operator report consists of three types of pages for each event:

1. Text description of the event indicating which elements to operate.
2. Drawing of the event.
3. (Optional) Form which flushing operator can complete to describe results of flushing for feedback.

In addition to the default drawing of the event, the user can create "Secondary Views" which may for example, zoom in to details of a complex intersection. To do this, right click on Report Views in the left pane and pick Add Secondary View. The draw a box around the extents of the secondary view and click Select New Report View.

The view that appears when the report is opened is called a Preview. With this preview it is possible to:

- Change page setup
- Print
- Export to a variety of file formats including pdf and txt file
- Transmit via email

The report can be saved and it is possible to zoom and pan within the document.

Modeling Tips

The paragraph presents some FAQs related to modeling water distribution networks with Bentley WaterCAD V8*i* . Also, please keep in mind that Bentley Systems offers workshops in North America and abroad throughout the year. These workshops cover these modeling topics in depths and many more in a very effective manner. The following modeling tips are presented:

- Modeling a Hydropneumatic Tank
- Modeling a Pumped Groundwater Well
- Modeling Parallel Pipes
- Modeling Pumps in Parallel and Series
- Modeling Hydraulically Close Tanks
- Modeling Fire Hydrants
- Modeling a Connection to an Existing Water Main
- Top Feed/Bottom Gravity Discharge Tank
- Estimating Hydrant Discharge Using Flow Emitters
- Modeling Variable Speed Pumps
- Resolving ‘Unbalanced Network’ Errors

Modeling a Hydropneumatic Tank

Hydropneumatic tanks can be modeled using a regular tank element and converting the tank pressures into equivalent water surface elevations. Based on the elevation differences, the tank's cross-sectional area can then be determined.

For example, consider a hydropneumatic tank that operates between 50 psig and 60 psig. The tank's storage volume is approximately 50 cubic feet.

The tank base elevation is chosen to be equal to the ground elevation, and the pressures are converted into feet of water (1 psi = 2.31 feet). It is apparent that the tank operates between levels of 115.5 feet and 138.6 feet. The difference between the levels is 23.1 feet, which brings us to a needed cross-section of 2.16 square feet.

Modeling a Pumped Groundwater Well

A groundwater well is modeled using a combination of a reservoir and a pump. Set the hydraulic grade line of the reservoir at the static groundwater elevation. The hydraulic grade line can be entered on the reservoir tab of the reservoir editor dialog box, or under the Reservoir Surface Elevation column heading in the Reservoir Report.

Pump curve data can be entered on the Pump Tab of the Pump Editor. The following example will demonstrate how to adjust the manufacturer's pump curve to account for drawdown at higher pumping rates. Drawdown occurs when the well is not able to recharge quickly enough to maintain the static groundwater elevation at high pumping rates.

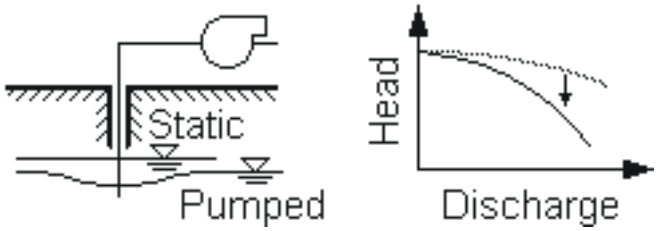


Figure 10-1: Pump Curve Accounting for Drawdown

EXAMPLE:

The pump manufacturer provides the following data in a pump catalog:

Head (ft.)	Discharge (gpm)
1260	0
1180	8300
1030	12400

Based on field conditions and test results, the following drawdown data is known:

Drawdown (ft.)	Discharge (gpm)
40	8300
72	12400

To account for the drawdown, the pump curves should be offset by the difference between the static and pumped groundwater elevations. Subtract the drawdown amount from the pump head, and use these new values for your pump curve head data.

The following adjusted pump curve data is based on the drawdown and the manufacturers pump data.

Head (ft.)	Discharge (gpm)
1260	0
1140	8300
958	12400

Modeling Parallel Pipes

With some water distribution models, parallel pipes are not allowed. This forces you to create an equivalent pipe with the same characteristics.

With this program, however, you can create parallel pipes by drawing the pipes with the same end nodes. To avoid having pipes drawn exactly on top of one another, it is recommended that the pipes have at least one vertex, or bend, inserted into them.

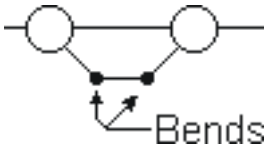


Figure 10-2: Pipe Bends

Modeling Pumps in Parallel and Series

Note: With pumps in series, it is actually more desirable to use a composite pump than to use multiple pumps in the network. When pumps shut off, it is easier to control one pump. Several pumps in series can even cause disconnections by checking if upstream grades are greater than the downstream grade plus the pump heads.

Parallel pumps can be modeled by inserting a pump on different pipes that have the same From and To Nodes. Pumps in series (one pump discharges directly into another pump's intake) can be modeled by having the pumps located on the same pipe. The following figure illustrates this concept:

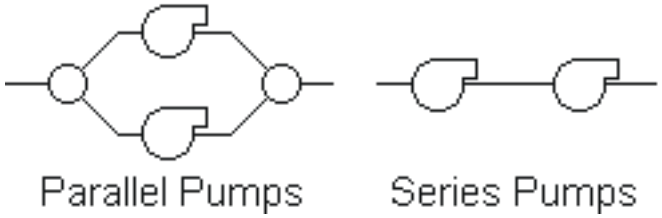


Figure 10-3: Pumps in Parallel and Series

If the pumps are identical, the system may also be modeled as a single, composite pump that has a characteristic curve equivalent to the two individual pumps. For pumps in parallel, the discharge is multiplied by the number of pumps, and used against the same head value. Two pumps in series result in an effective pump with twice the head at the same discharge.

For example, two pumps that can individually operate at 150 gpm at a head of 80 feet connected in parallel will have a combined discharge of 2•150 = 300 gpm at 80 feet. The same two pumps in series would pump 150 gpm at 2•80 = 160 feet of head. This is illustrated as follows:

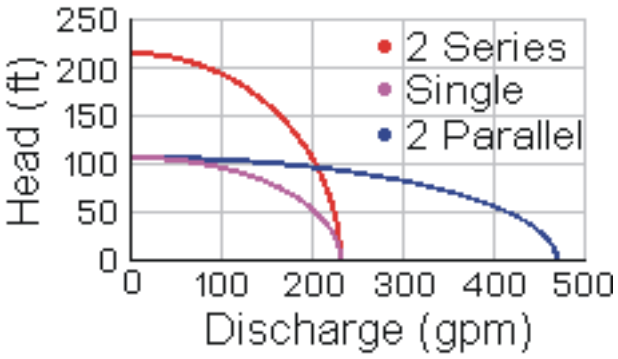


Figure 10-4: Pumps Curves of Pumps in Series and Parallel

Modeling Hydraulically Close Tanks

If tanks are hydraulically close, as in the case of several tanks adjacent to each other, it is better to model these tanks as one composite tank with the equivalent total surface area of the individual tanks.

This process can help to avoid fluctuation that may occur in cases where the tanks are modeled individually. This fluctuation is caused by small differences in flow rates to or from the adjacent tanks, which offset the water surface elevations enough over time to become a significant fluctuation. This results in inaccurate hydraulic grades.

Modeling Fire Hydrants

Fire Hydrant flow can be modeled by using a short, small diameter pipe with large Minor Loss, in accordance with the hydrant's manufacturer. Alternatively, hydrants can be modeled using Flow Emitters.

Modeling a Connection to an Existing Water Main

If you are unable to model an existing system back to the source, but would still like to model a connection to this system, a reservoir and a pump with a three-point pump curve may be used instead. This is shown below:



Figure 10-5: Approximating a Connection to a Water Main with a Pump and a Reservoir

The reservoir simulates the supply of water from the system. The Elevation of the reservoir should be equal to the elevation at the connection point.

The pump and the pump curve will simulate the pressure drops and the available flow from the existing water system. The points for the pump curve are generated using a mathematical formula (given below), and data from a fire flow test. The pipe should be smooth, short and wide. For example, a Roughness of 140, length of 1 foot, and diameter of 48 inches are appropriate numbers.

Please note that it is ALWAYS best to model the entire system back to the source. This method is only an approximation, and may not represent the water system under all flow conditions.

Qr = Qf * [(Hr/Hf)^.54]

Where:	Qr	=	Flow available at the desired fire flow residual pressure
	Qf	=	Flow during test
	Hr	=	Pressure drop to desired residual pressure (Static Pressure minus Chosen Design Pressure)
	Hf	=	Pressure drop during fire flow test (Static Pressure minus Residual Pressure)

EXAMPLE: DETERMINING THE THREE-POINT PUMP CURVE

1. The first point is generated by measuring the static pressure at the hydrant when the flow (Q) is equal to zero.

   Q = 0 gpm
   H = 90psi or 207.9 feet of head (90 * 2.31)

   (2.31 is the conversion factor used to convert psi to feet of head).

2. The engineer chooses a pressure for the second point, and the flow is calculated using the Formula below. The value for Q should lie somewhere between the data collected from the test.

   Q = ?
   H = 55 psi or 127.05 feet (55 * 2.31) (chosen value)

   Formula:

   Qr = Qf * (Hr/Hf)^.54
   Qr = 800 * [((90 - 55) / (90 - 22))^.54]
   Qr = 800 * [(35 / 68)^.54]
   Qr = 800 * [.514^.54]
   Qr = 800 * .69
   Qr = 558

Therefore,

Q = 558 gpm

3. The third point is generated by measuring the flow (Q) at the residual pressure of the hydrant.

   Q = 800 gpm
   H = 22 psi or 50.82 ft. of head (22 * 2.31)

   Pump curve values for this example:

Head (ft.)	Discharge (gpm)
207.9	0
127.05	558
50.82	800

Top Feed/Bottom Gravity Discharge Tank

A tank element in Bentley WaterCAD V8*i* is modeled as a bottom feed tank. Some tanks, however, are fed from the top, which is different hydraulically and should be modeled as such.

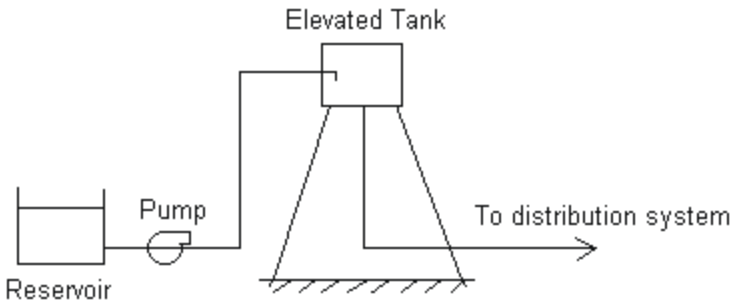


Figure 10-6: Top Feed/Bottom Gravity Tank

To model a top feed tank, start by placing a pressure sustaining valve (PSV) at the end of the tank inlet pipe. Set the elevation of the PSV to the elevation of the inlet to the tank. The pressure setting of the PSV should be set to zero to simulate the pressure at the outfall of the pipe.

Next, connect the downstream end of the PSV to the tank with a short, smooth, large diameter pipe. The pipe must have these properties so that the headloss through it will be minimal.

The tank attributes can be entered normally using the actual diameter and water elevations.

The outlet of the tank can then proceed to the distribution system.

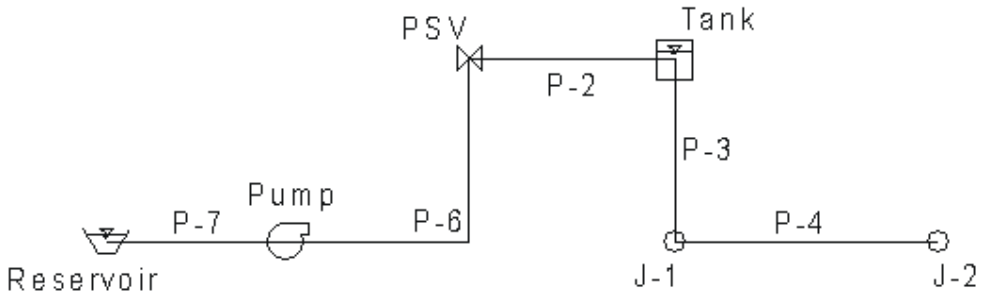


Figure 10-7: Example Layout

Estimating Hydrant Discharge Using Flow Emitters

Another way to model the discharge from a hydrant is to use flow emitters. A flow emitter relates the discharge to pressure immediately upstream of the emitter using:

$$Q = KP^n$$

Where:	Q	=	flow through hydrant (gpm, l/s)
	K	=	overall emitter coefficient (gpm/psi^n, l/s/m^n)
	P	=	pressure upstream of hydrant (psi, m)
	n	=	pressure exponent (0.5 for hydrant outlets)

The pressure exponent, n, is a variable that can be set in the Hydraulic Analysis Options section of the Calculation Options dialog box. The default value is 0.5, which should be used when using flow emitters to model hydrant outlets.

You should be able to model a hydrant as a flow emitter and enter the appropriate value for K. Not all of the energy available immediately upstream of the hydrant is lost, however. Instead, some of the energy is converted into increased velocity head, especially for the smaller (2.5 in, 63 mm) hydrant outlet.

In order to accurately model a hydrant, the model must be given an overall K value, which includes head loss through a hydrant and conversion of pressure head to velocity head. AWWA Standards C502 and C503 govern the allowable pressure drop through a hydrant. For example, the standards state that the 2.5 in. outlet must have a pressure drop less than 2.0 psi (1.46 m) when passing 500 gpm (31.5 l/s).

The energy equation can be written between a pressure gauge immediately upstream of the hydrant and the hydrant outlet:

$$K = \frac{1}{\left(\frac{1}{2gC_F c_F^2} \left(\frac{1}{D_O^4} - \frac{1}{D_P^4} \right) + \frac{1}{k^2} \right)^{1/2}}$$

Where:

v	=	velocity (ft./sec., m/s)
C_F	=	unit conversion factor (2.31 for pressure in psi, 1 for pressure in m)
c_F	=	unit conversion factor (2.44 for flow in gpm, diameter in inches, 0.0785 for flow in l/s, diameter in mm)
g	=	gravitation acceleration (ft./sec.^{2}, m/s^2)
k	=	pressure drop coefficient for hydrant
K	=	overall emitter coefficient
D_o	=	diameter of orifice
D_p	=	diameter of pipe

The difference between K and k is that K includes the terms for conversion of velocity head to pressure head. k is known, but K is the value needed for modeling.

A typical hydrant lateral in North America is 6 in. (150 mm) and typical outlet sizes are 2.5 in. (63 mm) and 4.5 in. (115 mm). Values for k vary from minimum values, which can be back calculated from AWWA standards, to much higher values actually delivered by hydrants. Values for K for a range of k values for 6 in. (150 mm) pipes are given below.

Table 10-4: Emitter K Values for Hydrants

Outlet Nominal (in.)	k gpm, psi	k l/s, m	K gpm/psi^n, l/s/m^n	K l/s, m
2.5	250-600	18-45	150-180	11-14
2-2.5	350-700	26-52	167-185	13-15
4.5	447-720	33-54	380-510	30-40

The coefficients given are based on a 5 ft. (1.5 m) burial depth and a 5.5 in. (140 mm) hydrant barrel. A range of values is given because each manufacturer has a different configuration for hydrant barrels and valving. The lowest value is the minimum AWWA standard.

Modeling Variable Speed Pumps

With Bentley WaterCAD V8*i* , it is possible to model the behavior of variable speed pumps (VSP), whether they are controlled by variable frequency drives, hydraulic couplings or some other variable speed drive. Workarounds that were previously used, such as pumping through a pressure-reducing valve, are no longer needed.

The parameter that is used to adjust pump speeds is the relative speed. The relative speed is the ratio of the pump's actual speed to some reference speed. The reference speed generally used is the full speed of the motor. For example, if the pump speed is 1558 rpm while the motor is a 1750-rpm motor, the relative speed is 0.89. This relative speed is used with the pump affinity laws to adjust the pump head characteristic curve to model the pump.

If only a steady state run is being made and the pump relative speed is known, the speed of the variable speed pump can be set in the General tab of the pump dialog box. However, if the conditions that control the pump are not known at the start or an EPS run is being made, then variable speed behavior must be described in more detail.

Modeling variable speed pumps includes:

- Types of Variable Speed Pumps on page 10-1015
- Pattern Based on page 10-1015
- Fixed Head on page 10-1015
- Controls with Fixed Head Operation on page 10-1016

Types of Variable Speed Pumps

The behavior of the VSP is set under the VSP tab within the pump dialog box. There are two ways to control a variable speed pump. One is to provide a Pattern of pump relative speeds. This is best used for cases where you are trying to model some past event where the pump speeds are known exactly or where the pump is not being controlled by some target head. This would be the case where human operators set speed based on a combination of time of day, weather and other factors.

The second type of control is Fixed Head control, where the pump speed is adjusted to maintain a head somewhere in the system. For water distribution pumping into a pressure zone with no storage, this is usually some pressure sensor on the downstream side of the pump. For wastewater pumping, the pump may be operated to maintain a constant wet well level on the suction side (i.e., flow matching).

To indicate that a pump is behaving as a VSP, first check the box next to **Variable Speed Pump?** at the top of the VSP tab. This will change the remaining boxes on the tab from gray to white.

Pattern Based

If you want to provide the actual pump relative speeds, **Pattern Based** should be selected from the VSP Type menu. The default pattern is Fixed, which corresponds to constant speed performance at a speed from the General tab.

Usually, you will want to specify a series of pump relative speeds. To do this, click the **Ellipsis (…)** button next to Pump Speed Pattern. This will open the Pattern Manager dialog box. Click the **Add** button, and the Pattern Editor dialog box will appear. From this dialog box, you can assign a label (name) to the new Pattern and complete the series of multipliers (i.e., relative speeds) versus time. Clicking **OK** twice will return you to the VSP tab.

A difficulty in using Pattern Based speeds is that the pattern that would work well for one scenario may not work well for other scenarios. For example, tanks will run dry or fill and shut off for a slightly different scenario than the one for which the pattern was created.

Fixed Head

Fixed head control is achieved by selecting Fixed Head from the VSP Type? menu. Once Fixed Head is selected, you must describe how the control is implemented.

You must identify a node that controls the pump. This is the node where some type of pressure or water level sensor is located. This can be done by:

- Using the menu and picking the node from the list

- Clicking the **Ellipsis (…)** button and using the Select Element dialog box.
- Clicking the Select From Drawing button and picking the node from the drawing.

In selecting the control node, you must choose a node that is actually controlled by the VSP. For example, the selected node must be in the same pressure zone (i.e., one that is not separated from the pump by another pump or PRV) and should not have a tank directly between the node and the pump.

You must then select the head to be maintained at that node. If the node selected for control is a tank, then the Target Head is set as the initial head in the tank. If a junction node is selected, the head must be a feasible head. If a physically infeasible head is given, the problem may not be solved or some unrealistic flow may be forced to meet this head (e.g., backward flow through pump).

You also have the option of setting the maximum relative speed of the pump, which would usually correspond to the rated speed of the motor. The default value for this is 1.0. You can have the model ignore this limit by placing a large value in the field for maximum speed.

Note: If the suction head is greater than target head, then pump head will be reported as zero and the speed value will not be meaningful.

Controls with Fixed Head Operation

Note: There should only be a single VSP serving a given pressure zone. If more than one VSP tries to use the same node as a control node, then the model will issue an error message and not solve. If you try to use two different nodes that are very close hydraulically, an error will also result.

When the relative pump speed reaches maximum speed (usually 1.0), the model treats the pump essentially as a constant speed pump. In the case of pumps controlled by a junction node, when the conditions warrant, the pump will once again behave as a VSP.

However, for pumps controlled by tanks, the pump will run at a maximum speed for the remainder of the EPS run, once they reach maximum speed. To get the pump to switch back to variable speed operation, you need to insert a control statement that switches the pump back to variable speed. Consider the example below:

PMP-1 tries to maintain 280 ft. discharge at node T-1 on the discharge side of the pump, but pump (PMP-1) switches to full speed when the flow is so great that it cannot maintain 280 ft. In that case, the water level drops below 280 ft. As demand decreases, the level increases until it reaches 280 ft., at which time variable speed operation begins again. To make this occur in the model, you must use a logical control to restore variable speed operation:

```
IF (HGL T-1 >= 280 ft) THEN (PMP-1 = ON)
```

Parallel VSPs

Variable speed pumps can also be modeled in parallel. If you use the Fixed Head pump type, both parallel VSPs must be set to the same target node. The program will attempt to meet the fixed head requirements you set using only one of the pumps. If the fixed head cannot be met with only one of the pumps, the second pump will be turned on, and the relative speed settings of the pumps will be adjusted to compensate.

Variable speed pumps (VSPs) can be modeled in parallel. This allows you to model multiple VSPs operated at the same speed at one pump station. To model this, a VSP is chosen as a “lead VSP”, which will be the primary pump to deliver the target head. If the lead VSP cannot deliver the target head while operating at maximum speed, then the second VSP will be triggered on and the VSP calculation will determine the common speed for both VSPs. If the target head cannot be delivered while operating both VSPs at the maximum speed, then another VSP will be triggered on until the target head is met with all the available VSPs.

All VSPs that are turned on are operated at the same speed. VSPs are to be turned off if they are not required due to a change in demand. If all standby VSPs are running at the maximum speed, but still cannot deliver the target head, the VSPs are translated into fixed speed pumps.

To correctly apply the VSP feature to multiple variable speed pumps in parallel, the following criteria must be met:

1. Parallel VSPs must be controlled by the same target node;
2. Parallel VSPs must be controlled by the same target head;
3. Parallel VSPs must have the same maximum relative speed factors;
4. Parallel VSPs must be identical, namely the same pump curve.
5. Parallel VSPs must share common upstream and downstream junctions within 3 nodes (inclusive) of the pumps in order for them to be recognized as parallel VSPs.

   If there are more than 3 nodes between the pumps and their common node, upstream and downstream, the software will treat them as separate VSPs. Since separate VSPs cannot target the same control node, this will result in an error message.

VSP Controlled by Discharge Side Tank

The improvement allows users to choose a tank at the downstream side of a pump as the control target. Once a user selects a tank as the control node for a VSP, the control target head is set to the initial tank head by default. The VSP algorithm will calculate the required relative pump speed to maintain the tank level. If the tank level drops below the target level, the VSP will be forced to increase the speed, up to the maximum allowable speed as specified, to meet the target tank level. If the tank level is greater than the target level, the VSP speed will be reduced or shut off to permit the tank supply system demand and thus the tank level can be gradually lowered to the target level.

To set up a discharge side tank as the VSP control node:

1. Click on a VSP or VPSB.
2. In the Properties editor, set the attribute Is Variable Speed pump? to True.
3. Set VSP Type as Fixed Head
4. Choose a desired discharge side tank as Control Node
5. Specify the maximum relative speed factor and set Is Suction Side Variable Speed Pump to False

Note: When the target level is missed due to either too high demand or too much inflow into the wet well, the VSP will be operating at the fixed speed until the target level can be reestablished, however, the reestablished target level may not be exactly the same as the initial target head. This is because the VSP is forced back by using the given time step, the pump is operated as a fixed speed pump to move the amount of water within one time step, so that the level cannot be exact unless the time step is small enough to ensure the exact amount of water is moved out the tank to maintain the exact target. The smaller the time step, the closer it will be to returning to the target.

VSP Controlled by Suction Side Tank

Similar to the function of a VSP controlled by a discharge side tank, a vsp can also be controlled by a tank at the upstream of pump, that is the suction side of a pump. This is the typical use case for a sewer forcemain sub-system, where a wet well (essentially a tank) is usually located at the suction side of a pump. In this case, the control target is to maintain a fixed water level at the wet well. When a VSP is installed at the down-stream side of a wet well to pump the flow out of the well and also to maintain a fixed wet well water level, WaterCAD V8*i* can be used to model the control scenario.

Unlike the vsp controlled by discharge side tank, when the wet well level is below the target level, suction side controlled vsp will slow down in speed to allow the water level to increase to the target level. When the wet well water level is above the target level, a vsp will speed up to move the flow out of well in order to reduce the water level at the wet well.

The workflow is the same as the VSP controlled by a discharge side tank, except that the user needs to set the attribute of Is Suction Side Variable Speed Pump to True in the property grid.

Note: When the target level is missed due to either too high demand or too much inflow into the wet well, the VSP will be operating at the fixed speed until the target level can be reestablished, however, the reestablished target level may not be exactly the same as the initial target head. This is because the VSP is forced back by using the given time step, the pump is operated as a fixed speed pump to move the amount of water within one time step, so that the level cannot be exact unless the time step is small enough to ensure the exact amount of water is moved out the tank to maintain the exact target. The smaller the time step, the closer it will be to returning to the target.

Fixed Flow VSP

Fixed flow VSP enables the user to model a pump that is controlled to deliver a desired amount of flow. This can be a typical control case when a pump is supplying water to an "open" system where a tank is located in the downstream distribution system. It is unlikely that a pump is expected to supply the fixed flow to a "closed" system where no tank is located at the downstream of a pump.

WaterCAD V8*i* facilitates the fixed flow VSP modeling. It automatically calculates the required pump speed, up to the maximum relative speed factor, to move the required flow through a pump. Multiple vsps can be in parallel and expected to deliver different target flows. To apply this feature, follow the steps as below.

1. Click on a VSP.
2. Set the attribute Is Variable Speed pump? to True.
3. Set VSP Type as Fixed Flow
4. Specify the maximum relative speed factor
5. Specify the Target Flow for the vsp

In the case of a VSPB, the target flow will be evenly divided among all the lead and lag VSPs.

Note: **In some cases, you may encounter a high-frequency oscillation effect when a tank is used as the control node. If this occurs, it is suggested that you use a node near the tank as the control node, rather than the tank itself.**

Resolving ‘Unbalanced Network’ Errors

For complex systems (e.g. with many pipes and a lot of controls) it can take more than the standard 40 iterations to converge on a good solution. In cases like this, sometimes increasing the number of **Trials** in the **Calculation Options** will allow the model to converge to a good solution. However we often find that models that give the ‘unbalanced network’ error have data entry errors (high friction coefficient, etc.) so it is always a good idea to check your data input carefully."

Pipe Renewal Planner

Pipe Renewal Planner provides the user with a tool to calculate a weighted score for each pipe based on whatever aspects the user chooses. Scoring pipes is highly system specific depending on the issues in that system and the availability of data. Pipe Renewal Planner can include any aspect that can be entered for a pipe or calculated for the pipe.

Scores that can be calculated for a pipe include:

1. Capacity
2. Criticality
3. Projected pipe breaks

Scores that can be based on properties include standard WaterCAD V8*i* properties such as:

1. Year installed
2. Material
3. Zone

Or User Data Extensions such as:

1. Type of surface activity
2. Depth of cover
3. Relation to water quality complaints

Each of the properties used above (e.g. capacity, material, and cover) is referred to as an aspect. The first set of aspects are calculated in special routines and are referred to as "Predefined Aspects" since there are WaterCAD V8*i* analyses that are used to determine the scores. See the Help for each of those individual aspects.

The overall process for determining the "Pipe Score", which is the final result of this analysis, is:

1. Build model with sufficient information to calculate aspect of interest
2. Optionally run capacity, criticality and pipe break analysis
3. Start Pipe Renewal Planner by selecting Analysis > Pipe Renewal Planner or picking the Pipe Renewal Planner button.
4. Pick the New button to create a new Pipe Renewal analysis
5. Select aspects to be used and weights for each
6. Set up scoring to convert raw score/property values into individual aspect scores
7. Compute Pipe Renewal Pipe Scores
8. Review results

Each of these steps is described in more detail below.

Pipe Renewal Planner - methods used

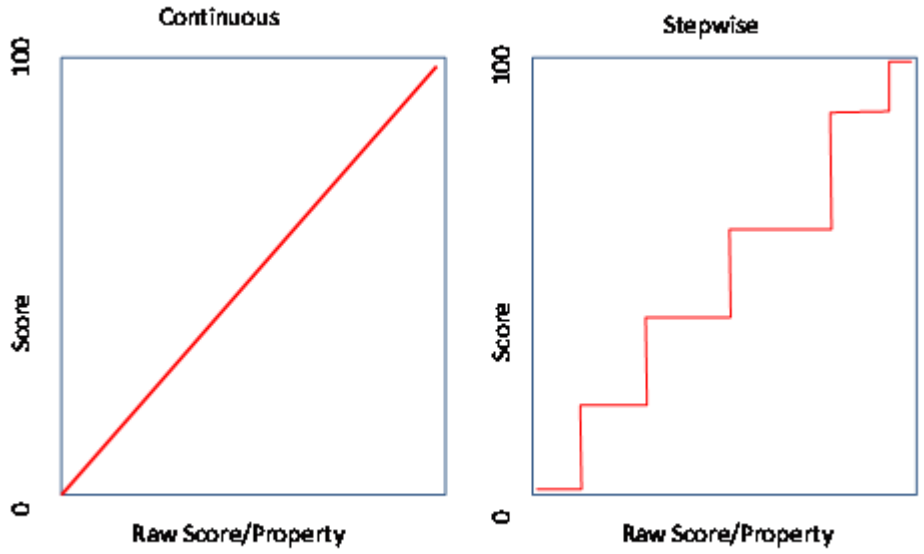
The result of the Pipe Renewal Planner analysis is a pipe score for each pipe. This is calculated for the j-th pipe using

$$\text{Score (j)} = \Sigma\, w_i R_{ij}$$

Where w_i is the weight for the i-th aspect and R_{ij} is the score for the j-th pipe for the i-th aspect.

The intent is that the individual scores (R values) are on a scale of 0 to 100 (100 being the worst). The w's should add up to 1 so that the overall score will also be on a 0 to 100 scale.

The scores for the individual aspects are determined on a continuous or a stepwise scale as appropriate for that type of aspect.



Aspects such as pipe break and criticality use the continuous function while user defined properties such as year installed and material use the stepwise function. The horizontal axis is described by some raw values such as pipe break rate in breaks/year/mile or maximum velocity (ft/sec) in pipe during fires or year installed.

Pipe Break: For the pipe break aspect, the user should run the Pipe Break Analysis to calculate the projected break rate for each pipe. The individual pipe break score is calculated as:

$$R_{ij} = \left(\frac{break_j}{breakmax} \right)$$

Where $break_j$ = break rate in j-th pipe, and breakmax = maximum break rate in all pipes.

Criticality: The criticality score is based on the shortfall in meeting demand as calculated by the WaterCAD V8*i* criticality analysis. Criticality may be based on taking an individual pipe element out of service or more accurately in taking a distribution segment out of the system (see criticality help for more discussion on this as well as details of calculating criticality below). The score for criticality is:

$$R_{ij} = 100\left(\frac{criticality_j}{criticalitymax}\right)$$

Where criticality is the shortfall due to an outage of the j-th pipe and criticalitymax is the greatest shortfall from any pipe.

Capacity (fire flow): Assigning fire flow scores to a pipe is somewhat more difficult in that fire flows are node, not pipe, properties. The goal is to identify which pipes serve as bottlenecks in the system. These are pipes which have high velocity when a downstream node fails the meet needed fire flow. The determination of a shortcoming in capacity is defined as the maximum difference between the target velocity and actual velocity for the worst fire flow event for each pipe. The user defines a velocity that would make a pipe a candidate for being a bottleneck (say 5 ft/s).

For each pipe, the raw score is defined as:

$r_j=max[v-v_t]$

Where v = velocity, ft/s, v_t = target velocity, ft/s

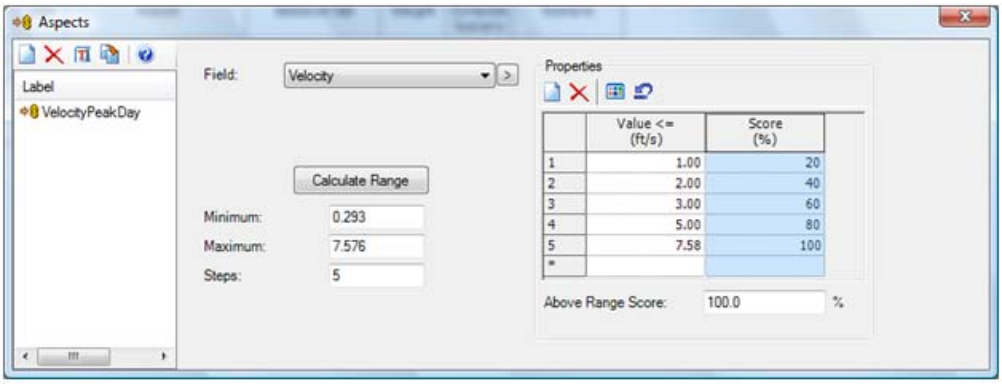
The scaled score for pipe j would be:

$$R_{ij} = 100\left(\frac{r_j}{rmax}\right)$$

Where rmax is the amount the velocity exceeds the target at the pipe with the highest velocity.

It may be necessary to eliminate small pipes (e.g. 2 in. pipes) from this calculation since they are not expected to carry fire flow. It may also be necessary to eliminate nodes from the fire flow analysis in areas where fire flows are not to be provided. Selecting the target velocity also involves some judgment in that too low of a value will point out some pipes that normally have a high velocity as being bottlenecks and too high of a will mean that virtually no pipes will have a non-zero value for R_{ij}.

Discrete aspect: In the case of aspects whose score is based on some pipe property, the user selects some function and manually enters the function using a table such as shown below:



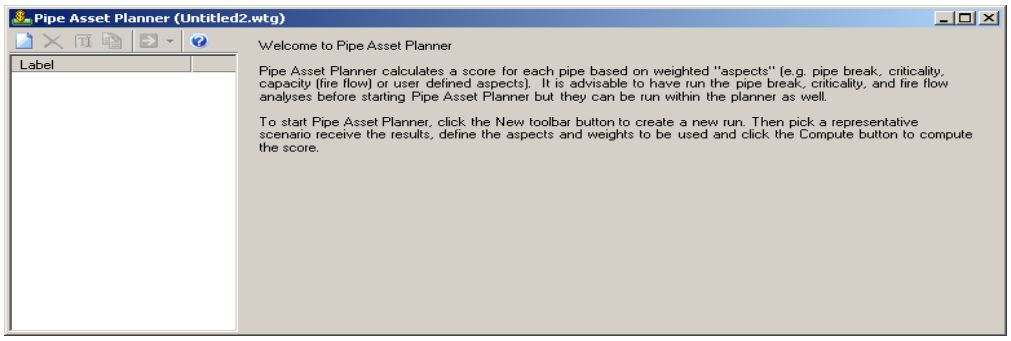
Using the Pipe Renewal Planner

Before using Pipe Renewal Planner, the user needs to identify which aspects will be used in scoring pipes and which properties are going to be used as a basis for calculating the aspect scores.

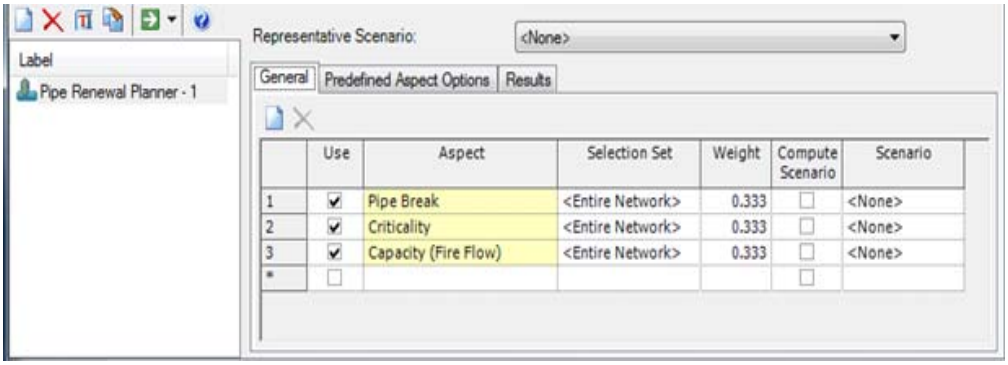
(It may be necessary to define new properties in User Data Extensions and import values for properties from external data sources using ModelBuilder or copy/paste features. In order to import values, it is essential that there exist a common key field shared by the WaterCAD V8*i* model and the external data source.)

Calculation of raw scores for aspects such as capacity (fire flow) and criticality (shortfall) can be time consuming such that it may be advisable to have already run these analyses before starting the Pipe Renewal Planner and noting which scenario was used. However, if any properties are changed that may affect scores, it may be necessary to rerun the scenario from within Pipe Renewal Planner.

The user can start Pipe Renewal Planner by selecting Analysis > Pipe Renewal Planner or picking the Pipe Renewal Planner button. This opens the welcome dialog if no analyses have already been run.



Select the New button on top of the left pane to create a new analysis. It opens with the following default values:



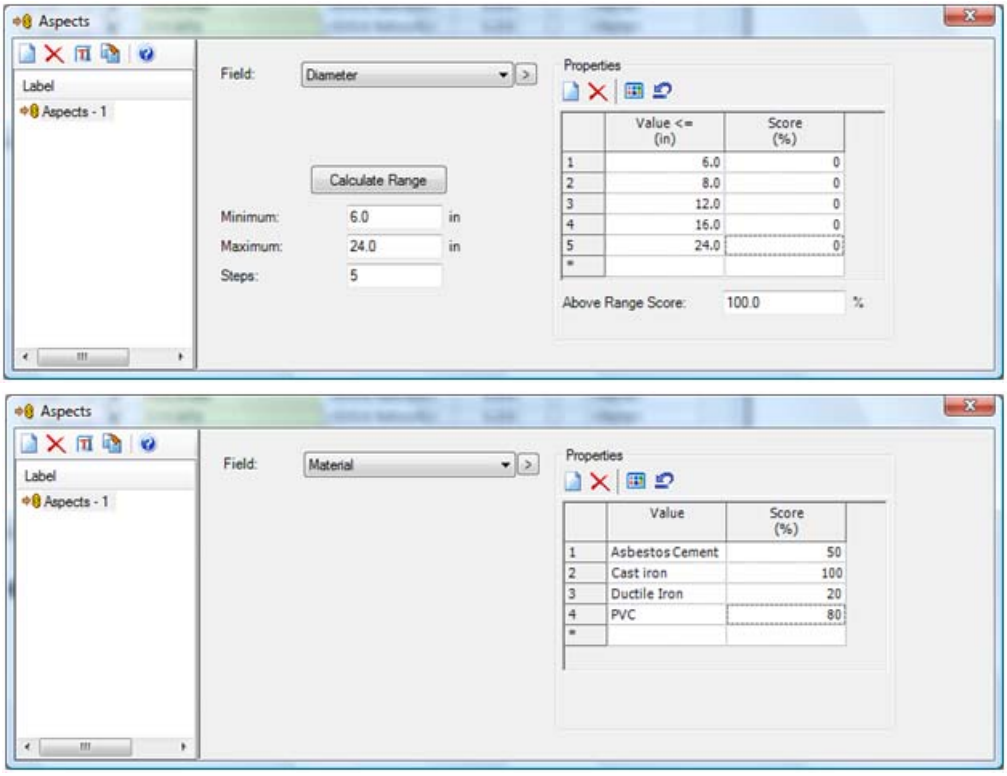
The user can rename the analysis by selecting the third button over the left pane.

The user should select the Representative Scenario which need not necessarily be the current scenario. This scenario will be used as the source of property values and the location to save results except for those places where another scenario is explicitly called out.

General Tab: In the General tab in the right pane the user can create new aspects or delete aspects using the buttons on top of the dialog.

The Use button determines which aspects are to be included in the pipe score calculation as indicated by the check.

Under the Aspect column, the user can define new aspects. The default Aspects - Pipe Break, Criticality and Capacity (Fire Flow) -- are automatically included in the list although they can be deleted. To create a new Aspect, click inside a blank cell in the Aspect column and select the ellipse (…) button. This will open the dialog below where the scoring for the new aspect can be defined by first selecting the New button, then naming the Aspect.



The user then picks which field is to be used as the basis for this Aspect, initializes the values and sets the scores. If the property is a numerical value, then the value in the Value column is the upper limit of the range (above) while if the property is text, the list of possible text values is displayed (below).

The Selection Set column determines whether the Pipe Renewal Planner will be run for the entire network (default) or some previously defined selection set of pipes.

The Weight column is the place where the user defines the weights assigned to each aspect. Ideally, the weights should add up to 1 but the user may use some other weighting system.

The Compute Scenario box when checked means that WaterCAD V8*i* will recalculate the indicated scenario when it calculates the Pipe Score. If unchecked, the Pipe Renewal Planner will use the most recent results from that scenario.

The Scenario column indicates which scenario is to be used to calculate the raw score for that Aspect. It is important that the user pick the correct type of scenario. For example, if the Aspect is criticality, the scenario selected should be one containing the results of a criticality run.

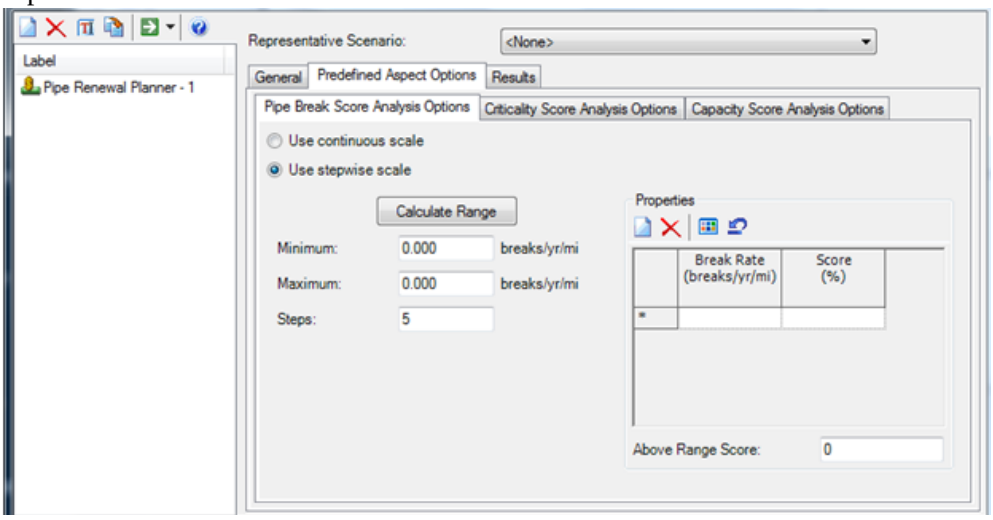
Predefined Aspects Tab: The Predefined Aspects Options tab gives the user additional control over the handling of the three predefined aspects - Pipe breaks, Criticality and Capacity. In each of those sub-tabs, the user can decide whether to calculate the score on a continuous scale (default) or set up some stepwise function to convert the raw score into a scaled score to the overall pipe score. The user indicates this by selecting:

Use continuous scale

Or

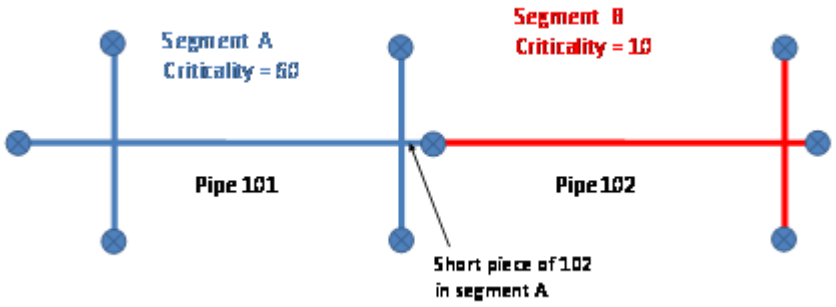
Use Stepwise scale

If the user selects the continuous scale, then no additional information is necessary. If the user selects the stepwise scale, then he must define the scale as done for other aspects.



The criticality and capacity score provide the user with additional capability to specify some additional options.

In calculating the criticality score, the shortfall may be calculated based on distribution segments rather than pipe elements. (Segments are the minimum portion of the system that can be isolated by valving. See help topic on segments.) There is not a one-to-one association between segments and pipes. A pipe may be made up of several segments depending on valving. The user has the ability to control how the segment shortfall is transformed into pipe shortfall. In the figure below, there are two segments than overlap pipe 102-a short one and a long one.



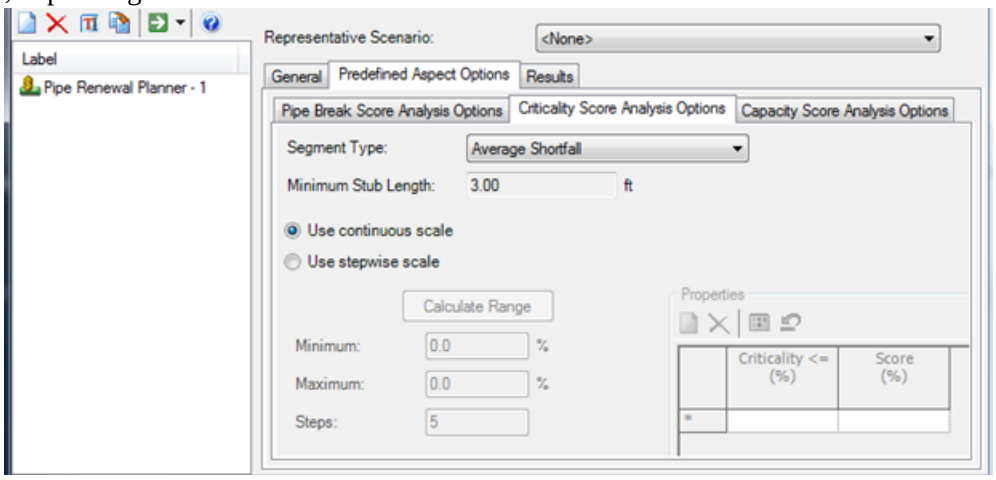
The user has three ways to handle multiple segments:

1. Use the average shortfall weighted by the length of each segment (default)
2. Ignore small segments below a certain size (called minimum stub length)
3. Use the shortfall corresponding to the worst segment in the pipe

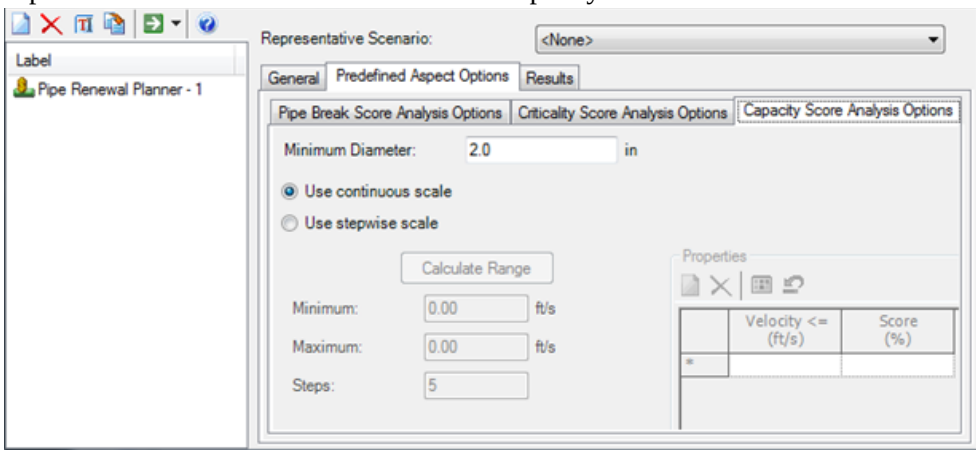
For the example above, suppose pipe 102 is 200 ft long and 195 ft are in Segment B (criticality = 10) while the remaining 5 ft are in segment A (criticality = 60). The corresponding scores would be:

1. (195/200)10 + (5/200)60 = 11.25
2. 10 (if minimum stub length is greater than 5 ft)
3. 60

, depending on the user's choice.



The capacity score as described in the "Pipe Renewal Planner - methods used" topic, is based on the maximum extent that the velocity exceeds the target velocity in a fire flow analysis. Because some pipes are small and not intended for fire flow, those pipes can be excluded from the analysis using the minimum diameter value (default = 2 in). Pipes that small or smaller will not have a capacity score calculated for them.



The velocity used in the calculate is the velocity that will occur when the residual pressure meets the required residual. For pipes with large capacity, this value will be much greater than the needed fire flow. If the user wants the velocity to simply meet the needed fire flow, then the "Fire Flow (Upper Limit)" parameter in the fire flow alternative should be set to a value just slightly above the needed fire flow.

Results Tab

To run the pipe scoring calculation, the user would pick the green compute button on the top of the left pane. To simply validate that the calculation is runable, pick the small drop down arrow next to the compute button and pick Validate.

Once the run is complete, a summary results table is displayed with the following columns:

- Pipe ID and Label
- **Pipe Score** - The overall pipe score which is a weighted sum of the individual aspect scores. A higher value indicates a pipe with potential problems in need of repair, rehabilitation, replacement or some other remedial action.

  Scores are generally presented on a 0 to 100 scale unless the user has set up some different scaling. This is followed by summaries for each of the aspects used:
- **Raw score pipe break (breaks/yr/mi)** -The result for the pipe break analysis.
- **Score Pipe Break** - The score for the pipe break aspect on a 0-100 scale.
- **Raw score criticality** - The percent shortfall for that pipe being taken out of service as calculated in the associated criticality scenario.
- **Score Criticality** - The score for criticality on a 0 to 100 scale.
- **Raw Score Capacity** - The maximum velocity corresponding to the fire flow analysis scenario.
- **Score Capacity** - The score for capacity on a 0 to 100 scale.

  The next several columns contain a pair of columns for each user created aspect if there are any. The first column is the raw score for the property while the second is the score on a 0 to100 scale.
- The final columns contain the diameter, length, material and installation year for each pipe.

Pipe Break Analysis

Pipe Break Analysis enables the user to calculate a projected pipe break rate and some auxiliary costs based on historical pipe break data. These values can be viewed directly or can be used as part of Pipe Renewal Planner to determine the overall condition score for a pipe.

The primary input information used in the pipe break analysis is a history of pipe breaks. In order to assign a historical pipe breaks to a pipe in the model, it is essential that the break history and water model share some common key fields.

The pipe break history can be in one of two formats

1. A table with one record per pipe consisting of pipe id in one column and the number of breaks in another column (Pipe Break Table - Type 1)
2. A table with one record per pipe break (failure) with the pipe id serving as one of the fields in each record (Failure History - Type 2)

The user can also create the first type of input table by manually entering the historical number of breaks at each pipe.

It is the user's responsibility to assign breaks to pipes and screen the data to eliminate historical breaks that are not to be part of the analysis such as service line or hydrant breaks or breaks caused by contact with construction equipment.

The break history is converted into break rates for individual pipes using

```
Individual break rate (break/yr/mi) = No. of breaks /
[(length of history)(length of pipe)]
```

The user can also calculate the break rate for the group of pipes that a pipe belongs to using:

```
Group break rate (break/yr/mi) = No. of breaks/[length
of history)(length of pipe in group)]
```

The projected break rate is a weighted sum between the individual and group break rate according to:

```
Projected break rate = a (Individual break rate) + (1-
a) (Group break rate)
```

The user controls the value of 'a' to be used.

The overall steps in conducting a pipe break analysis consists of

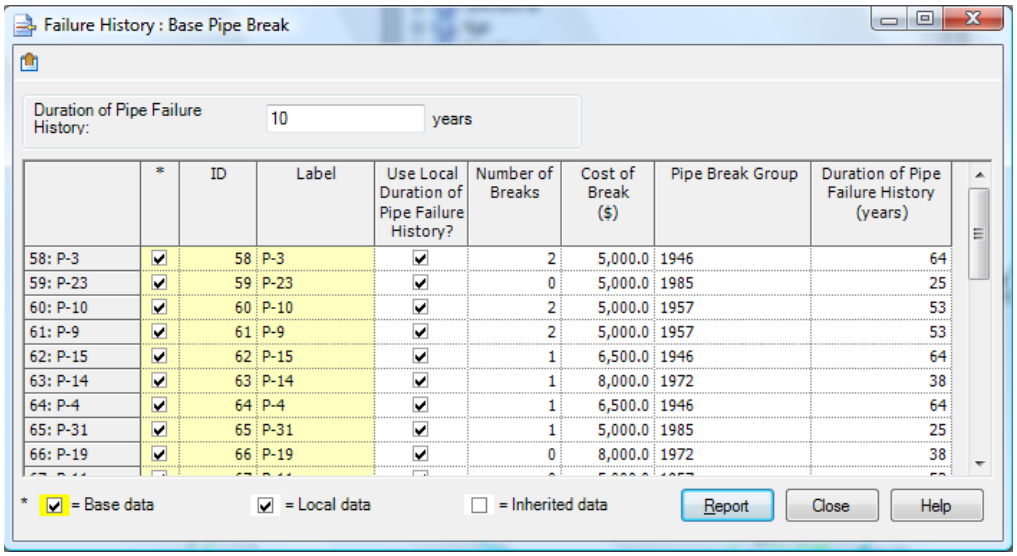
1. Creating a model (preferably an all-pipes model or a skeletonized model)
2. Setting up scenario with a pipe break alternative
3. Starting Pipe Break Analysis
4. Importing/creating pipe break history data
5. Creating pipe break groups (if group feature is used)
6. Setting other analysis options (e.g. individual vs. group rates, analysis period)
7. Run break projection
8. View results
9. Use results in Pipe Renewal Planner

All WaterCAD V8*i* symbology/visualization tools can be used to review results including property grid, Flex Tables, color coding, and annotation.

Running Pipe Break Analysis

The user should first create a WaterCAD V8*i* model and assemble pipe break history data (see Pipe Break History Import below).

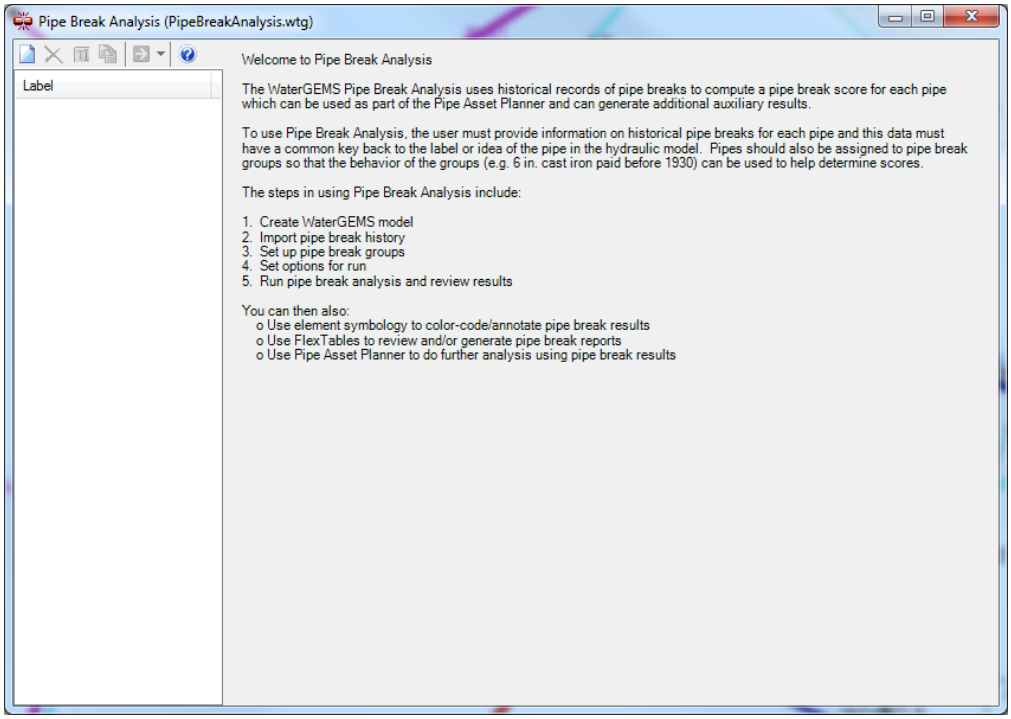
Decide on which scenario and Failure History Alternative, the pipe break analysis will use. Review any data that may be included in the failure history. Failure history data may be entered in the Failure History Alternative or in the Pipe Break Manager. The only value that must be entered here is the "Duration of Pipe Failure History" which will be used as the global default if the user does not override it with a "Local" value.



	*	ID	Label	Use Local Duration of Pipe Failure History?	Number of Breaks	Cost of Break ($)	Pipe Break Group	Duration of Pipe Failure History (years)
58: P-3	☑	58	P-3	☑	2	5,000.0	1946	64
59: P-23	☑	59	P-23	☑	0	5,000.0	1985	25
60: P-10	☑	60	P-10	☑	2	5,000.0	1957	53
61: P-9	☑	61	P-9	☑	2	5,000.0	1957	53
62: P-15	☑	62	P-15	☑	1	6,500.0	1946	64
63: P-14	☑	63	P-14	☑	1	8,000.0	1972	38
64: P-4	☑	64	P-4	☑	1	6,500.0	1946	64
65: P-31	☑	65	P-31	☑	1	5,000.0	1985	25
66: P-19	☑	66	P-19	☑	0	8,000.0	1972	38

Start pipe break history by selecting Analysis > Pipe Break Analysis (or selecting pipe break button).

If no analyses have been run, a Welcome dialog will be displayed with some directions.



If analyses have already been created, they will be displayed in the left pane.

To start creating an analysis, select the New button on top of the left pane.

On creating a new analysis, the user will see the table below which lists all the pipes in the model and enables the user to enter individual pipe break data manually. It also provides the user with the ability to select which scenario the pipe break analysis will reference. This scenario is used for pipe properties and active topology on input and is the scenario where pipe break results are saved.

Pipe Breaks Tab

Usually the user will want to import the pipe break history from an external data source. To do this, it is essential that there exist a common key field in both the pipe break history and the model. If pipe break data are only available by street address, they need to be associated with a pipe asset before they can be imported.

There are two formats from which pipe break data can be imported.

Pipe Break Table (Type 1): For each pipe there is a single record with the pipe id and the number of breaks in the pipe history. If the length of the pipe history is different than the default specified in the "Duration of Pipe Failure History" in the Failure History Alternative, this value can also be specified at this time. The table should look like this:

Table 10-5: Pipe Break Table Example

Pipe ID	Breaks	History, yr
137	3	10
219	0	10
22	1	10

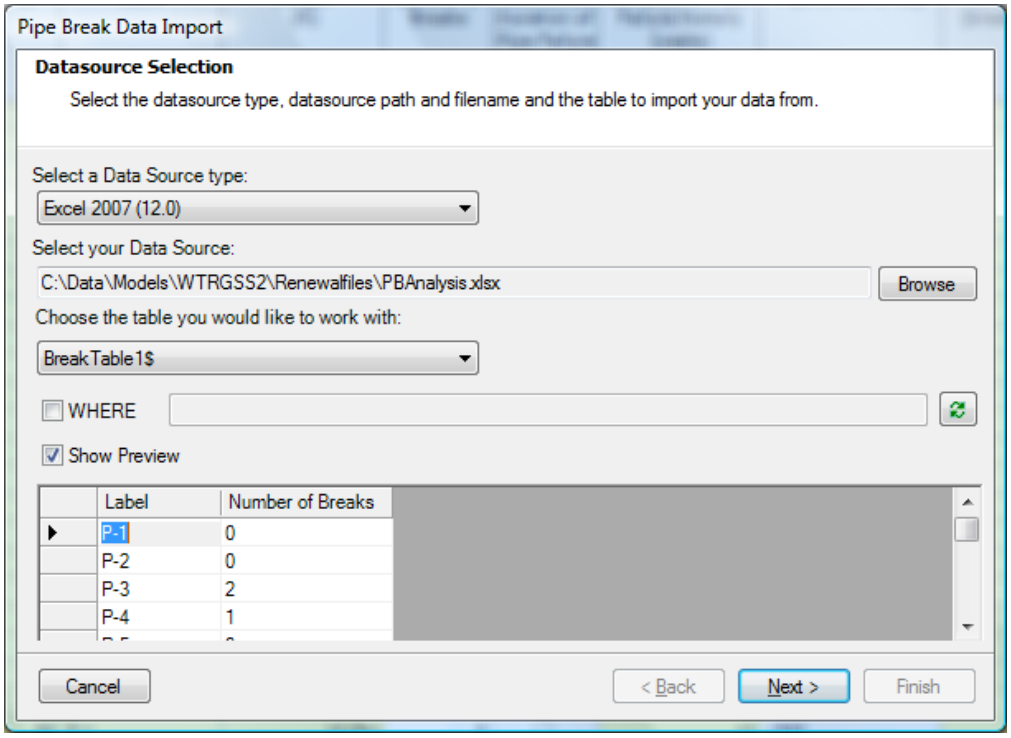
Failure History (Type 2): For each pipe break event, there is a single record. The only value used is the Pipe ID and the import routine counts the number of records with that Pipe ID and sums them to determine the number of breaks. There may be a great deal of additional information available in the break record as show below but that information is not imported.

Table 10-6: Failure history Example

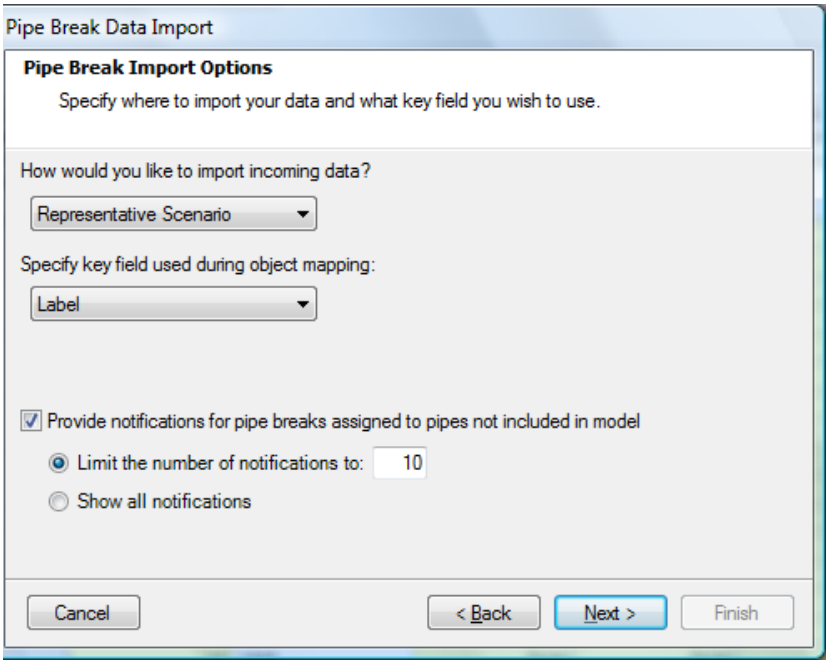
Pipe ID	Date of Break	Type of Break	Duration (hr)	Address
137	22 Sept 2008	Circumferential	2	3 South St.
137	8 Apr 2001	Longitudinal	3	101 Main St.
22	2 May 2004	Corrosion Hole	5	17 East St.
137	5 Oct 2000	Unknown	1	7 Adams Ave.

To start the import of either type of table, select the button [show button] at the top left corner of the right pane. This will open a Data Source Selection dialog where the user will identify the type of data source, the data file, and the table within the data file.

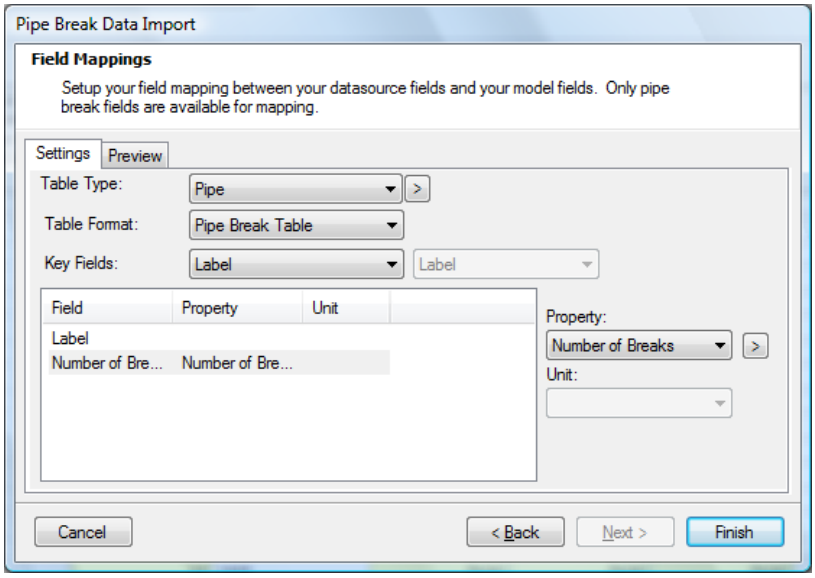
Pipe Break History Import



The user then selects options as to which scenario the data will be imported into, which field in the model will be matched to the key field in the data source and whether the user wants to be notified if there are pipes in the data source that are not in the model (the user can limit the number of these warnings so as not to get a large number initially as the data are being cleaned up).



The user must name the type of table (default pipe) and the key field in the data source, and then map individual fields in the data source to properties in the model. The Table format entry is where the user indicates if the data are a Pipe Break Table (Type 1) or Failure History (Type 2).



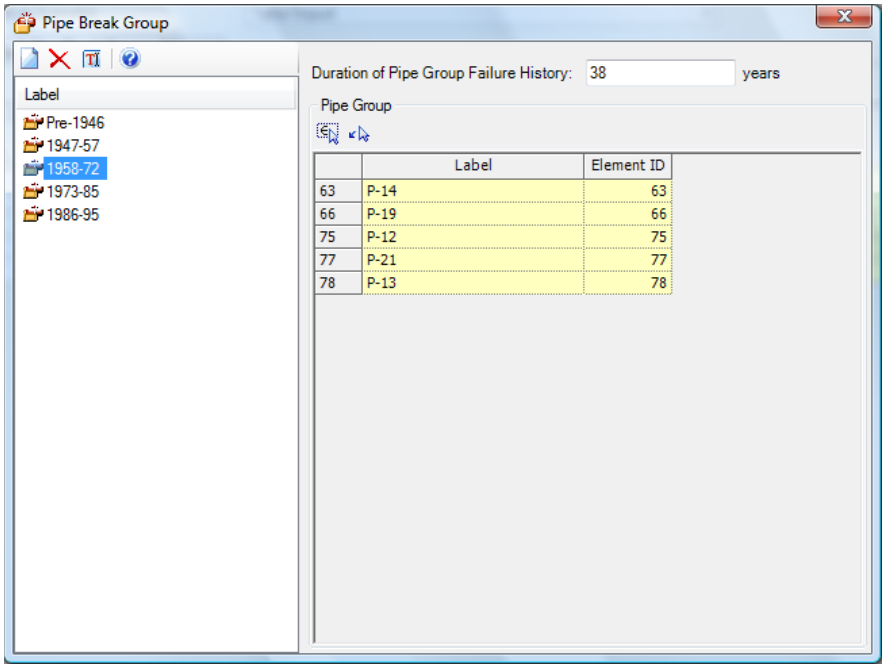
Once the fields from the source file have been mapped to properties of the model, the user selects Finish. A brief report indicates the number of pipes which have been updated with break data.

Pipe Break Groups Tab

The pipe break groups tab enables the user to create and name pipe break groups. The dialog to create groups is started from the button on the top left of the right pane.

This opens up the Pipe Break Group dialog where the user can add pipes to a group by either

1. Picking a previously created selection set using the selection set button.
2. Picking pipes individually from the drawing using the select from drawing button.

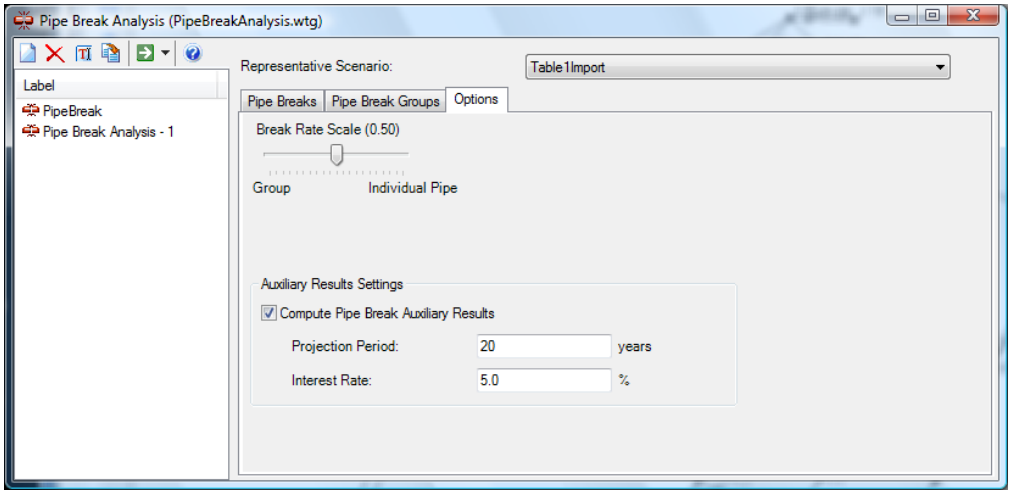


The user can also create and modify Pipe Break Groups by selecting Components > Pipe Break Groups.

The assumption is that pipes in a group have similar properties with respect to pipe breakage. These properties would include similar age, material, laying condition and loading and period of break records. It is usually best to create selection sets of such pipes before starting the pipe break analysis. Name the group with a label that reflects the pipes in the group. If a pipe is not assigned to a group, its individual break rate will be used as the scaled break rate.

Options Tab

In the options tab, the user can select the extent to which the overall scaled pipe break rate for the pipe is based on the individual pipe's history (a value near 1) or the group's history (a value near 0). Moving the slider to the left, lowers 'a' and increases the importance of the group while moving it to the right increases 'a' and decreases the importance of the group.

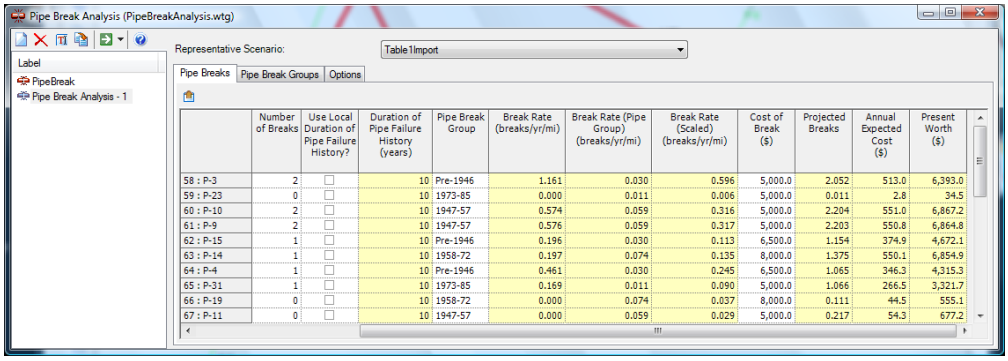


The auxiliary results settings controls optional calculations such as the projected number of breaks and present worth of break costs. These calculations will only be carried out when the "Compute Pipe Break Auxiliary Results" box is checked.

The projection period and interest rate are used in projecting breaks and economic parameters.

Results

To obtain results from the Pipe Break Analysis, select the green compute button on top of the left pane. (To avoid confusion, it is best that the Current scenario be the Representative scenario because the results are stored with the Representative scenario.)



The results that are calculated by the Pipe Break Analysis include:

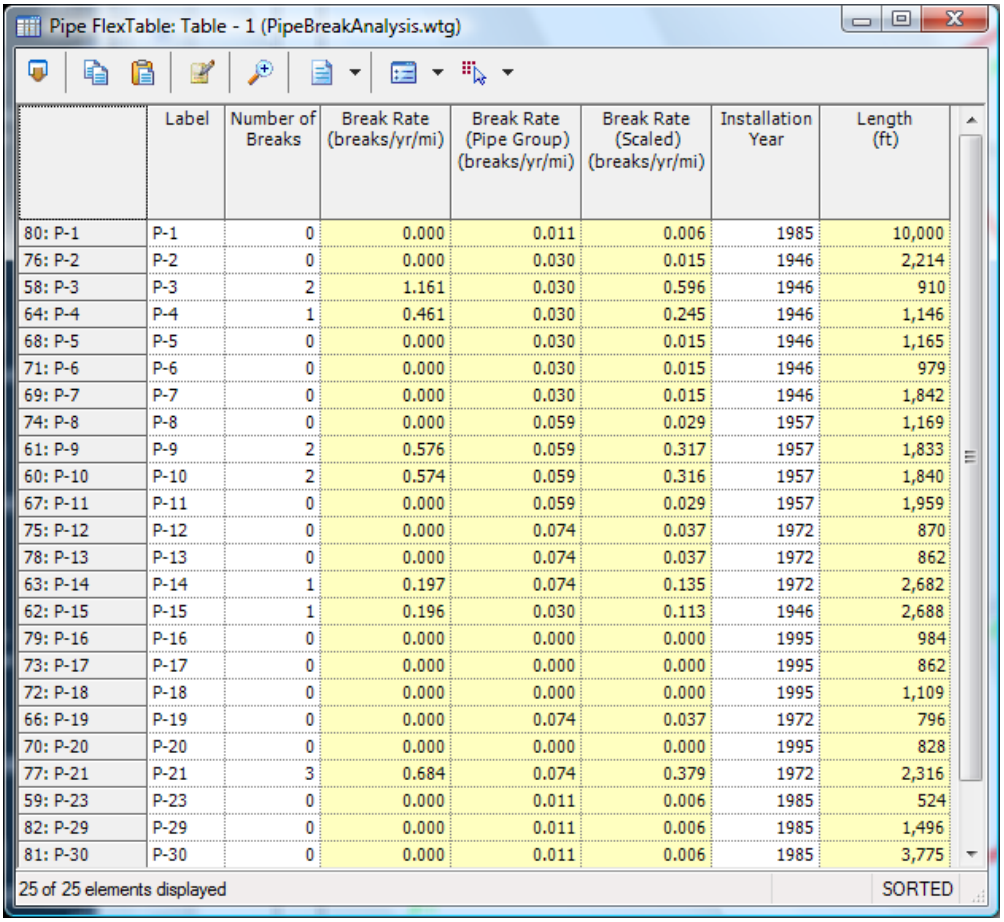
- **Break rate (breaks/yr/mi)** - based on length and number of breaks for individual pipe over the duration of break history for that pipe
- **Break rate (Pipe Group) (breaks/yr/mi)** - based on the number of breaks and total length of pipe in the group that this pipe belongs to over the duration assigned in the pipe group dialog.
- **Break Rate (Scaled) (breaks/yr/mi)** - based on the weighted sum of the individual pipe break rate and the break rate for the group that the pipe belongs.
- **Projected breaks** - the product of the scaled break rate, the projection period and the length of pipe. Estimate of the number of breaks over the projection period assuming that past break rates persist.
- **Annual cost** - the product of the scaled break rate, the length of pipe and the cost per break. Estimate of the annual cost of breaks.
- **Present worth** - the product of the scaled break rate, the length of pipe and the cost per break multiplied by the series present worth factor. Estimate of the present worth of all break costs over the projection period.

Viewing Results

In addition to viewing the results of the Pipe Break Analysis in the Pipe Break Manager, the user can view results using any of the other WaterCAD V8*i* features including the property grid, flex tables, color coding, or annotation.

In viewing results, it is important to remember that the results are stored with the Representative Scenario associated with the Pipe Break Analysis and the WaterCAD V8*i* display is associated with the currently active scenario. To view the results, make the Representative Scenario active.

If Flex Tables are to be used to view results, it is useful to create a flex table only containing pipe break related properties, excluding hydraulic results, as shown below, and making that a project or shared flex table.

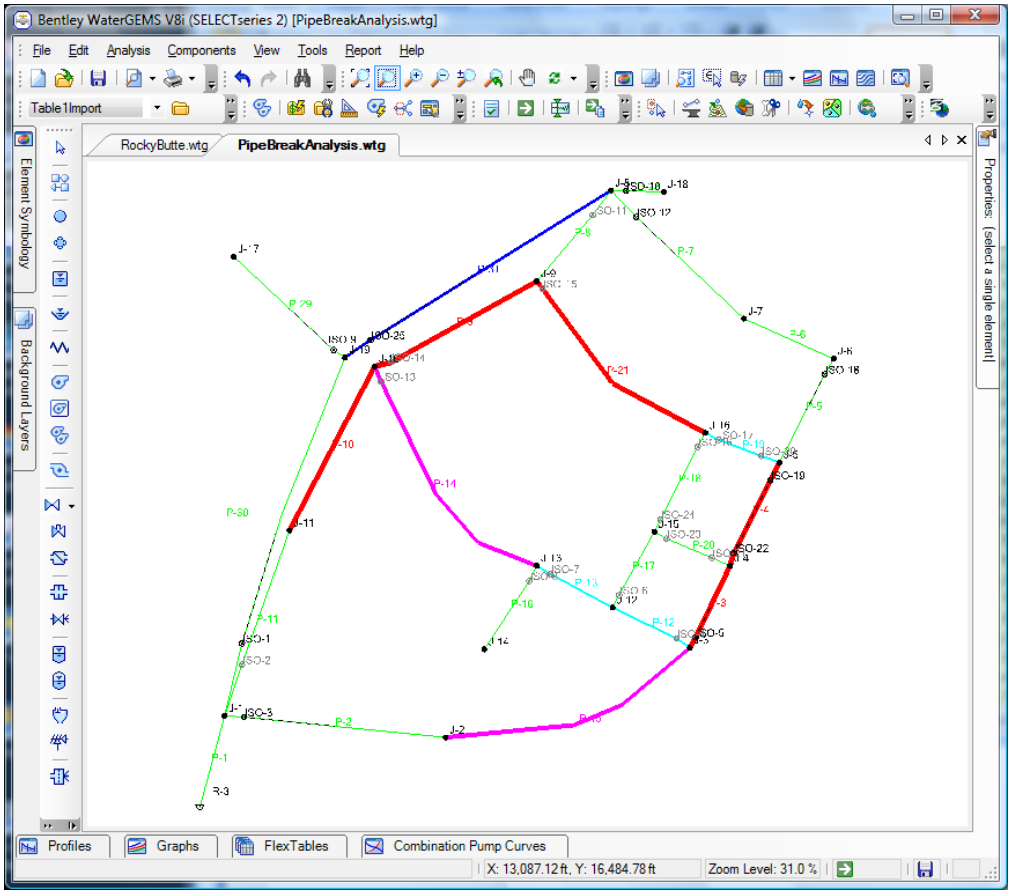


Pipe FlexTable: Table - 1 (PipeBreakAnalysis.wtg)

	Label	Number of Breaks	Break Rate (breaks/yr/mi)	Break Rate (Pipe Group) (breaks/yr/mi)	Break Rate (Scaled) (breaks/yr/mi)	Installation Year	Length (ft)
80: P-1	P-1	0	0.000	0.011	0.006	1985	10,000
76: P-2	P-2	0	0.000	0.030	0.015	1946	2,214
58: P-3	P-3	2	1.161	0.030	0.596	1946	910
64: P-4	P-4	1	0.461	0.030	0.245	1946	1,146
68: P-5	P-5	0	0.000	0.030	0.015	1946	1,165
71: P-6	P-6	0	0.000	0.030	0.015	1946	979
69: P-7	P-7	0	0.000	0.030	0.015	1946	1,842
74: P-8	P-8	0	0.000	0.059	0.029	1957	1,169
61: P-9	P-9	2	0.576	0.059	0.317	1957	1,833
60: P-10	P-10	2	0.574	0.059	0.316	1957	1,840
67: P-11	P-11	0	0.000	0.059	0.029	1957	1,959
75: P-12	P-12	0	0.000	0.074	0.037	1972	870
78: P-13	P-13	0	0.000	0.074	0.037	1972	862
63: P-14	P-14	1	0.197	0.074	0.135	1972	2,682
62: P-15	P-15	1	0.196	0.030	0.113	1946	2,688
79: P-16	P-16	0	0.000	0.000	0.000	1995	984
73: P-17	P-17	0	0.000	0.000	0.000	1995	862
72: P-18	P-18	0	0.000	0.000	0.000	1995	1,109
66: P-19	P-19	0	0.000	0.074	0.037	1972	796
70: P-20	P-20	0	0.000	0.000	0.000	1995	828
77: P-21	P-21	3	0.684	0.074	0.379	1972	2,316
59: P-23	P-23	0	0.000	0.011	0.006	1985	524
82: P-29	P-29	0	0.000	0.011	0.006	1985	1,496
81: P-30	P-30	0	0.000	0.011	0.006	1985	3,775

25 of 25 elements displayed SORTED

Color coding is very useful for identifying which pipes are likely to be troublesome in terms of future pipe breakage.



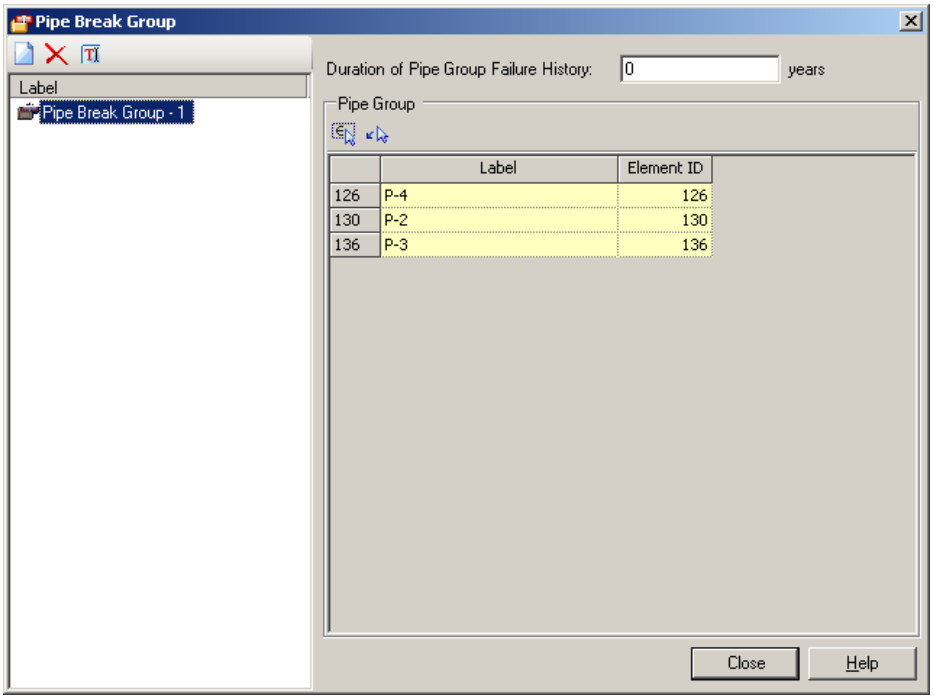
The results of pipe break analysis are useful in themselves but they also serve as one of the inputs to Pipe Renewal Planner.

Pipe Break Group Dialog Box

The Pipe Break Group dialog allows you to add pipes to a pipe break group by either:

1. Picking a previously created selection set using the selection set button.
2. Picking pipes individually from the drawing using the select from drawing button.

The assumption is that pipes in a group have similar properties with respect to pipe breakage. These properties would include similar age, material, laying condition and loading and period of break records. It is usually best to create selection sets of such pipes before starting the pipe break analysis. Name the group with a label that reflects the pipes in the group. If a pipe is not assigned to a group, its individual break rate will be used as the scaled break rate.



The dialog consists of a list pane on the left that displays all of the pipe breaks that have been created for the current project and the detail pane on the right that displays the pipes that are included in the group that is currently highlighted in the list pane, along with the following controls:

New 	Creates a new pipe break group.
Delete 	Deletes the currently highlighted pipe break group.
Rename 	Renames the currently highlighted pipe break group.
Add Pipes From Selection Set 	Allows you to add pipes to a group using a previously created selection set.
Add Pipes From Drawing 	Allows you to add pipes to a group by picking them in the drawing view.

Pick A Selection Set Dialog Box

This dialog allows you to choose a predefined selection set. Select the desired selection set from the lkst and click OK.



Calibrating Your Model with Darwin Calibrator 11

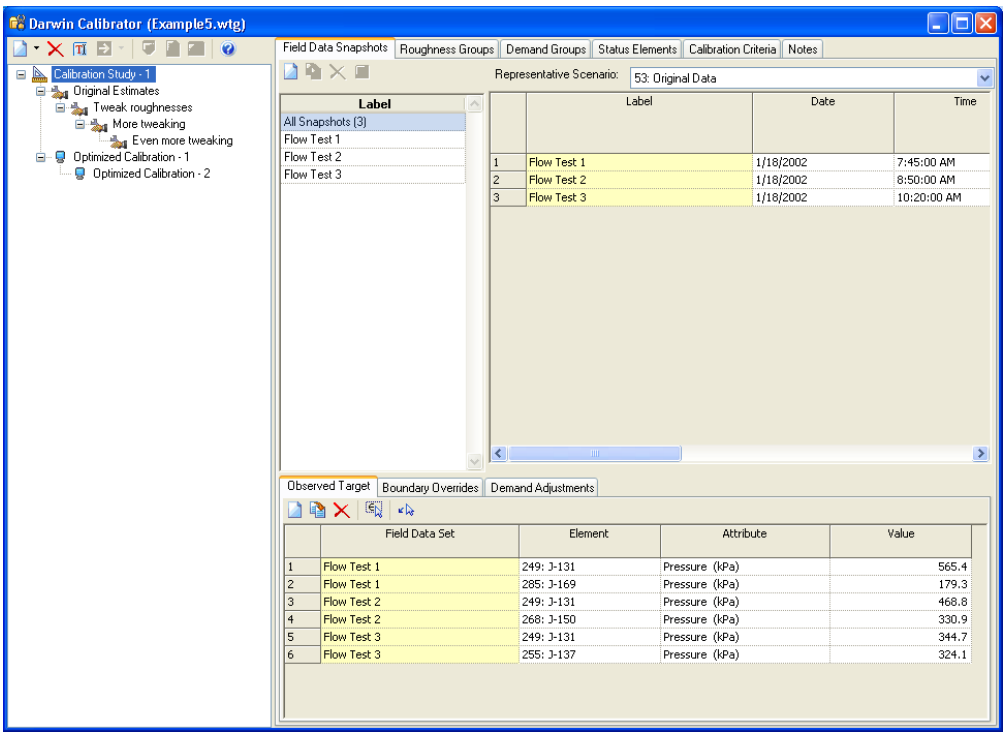
Note: **Calibrator (as well as Designer and Skelebrator) are components that initialize their data when first used, so one needs to at least open the component for those database fields to be created in the current model.**

As an example, if you are trying to use ModelBuilder to import calibration data but have never opened Calibrator in this particular model, you will not see the "Field Data Snapshot" model type in the dropdown list for Table Type. This is because that database type and its associated fields haven't been

initialized yet. You would click on Analysis>Darwin Calibrator first in the main menu. Once this is done, the Field Data Snapshot and other Calibrator related fields are created, and those options will then appear in the ModelBuilder dialogs.

The Bentley WaterCAD V8*i* Darwin Calibrator provides a history of your calibration attempts, allows you to use a manual approach to calibration, supports multiple field data sets, brings the speed and efficiency of genetic algorithms to calibrating your water system, and presents several calibration candidates for you to consider, rather than just one solution. You can set up a series of Base Calibrations, which can have numerous Child Calibrations that inherit settings from their parent Base Calibrations.

Use Base and Child Calibrations to establish a history of your calibration trials to help you derive a list of optimized solutions for your water system. Inheritance is not persistent. If you change the Base Calibration, the change does not ripple down to the Child Calibrations.



You can adjust your model to better match the actual behavior of your water distribution system by using the Darwin Calibrator feature. It allows you to make manual adjustments on the model as well as adjustments using genetic algorithm optimization.

The left pane of the Darwin Calibrator dialog box displays a list of each calibration study in the current project, along with the manual and optimized runs and calculated solutions that make up each study.

The following controls can be found above the list pane:

New

Clicking the New button opens a submenu containing the following commands:

- **New Calibration Study** - Creates a new calibration study.
- **New Optimized Run** - Creates a new optimized run. Use this command if you want Bentley WaterCAD V8*i* to efficiently process and evaluate numerous trial calibrations of your water system. You can set the optimized calibration to deliver several solutions for you to review.
- **New Manual Run** - Creates a new manual run. Use this command if you want to test fitness by adjusting roughness, demand, or status manually. If you have specific solutions in mind, Manual Calibration might let you quickly narrow-down or refine the number and measure of adjustments before you use the genetic algorithm.

Delete

Deletes the calibration study, manual run, or optimized run that is currently highlighted in the list pane. Deleting a study will also delete all runs that are a part of that study. Deleting a run will also delete any child runs based on it.

Rename

Renames the calibration study, manual run, or optimized run that is currently highlighted in the list pane.

Compute	Opens a submenu containing the following commands: • **Compute**: Computes the optimized or manual run that is currently highlighted in the list pane. • **Hierarchy**: Computes the highlighted optimized or manual run as well all the optimized or manual runs branching from it hierarchically. • **Children**: Computes the highlighted optimized or manual run as well as all the calibration runs derived from it. • **Batch Run**: Opens the Batch Run dialog, allowing you to select multiple runs to compute together.
Export to Scenario	Opens the Export to Scenario dialog box, allowing you to export the solution that is currently highlighted in the list pane to a new or existing scenario, alternative, and/or set of alternatives.
Report	Opens the Report Viewer, which displays a detailed report of the solution that is currently highlighted in the list pane.
Graph	Opens the Correlation Graph dialog box, which displays a graph of the solution that is currently highlighted in the list pane.
Help	Opens the online help.

The right side of the dialog contains controls that are used to define settings and input data for Calibration Studies and their component Manual and Optimized Runs. The controls available on the right side of the dialog box will change depending on what is highlighted in the list pane:

Calibration Studies

Optimized Runs

Manual Runs

Calibration Solutions

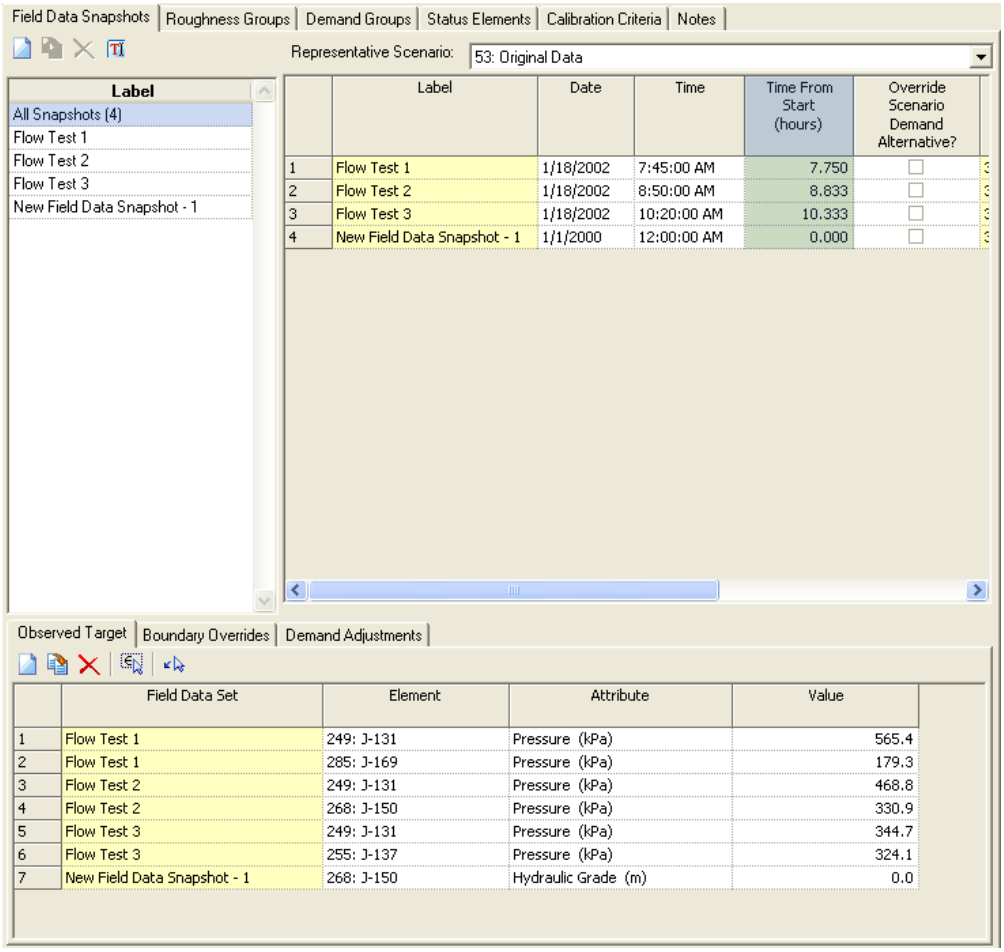
Calibration Studies

A Calibration Study is the starting point for all calibration operations. A Calibration study consists of the following components:

- Field Data Snapshots Tab
- Adjustment Groups
  - Roughness Groups
  - Demand Groups
  - Status Elements
- Calibration Criteria
- Notes (Optional).

Field Data Snapshots Tab

The Field Data Snapshots tab allows you to input observed field data for the calibration study that is currently highlighted in the list pane.



The following controls, located above the Field Data Snapshots list pane, allow you to manage your field data snapshots:

New
- Creates a new field data snapshot.

Duplicate
- Duplicates the currently highlighted field data snapshot.

Delete
- Deletes the currently highlighted field data snapshot.

Rename
- Renames the currently highlighted field data snapshot.

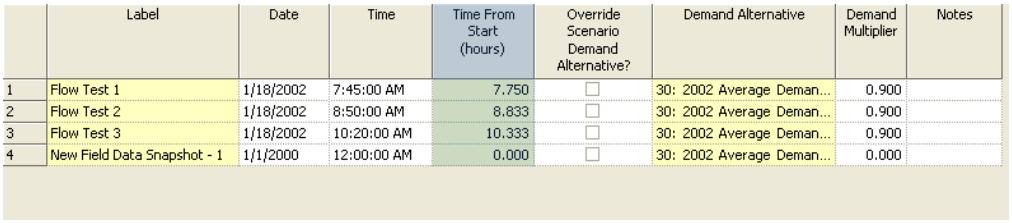
After a field data snapshot has been created, highlighting it in the list pane allows you to define or modify the following data:

Representative Scenario



Choose the scenario that will be used as the base data for the calibration study.

Snapshot Data



	Label	Date	Time	Time From Start (hours)	Override Scenario Demand Alternative?	Demand Alternative	Demand Multiplier	Notes
1	Flow Test 1	1/18/2002	7:45:00 AM	7.750	☐	30: 2002 Average Deman...	0.900	
2	Flow Test 2	1/18/2002	8:50:00 AM	8.833	☐	30: 2002 Average Deman...	0.900	
3	Flow Test 3	1/18/2002	10:20:00 AM	10.333	☐	30: 2002 Average Deman...	0.900	
4	New Field Data Snapshot - 1	1/1/2000	12:00:00 AM	0.000	☐	30: 2002 Average Deman...	0.000	

Enter the following Snapshot data:

Label — Enter a label for the field data snapshot.

Date — Set the date of the observations and field tests.

Time — Set the time of the observations and field tests. When using the pull down menu to select a time using the up and down arrows, hit the Enter key when you have selected the time you want to accept the change.

Time from Start — Displays the time difference from the time you set for the field data set to the time defined as the start of the scenario.

Override Scenario Demand Alternative? — Check this box to override the displayed Demand Alternative and use a different demand alternative or to use the specified Demand Multiplier. Clear this check box if you want to use the displayed alternative or if you do not want to use the Demand Multiplier.

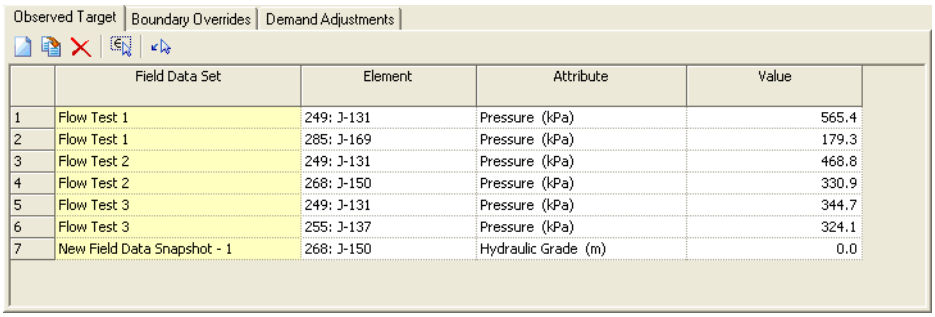
Demand Alternative — Displays the Demand Alternative associated with the selected set of observations. If the Override Scenario Demand Alternative? box is checked, you can choose a different demand alternative here.

Demand Multiplier — Set a demand multiplier that is applied to your water model. For example, if you have knowledge that your demand is higher or lower by a specific percentage, you can set that value here. If the multiplier is set to zero, the demand will also be zero. By default this value is set to 1.

Notes — Use the Notes field to enter any comments you want saved with the field data snapshot.

Note: Field data set time is important since Calibrator uses the specified time to determine nodal demands from the represenative scenario by applying pattern multipliers for the specified times. To that end be sure to specify the time that corresponds to the time the field data was acquired.

Observed Target



	Field Data Set	Element	Attribute	Value
1	Flow Test 1	249: J-131	Pressure (kPa)	565.4
2	Flow Test 1	285: J-169	Pressure (kPa)	179.3
3	Flow Test 2	249: J-131	Pressure (kPa)	468.8
4	Flow Test 2	268: J-150	Pressure (kPa)	330.9
5	Flow Test 3	249: J-131	Pressure (kPa)	344.7
6	Flow Test 3	255: J-137	Pressure (kPa)	324.1
7	New Field Data Snapshot - 1	268: J-150	Hydraulic Grade (m)	0.0

The Observed Target tab allows you to input calibration target values (node pressure and hydraulic grade line, as well as pipe flows) that the calibration operations will be attempting to match. Each row in the table represents a single target observation. The following controls are available in this tab:

New — Creates a new target observation for the Field Data Snapshot that is currently highlighted in the list.

Duplicate — Makes a copy of the currently highlighted target observation for the Field Data Snapshot that is currently highlighted in the list.

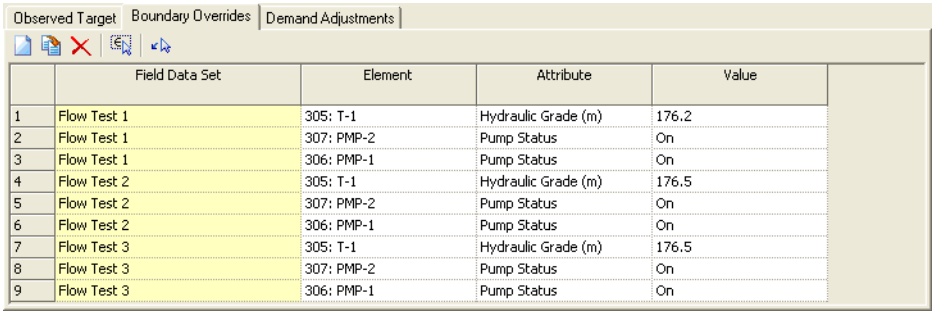
Delete — Deletes the currently highlighted target observation.

Initialize Table from Selection Set	Opens the Initialize From Selection set dialog, allowing you to choose a selection set. After a selection set is specified, this command generates a target observation for each element in the selection set.
Select From Drawing	Opens the Select dialog box, allowing you to select elements in the drawing view.

For each target observation, the table contains the following columns:

Field Data Set	Displays the field data set to which the target observation belongs.
Element	Select the element for which you want to enter observed data.
Attribute	Select the attribute for which you have observed data. Different attributes are available for each element type.
Value	Select a value from the drop-down list or enter in a value for the selected attribute.

Boundary Overrides



	Field Data Set	Element	Attribute	Value
1	Flow Test 1	305: T-1	Hydraulic Grade (m)	176.2
2	Flow Test 1	307: PMP-2	Pump Status	On
3	Flow Test 1	306: PMP-1	Pump Status	On
4	Flow Test 2	305: T-1	Hydraulic Grade (m)	176.5
5	Flow Test 2	307: PMP-2	Pump Status	On
6	Flow Test 2	306: PMP-1	Pump Status	On
7	Flow Test 3	305: T-1	Hydraulic Grade (m)	176.5
8	Flow Test 3	307: PMP-2	Pump Status	On
9	Flow Test 3	306: PMP-1	Pump Status	On

Observed boundary conditions such as tank level, pump status and speed and valve settings are entered in the Boundary Overrides tab. Each row in the table represents a single boundary override. The following controls are available in this tab:

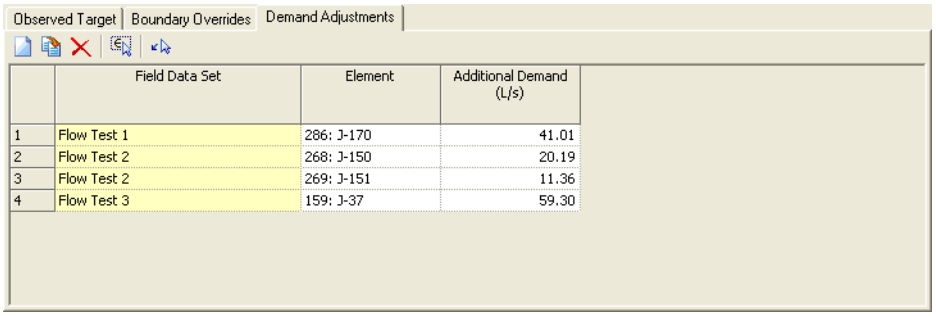
New	Creates a new boundary override for the Field Data Snapshot that is currently highlighted in the list.

Duplicate	Makes a copy of the currently highlighted boundary override for the Field Data Snapshot that is currently highlighted in the list.
Delete	Deletes the currently highlighted boundary override.
Initialize Table from Selection Set	Opens the Initialize From Selection set dialog box, allowing you to choose a selection set. After a selection set is specified, this command generates a boundary override for each applicable element in the selection set.
Select From Drawing	Opens the Select dialog box, allowing you to select elements in the drawing view.

For each boundary observation, the table contains the following columns:

Field Data Set	Displays the field data set to which the boundary override belongs.
Element	Select the element for which you want to enter a boundary override.
Attribute	Select the attribute for which you have a boundary override. Different attributes are available for each element.
Value	Select a value from the drop-down list or type in a value for the selected attribute.

Demand Adjustments



Use the Demand Adjustments tab to adjust demand for individual elements, such as flow from a hydrant. Additional demands (e.g., fire flow tests) are in addition to, not in lieu of, demands already calculated from pattern multipliers. Each row in the table represents a single demand adjustment. The following controls are available in this tab:

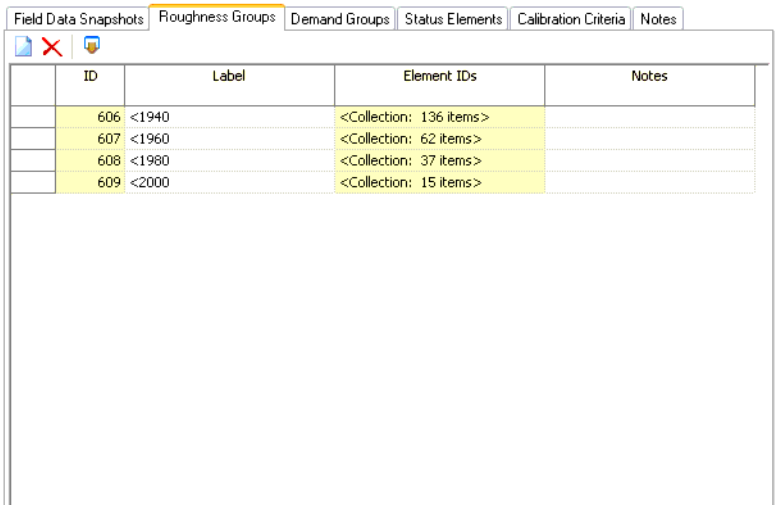
New	Creates a new demand adjustment for the Field Data Snapshot that is currently highlighted in the list.
Duplicate	Makes a copy of the currently highlighted demand adjustment for the Field Data Snapshot that is currently highlighted in the list.
Delete	Deletes the currently highlighted demand adjustment.
Initialize Table from Selection Set	Opens the Initialize From Selection set dialog, allowing you to choose a selection set. After a selection set is specified, this command generates a demand adjustment for each applicable element in the selection set.
Select From Drawing	Opens the Select dialog, allowing you to select elements in the drawing view.

For each demand adjustment, the table contains the following columns:

Field Data Set	Displays the field data set to which the demand adjustment belongs.
Element	Select the element for which you want to enter a demand adjustment.
Additional Demand	Type in a value for the demand adjustment.

Adjustment Groups

Adjustment groups are groups of elements whose attributes are adjusted together during the calibration process. You must be careful to group similar elements and not dissimilar ones. You can adjust the properties for a group as a whole but not for individual members of the group.



There are three kinds of adjustment groups, each of which are created and modified in their respective calibration study settings tab:

Roughness Groups - Add, edit, delete, or rename Roughness adjustment groups in the Roughness tab. Each roughness group should comprise elements that have similar attributes, such as pipes in a location of a similar material and age. Adjustments made to a group are applied to every element in the group. Click the Export Groups button to export the Calibration Group ID data to an automatically created user defined attribute. All elements within a calibration group will have an identical Calibration Group ID. This allows you to color code by calibration roughness group.

Demand Groups - Add, edit, delete, or rename Demand adjustment groups in the Demand tab. Adding Demand Calibration adjustment groups introduces more unknowns into a calibration problem. If available, you should enter more accurate demand data into your Bentley WaterCAD V8*i* model, rather than adding Demand Adjustment Groups. Consider creating Demand Groups based on usage patterns. Click the Export Groups button to export the Calibration Group ID data to an automatically created user defined attribute. All elements within a calibration group will have an identical Calibration Group ID. This allows you to color code by calibration demand group.

You can automatically create demand groups from selection sets using the Group Generator. To open the Group Generator click the Create Multiple Design Groups button.

Status Elements - Add, edit, delete, or rename Status Element adjustment groups in the Status Elements tab. Status indicates whether a pipe is open or closed. GA-optimized calibration will identify the status of each pipe within the status group so that the chosen objective function is minimized. Status groups are generally used when a particular area of the system is believed to contain a closed pipe or valve. We recommend that Status Groups comprise, at most only a few pipes, or one pipe. Click the Export Groups button to export the Calibration Group ID data to an automatically created user defined attribute. All elements within a calibration group will have an identical Calibration Group ID. This allows you to color code by calibration status group.

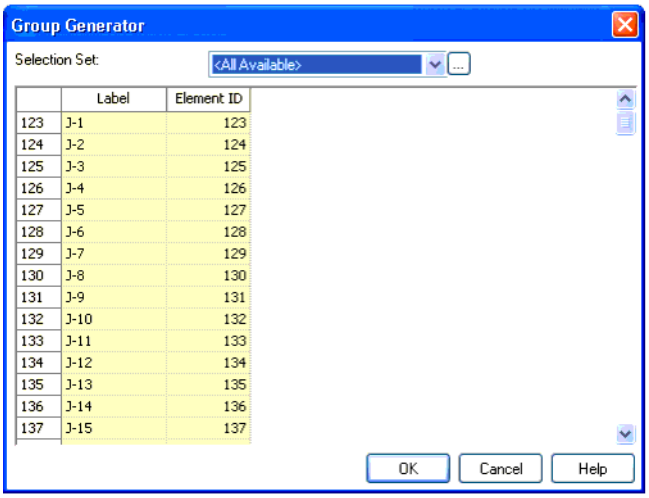
Each adjustment group tab consists of a table that lists the adjustment groups, a New button to add groups to the table, and a Delete button to remove the currently selected group from the table. The table consists of the following columns:

ID	The automatically assigned ID of the adjustment group.
Label	The user-defined name of the adjustment group. To change the label, click on it and type a new name.
Element IDs	The elements that are contained within the adjustment group. Clicking the ellipsis button in this field will open the Selection Set dialog, which allows you to add and remove elements by selecting them in the drawing view.
Notes	Use the Notes field to enter any comments you want saved with the adjustment group.

Tip: **Decide on your Adjustment Groups first and then collect the Field Data to support the number or groups, rather than letting available data determine how many Adjustment Groups you have.**

Group Generator Dialog Box

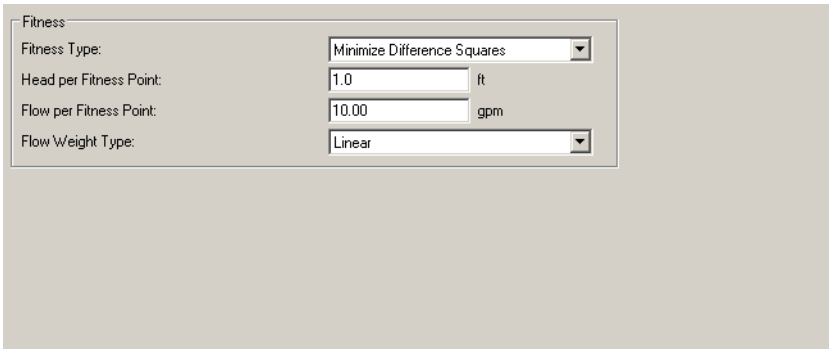
The Group Generator allows you to automatically create multiple design groups based on existing selection sets, or by selecting a group of elements from the drawing.



The dialog consists of a list of elements that will be used to create demand groups (one element per group) and a menu that allows you to select the elements that are included in the list. The menu contains a list of all existing selection sets. Click the elipsis button to select elements from the drawing directly. When the list contains all of the elements that you want to be included in demand groups, click OK.

Calibration Criteria

Use the Calibration Criteria tab to set up how the calibrations are evaluated.



The options you specify are applied to every calibration trial in the Calibration Study. The Calibration Criteria tab contains the following controls:

- **Fitness Type -** Select the Fitness Type you want to use from the drop down list. In general, regardless of the fitness type you select, a lower fitness indicates better calibration. Fitness Types include: Minimize Difference Squares, Minimize Difference Absolute Values, and Minimize Maximum Difference. For more information, see Calibration Criteria Formulae.
  - **Minimize Difference Squares -** Uses a calibration designed to minimize the sum of squares of the discrepancy between the observed data and the model simulated values. (Model simulated values include hydraulic grades and pipe discharges.) This calibration favors solutions that minimize the overall sum of the squares of discrepancies between observed and simulated data.
  - **Min. Diff. Absolute Values -** Uses a calibration designed to minimize the sum of absolute discrepancy between the observed data and the model simulated values. This calibration favors solutions that minimize the overall sum of discrepancies between observed and simulated data.
  - **Minimize Max. Difference -** Uses a calibration designed to minimize the maximum of all the discrepancies between the observed data and the model simulated values. This calibration favors solutions that minimize the worst single discrepancy between observed and simulated data. Note that the Minimize Maximum Difference Fitness Type is more sensitive to the accuracy of your data than other Fitness Types.
- **Head/Flow per Fitness Point -** Head and Flow per Fitness Type provide a way for you to weigh the importance of head and flow in your calibration. Set these values such that the head and flow have unit equivalence. You can give higher importance to Head or Flow by setting a smaller number for its Per Fitness Point Value.
- **Flow Weight Type -** Select the type of weight used: None, Linear, Square, Square Root, and Log. The weighting type you use can provide a greater or lesser fitness penalty.

  In general, measurements with larger flow carry more weight in the optimization calibrations than those with less flow. You can exaggerate or reduce the effect larger measurements have on your calibration by selecting different weight types. For example, using no weighting (None) provides no penalty for measurements with lesser flow versus those with greater flow. Using log and square root reduces the fitness penalty for measurements with lesser flow, and using linear or square increases the fitness penalty for measurements with less flow.

  Note: If you change the Calibration Options, any fitness values you get are not comparable to fitness values obtained using different Calibration Options settings.

Calibration Criteria Formulae

The following formulae are used for Minimize Difference Squares, Minimize Difference Absolute Values, and Minimize Maximum Difference.

$$\frac{\sum_{np=1}^{NH} w_{nh}\left(\frac{Hsim_{nh} - Hobs_{nh}}{Hpnt}\right)^2 + \sum_{nf=1}^{NF} w_{nf}\left(\frac{Fsim_{nf} - Fobs_{nf}}{Fpnt}\right)^2}{NH + NF}$$

Figure 11-1: Minimize Difference Squares:

$$\frac{\sum_{np=1}^{NH} w_{nh}\left|\frac{Hsim_{nh} - Hobs_{nh}}{Hpnt}\right| + \sum_{nf=1}^{NF} w_{nf}\left|\frac{Fsim_{nf} - Fobs_{nf}}{Fpnt}\right|}{NH + NF}$$

Figure 11-2: Minimize Difference Absolute Values

$$\max\left\{\max_{nh=1}^{NH} w_{nh}\left|\frac{Hsim_{nh} - Hobs_{nh}}{Hpnt}\right|, \max_{nf=1}^{NF} w_{nf}\left|\frac{Fsim_{nf} - Fobs_{nf}}{Fpnt}\right|\right\}$$

Figure 11-3: Minimize Maximum Difference

where W_{nh} and W_{nf} represent a normalized weighting factor for observed hydraulic grades and flows respectively. They are given as:

$$W_{nh} = \frac{Hobs_{nh}}{\sum Hobs_{nh}}$$

$$W_{nf} = \frac{Fobs_{nf}}{\sum Fobs_{nf}}$$

The weighting factors may also take many other forms, such as no weight (equal to 1), linear, square, square root and log functions. Other variables include:

- **$Hobs_{nh}$** designates the nh-th observed hydraulic grade.
- **$Hsim_{nh}$** is the nh-th model simulated hydraulic grade.
- **$Fobs_{nf}$** is the observed flow.
- **$Fsim_{nf}$** is the model simulated flow.
- **Hp_{nt}** notes the hydraulic head per fitness point.
- **Fp_{nt}** is the flow per fitness point.

- **NH** is the number of observed hydraulic grades.
- **NF** is the number of observed pipe discharges.

Optimized Runs

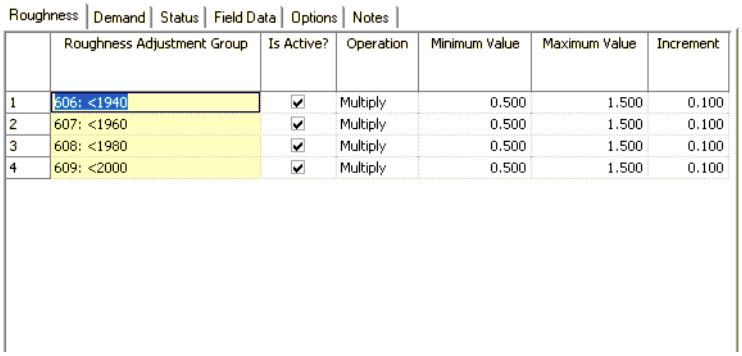
A genetic-algorithm Optimized Run consists of categorized data split among the following tabs:

- Roughness Tab
- Demand Tab
- Status Tab
- Field Data Tab
- Options Tab
- Notes Tab

Note: The Roughness, Demand, and Status tabs display the groups you added when setting up your Adjustment Groups (for more information, see Adjustment Groups). If a tab is empty, then you did not create a group for the condition represented by that tab.

Roughness Tab

The Roughness tab allows you to select the roughness adjustment groups (which were defined in the Calibration Study) and the parameters to use during the optimized run.



	Roughness Adjustment Group	Is Active?	Operation	Minimum Value	Maximum Value	Increment
1	606: <1940	☑	Multiply	0.500	1.500	0.100
2	607: <1960	☑	Multiply	0.500	1.500	0.100
3	608: <1980	☑	Multiply	0.500	1.500	0.100
4	609: <2000	☑	Multiply	0.500	1.500	0.100

The Roughness tab consists of a table containing the following columns:

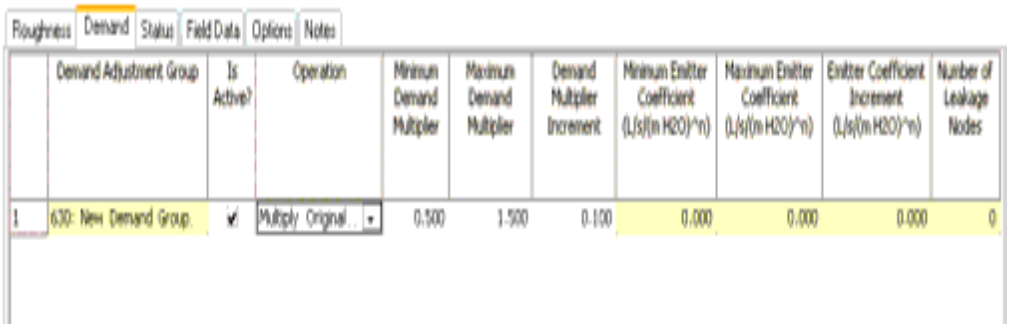
- **Roughness Adjustment Group -** Displays the name of the roughness adjustment group.
- **Is Active? -** If this box is checked, the associated adjustment group will be considered during calibration. If the box is cleared, it will be ignored.

- **Operation** - Select the operation you want the calibration to perform.
- **Minimum Value** - Enter the minimum value that you want the genetic algorithm to use as a lower boundary when calculating fitness solutions.
- **Maximum Value** - Enter the maximum value that you want the genetic algorithm to use as an upper boundary when calculating fitness solutions.
- **Increment** - Set the increment as the intervals at which you want the GA to test. Try to choose an increment that gives the least number of possible alternatives. You may need to decrease the range between your upper and lower limits to do this.

Note: When using Darcy Wesibach as the headloss formula and using the SET option for applying roughnesses to calibration groups, the expected unit of the input for Darcy Weisbach e is millifeet.

Demand Tab

The Demand tab allows you to select the demand adjustment groups (which were defined in the Calibration Study) and the parameters to use during the optimized run.



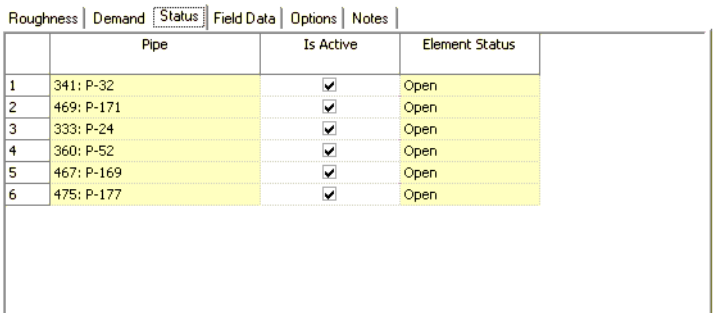
The Demand tab consists of a table containing the following columns:

- **Demand Adjustment Group** - Displays the name of the demand adjustment group.
- **Is Active?** - If this box is checked, the associated adjustment group will be considered during calibration. If the box is cleared, it will be ignored.
- **Operation** - Select the operation you want the calibration to perform.
- **Minimum Demand Multiplier** - Enter the minimum demand multiplier that you want the genetic algorithm to use as a lower boundary when calculating fitness solutions. This field will only be editable for **Multiply Original Demand** Operations.

- **Maximum Demand Multiplier -** Enter the maximum demand multiplier that you want the genetic algorithm to use as an upper boundary when calculating fitness solutions. This field will only be editable for **Multiply Original Demand** Operations.
- **Demand Multiplier Increment -** Set the increment as the demand multiplier intervals at which you want the GA to test. Try to choose an increment that gives the least number of possible alternatives. You may need to decrease the range between your upper and lower limits to do this. This field will only be editable for **Multiply Original Demand** Operations.
- **Minimum Emitter Coefficient** - Enter the minimum emitter coefficient that you want the genetic algorithm to use as a lower boundary when calculating fitness solutions. This field will only be editable for **Set Emitter Coefficient** and **Detect Leakage Node** Operations.
- **Maximum Emitter Coefficient** - Enter the maximum emitter coefficient that you want the genetic algorithm to use as an upper boundary when calculating fitness solutions. This field will only be editable for **Set Emitter Coefficient** and **Detect Leakage Node** Operations.
- **Emitter Coefficient Increment** - Set the increment as the emitter coefficient intervals at which you want the GA to test. Try to choose an increment that gives the least number of possible alternatives. You may need to decrease the range between your upper and lower limits to do this. This field will only be editable for **Set Emitter Coefficient** and **Detect Leakage Node** Operations.
- **Number of Leakage Nodes** - The maximum number of leakage nodes possible for the demand group when calculating fitness solutions. This field will only be editable for **Detect Leakage Node** Operations.

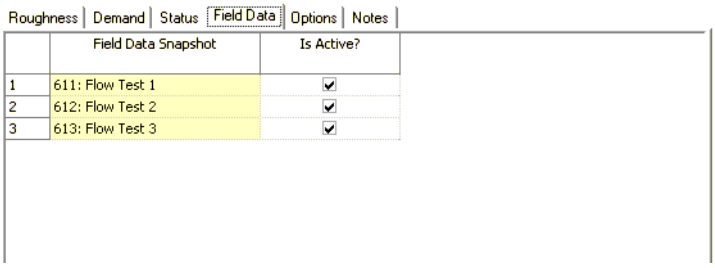
Status Tab

Use the Status tab to see the initial status of each of the pipes in each of the Status Element adjustment groups which were defined in the Calibration Study. For each of the elements, if the Is Active? box is checked, the associated element will be considered during calibration. If the box is cleared, it will be ignored.



Field Data Tab

The Field Data tab displays all the field data snapshots you have entered for the calibration. Click the Is Active? check box next to the name of each of the field data snapshots you want to use for the calibration trial. Field data snapshots that have unchecked boxes next to them will not be used to test fitness when you Compute.



Options Tab

Use the Options tab to refine how Bentley WaterCAD V8*i* applies the genetic algorithm (GA) to your optimized calibration trials.

Options

- **Reset -** Click **Reset** to restore the software default values for the Darwin Calibration Options.

- **Fitness Tolerance -** Set the precision with which you want the optimized calibration to calculate fitness. As with many of these settings, you should determine a tolerance that balances accuracy and speed for your water models. Fitness Tolerance works in conjunction with Non-Improvement Generations.
- **Maximum Trials -** Set the maximum number of calibration trials you want the Optimized Calibration to process before stopping.
- **Non-Improvement Generations -** Set the number of maximum number of non-improvement generations you want the GA to process without calculating an improved fitness. If the Optimized Calibration makes this number of calculations without finding an improvement in fitness that is better than the defined Fitness Tolerance, the calibration will stop. Non-Improvement Generations works in conjunction with Fitness Tolerance.
- **Solutions to Keep -** Set the number of fitness solutions that you want to keep. Rather than presenting you with only one solution, Bentley WaterCAD V8*i* presents you with a customizable number of solutions, so you can review them manually.

  Note: Larger values for maximum trials and non-improvement generations will make the optimization run longer. You may want to start with fairly low numbers and then gradually increase the numbers in subsequent runs as you want to ensure better solutions. If a run seems to be taking a long time, you may click the Stop button to stop the optimization.

- Leakage Detection Penalty Factor -

Advanced Options

The Advanced Options let you customize how the genetic algorithm (GA) performs. Since genetic-algorithm optimization is a randomly guided search algorithm, different parameter values may yield a slightly different set of solutions, which can be used for a sensitivity study of your model calibration.

Note that all values must be positive, not negative. Recommended values are based on maximizing speed and efficiency.

- **Reset -** Click **Reset** to restore the software default values for the options.
- **Maximum Era Number -** Lets you controls the number of outer loops the genetic algorithm (GA) uses. Each outer loop runs over the number of generations with the same population size. A large value for maximum era number will make the optimization run longer than a smaller number would. You might want to start with a low number and increase the number in subsequent runs.

  The allowable range for values is greater than or equal to 1. If you use 0 or less, the Optimized The GA uses values based on what is set for Maximum Trials and Non-Improvement Generations.

- **Era Generation Number** - Sets the number of generations of each inner loop the GA uses.

  The allowable range for values is greater than or equal to 1. If you use 0 or less, the Optimized The GA uses values based on what is set for Maximum Trials and Non-improvement Generations.

- **Population Size** - Sets the number of GA solutions in each generation. Increasing Population Size results in a longer time for each generation and more solutions to be evaluated.

  The allowable range for values is from 50 to 500. We recommend you use a range of 50 to 150.

- **Cut Probability** - Sets the probability that a GA solution will be split into two pieces. Setting this value closer to 100% increases the number of cuts made and reduces the average string (chromosome) length. Increasing Cut Probability causes solutions to vary more widely from one generation to the next, whereas decreasing this results in more marginal changes.

  The allowable range for values is between 0% and 100%, not inclusive. We recommend you use a value less than 10%.

  Setting the Splice probability closer to 100% increases the demand on system RAM. If you are getting out-of-memory errors when using GA Optimization, try reducing the Splice Probability closer to 0% and try increasing the Cut Probability away from 0%.

- **Splice Probability** - Sets the probability that two GA solutions will be joined together. A Splice Probability set close to 100% results in long solution strings, which increases the mixing of alleles (genes) and improves the variety of solutions.

  The allowable range for values is between 0% and 100%, not inclusive. We recommend you use a range from 50% to 90%.

- **Mutation Probability** - Sets the probability that a GA solution is randomly altered. A value closer to 100% causes the solutions to contain more randomization than values closer to 0%.

  The allowable range for values is between 0% and 100%, not inclusive. We recommend you use a value less than 10%.

- **Random Seed** - Lets you set the random number generator to a new point. Changing this value and leaving all other parameters as-is will yield a different solution set.

  The allowable range for values is from 0 to 1, inclusive.

- **Penalty Factor** - In Darwin Designer, use a penalty factor to help find the solution. A high penalty factor causes the GA to focus on feasible solutions, which do not violate boundaries of pressure and flow. A low penalty factor (50,000 or so) permits the GA to consider solutions that are on the boundary between feasible

and infeasible solutions, possibly violating pressure or flow boundaries by a small amount. Because the optimal solution often resides in the boundary between feasible and infeasible solutions, a high penalty factor causes the GA to find a feasible solution quickly but is less likely to find the optimal solution.

From a practical standpoint, you might consider starting with a high penalty factor and working towards a lower penalty factor as you pursue an optimal solution.

Notes Tab

Type any notes that you want associated with the calibration.

Manual Runs

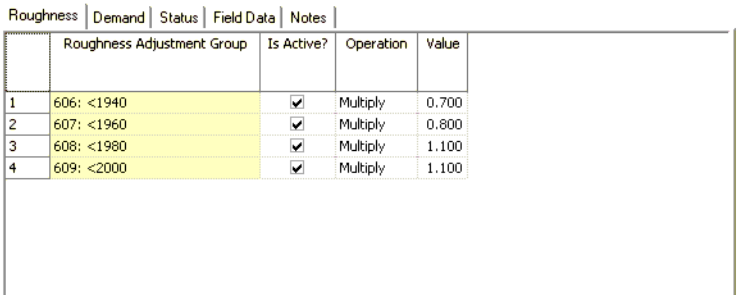
A Manual calibration run consists of categorized data split among the following tabs:

- Roughness Tab
- Demand Tab
- Status Tab
- Field Data Tab
- Notes Tab

Note: The Roughness, Demand, and Status tabs display the groups you added when setting up your Adjustment Groups (for more information, see Adjustment Groups). If a tab is empty, then you did not create a group for the condition represented by that tab.

Roughness Tab

The Roughness tab allows you to select the roughness adjustment groups (which were defined in the Calibration Study) and the operations to perform during the manual run.

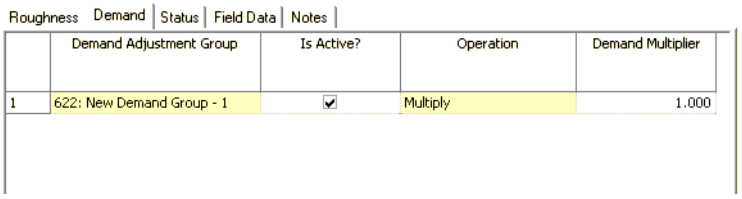


The Roughness tab consists of a table containing the following columns:

- **Roughness Adjustment Group -** Displays the name of the roughness adjustment group.
- **Is Active? -** If this box is checked, the associated adjustment group will be considered during calibration. If the box is cleared, it will be ignored.
- **Operation -** Select the operation you want the calibration to perform.
- **Value -** Type the value you want to be used in conjunction with the operation during the manual calibration run.

Demand Tab

The Demand tab allows you to select the demand adjustment groups (which were defined in the Calibration Study) and the parameters to use during the optimized run.

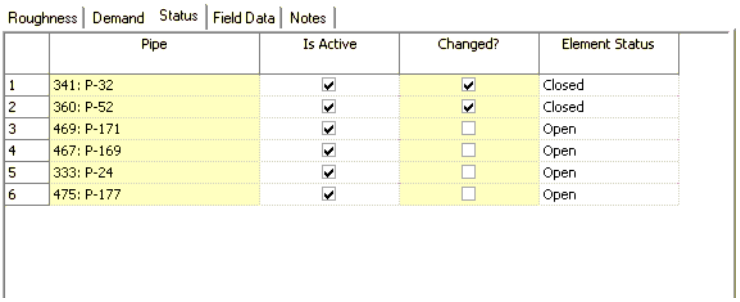


The Demand tab consists of a table containing the following columns:

- **Demand Adjustment Group -** Displays the name of the demand adjustment group.
- **Is Active? -** If this box is checked, the associated adjustment group will be considered during calibration. If the box is cleared, it will be ignored.
- **Operation -** Select the operation you want the calibration to perform.
- **Demand Multiplier-** Type the value you want to be used in conjunction with the operation during the manual calibration run.

Status Tab

Use the Status tab to view and modify the initial status of each of the pipes in each of the Status Element adjustment groups which were defined in the Calibration Study.

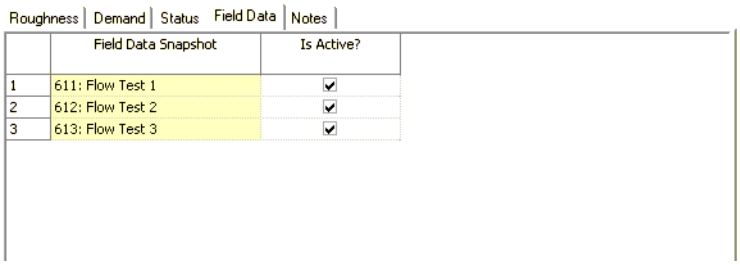


For each of the elements, if the Is Active? box is checked, the associated element will be considered during calibration. If the box is cleared, it will be ignored.

To change the initial status of a pipe, click the associated Element Status field and select the new status. When an initial status has been changed, the associated Changed? check box will be checked.

Field Data Tab

The Field Data tab displays all the field data snapshots you have entered for the calibration. Click the Is Active? check box next to the name of each of the field data snapshots you want to use for the calibration trial. Field data snapshots that have unchecked boxes next to them will not be used to test fitness when you Compute.



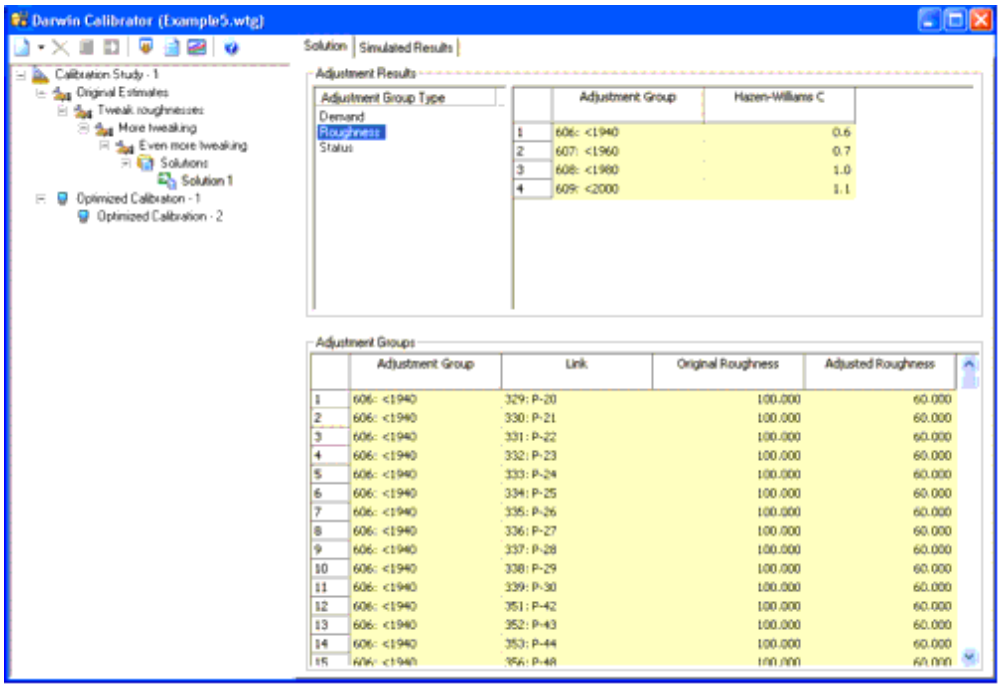
Notes Tab

Enter any notes that you want associated with the calibration.

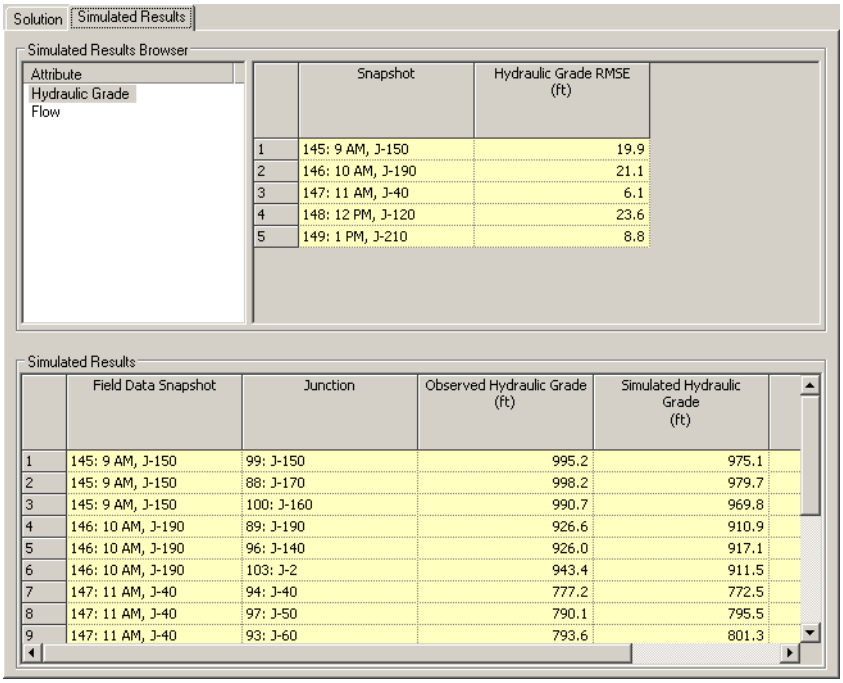
Calibration Solutions

After computing an optimized or manual run, one or more solutions will appear in the calibration study list pane. Highlighting a solution makes the following tabs available on the right side of the dialog:

Solution Tab - The Solution tab displays the adjusted values for each adjustment group along with a comparison of the original and adjusted value for each element within each adjustment group. The solution results are filtered by Adjustment Group Type; click the desired type in the Adjustment Group Type pane.



Simulated Results Tab - The Simulated Results tab displays the simulated HGL or flow against the observations you recorded in your field data and the difference between the observed and simulated values. The solution results are filtered by attribute type; click the desired type in the Attribute pane.

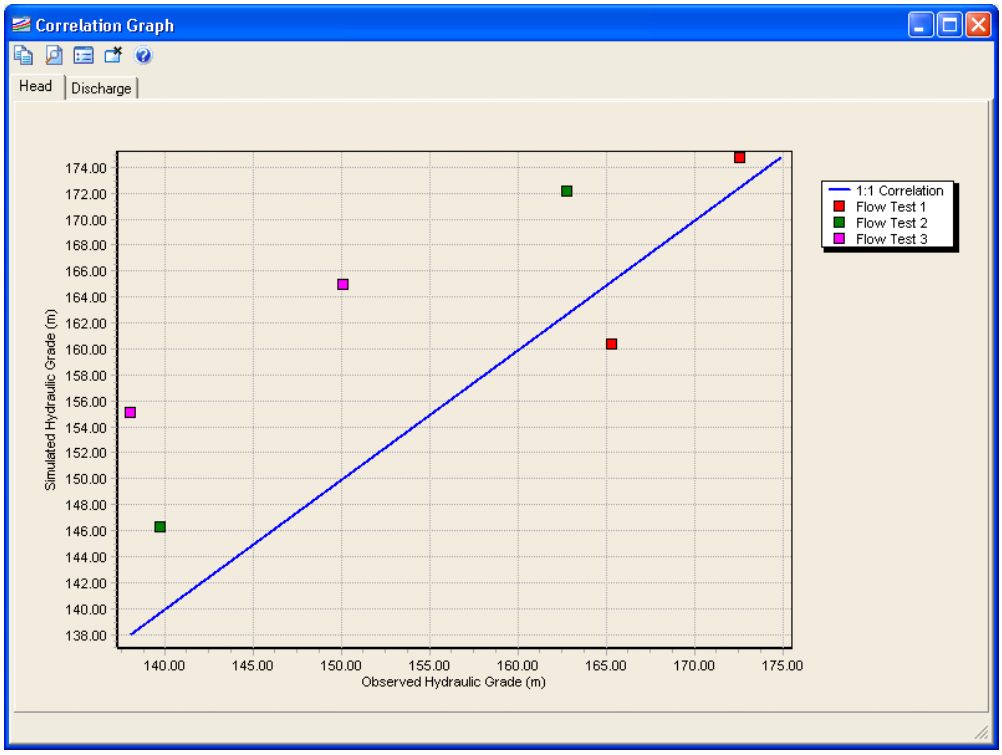


Additionally, when a solution is highlighted in the calibration study list pane, the following controls become available:

- **Export to Scenario** - Click the **Export to Scenario** button to export the currently selected Calibration solution to the water flow model. This opens the Export Calibration to Scenario dialog box (for more information, see Calibration Export to Scenario Dialog Box on page 11-1073).
- **Report** - Click the **Report** button to display a print preview of the solutions data window.
- **Graph** - Click **Graph** button to see a graph of your observed data sets versus the HGL correlation between the Simulated and Observed HGL.

Correlation Graph Dialog Box

This dialog displays a graph that shows the correlation between the Simulated and Observed HGL.



Copy: Copies the current graph to the clipboard.

Print Preview: Displays a preview of the graph as it will look when printed.

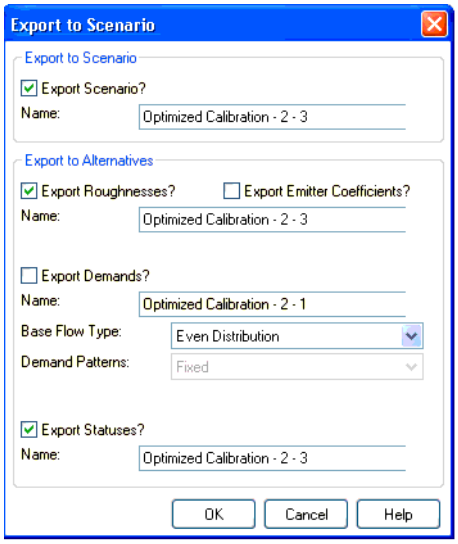
Options: Opens the chart options to allow the graph display to be customized.

Close: Closes the graph window.

Help: Opens the help for the Correlation Graph dialog box.

Calibration Export to Scenario Dialog Box

Use the Calibration Export to Scenario dialog box to apply the results of your Optimized Calibration or Manual Calibration to your water model.



Export Scenario?

Check the Export Scenario? box to export the calibration solution to a new scenario. You can change the default name of the new scenario by typing a different one in the Name field. If you export to a scenario and do not export to an alternative (by unchecking the associated box or boxes), the data for that alternative type will be exported to the Base alternative.

Export Alternatives:

Choose which types of data to export to new alternatives. You can rename the newly created alternatives by typing over the default name.

Choose to export **Rougnesses** to the **Physical** alternative by checking the **Export Roughnesses?** box.

Choose to export **Emitter Coefficients** to the **Physical** alternative by checking the **Export Emitter Coefficients?** box.

When exporting to **Demand** alternative, you are able to choose how the adjusted demand (the difference between the total calibrated demand and the original demand) is exported by selecting **Base Flow Type** of **Even Distribution** or **Assign One Base Flow**. If **Even Distribution** is selected, the adjusted demand

is evenly distributed to all of the base demand components as differentiated by demand patterns for a node. If **Assign One Base Flow** is selected, the adjusted demand is exported to the user-selected base demand component as differentiated by demand pattern.

Choose to export **Statuses** to the **Initial Settings** alternative by checking the **Export Statuses?** box.

OK/Cancel: Click **OK** to export your calibration or Cancel to close the dialog box without exporting your calibration.

Importing Field Data into Darwin Calibrator Using ModelBuilder

Darwin field data snapshots can be imported via ModelBuilder, the field data needs to be prepared in a certain format for a different collection of data. Let's take Excel as a data source example; the import process from other data sources will be very similar to this too.

Import Snapshots

Multiple snapshots can be imported into calibration study in Darwin Calibrator; the data should be prepared in a format as in the table below:

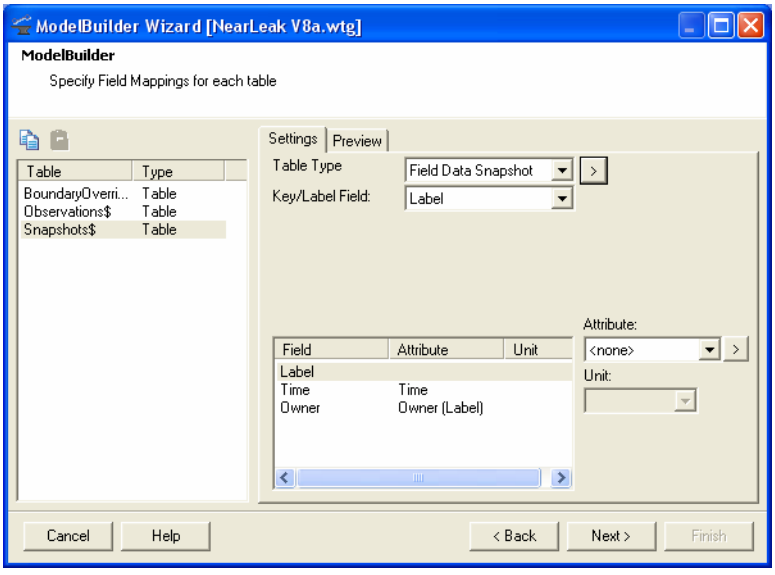
Snapshot Label	Time	Owner
highupstream leak hr 18test 2	18:00	New Calibration Study - Imported Data
highupstream leak hr 5test	5:00	New Calibration Study - Imported Data
even leak hr 8test	8:00	New Calibration Study - Imported Data

Snapshot Label	Time	Owner
even leak hr 18test	18:00	New Calibration Study - Imported Data
highupstream leak hr 8test	8:00	New Calibration Study - Imported Data
highdownstream leak hr 8test	8:00	New Calibration Study - Imported Data
highdownstream leak hr 18test	18:00	New Calibration Study - Imported Data

Once the data source is connected within ModelBuilder, make sure that the attribute is correctly mapped as follows.

1. Highlight the Snapshot table in the left panel
2. Select Field data Snapshot for Table Type under Setting Tab on the right
3. Map the correct attribute for the snapshot data fields.

Example is given as below.



Import Observed Target

The observed targets are the attributes to be matched for the calibration.

The data needs to be prepared as in the table below:

Field Data Snapshot Label	Element Label	Junction Attribute	Pipe Discharge (L/s)	Junction HGL (m)	Element Type
even leak hr 8test	xx3	Hydraulic Grade	0	276.18	Node
even leak hr 8test	xx9	Hydraulic Grade	0	288.68	Node
even leak hr 8test	xx8	Hydraulic Grade	0	288.68	Node
even leak hr 5test	xx1	Hydraulic Grade	0	292.99	Node
even leak hr 5test	xx7	Hydraulic Grade	0	297.58	Node
even leak hr 5test	xx9	Hydraulic Grade	0	296.77	Node
even leak hr 5test	aa		13464.96	0	Pipe
even leak hr 18test	xx3	Hydraulic Grade	0	259.84	Node
even leak hr 18test	xx4	Hydraulic Grade	0	262.17	Node
even leak hr 18test	xx3	Hydraulic Grade	0	280.73	Node
highupstream leak hr 8test	xx7	Hydraulic Grade	0	292.13	Node
highupstream leak hr 8test	aa		26929.89	0	Pipe
highupstream leak hr 8test	xx6	Hydraulic Grade	0	292.15	Node
highupstream leak hr 5test	xx7	Hydraulic Grade	0	297.91	Node
highupstream leak hr 5test	xx4	Hydraulic Grade	0	295.03	Node

To make the mapping for import observed target data, do the following:

1. Highlight Observations (Excel data sheet contains observed target data) Table on the left
2. Select Field data Snapshot, Observed Target for Table Type under Settings Tab
3. Select Field Data Snapshot Label as Key/Label Field
4. Map the data fields correctly as shown previously.

Continue going through the ModelBuilder steps as normal to import the data into Darwin Calibrator.

GA-Optimized Calibration Tips

Darwin Calibrator employs a powerful competent genetic algorithm search method based on the principles of natural evolution and biological reproduction. This kind of search algorithm is well suited to optimization of problems of a non-convex and multiple local-optimal solution nature. Calibration of a hydraulic model falls into this problem category and, as a result, a GA-optimization based search tool, such as Darwin Calibrator, is a sound choice for hydraulic model calibration.

Despite all the good features of GA there are, however, some issues to consider:

- A solution is fitter only in relation to other known solutions and, consequently, a GA has no test for true optimality. As a GA only knows the best solution relative to others, a GA has no precise rule for when to stop. This means that heuristic methods must be used to determine whether to stop a GA run. In Darwin Calibrator you can set a GA run to stop either by:
  - Clicking **Stop.**
  - Setting a maximum number of trial solutions.
  - Setting a maximum number of non-improvement generations, whereby if the fitness of the best solution does not improve by more than a specified tolerance in a set number of generations, then the GA stops.
- A GA is a non-deterministic method that relies to a certain extent on its initial random population (starting locations in the solution space). Thus, each GA run performed may produce different solutions. (If you keep all GA parameters and fitness settings the same, the method is deterministic and will produce identical solutions every time.) Given the fact that a GA has no true test for optimality, after stopping a GA and producing a particular result, there is always the possibility that if you run the GA again you may find a better solution. In fact, it is good practice to run a GA a number of times, each time modifying something about the GA

run (e.g., GA parameters, fitness weightiness, or adjustment group settings), in order to produce another set of potentially better results. At a minimum, the random number seed should be changed for each individual run so that the GA search initiates differently and therefore concludes differently.

- The GA calculates fitness of each trial solution according to the defined objectives for the optimization problem. GA only uses objective means to decide what constitutes a fit solution and what constitutes a less fit solution. The GA has no way of subjectively assessing a solution other than the methods (weightings) built into the definition of the fitness calculation. The best solution found by a GA shouldn't be blindly accepted as being correct. To any single optimization problem there are likely to be many solutions that closely match the required objectives. Due to the fact that the GA has no concept of what constitutes a fit solution, other than its performance against the defined objectives, the GA may produce solutions that are impractical. That is, the GA cannot think for the engineer, it can only search the combination of choices that are presented to it. If the engineer doesn't provide the GA with high quality data and enough or sufficiently flexible options to consider, then the GA may not be able to find a satisfactory solution. Conversely if the GA is presented with too many possibilities to try (e.g., in Darwin Calibrator, if you define excessively large adjustment group ranges combined with small adjustment increments and a large number of adjustment groups), then the efficiency of the GA search is reduced, and the likelihood that the GA will find the correct answer is also greatly reduced. GA is a highly sophisticated search technique, but despite all of its great features, GA still must be used with a degree of engineering judgment and skill. Only then can the engineer expect the GA to find solutions that are not only fit but are practical and likely to represent the real life situation as accurately as possible.
- Uncertainty in field observations should be assessed before these observations are used in an optimization. It is not uncommon for errors in measurement of head loss to be on the same order of magnitude or larger that the actual head loss (Walski, 2000). Such values should not be used in calibration because the calibration algorithm will dutifully try to match the field observations even if they are erroneous. To ensure that head loss is adequate to exceed measurement error, it is helpful to collect data when velocities in pipes are appreciable. In some systems sized for fire protection, demands (and velocities and head losses) are so low most of the time that head loss measurements are meaningless, other than to check pressure gage elevations. Another problem that occurs when calibrating a model is that some of the parameters determined are fixed and knowable at the time the data were taken (roughness, valve status), while others are merely a random observation from a stochastic process (water use). If a C-factor is determined as 90, then that value will be true in the not to distant future. If water use during a pressure observation is determined to be 100 gpm (6.3 l/s), is that value the demand that should be used in modeling, given that it is only one observation from a distribution? The actual water determined from calibration may not be the best value to use for representing the current year status of the system. You need to decide if the water use observed during calibration is the water use that should be used as a basis for future modeling.

Darwin Calibrator Troubleshooting Tips

If you've found your way to this section, then you are probably looking for an answer to a problem that you cannot find elsewhere. Please refer to the list below if you are having problems running Darwin Calibrator (you keep getting unsatisfactory solutions) or if you receive this message while running a calibration: **The calibration engine was unsuccessful. See the help system for troubleshooting tips**.

If you are receiving the **engine unsuccessful** message, try the following:

- Take note of the error message that is provided along with the calibration engine was unsuccessful message. It may provide a clue as to why your calibration didn't run and save you from having to go any further through this list!
- Ensure that the scenario model upon which the calibration is based will run properly in Bentley WaterCAD V8*i* . Select **Analysis > Compute**, select the steady state button, and click **GO**. If the run obtains either a yellow or green light, then the hydraulic model runs and this is not the problem.
- Ensure that all your roughness and demand group settings are valid and reasonable. For example, ensure that roughness adjustments and/or demand adjustments are not such that your hydraulic model might have difficulty converging. For example, make sure that you are not allowing demands to be set too high or pipes too rough, causing excessive amounts of head loss.
- If you have a large number of pipes assigned to status groups, review the need to include all of those pipes as status decisions and try to minimize the number of pipes in status groups.

  Note: Virtual memory settings should only be adjusted by advanced users or system administrators.

- You may be experiencing low system memory. When running Darwin Calibrator, be sure to close any other unused applications and if adjusting advanced GA parameters ensure that you are using a cut probability of more than a few percent, and a splice probability of less than 90 percent. If your system doesn't have much RAM (<128Mb), you may also wish to increase the amount of allocated virtual memory that your system is using. Windows 98/ME users should let Windows manage virtual memory, however, Windows NT4/2000/XP users may wish to increase the size of their system paging file. Please see your Microsoft Windows documentation for information on virtual memory settings specific to your operating system.

If you are having problems getting reasonable calibration solutions, try the following:

- Ensure that the Time field for each of your field data measurement sets corresponds to the time of day that your measurements were taken. The reason being that the time entered in your field data set is used to determine demand multipliers (from hydraulic patterns), which are in turn used to calculate the junction demands that will be simulated within the GA calibration engine. (The demand at a junction during a GA calibration run is the product of its baseline demands and the demand factors at the time specified for the field data set.) Pump settings and control settings, etc., are also determined from the time setting you specify. Demand multiplier adjustments and additional junction demands (e.g., fire flow tests) are in addition to, not in lieu of, junction demands already calculated from pattern multipliers. Also note that a steady state run in Bentley WaterCAD V8*i* will run with only junction baseline demands applied, whereas a GA calibration run based on a steady state scenario will still use pattern multipliers for the specified time.
- Modifying the status of a link can have significant effects on hydraulic results and your chances of finding good calibration solutions. If you are using a number of status group adjustments, you should review why you need those adjustment groups. It may be better to experiment with these kinds of adjustments manually, or get somebody to find out whether that valve really is closed and remove the status decision from the GA calibration. In general, try to keep status adjustment decisions to a minimum.
- Make sure that your adjustment groupings are logical. For example, junctions are grouped by similar pattern or demands for demand groups and pipes are grouped by similar size, age and location for roughness groups.
- Ensure that you do not have too many adjustment groups or the allowable ranges and increments for those groups do not allow too many choices for each group. For example, a roughness group allowed to vary between a Hazen-Williams C of 80 and a Hazen-Williams C of 130, with an increment of 0.1 equates to 500 different possible roughness settings for one group. This is far too high! Try to choose lower and upper bounds, and an increment that will give you no more than 10-12 possible values. If need be, you can start off with course settings (say 80 to 130 with an increment of 5) initially, and gradually refine the allowable range and increment to refine your calibration solutions. This applies to both roughness adjustment groups and also to demand adjustment groups.
- Make sure that you have sufficient and quality field data and that it has been entered correctly. In general, it is a good idea to have as many (or more) field data measurements as adjustment groups for the calibration, or else your calibration problem is under-specified. This means that there is likely to be multiple calibration solutions that produce the same or very similar hydraulic results (e.g., solutions that exhibit compensating errors). In theory, there is only one correct solution, however, due to limits observed for many practical model calibrations, the more quality field data you can provide, the better chance you have of finding a solution that is close to the real situation. When assessing the number of field

observations that you have, consider that each individual observation should contribute unique and accurate information to the calibration. For example, pressure measurements made at two junctions in different parts of the distribution system are likely to be more valuable than two measurements made at locations close to each other in the distribution system. In fact, the two measurements taken at points close together may only be as good as one measurement. That is, both measurements say the same thing about the system. Simply, the field data you collect and enter into Darwin Calibrator should be data that represents times when your system is experiencing high demand, even if it is only the result of such activities as fire flow tests. The reason for this is that during times of normal demands, the head loss across the system is usually on the same order of magnitude as the error in measuring head loss. Therefore, small errors in measurement can lead to huge errors in roughness coefficient or demand.

- Make sure that you haven't entered field data observations that are made impossible to achieve by any observed boundary conditions, such as an observed grade out for a PRV set to a different grade.

  Note: Tank levels, pump speed settings, valve settings, and reservoir HGL are all used by the calibration engine as boundary conditions and as such these field data entries will not appear in the calibration report summary. That is, these quantities are set as fixed in the calibration simulations and the calibration does not try to match these data. All other quantities are used as observed quantities that the calibration engine tries to match by adjusting parameters defined in your adjustment groups.

- Make sure you are using the correct boundary conditions. If you have entered observations for tank levels etc., ensure that you have not made any errors in entering the data.

Optimizing Capital Improvement Plans with Darwin Designer

12

Darwin Designer

Design Study

Optimized Design Run

Manual Design Run

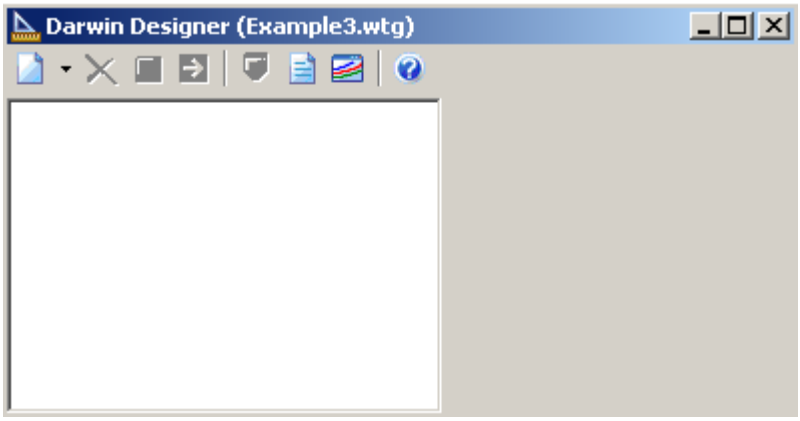
Manual Cost Estimating

Darwin Designer

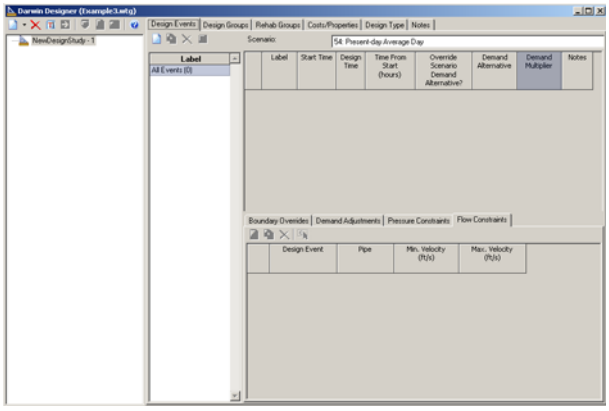
Darwin Designer allows you to design new pipe layouts or pipe rehabilitation for existing pipes. A genetic-algorithm based approach avoids a manual trial and error approach to finding the most efficient design. Solutions and costs calculated using Darwin Designer can be exported back to any scenario.

To open Darwin Designer

1. Start Bentley WaterCAD V8*i* .
2. Go to Analysis > Darwin Designer.



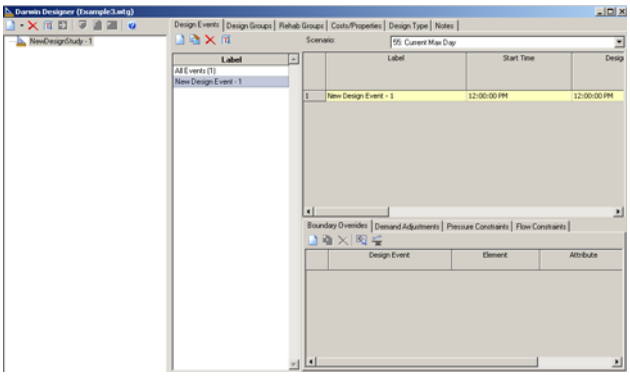
3. Click New Designer Study.



Design Study

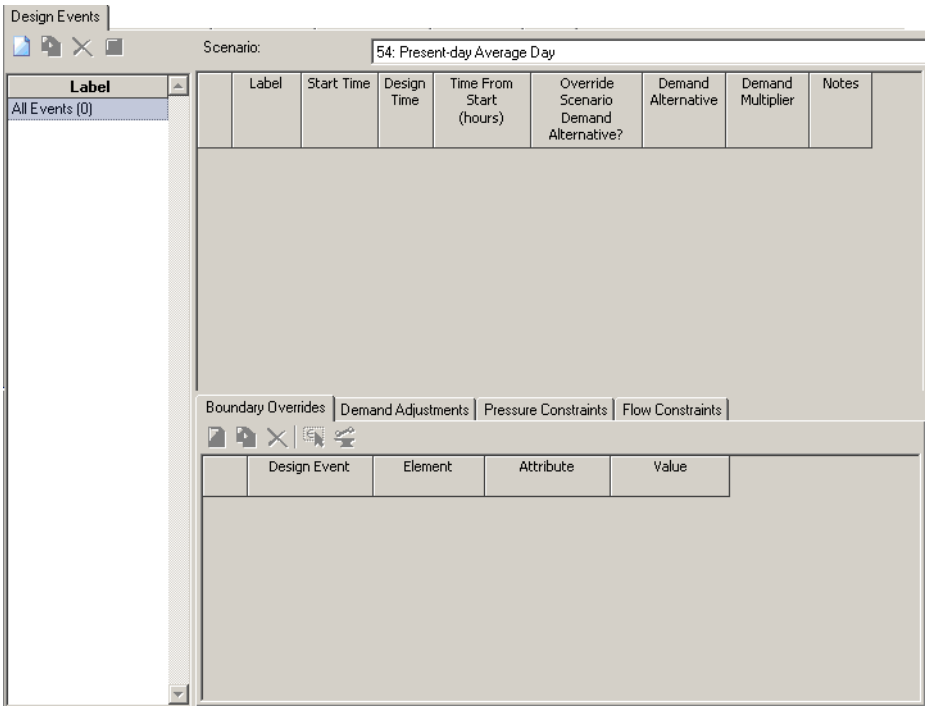
A design study is a top-level grouping of the pipe design and rehabilitation you want to do for one complete design project. A design study should be used to represent a real project unit, such as a system expansion, main replacement, system augmentation, etc. For different or unrelated projects—such as a main replacement project and a project to design a new service area—you should use different, new design studies.

To start using Darwin Designer, you must first create a design study. All Darwin Designer data exists within design studies.

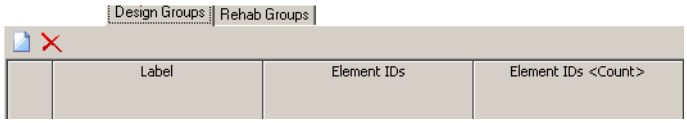


A design study includes the following

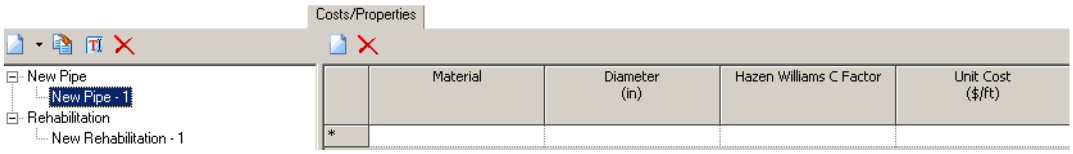
1. A description of the events that serve as the basis for design.

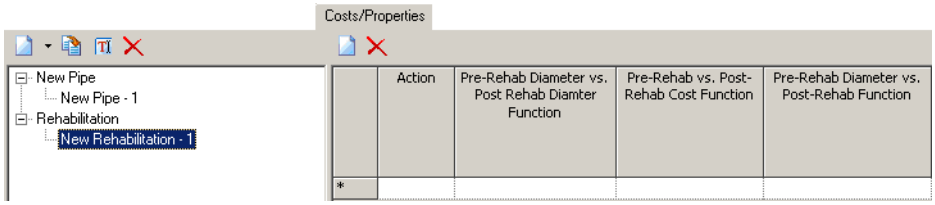


2. A set of pipes being sized or rehabilitated.
3. Constraints you must meet, which are defined in a design event.
4. A range of design sizes or rehabilitation options.

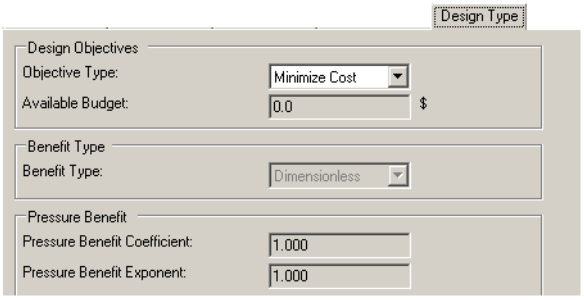


5. Cost data for use in the optimization.





6. Genetic algorithm options.
7. A number of design runs to test the design.



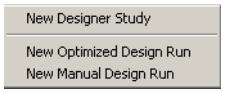
8. The results of design runs.

It is apparent that one or more of these items will be different between different design studies, hence the ability to create as many design studies as you need.

You can create more than one design study. Each design study can include one or more design runs. Each design run is manual or optimized. The particular events and groups are specified by making them active. You may create many design runs within a design study.

In the design study, create the groups of pipes for design and rehabilitation, define the design/rehab options (costs and sizes, etc.), and define constraints and parameters for your designs. These items get used in the design runs and the computations that produce your design results.



New 	 • New Designer Study - More than one design study can be added and design studies are not related. • New Optimized Design Run - Add an optimized design run. Optimized design runs use a genetic algorithm. • New Manual Design Run - Add a manual design run for specific solution alternatives for trial-and-error calculations.
Delete 	Click to delete the selected design study.
Rename 	Click to change the name of the selected design study.
Compute 	Click to compute the run.

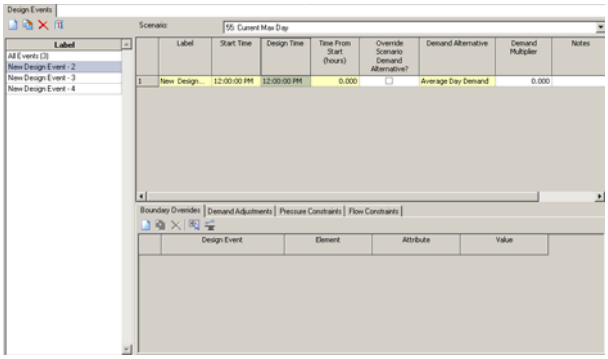


Export to Scenario 	Click to export your results as an alternative to your WaterCAD V8*i* scenario. Export creates a new scenario and then can export the following data to alternatives. • Physical Alternative data: diameter, roughness, and material. • Active Topology Alternative: If the pipe diameter is 0, the pipe is made inactive in the active topology alternative.
Report 	Click to present the data in the Report Viewer.
Graph 	Click to display a graph of the results.
Help 	Click to open WaterCAD V8*i* Help.

Design Events tab

In producing a system design, the design must typically achieve some objective or objectives. Generally, a design must supply some specified demands, while concurrently meeting specified performance criteria, subject to specific boundary conditions, such as tank levels, or emergency conditions.

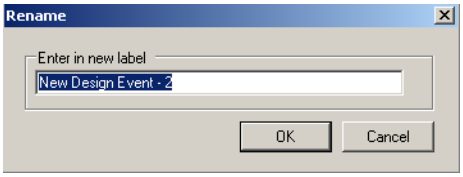
Use Design Events to create or edit design events used as parameters for your designs or rehabilitation of systems. Design events are used to define the requirements of your designs. Design events include information about the demand conditions a design must satisfy, the performance requirements or constraints a design must meet (in the form of pressure and flow constraints), and also the boundary conditions under which the design must achieve the previous two goals.



In order to create a design using Darwin Designer you need at least one design event, however, in many cases you will use more than that. A design event represents a single time step hydraulic analysis that will be analyzed by Darwin Designer.



New 	Click to add a new design event.

Duplicate 	Click to create a copy of the selected design event. This can be an efficient way to create a new design event that has many of the attributes of an existing event.
Delete 	Click to delete the selected design event.
Rename 	Click to change the name of the selected design event. When the rename box opens, type in the new name, and then click OK. 
Scenario	Select the scenario that should be used for the design and calculations. The menu displays scenarios that have already been defined in your project. 

Scenarios

The scenario selected is what Darwin Designer will base its designs. The scenario must contain any and all data that will be considered for design purposes. It must be either a Steady State or EPS scenario.

The types of data that this includes

- Topological data, such as the locations of existing and possible new facilities. Pipes that do not currently exist (Designer will be used to size them); it is recommended that you model them as open pipes with small diameters (e.g., 0.01 inches or 0.01 mm). It is also advisable to adopt a naming convention, such as FP-1, FP-2 (Future Pipe) or GA-P-1, GA-P-2. It is also possible to consider the inclusion/exclusion of other facilities using topological data.
- Physical data, such as pipe diameters, lengths, tank diameters, elevations, etc.
- Initial Settings data, such as tank levels, control valve statuses, etc.
- Demand data, such as loading patterns, nodal demands, fire flows (as nodal demands).

After you select a scenario, it is possible within Darwin Designer to set up multiple design events that specify differences over and above the scenario. It is possible to specify additional demands and also different boundary conditions. In this way, you can set up a suite of design events that capture the design requirements of the project. As an example, the scenario might reference peak hour demands. In this case, you could set up a design event that uses the scenario unchanged to ensure the design meets peak hour flows, and then you could add in additional design events that specify fire flows (additional demands) or emergency conditions, such as pipe breaks (boundary conditions).

The first component of a design study is the design event that is being analyzed. It is in the design event that you describe the flows that must be delivered and the constraints that must be met.

There are several different ways to modify or overwrite the demands in the representative scenario.

- Override Scenario Demand Alternative—This option allows selecting a new demand alternative to use in lieu of the demand alternative referenced by the representative scenario. In this way, you can set up all of your different demand cases in Bentley WaterCAD V8*i* before starting Darwin Designer, and then reference them by selecting **Override Scenario Demand Alternative** and selecting the appropriate demand alternative. Using this option eliminates the need for the following options but does not preclude their use.
- Adjust demands with a fixed multiplier—In some cases, the demands for the representative scenario might be for an average day and you would like to adjust them for a peak hour. To do so, enter a demand multiplier to adjust it. Note that the multiplier you should enter is the value needed to adjust the demands at the specified time to the desired value. Assuming that the time from start was already 7 hours, which equated to 7 a.m. in a particular model, and you want to adjust demands up to the 9 p.m. peak. Rather than enter the 9 p.m. peak multiplier, you should enter the ratio of the 7 a.m. multiplier and the 9 p.m. multiplier. For example, if the 7 a.m. multiplier is 1.3 and the 9 p.m. multiplier is 1.6, then 1.23 should be used as the demand multiplier. This is illustrated as follows:

  1.3 x 1.23 = 1.6

  Thus it is true to say that the demand for any single junction is calculated by:

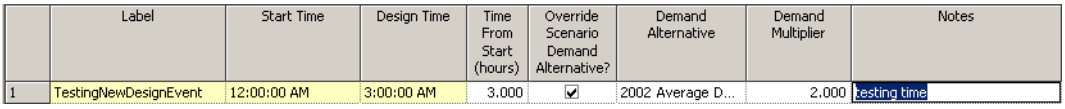
  Qc = Qb * DMt * DM

  Where: Qc = calculated flow

  Qb = base flow

  DMt = demand multiplier at time t (Time from start) determined for demand patterns

  DM = specified demand multiplier (default is 1.0)



	Label	Start Time	Design Time	Time From Start (hours)	Override Scenario Demand Alternative?	Demand Alternative	Demand Multiplier	Notes
1	TestingNewDesignEvent	12:00:00 AM	3:00:00 AM	3.000	☑	2002 Average D...	2.000	testing time

Label	The name of the event.
Start Time	The time at which the scenario is set to begin. This is the clock time for the start of the hydraulic simulation defined as part of the representative scenario calculation properties.
Design Time	Scenario start time plus time from start. This is the clock time that the Time From Start value represents.
Time from Start (hours)	Only adjustable when the representative scenario is set for EPS, the time from start specifies the time to use as the basis of design. That is, for a model with a scenario start time of 12:00:00AM, a time from start value of 7 equates to 7:00:00AM. The result is that Darwin Designer will, for the current design event, simulate demands as the base demands multiplied by their respective pattern multipliers at 7:00:00AM. In short, the demands at 7 a.m. are used. It is easy to see that you can set up multiple design events that consider demands at different times in the day, simply by adjusting the Time From Start value.
Override Scenario Demand Alternative?	Select this check box to override the displayed Demand Alternative and to use the Demand Multiplier. Clear this check box if you do not want to use the Demand Multiplier.
Demand Alternative	Displays the Demand Alternative associated with the selected set of observations.
Demand Multiplier	Set a demand multiplier that is applied to your water model at that time from start. For example, if you have knowledge that your demand is higher or lower by a specific percentage, you can set that value here.
Notes	Type information to be stored on this design event.

Boundary Overrides tab

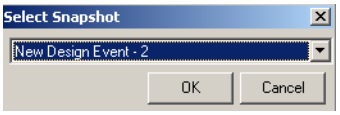
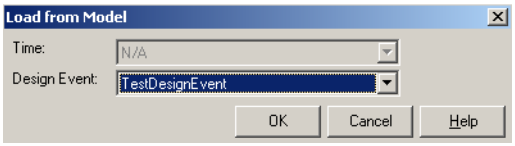
Boundary overrides are explicitly specified for each design event and used for evaluating a trial design solution for a design event.

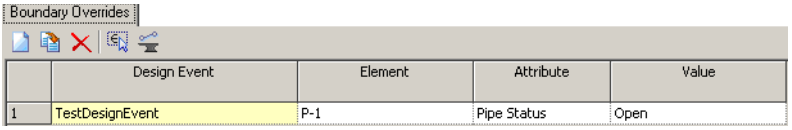
Boundary conditions can be used to override initial settings from the design representative scenario for a design event. For example, if you want to simulate a pipe break, you can set the status of a pipe to closed for a pipe-outage design event. Similarly, valve settings can be applied, tank levels, and so on. Without a specified boundary condition for a design event, Darwin Designer will apply the initial settings from the representative scenario when evaluating the corresponding design event.

When calculating an EPS model to get boundary conditions, Darwin Designer uses the sizes, demands, etc., that are present in the representative scenario. If the representative scenario includes lots of unsized pipes, then you will need to override the appropriate boundary conditions (such as, a tank in a new part of the model). If you do not specify a time step on the Demand Adjustments tab, the initial conditions at time 0 will be used.

You only need to explicitly state a boundary condition if you wish to change it from the default. Do not try to look at boundary conditions by selecting All Pipes or All Pumps because this sets all pipes to Closed or all pumps to Off.



New 	Click to add a new design event. Opens the Select Snapshot box where you can select a new design event or an existing design event.  Click OK after you make a selection.
Duplicate 	Click to create a copy of the selected design event. This can be an efficient way to create a new design event that has many of the attributes of an existing event.
Delete 	Click to delete the selected design event.
Initialize Table from Selection Set 	Click to open the Initialize Table from Selection Set box where you can choose the Selection Set and the Design Event.  Click OK to run.
Load from Model 	Click to open the Load from Model box. Load settings and conditions for your elements at a time from start that you specify. For example, if your peak time is 6 pm, you can load the settings for your elements from the model at that time.  Click OK to run.

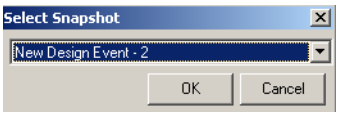


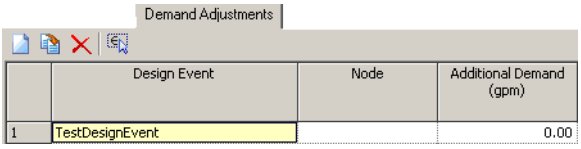
Design Event	The name of the event.
Element	Click the ellipsis to select from the drawing the type of element to set a boundary condition: pump, tank, pipe, or valve.
Attribute	The attribute list reflects your selection of an element type.
Value	Open, Closed, On, Off, or a numeric value depending on the selected attribute.

Demand Adjustments tab

The sizing of pipes in designer is driven by demands. By default, the demands used will be those associated with the representative scenario. However, you may want to use different demands, such as fire flows or peaks.



New 	Click to add a new design event. Opens the Select Snapshot box where you can select a new design event or an existing design event.  Click OK after you make a selection.
Duplicate 	Click to create a copy of the selected design event. This can be an efficient way to create a new design event that has many of the attributes of an existing event.
Delete 	Click to delete the selected design event.
Initialize Table from Selection Set 	Click to open the Initialize Table from Selection Set box where you can choose the Selection Set and the Design Event.  Click OK to run.

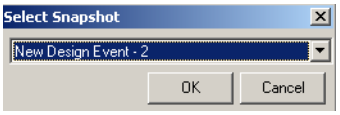


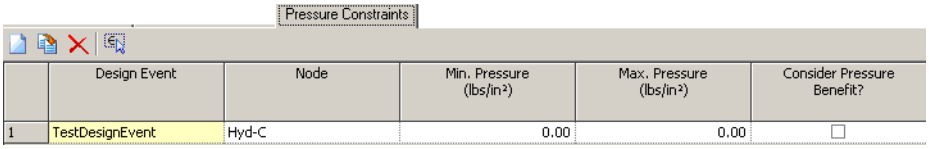
Design Event	The name of the event.
Node	Click the ellipsis to select the node from the drawing.
Additional Demand	Fire flows or other special cases can be achieved by adding demand adjustments to individual junctions: by selecting the junction and specifying the additional demand. If necessary, demands can also be subtracted by specifying a negative number. Be sure to enter demands in the correct flow units.

Pressure Constraints tab

Use this tab to define pressure constraints for all junctions or a set of junctions.



New 	Click to add a new design event. Opens the Select Snapshot box where you can select a new design event or an existing design event.  Click OK after you make a selection.
Duplicate 	Click to create a copy of the selected design event. This can be an efficient way to create a new design event that has many of the attributes of an existing event.
Delete 	Click to delete the selected design event.
Initialize Table from Selection Set 	Click to open the Initialize Table from Selection Set box where you can choose the Selection Set and the Design Event.  Click OK to run.

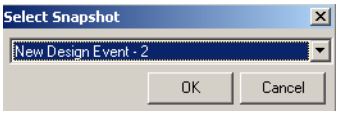


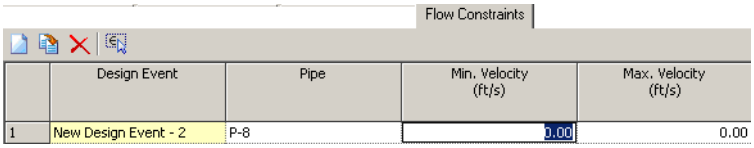
Design Event	The name of the event.
Node	Click the ellipsis to select the node from the drawing.
Min. Pressure	Set a minimum pressure that you require for the selected set of junctions. Violations of this boundary are displayed when you calculate your network.
Max. Pressure	Set a maximum pressure that you require for the selected set of junctions. This value cannot be lower than the minimum pressure you set. You can set this to an unusually high value if you are unconcerned with maximum pressure. Violations of this boundary are displayed when you calculate your network.
Consider Pressure Benefit?	Select this check box if you want the genetic algorithm to consider the benefits provided to your design by higher system pressures.

Flow Constraints tab

Use this tab to define flow boundary conditions for a junction or set of junctions.



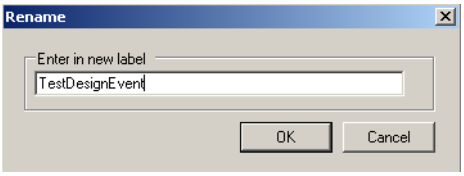
New 	Click to add a new design event. Opens the Select Snapshot box where you can select a new design event or an existing design event.  Click OK after you make a selection.
Duplicate 	Click to create a copy of the selected design event. This can be an efficient way to create a new design event that has many of the attributes of an existing event.
Delete 	Click to delete the selected design event.
Initialize Table from Selection Set 	Click to open the Initialize Table from Selection Set box where you can choose the Selection Set and the Design Event.  Click OK to run.



Design Event	The name of the event.
Pipe	Click the ellipsis to select the pipe from the drawing.
Min. Velocity	Set a minimum velocity that you require for the selected set of pipes. Violations of this boundary are displayed when you calculate your network.
Max. Velocity	Set a maximum velocity that you require for the selected set of pipes. You can set this to an unusually high value if needed. Violations of this boundary are displayed when you calculate your network.
Consider Pressure Benefit?	Select this check box if you want the genetic algorithm to consider the benefits provided to your design by higher system pressures.

To create a new Design Event

1. Select the Scenario to base your design.
2. Click New  .
3. Select the new event in the Label field and click rename 
4. Type a name for the design event and then click OK.



5. Enter the data to define the design event.

Design Groups tab and Rehab Groups tab

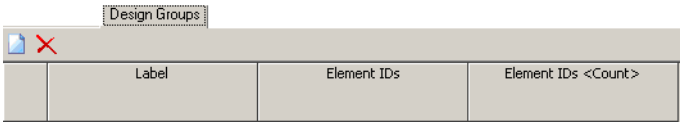
Darwin Designer determines the size or rehab action for pipes. It is unlikely, however, that a large pipeline will change diameter every block along its route. Plus, if fewer pipes were being sized, optimization will happen faster than if a larger number of pipes were sized. Therefore, Darwin Designer uses the idea of a pipe group or rehab

group to group pipes that will attract the same design decision. At the end of a run, all of the pipes in the same design group are given the same diameter, and all of the pipes in the same rehab group receive the same rehab action. This is both logical and more efficient from a computational standpoint.

For a pipe to be considered a candidate for design or rehab, it must be placed in a group. This is done on the Design Groups or Rehab Groups tab when the Design Study is highlighted. (When the Design Run is highlighted, you choose which groups are to be considered during that run.)

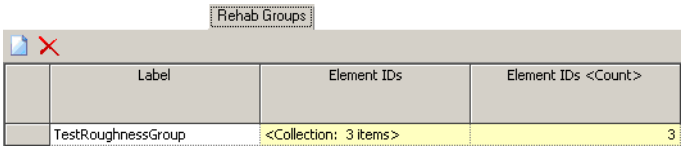
You must insert at least one pipe in each design group. There is no absolute rule for deciding which pipes belong in a given group. Usually it is the set of pipes that will be laid with the same diameter and at the same time, but it can also be smaller groups than that, and in the case of smaller design problems or academic exercises, it may be only 1 pipe per group, which is easily expedited with the **Create Multiple Design Groups** selection. The down side of adding every pipe to its own group, however, is that this can be computationally inefficient and potentially leads to a pipeline that is say 12 in. for one block, 8 in. for the next, 6 in. the next, etc., which may be a theoretically least-cost design but is not a solution that is likely to be installed. Ultimately the choice comes down to a trade-off between number of pipe groups (and size of the optimization problem) versus constructability of the design through the potential for different pipe sizes adopted for each group.

Design Groups tab



New 	Click to add a new demand group.
Delete 	Click to delete the selected demand group.
Label	Type in the field to rename the demand group.

Rehab Groups tab



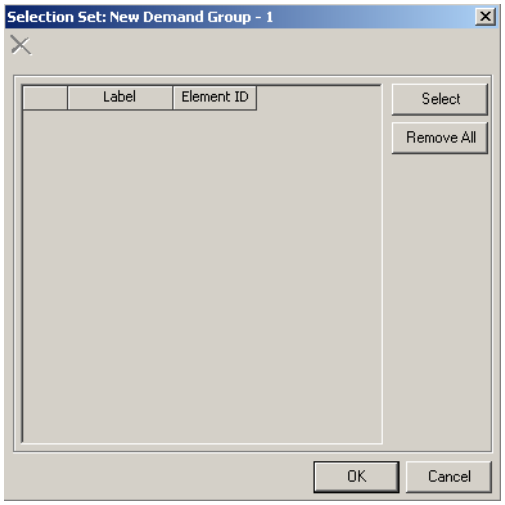
New 	Click to add a new roughness group.
Delete 	Click to delete the selected roughness group.
Label	Type in the field to rename the roughness group.

To add a new design or rehab group

1. Click New  .
2. Type in the Label field to rename the demand group.
3. In the Element ID field, click the ellipsis to select the pipes included in the group.



4. The Selection Set box opens.



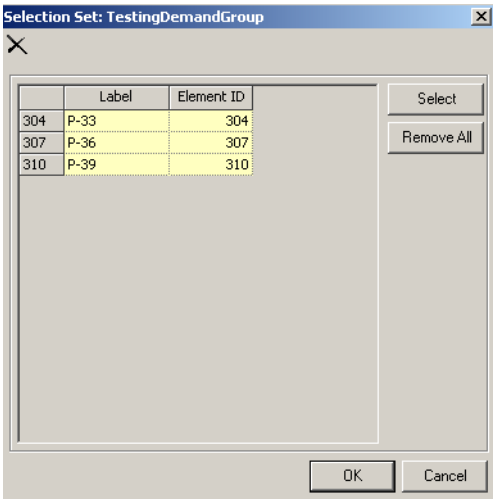
Click Select.

5. Use the Select box to either choose items from the drawing to include in the group, or click Query to build a query for this group.

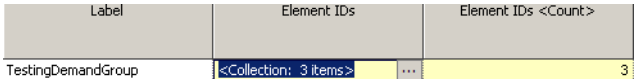


Click Done  when finished.

6. Click OK to create the group or Cancel to exit without creating the group.



7. The Element ID field will show the new Collection and the Element IDs <Count> field will show the number of pipes in the group.

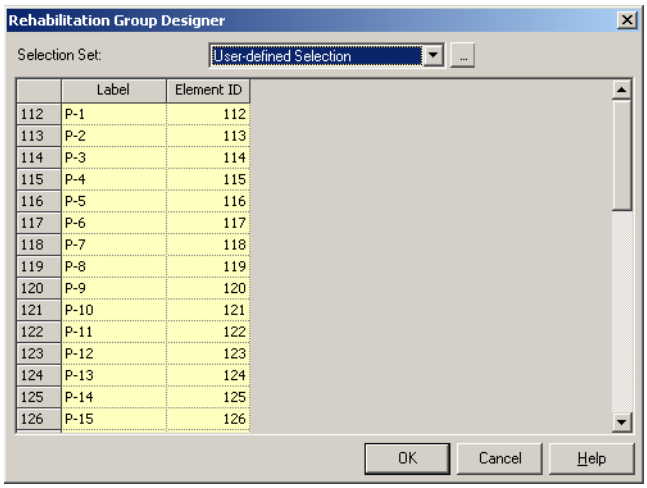


To make changes to a design or rehab group

1. Click the ellipsis  in the Element ID field.
2. In the Selection Set box, you can either remove the pipes and/or junctions you want to include in your group, or add additional pipes and/or junctions.
3. After you have selected the elements, click OK to apply your changes to the group or click Cancel to exit without making any changes.

Rehabilitation Group Designer Dialog Box

This dialog allows you to create rehabilitation groups by selecting elements from the drawing or by using predefined selection sets.

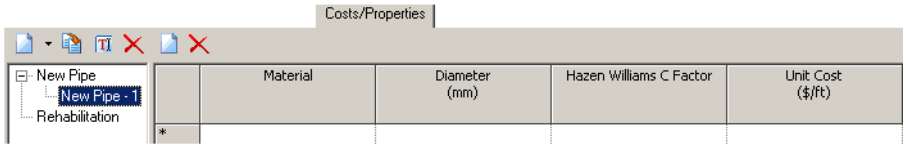


Choose a predefined selection set from the Selection Set list, use all elements by selecting <All Available>, or click the ellipsis button to pick a group of elements from the drawing. The elements that are part of the group are displayed in the list pane at the bottom of the dialog.

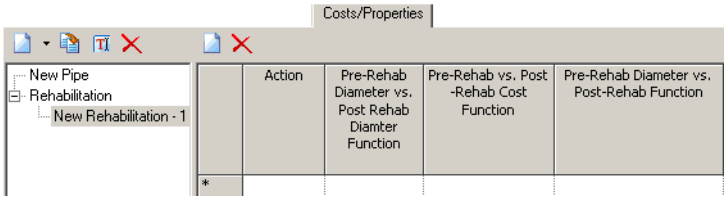
Costs/Properties tab

Costs/Properties are used by Darwin Designer to determine the hydraulic effect and calculate the capital cost of the solutions it generates. Cost/Properties come in two types: Design Option Groups (new pipes) and Rehab Option Groups (rehabilitation actions).

Design options (new pipe sizes and associated roughness, material type and unit cost) are defined by adding design option groups.



Rehab Options (rehab actions and associated post action functions) are defined by adding rehab option groups.



Each option group contains a set of options that Darwin Designer can select from in order to create its hydraulic solutions. Design Option Groups are used where you are designing a new system or part of a system and brand new pipes need to be installed. Rehab Option Groups are used when you are examining the effect of rehabilitating (cleaning, lining, etc.) existing pipes.

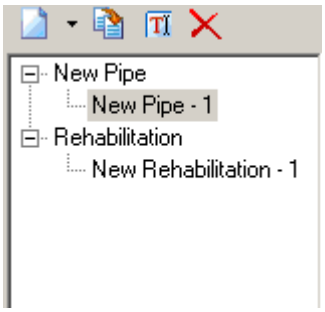
Adding and Editing Design Option Groups

Design Option Groups are used to define a selection of pipes that can be used in your design. You may choose to use as much or as little detail as you wish. For example, for a rough cut design, you may simply wish to use nominal diameters and estimated unit rates, but for a detailed design you may wish to use internal pipe diameters and even distinguish between different materials. The new pipe option group is set up to allow you to adopt either approach.

In setting up option groups, you can set up as many groups as needed to describe the different cost situations in your project. For example, you may decide that you have three different cost types that need to be considered: Residential, Greenfields and Commercial. In this case, you can set up three different option groups to reflect the different in-ground costs for each of the three different cost types. For example, Greenfields would be cheaper than Residential, where the additional costs of breaking the road and resurfacing need to be included. Not all groups need to include the same pipe sizes either, so you may choose to use different option groups as a way of limiting certain pipe groups to being able to attain only certain sizes. For example, there is not much point allowing a transmission main to be sized as a 6-in. pipe, where a consumer connection pipe might be acceptable as a 6-in. pipe.

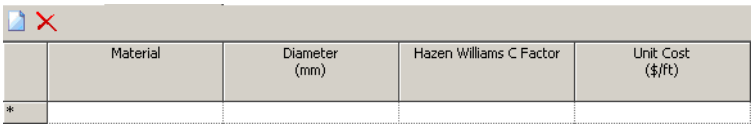
Darwin Designer has the ability to not only size new pipes from a range of possible available pipe sizes, but it can also determine whether a particular pipe needs to be constructed at all. To get Designer to determine whether a pipe needs to be constructed at all, simply add a zero diameter option to the pipe option group. The zero diameter option should also attract a cost of zero (in this case, roughness is redundant). The zero size option can be used to size parallel pipes and it can also be used to determine the optimal design layout, whereby more pipes are being sized than are necessary to service all demands.

For pipes that are essential for service and that must be sized, define and use a pipe-option group that contains no zero diameter option.



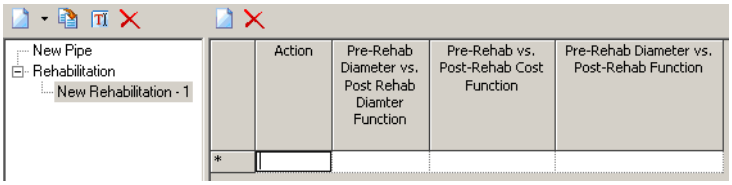
New 	Click to add a new option group. 
Duplicate 	Click to create a copy of the selected option group. This can be an efficient way to create a new option group that has many of the attributes of an existing event.
Rename 	Click to change the name of the selected option group.
Delete 	Click to delete the selected option group.

For Design Option Groups



New/ Delete 	Click New or Delete to add or remove rows from the table.
Material	Click the ellipsis to open the Engineering Libraries box to select the pipe material.
Diameter	Type a diameter for the pipe.
Hazen Williams C Factor	Type the roughness value for the pipe.
Unit Cost	Type the unit cost value for the pipe.

For Rehab Option Groups



New/ Delete 	Click New or Delete to add or remove rows from the table.
Action	Type the name of the rehabilitation action you are creating.
Pre-Rehab Diameter vs. Post Rehab Diameter Function	Select or create the function to use for the rehabilitation action you are creating. This function describes the pre- and post-rehabilitation pipe diameters. You must create at least one function for pre-rehabilitation diameter versus post-rehabilitation diameter.
Pre-Rehab vs. Post-Rehab Cost Function	Select or create the function to use for the rehabilitation action you are creating. This function describes the cost of the action per length for pipe of a given pre-rehabilitation diameter. You must create at least one function for diameter versus cost.
Pre-Rehab Diameter vs. Post Rehab Function	Select or create the function to use for the rehabilitation action you are creating. This function describes the pre-rehabilitation diameter versus the post-rehabilitation pipe roughness. You must create at least one function for diameter versus roughness.

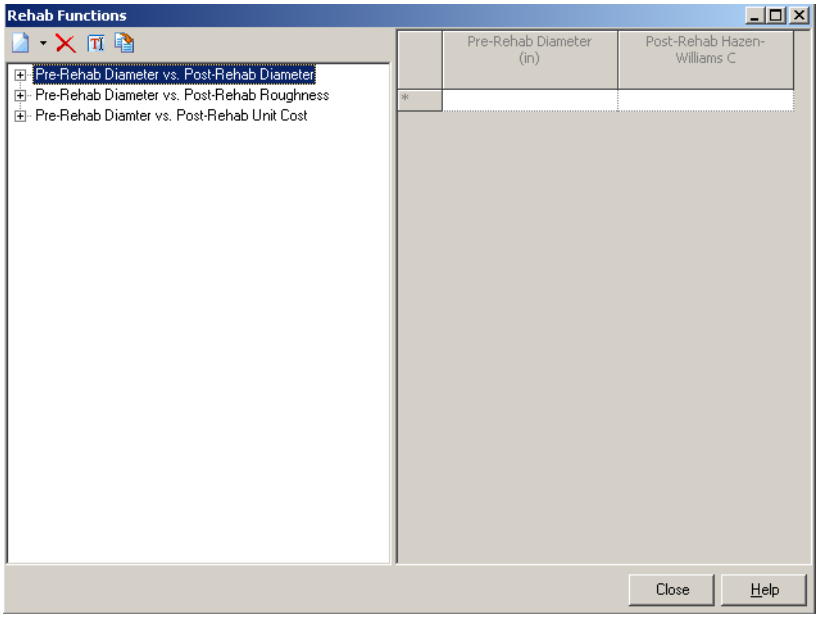
Rehab Option Groups are used to define the selection of rehab actions that can be used in the design. You may choose to use as much or as little detail as you want. You can set up as many groups as you need for different cost types, and not all groups need to include the same rehabilitation options.

Rehab option groups define the selection of rehab actions that can be used in the design. There can be as much detail as needed, as many groups have different cost types, and not all groups need to include the same rehab options.

In setting up option groups, you can set up as many groups as needed to describe the different cost situations in your project.

To define a rehab option group

1. Click New > Rehab Option Group or right-click Rehabilitation > New Rehabilitation.
2. Click to rename and type the name.
3. Type a name in the Action field.
4. Select the three functions that describe the pre- and post-rehabilitation conditions. You must select one of each type of function for a rehabilitation action.
   a. Click the arrow to select a previously defined function.
   b. Or click the Ellipsis (…) to open the Rehab Function manager where you can define a new function.



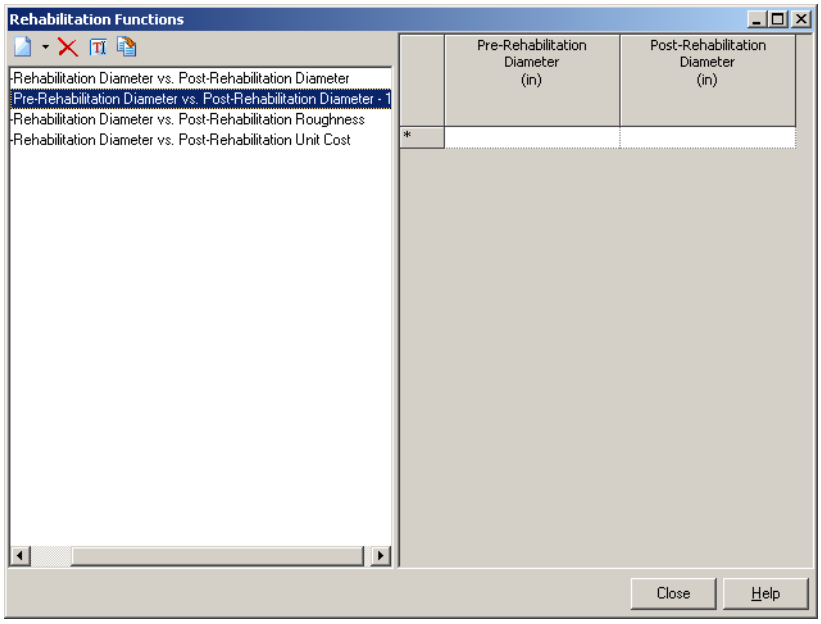
5. As needed, click New or Delete to add and remove rows.
6. Create as many rehabilitation actions as needed.

Rehabilitation Functions

Use the Rehabilitation Functions manager to create a rehabilitation function.

To create a rehabilitation function from within a table in the Cost/Properties tab

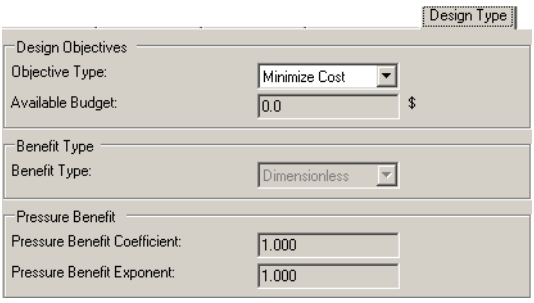
1. Click in one of Pre-Rehab fields and click the ellipsis (…) to open the Rehab Functions manager.



2. Click New to open the menu and select one of the options.
3. Type in the necessary information in the corresponding field.
4. Click Close.

Design Type tab

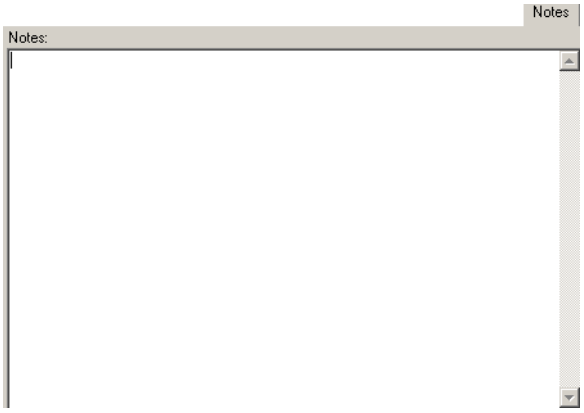
The Design Type tab allows you to design and weigh benefits so the genetic algorithm knows better what your design priorities are.



Design Objectives	Objective Type - the overall priority of the design. Select one of the following: • Minimize Cost sets price as your primary concern and the genetic algorithm will consider costs most heavily. • Maximize Benefit sets the performance of the system as the highest priority. The system performance is measured by the pressures at specified junctions using pressure benefits. • Multi-Objective Trade-off allows the genetic algorithm to consider where the best compromise lies between cost and pressure benefit. This selection has higher computational requirements than the other design types. Available Budget - Type a dollar amount. This field is not available for Minimize Budget.
Benefit Type	Select Dimensionless or Unitized benefit for Maximized Benefit or Multi-Objective Trade-off. • Dimensionless - If pressure improvement is not a primary concern, dimensionless benefit considers the ratio of pressure improvement to minimum pressure for selected junctions. • Multi-Objective Trade-off - If you are looking for a specific pressure improvement from your system, unitized benefit considers the average pressure increase for selected junctions.
Pressure Benefit	Set the Pressure Benefit Coefficient and the Pressure Benefit Exponent. These increase the weighted value of pressure in your network. Exponent has a larger affect on the weighted value than the same number for the coefficient.

Notes Tab

Use the Notes tab to type comments about your project and read things like log entries and dates.



Initialize Table From Selection Set Dialog Box

This dialog is used to load data from an existing selection set into the current table. The dialog consists of the following controls:

In Designer:

Selection Set - This menu contains a list of selection sets. Choose the one that contains the data you want to load.

Design Event - This menu contains a list of the design events. Choose the destination for the selection set data initialization.

In Darwin Calibrator:

Selection Set - This menu contains a list of selection sets. Choose the one that contains the data you want to load.

Owner Element - This menu contains a list of the field data snapshots. Choose the destination for the selection set data initialization

Load From Model Dialog Box

Click to open the Load from Model box. Load settings and conditions for your elements at a time from start that you specify. For example, if your peak time is 6 pm, you can load the settings for your elements from the model at that time.

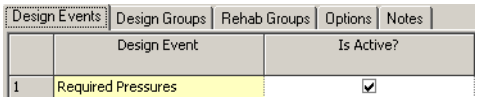
Optimized Design Run

As part of any design study, you will want to make numerous design runs. A design run is a single, complete solution of the problem consisting of the design events, groups, and other options plus the results of the design run.

The way that you decide to use an event or a constraint is to make it active by checking a box. You must have at least one active design event and one active design or rehab group to make up a design run.

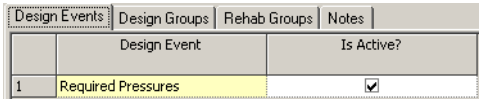
To create a design run, right-click the design study that the run is to be part and choose:

- Add a new optimized design run.



or

- Add a new manual design run.



or

- Select an existing design and duplicate it.

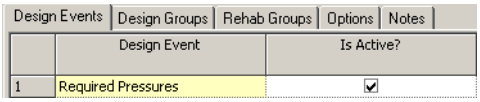
Each time you want to run an optimization, you can create a new run or edit an existing run.

Design runs can either be GA optimized or manual runs. A GA optimized design run uses genetic-algorithm optimization to optimize the selected objective (e.g., minimize cost) for your design. A manual design run allows you to make a single selection of pipe sizes and/or rehabilitation actions in order to evaluate the specified design against the same criterion as a GA optimized design. The difference between the two kinds of run is that a manual run does not use GA optimization, and it executes a single solution evaluation using the pipe sizes and rehabilitation options that you selected.

Note: Darwin Designer prevents runs from executing that would only result in a handful of unique combinations of trial solutions. In such a case it's more efficient to enumerate all possible solutions manually.

Design Events tab

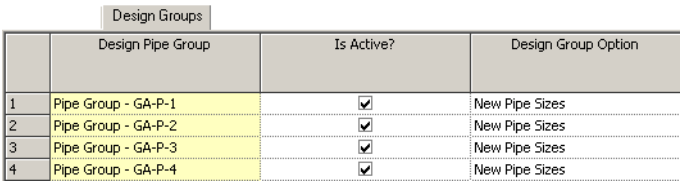
The Design Events tab displays a list of the events you have set up. Select the check boxes to set as Active those criteria that you want to be used in the calculation of your design run. Your design run must have at least one active design event in order to be calculated without error.



Design Events	Lists the design event.
Is Active?	Select the check box for the design events to be included in the current design run.

Design Groups tab

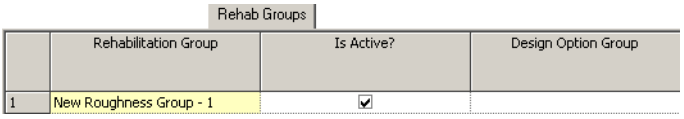
You must have at least one active design or rehab group set to a valid design or rehab option group.



Design Pipe Group	Lists the names of the design pipe groups.
Is Active?	Select the check box for the design groups to be included in the current design run.
Design Group Option	For each design group, you must select the design option group (set of possible pipe sizes) you want to use.

Rehab Groups tab

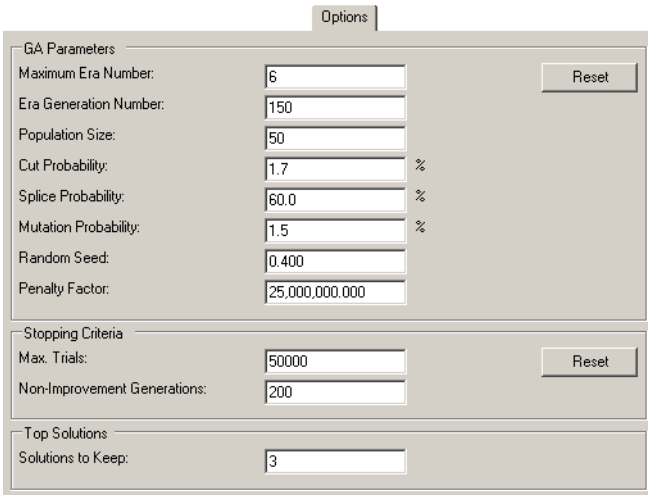
You must have at least one active rehab group set to a rehab option group.



Rehabilitation Group	Lists the names of the roughness groups.
Is Active?	Select the check box for the design groups to be included in the current design run.
Design Option Group	For each design group, you can select the design option group you want to use.

Options tab (Optimized Run only)

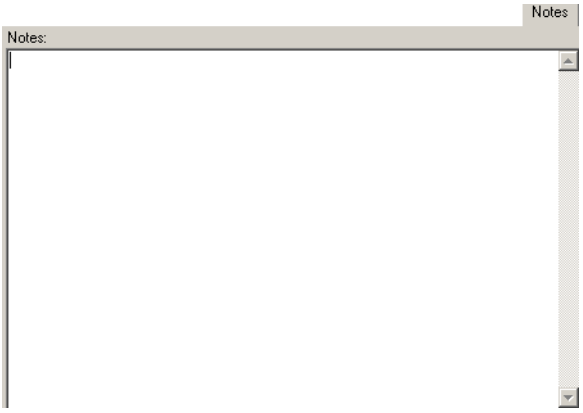
The Options tab is where you define the parameters for the genetic algorithm. Options relate to optimized design runs only and therefore are not available for manual design runs. Use these settings to fine-tune the way the GA finds results. If adjusting a particular GA control gives you better results, pursue the approach to maximize your design.



Stopping Criteria	• Max. Trials - Set the maximum number of calibration trials you want the GA to process before stopping. • Non-Improvement Generations - Set the number of maximum number of non-improvement generations you want the GA to process without calculating an improved fitness. If the GA makes this number of calculations without finding an improvement that is better than the defined Fitness Tolerance, the GA will stop. Non-Improvement Generations works in conjunction with Fitness Tolerance.
Top Solutions	• Solutions to Keep - Select the number of solutions you want to keep. For a design type of Minimize Cost or Maximize Benefit, Darwin Designer retains the top feasible solutions according to the value of the objective function. If the user-specified number of top solutions is greater than the number of feasible solutions found, Darwin Designer reports all the feasible solutions found.

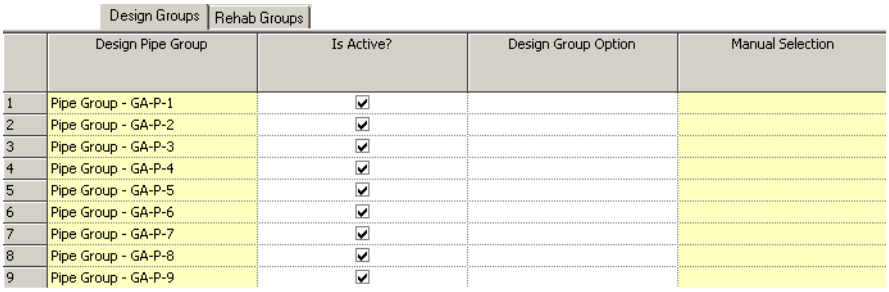
Notes Tab

Use the Notes tab to type comments about your project and read things like log entries and dates.



Manual Design Run

Manual selections are used to force Darwin Designer to use specific designs in calculating costs of a network. The difference between a manual design run and an optimized design run is the Manual Selection column in the Design Groups and Rehab Groups tab for the run. After you select a table to use for a group, you then must set that group to use a specific pipe size or specific rehabilitation action.



	Design Pipe Group	Is Active?	Design Group Option	Manual Selection
1	Pipe Group - GA-P-1	☑		
2	Pipe Group - GA-P-2	☑		
3	Pipe Group - GA-P-3	☑		
4	Pipe Group - GA-P-4	☑		
5	Pipe Group - GA-P-5	☑		
6	Pipe Group - GA-P-6	☑		
7	Pipe Group - GA-P-7	☑		
8	Pipe Group - GA-P-8	☑		
9	Pipe Group - GA-P-9	☑		

Examples of why you might use a manual design

- You might use a manual design to test some hand calculations you have made or to reproduce an optimized design that you want to force manual overrides.
- You could create a manual design run in which you force the groups of pipes to specific sizes.
- You might create a rehabilitation design that forces groups to use specific actions.

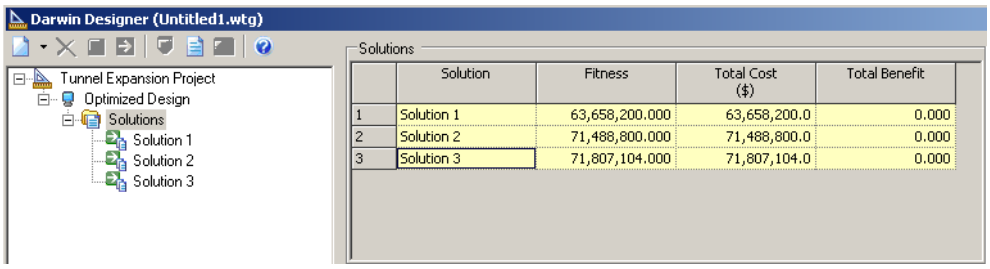
Design Pipe Group (Design Groups tab)	Lists the names of the design pipe groups.
Rehabilitation Group (Rehab Groups tab)	Lists the names of the roughness groups.
Is Active?	Select the check box for the design groups to be included in the current design run.
Design Option Group	For each design group, you can select the design option group you want to use.
Manual Selection	Forces a particular action for the selected group.

Note: You must have at least one active design or rehab group set to a valid design or rehab option group.

Compute the Design Run

After you set up your design run, click Compute  to compute the results of your design.

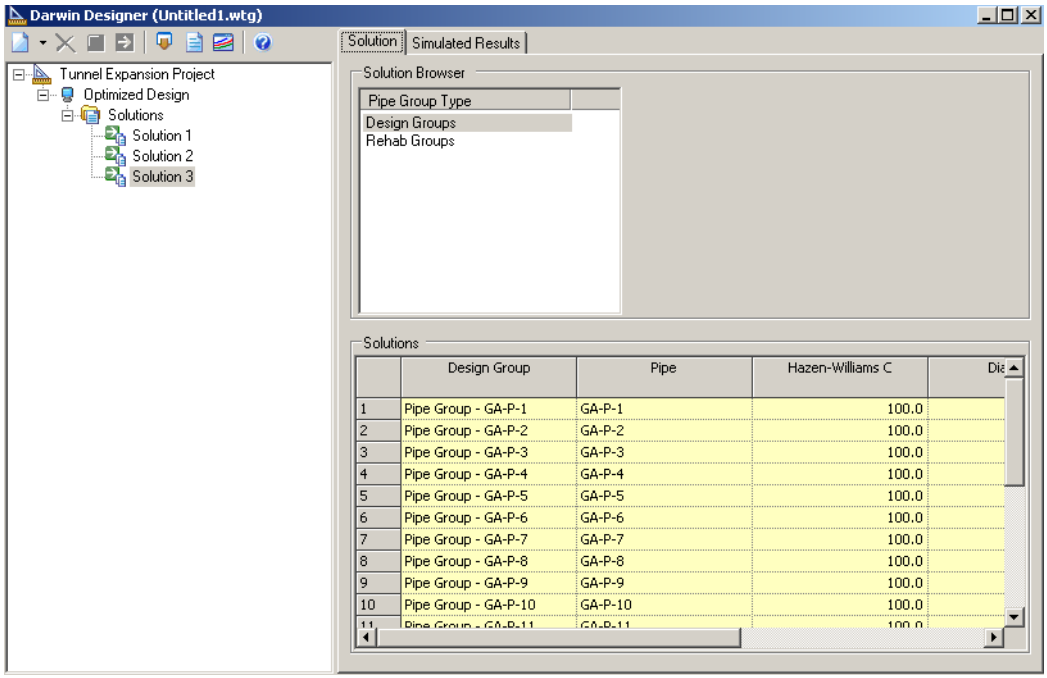
After you have computed your design run, Solutions is added to the project list.



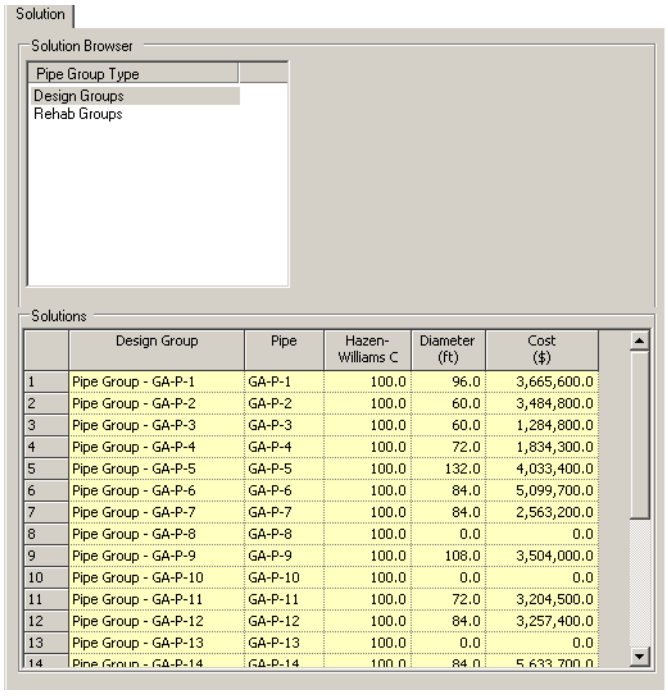
Solution	The list of solutions.
Fitness	Fitness is the overall score given a solution by Darwin Designer. For Minimize Cost solutions, a lower fitness is best. Otherwise, higher fitness indicates the best solution.
Total Benefit	This only has a value for Maximize Benefit and Multi-Objective Trade-off calculations. This is a score of the calculated benefits, with a higher value indicating more benefit in terms of improved network pressure.
Total Cost	Total Cost displays the sum of rehabilitation and design costs.

To view more information on the Solution

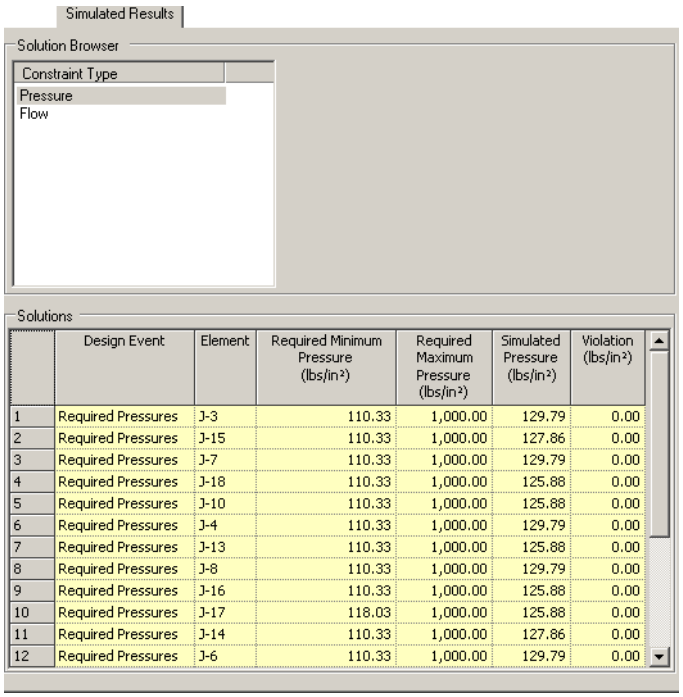
1. Click on one of the Solutions to view the Solution Browser.



2. Click the Solution tab to view Pipe Group Type information for Design Groups and Rehab Groups.



3. Click the Simulated Results tab to view Constraint Type information on Pressure and Flow.



	Design Event	Element	Required Minimum Pressure (lbs/in²)	Required Maximum Pressure (lbs/in²)	Simulated Pressure (lbs/in²)	Violation (lbs/in²)
1	Required Pressures	J-3	110.33	1,000.00	129.79	0.00
2	Required Pressures	J-15	110.33	1,000.00	127.86	0.00
3	Required Pressures	J-7	110.33	1,000.00	129.79	0.00
4	Required Pressures	J-18	110.33	1,000.00	125.88	0.00
5	Required Pressures	J-10	110.33	1,000.00	125.88	0.00
6	Required Pressures	J-4	110.33	1,000.00	129.79	0.00
7	Required Pressures	J-13	110.33	1,000.00	125.88	0.00
8	Required Pressures	J-8	110.33	1,000.00	129.79	0.00
9	Required Pressures	J-16	110.33	1,000.00	125.88	0.00
10	Required Pressures	J-17	118.03	1,000.00	125.88	0.00
11	Required Pressures	J-14	110.33	1,000.00	127.86	0.00
12	Required Pressures	J-6	110.33	1,000.00	129.79	0.00

The Design Groups tab in the Solutions area displays

- Design group name
- Pipe label
- Hazen-Williams C
- Diameter
- Cost.

The Rehab Groups tab in the Solutions area displays

- Rehabilitation group name
- Pipe label
- Design Rehabilitation action taken
- Cost.

The Pressure tab in the Solutions area displays information about junction pressures

- Design event name

- Element
- Required minimum pressure
- Required maximum pressure
- Simulated pressure
- Violation - any calculated pressures that fall below the minimum or above the maximum (as a negative number if below the minimum, as a positive one if above the maximum).

The Flow tab in the Solutions area displays information about junction pressures

- Design event name
- Element
- Minimum velocity
- Maximum velocity
- Simulated Flow
- Violation - any calculated velocities that fall below the minimum or above the maximum (as a negative number if below the minimum, as a positive one if above the maximum)

Report Viewer

You can view, print, and search reports you create about your optimization.

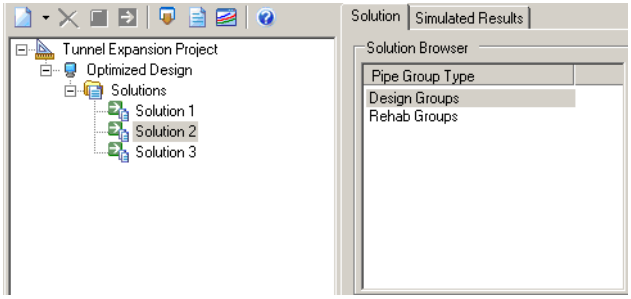
You can select the following options from within the Report Viewer:

Print 	Prints your report to an installed printer.
Copy 	Copies the report to the clipboard to paste into another program.
Find 	Searches for text in your report. Report Viewer highlights the text as it finds it.

Single/Multiple Page 	Displays one of your report pages or several pages at once.
Zoom Out/Zoom In 	Magnifies or reduces the display of your report for better viewing.
Previous Page/Next Page 	Pages through your report. You can also use the <Page Up> and <Page Down> keys on your keyboard.
Backward/Forward 	Navigates between pages you have just viewed.

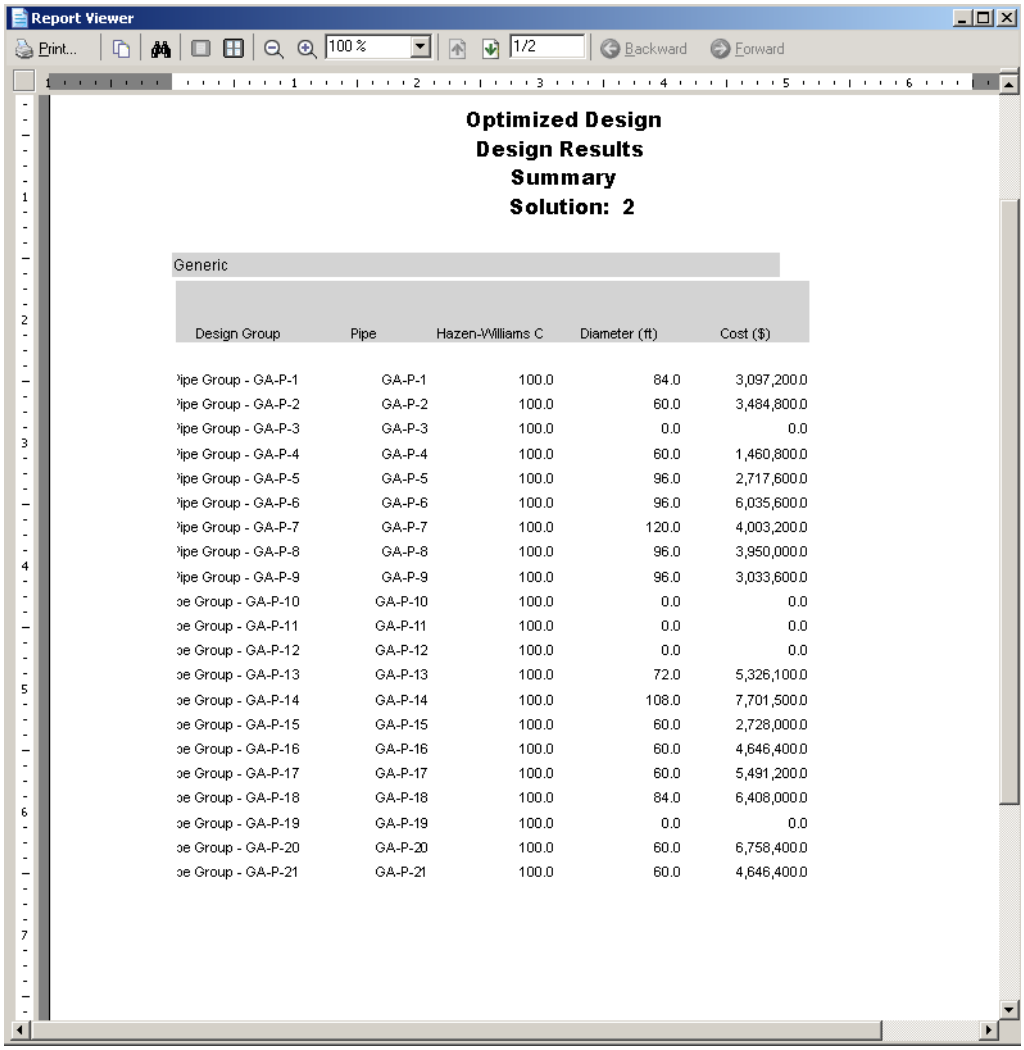
To create a report of your solution

1. Select a Solution and in the Solution Browser select Design Groups.



2. Click Report  .

3. The Report Viewer opens.



Graph Dialog Box

You can create two graphs from your Darwin Designer calculations.

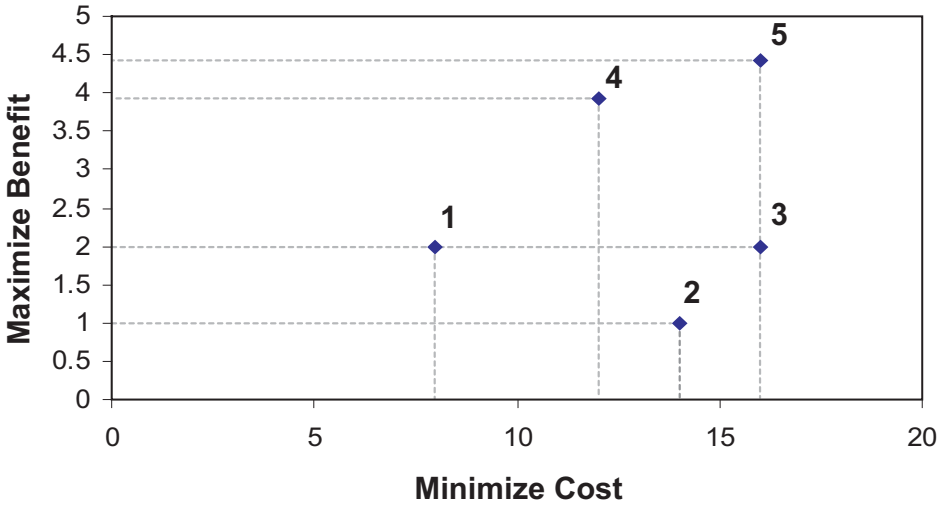
- **Pareto Optimal Plot**—Shows Benefit versus Cost for your calculations, provided you have used Maximum Benefit or Multi-Objective Trade-off Design Parameters.
- **Pipe Size Usage Plot**—Shows the total length of pipe of a certain diameter used by the solution.

Copy 	Copies the current graph as a raster (bitmap) image to the clipboard.
Print Preview 	Opens the Print Preview window where you can view how the graph will look before you print it.
Options 	Opens the TeeChart Editor where you can change the appearance of the graph.
Close 	Closes the graph.
Help 	Opens WaterCAD V8*i* Help.
Copy 	Copies the current graph as a raster (bitmap) image to the clipboard.
Print Preview 	Opens the Print Preview window where you can view how the graph will look before you print it.

About Pareto Optimal Plots:

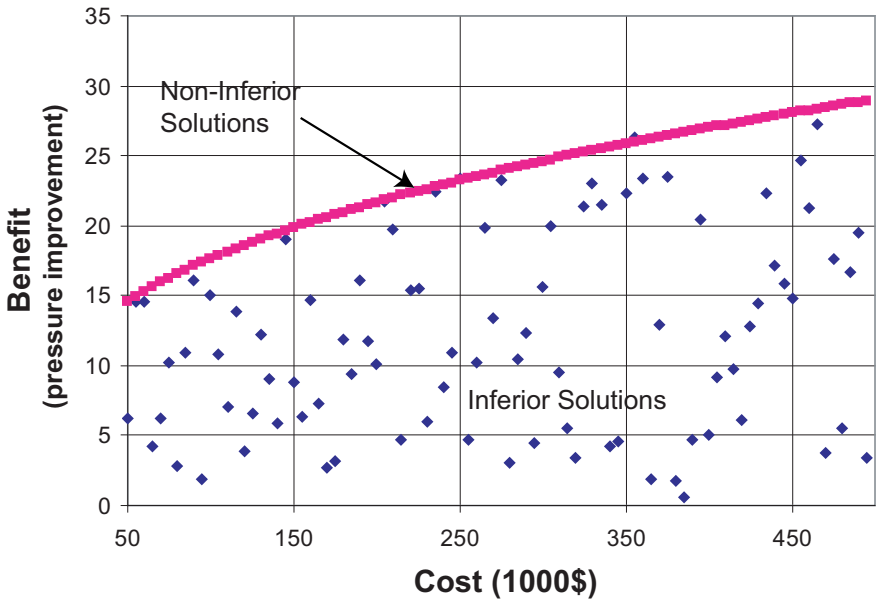
When there is more than one objective in a design, it is seldom possible to say that one solution is clearly the best of all because it may be better than another solution with regard to one objective measure but worse on another objective. (Although, there are many solutions that are clearly inferior. That is, there are other solutions that are better than an inferior with regard to all objectives.)

For instance, as illustrated in Non-Inferior Solutions vs. Inferior Solutions, solution 1, 4, and 5 give lower cost and greater benefit than solution 2 and 3, thus solution 1, 4, and 5 are better (not worse) than both solution 2 and 3. Solution 1, 4, and 5 are often referred as non-inferior or non-dominated solutions, while solution 2 and 3 are called inferior or dominated solutions.



Non-Inferior Solutions vs. Inferior Solutions

When you choose to do cost-benefit trade-off design, Darwin Designer minimizes the cost and maximizes the benefit. Both objectives conflict, because minimizing the cost of a design diminishes the benefit instead of improving it. Darwin Designer searches for non-inferior solutions. Non-inferior, or Pareto optimal (after Pareto, an Italian economist), solutions are the set of solutions for which no solution can give a better value of one objective without having a worse value for another objective, as shown in A Plot of Pareto Optimal Front.



A Plot Of Pareto Optimal Front

For example, one solution may cost $5 million and have a pressure benefit of 2 (high is good), while another may cost $6 million and have a pressure benefit of 2.2. Neither is clearly superior but neither is clearly inferior; they are both non-inferior to one another.

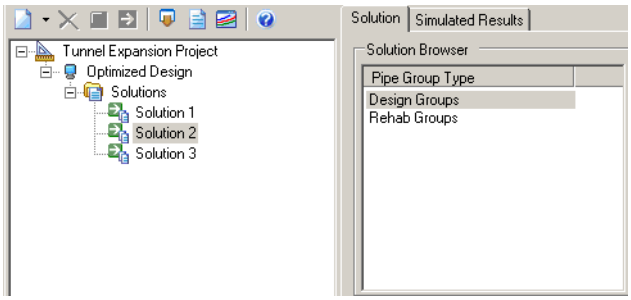
When working with multiple objectives, there is not likely to be a single solution that is superior for all objectives. Therefore, when multiple objectives are involved, you must chose between a number of non-inferior solutions.

Darwin eliminates the thousands of inferior solutions and provides two ways to compare non-inferior solutions:

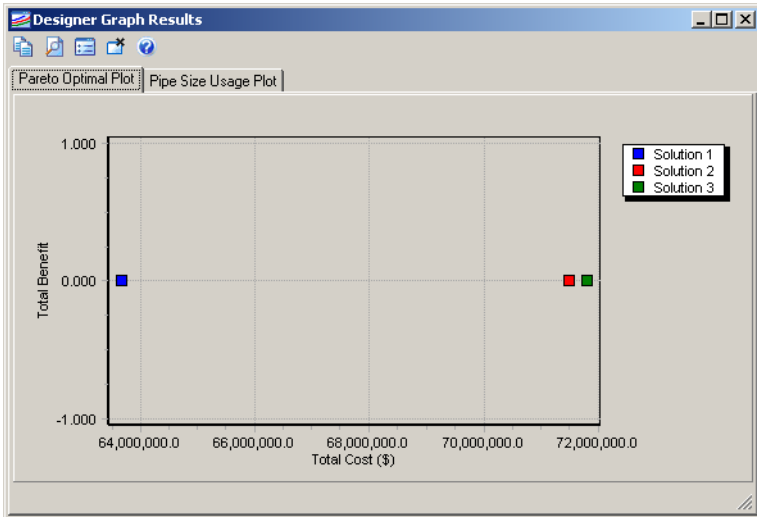
1. Solution comparison table.
2. Pareto optimal plot.

To create a graph of your solution

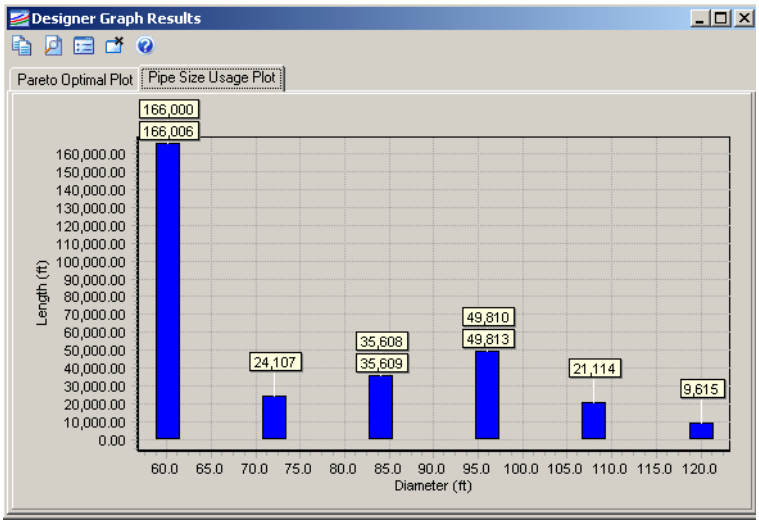
1. Select a Solution and in the Solution Browser select Design Groups.



2. Click Graph  .



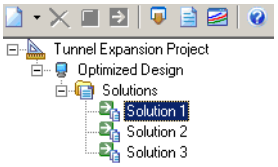
3. The Graph opens the Pareto Optimal Plot. Click the Pipe Size Usage Plot to view that graph.



Export to Scenario

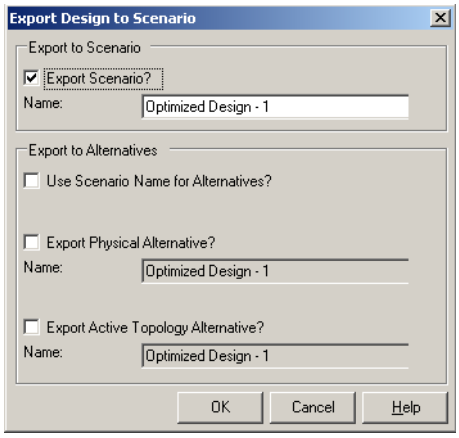
Use Export to Scenario to pass your results and optimized network for use in Bentley WaterCAD V8*i* .

1. Expand the Solutions folder and select one of the solutions to export.

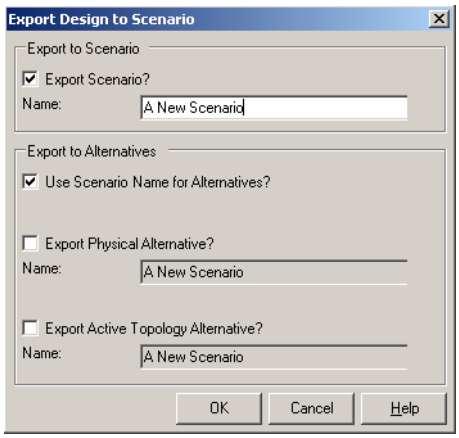


2. Click Export to Scenario  .

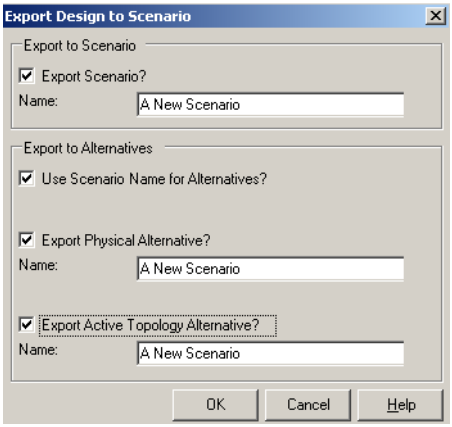
3. The Export Design to Scenario dialog box opens.



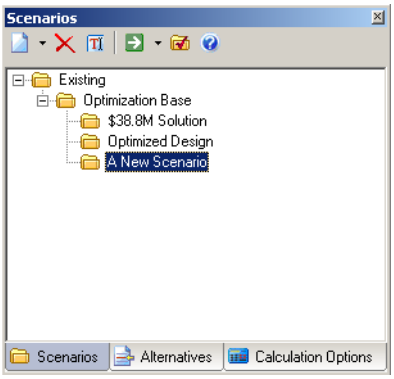
4. By default, Bentley WaterCAD V8*i* uses the name of the design run as the name for the scenario and alternatives you export. In order to rename the scenarios and alternatives using the same name, not the design run name, check the Use Scenario Name for Alternatives box and type in the Export to Scenario Name field; the text boxes for the alternatives will match what you type.



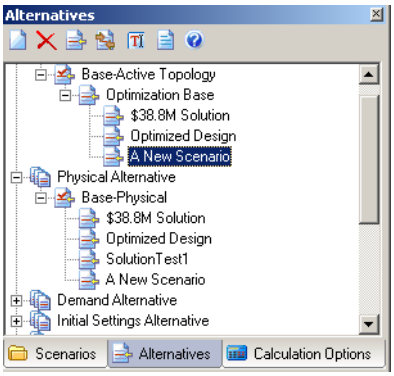
5. Select the check boxes for the items to export.



6. Click **OK** to export the scenarios and alternatives.
7. To view the exported scenario go to Analysis > Scenarios



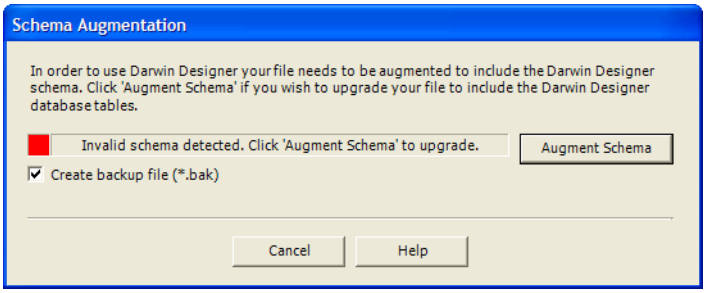
8. To view the exported alternatives, click on the Alternatives tab in the Scenario manager.



Note: If you export a Designer solution to the scenario manager, the extra demand adjustments and boundary (initial) conditions aren't exported (only physical properties, active topology, and capital cost alternatives can be exported). Given this, to recreate simulation runs that are equivalent to each Design Event, it is necessary for you to build a corresponding demand and initial alternative that reflects any additional demand adjustments and any boundary conditions.

Schema Augmentation

The Schema Augmentation dialog box opens if the Bentley WaterCAD V8*i* file does not contain the Darwin Designer schema.



A schema is the series of tables and table cells that contain your data. A schema change typically means a table or table cells have been added, usually by an update to the software.

When you use Schema Augmentation, Bentley WaterCAD V8*i* adds any missing tables to the schema of the file you are using. Updating a schema should not damage your data but we do recommend you create a backup. Select the **Create backup: *.bak** check box to create a backup of your existing database. It will be saved in its current directory but will have **.BAK** appended to the filename.

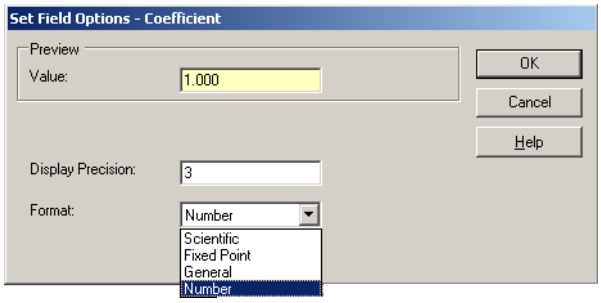
To restore the backup, delete or move your current .sqlite file and then rename your backup file by deleting the **.BAK** extension, so the extension becomes only **.sqlite**.

Set Field Options

Right-click on the Demand Multiplier field 

.

You can set the value, precision, and format for the data:



Scientific:	Scientific numbers use the form, 1.111 E+111.
Fixed Point:	Fixed point numbers use the form 111.111.
General:	General format uses the most compact of either fixed-point or scientific notation
Number:	Numbers use the form 1,111,111.111, where number separators are used.

Verification Summary

If you try to calculate a network using invalid Darwin Designer settings, the Designer Data Verification Summary displays. This dialog box means that there are some invalid settings in your run that prevent Darwin Designer from calculating your solution.



If the Designer Engine Error Message opens

- Do your groups reference elements that are inactive in your Representative Scenario? Check the scenario you are using. Make sure your scenario uses only active pipes.
- Does your design run have an Active Design Event? It should.

- Do you have active design groups that are assigned to valid design option tables? You need at least one active design group that corresponds to a design option table.
- Is it possible that elements have been deleted from the model from another client application? If so, close Darwin Designer and re-open it. Darwin Designer will update itself based on the latest GEMS model, deleting any references to deleted elements.

Manual Cost Estimating

With version 8 of Bentley WaterCAD V8*i* , construction cost estimating for piping has been moved to the Darwin Designer.

Cost calculations are performed in WaterCAD V8*i*/GEMS in Darwin Designer based on the formula:

Cost = Unit Cost x Length

for each pipe element, where the unit cost is a function of the pipe diameter. The total costs are the sum of the costs for each element.

The user specifies the cost functions and has the option of having different cost functions for different locations (e.g. new developments, central city, stream crossing). The user must identify which pipes are to be included in the estimate and which pipes are assigned to each cost function.

An overview of the steps consists of:

1. Create scenario(s)
2. Start Darwin Designer
3. Create cost functions
4. Identify groups of pipe to use each function
5. Pick scenario
6. Pick pipes to be include in this cost calculation
7. Run cost calculation

The detailed steps are listed below.

Initiating Costing Runs

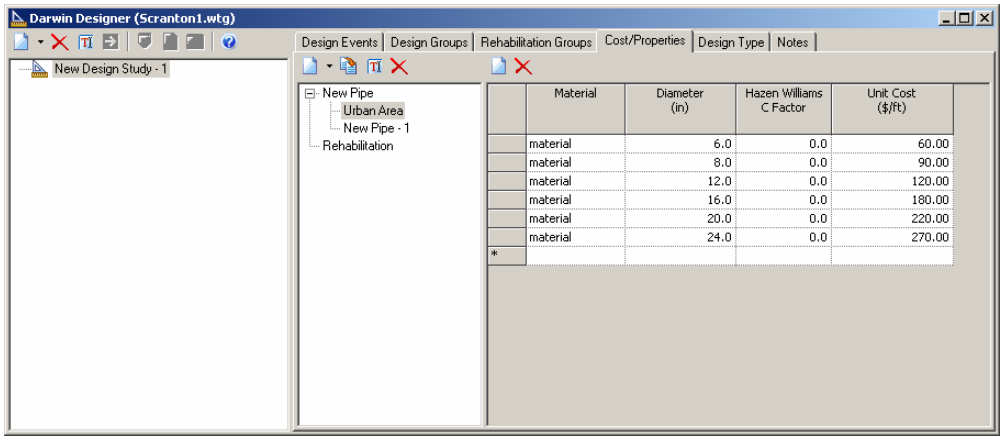
Unless the user wants to manually enter pipe diameters in the cost estimating run, the user should have already created the scenarios for which the costs are to be run before entering Darwin Designer.

To develop a cost estimate for new piping, start Darwin Designer using Analysis > Darwin Designer and create a New Design Study, if none exists, by picking New > Create Design Study above the left pane. (Users with a limited features version of WaterCAD V8*i* may not be able to use all the optimization features in Darwin Designer but will be able to use manual cost estimating.)

Building A Cost Function

The first step is creating unit cost functions to be used in the cost estimating.

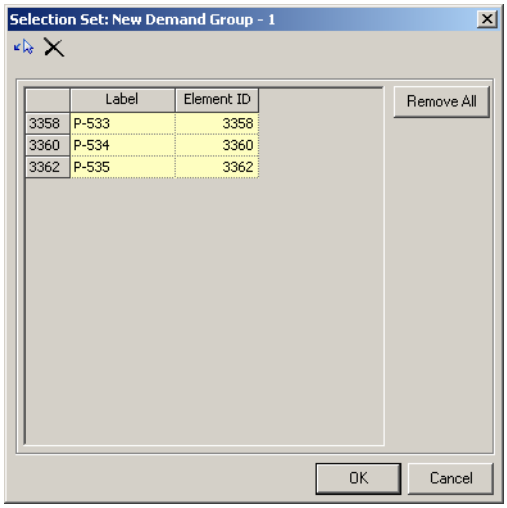
Click the Cost/Properties tab from the right pane and click the New button in the right pane to create a new cost function. It is advisable to give each function a more useful name than the default "New Pipe-1". For example use "congested urban area", "new subdivision," "state highway", or "open field" as cost function names.



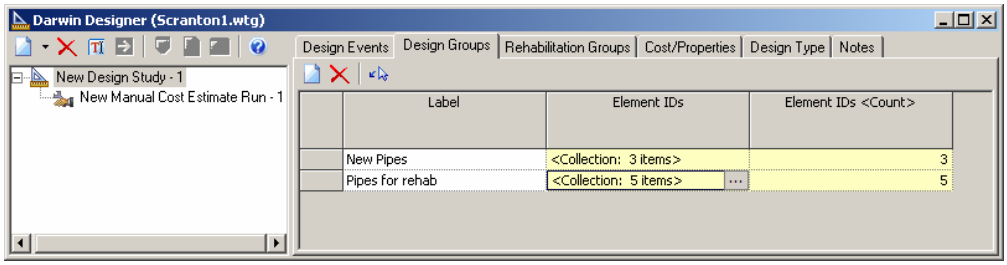
There must be a unit cost for each diameter that is included in the cost calculation. No interpolation is done. For example, if a 10 in. (250 mm) pipe is included in the scenario for which costs are calculated but a unit price for a 10 in. pipe is not included in the cost function, the cost calculation will fail and an error "Unable to match at least one scenario derived pipe diameter to the specified cost table" will appear under user notifications. To correct this, add the unit cost for that diameter.

Identifying Elements for the Cost Calculation

To identify pipes to include in the cost calculation, click the Design Group tab and assign a name to the group. Then in the Element ID column, create a group by clicking the ellipsis (...) button and selecting the pipes from the drawing to be included in this group. Once done, click the green check and the list of elements appears.



Each group should be created so that the individual pipes in the groups will share the same cost function.



When doing manual cost estimating, there is no need to use the tabs for Design events, Rehabilitation Groups, Design Type or Notes.

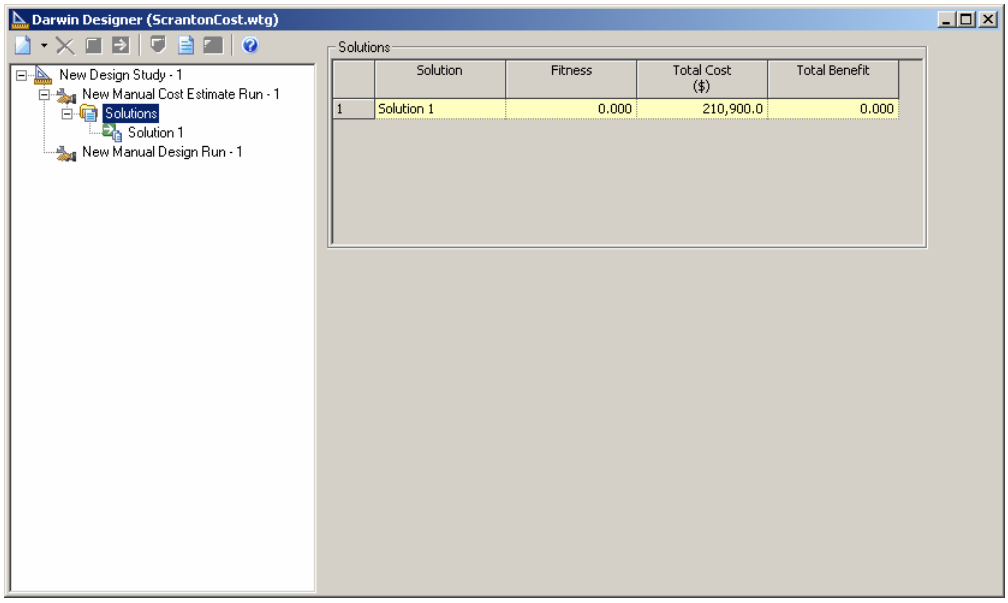
Calculating Costs

To perform the cost calculation, select New > New Manual Cost Estimate Run from above the left pane.

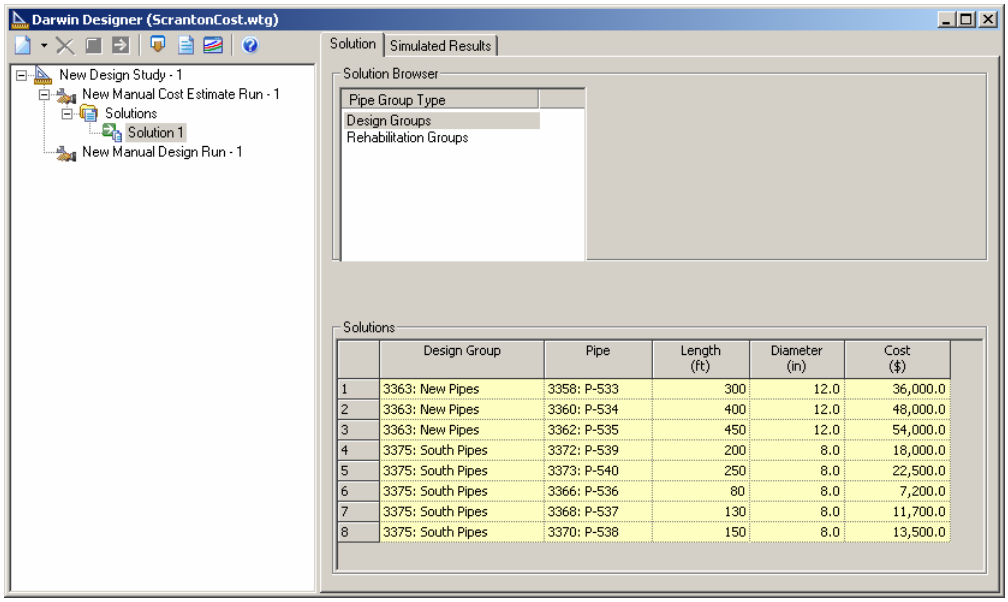
Then select which groups are to be included by checking "Is active" for those groups, the cost function to use for each group, and the diameter for each group. When the boxes under **Is Active?** Are checked, the corresponding pipe group is included in the cost calculation

By default, the check box labeled "Use Diameters from Representative Scenario" is checked. This means that costs are based on the diameter from the current scenario for any pipes in the groups that are checked and the column labeled "Manual Selection" is not used. If this box is unchecked, the user must enter the diameter in the "Manual Selection" column in the dialog.

To perform the cost calculation, click the green Go arrow button above the left pane. When the calculation is complete, click Close in the calculation progress dialog box and the results will appear under Solution. When the calculations are complete, two new lines will appear in the left pane, one titled Solutions which gives the total cost summed over all elements, and a second called Solution 1 which gives the cost of each pipe. There will only be a single solution for a manual cost run. The Solutions display looks like the one below.



A detailed breakdown by pipes is given by picking Solution 1.



	Design Group	Pipe	Length (ft)	Diameter (in)	Cost ($)
1	3363: New Pipes	3358: P-533	300	12.0	36,000.0
2	3363: New Pipes	3360: P-534	400	12.0	48,000.0
3	3363: New Pipes	3362: P-535	450	12.0	54,000.0
4	3375: South Pipes	3372: P-539	200	8.0	18,000.0
5	3375: South Pipes	3373: P-540	250	8.0	22,500.0
6	3375: South Pipes	3366: P-536	80	8.0	7,200.0
7	3375: South Pipes	3368: P-537	130	8.0	11,700.0
8	3375: South Pipes	3370: P-538	150	8.0	13,500.0

Advanced Darwin Designer Tips

1. **How do I consider fire flows in my design?**

   You may consider fire flows by one of two methods:

   a. Use the demand adjustments feature in the required design event to add additional demand to the specific junctions at which fires are to be fought.

   b. In Bentley WaterCAD V8*i* , create a child demand alternative of the demand alternative referenced by the representative scenario, and then add the fire flows as fixed pattern flows to the appropriate junctions. Next, in Darwin Designer, set up a design event and select the **Override Scenario Demand Alternative** check box, and select the new child demand alternative you created.

   Of the two methods, the second one is preferred, since, after you have exported your design from Darwin Designer to a new scenario, you will most likely want to verify the performance of the design directly within Bentley WaterCAD V8*i* . If you have used method one to add fire flows, then you will have to add those fire flows to your current (or new) demand alternative in order to simulate the design against the same demands as in your design event. If you had used method two, however, then you would not need to create any additional demand alternatives, since you had already done that.

2. **Where should I set fire flows in my system to achieve a good design?**

Fire-flow design event can be set up by using one of two methods in Question 1. To achieve a good design, you need to ensure that a design can funcion under the most important fire-fighting scenarios. This will be different from system to system. When you set a fire-flow design event, Darwin Designer optimizes the system capacity (pipe sizes) to meet the additional demand requirement for the portion of a system where a fire flow is set up. The other portion of the system may have inadequate capacity. To improve the system-wide emergency response capability, it is recommened that fire flows are set at the outskirts of a distribution grid; this will allow Darwin Designer to optimize the systemwide supply capacity.

3. **How do I consider emergency conditions and facility outages?**

   Emergency conditions, such as pipe breaks and facility outages, can be handled in Darwin Designer by using the boundary-conditions feature of a design event to close pipes that would normally be open. For example, you may want to consider the effect of a water treatment plant being out of service. This can be achieved by adding any connecting pipes to the design-event boundary conditions and setting their status to closed.

4. **Designer only sizes or rehabilitates pipes. How can I consider the inclusion of new facilities?**

   Selection of new facilities may be achieved by using various modeling techniques, an example of which follows.

   Selecting the location of a new tank:

   a. You can select the location of a new tank modeling the new proposed tank in the representative scenario. Given a specific tank location you will need to enter the tank elevation, diameter, and other size information as if it existed—but, connect the tank to the system with a short small diameter pipe. Give the new pipe an obvious label such as **New Tank Connector**.

      The pipe that connects the tank to the system should have a length of 1 and a diameter of 0.01.

   b. Create a new Design group and label it as **New Tank Connector**, or something similar, and add the connecting pipe to the new group.

   c. In Darwin Designer, create a new pipe option group, label it **New Tank**, or something similar, and add the following data:

Diameter	Cost
0	0
X	Cost of Tank

      Where, X is some large diameter sufficient for the expected flows to and from the tank.

d. In your local design run group, enable the new pipe group by clicking **Active** and select the **New Tank** option group.

Darwin Designer can now connect the tank to the system and incur the cost specified in the above table, or it will construct a 0 diameter pipe (no pipe) and the tank will not be included in the system. Note that it is up to you to make sure that sufficient demand cases are investigated to verify the tank's design and that tank operation is independently verified through an EPS simulation.

Using similar logic Designer could be used to consider the inclusion exclusion of pump stations, valves, water treatment facilities, reservoirs and so on.

5. **Designer keeps coming up with strange results. What am I doing wrong?**

There are a number of things that could be causing you get strange or unexpected results with Darwin Designer. Before calling technical support, please take the time to review this list to see if any of these things may apply to you.

a. **Make sure you are using the correct design data.** Make sure you are using the correct representative design scenario and that scenario includes all pipes to be sized by Darwin Designer.

b. **Make sure that the representative design scenario runs successfully within** Bentley WaterCAD V8*i* **.** If it does not, then Designer will not be able to function correctly.

c. **Make sure that the correct demands are present.** For EPS representative scenarios, make sure your patterns are correct and that you are using the correct time from start value in your design events.

d. **Make sure that you have applied the correct and necessary boundary conditions.** For example, if you are designing for a 7 a.m. peak-flow condition, make sure that you have boundary conditions specified for all necessary tank levels, pump operation, etc. For designs that include a significant amount of new infrastructure or completely new designs, tank levels have to be assumed tank levels.

e. **Make sure that the range of pipe sizes and rehab actions you are using are reasonable.** For example, make sure that you are allowing Darwin Designer a sufficient range of pipe diameters to come up with a reasonable design. While Darwin Designer does perform an initial feasibility check (it uses the largest pipe sizes and checks minimum pressures), too few pipe choices may artificially restrict the flexibility of the optimization. Conversely, too many choices may affect the convergence of the optimization on to a good solution. It doesn't make sense, for example, to allow a rising main from a pump station to be 6 in. or 8 in.

f. **Make sure that you have a reasonable number of design and/or rehab groups.** As an extreme example, consider that every pipe to be design was in the same group. Then the only possible solution that the optimization can arrive at is to construct all of the pipes the same size. While it may still be

possible to find a feasible solution, only having a single design group will restrict the flexibility of the optimization and the ability of Darwin Designer to find cheaper solutions. Conversely, too many design groups will hinder the convergence of the optimization and result in sub-optimal solutions. A good number of design groups will depend on the actual model and design situation, but would lie somewhere between 10 and 100.

g. **Make sure you have sufficient and reasonable design constraints in place.** The genetic algorithm optimization engine in Darwin Designer is very powerful. If the objective of the optimization is to minimize cost, the optimization engine will do everything in its power to minimize cost including unwanted things that may not have been disallowed by the designer. The worst case scenario is a design with no constraints. If the design does not have any performance requirements, then the cheapest design is no design at all.

   The optimization algorithm only knows the problem that is defined for it, and to that end if you wish to get meaningful designs from Darwin Designer, you need to constrain your designs appropriately. The idea is to set up design constraints that corner the optimization algorithm into a region of the solution space (region of all possible solutions) that makes the most practical sense.

   Design constraints can be applied in Darwin Designer by pressures (max. and min.) and also pipe velocities (max. and min.). An example of an impractical situation in a hydraulic model might be a 1 MG tank that is draining at far too high a rate. In order to save costs on constructing pipes to a more distant source, the optimization algorithm may over-use a closer water source.

   Another example of a design constraint—other than the pressure and flow constraints—is the number of design events (and hence demand/operational cases) that the design must meet. The optimal solution to a single demand case does not fully reflect the real system operating scenarios. If a single load condition is used along with a zero-diameter as one of possible sizes in a option group, it will most likely result in a branched network design. Thus, it is necessary for reliability reasons to design systems for multiple demand conditions.

   It is up to the engineer to recognize any impracticality of an optimized design and set up the necessary design constraints to prevent that type of design from being feasible, thus removing that design possibility from the grasp of the optimization algorithm.

6. **How do I include a special cost, such as the cost of a highway crossing or interconnection in my design?**

   To do this you need to do three things:

   a. Group together the pipes that will attract the special cost. These pipes can be each in their own groups or all in one group, but they should be grouped such that they are separate from pipes that won't attract the special cost.

   b. Create a option group (new pipe or rehabilitation option group) that includes the special cost premiums.

   c. Assign the special option groups to the associated design groups locally, for the design run you wish to use with the special costs.

7. **Designer keeps coming up with pipe sizes that change up or down in size. I wouldn't construct such a design; what can I do?**

   Darwin Designer applies a competent genetic algorithm to optimize the design. It does not require or have any domain-specific knowledge about the water system, which ensures it is a generic tool, but also causes some side-effect for some design cases—like giving up-or-down pipe sizes. In particular, the solutions are evaluated by comparing the fitness values of solutions. Darwin Designer will assume a pipeline with pipe sizes that go up and down (to meet required pressures as closely as possible) is better than one that has a constant size that exceeds the pressures at some locations, since there is no specific penalty assigned to the fitness of a solution that has pipes that change up and down in size. It is, therefore, up to you to control the eventual design and this can be done by different means, as follows:

   a. The first means is simply to make manual adjustments to a design after Darwin Designer has finished, in order to clean up the design and make it a practical design. Cleaning up a design may technically move you away from the cheapest design, but an inexpensive design that won't be constructed is of little use. You may find that not much cleaning up is necessary. Quick edits to diameters or rehab actions like can be performed effectively in Darwin Designer by using a manual design run.

   b. Another thing to consider when analyzing a Darwin Designer design is whether the chosen pipe sizes are a function of the lengths of pipe in your model.

      To better illustrate this concept, consider a run of four pipes in series, each with different lengths. For these four pipes, the controlling pressure is the downstream-most junction, and all intermediate junctions are well above the required pressure. Now, after Darwin Designer finishes designing the run of pipe, it selects the first pipe as a 16 in., the second as 12 in., the third as 16 in. and the fourth as 12 in. It is unlikely that this design would be constructed as-is, but if the pipes themselves represented sufficient length of pipe, then it may be practical to construct a portion of the pipeline as 16 in. and a portion as 12 in. If this is the case, then you need to look at the model to determine why Darwin Designer is changing the third pipe back up to 16 in. It may be

that since the downstream-most junction is the only controlling node, that Darwin Designer is merely trying to achieve the right head-loss in the total pipe length, by choosing the length of pipe that should be 16 in. and the length that should be 12 in. Of course, it is still constrained by the individual pipe lengths in the model, but if they are different, the optimization algorithm will use this fact to its advantage. In this case, it may very well be that Designer is saying construct a total of 1500 ft. of 16-in. and 1000 ft. of 12-in. pipe, and not necessarily 850 feet of 16-in., 600 feet of 12-in., 650 feet of 16-in., and 400 feet of 12-in. pipe in sections. Use engineering judgment when analyzing the results.

c. Another means of achieving more constructible designs from Darwin Designer is to group in the same group pipes that would be constructed the same size. For example, a rising main would most likely be constructed a single size, and it would thus make sense to include all the model pipes that make up the rising main in the same design group. What you don't want to do by grouping pipes is artificially design the system even before you have had a chance to optimize it.

8. **When sizing new pipes, Darwin Designer can choose a zero-size, which means, do not construct that pipe. Is it possible to do a similar thing for rehabilitation actions?**

   It is possible to do the same thing for rehabilitation actions. To create a rehabilitation action that represents a Do Nothing option, simply follow these steps:

   a. Create a pre-rehab diameter versus post-rehab diameter function that defines at least two diameters that cover the domain of diameters in your model. For example, mi.n pipe size through max. pipe size and make the pre-rehab diameter the same as the post-rehab diameter. This function will define that the diameter of any single pipe remains the same before and after the rehab action.

   b. Create a diameter versus unit cost function that defines at least two diameters that cover the domain of diameters in your model. E.g., min. pipe size through max. pipe size and make the cost for each diameter zero. This function will thus define that the cost for the rehab action, regardless of pipe size is zero.

   c. Create a pre-rehab diameter versus post-rehab roughness function that defines at least two diameters that cover the domain of diameters in your model. E.g., min. pipe size through max. pipe size and make the post-rehab roughness, the roughness of the current pipes to which the Do Nothing option will be an option. This function will thus define that the resulting roughness stays the same as the original values.

   Create a Do Nothing rehab action that references each of the above functions. If selected by Designer, the Do Nothing action will leave the same diameter, cost nothing, and leave the same roughness: in effect, doing nothing.

9. **Do I have to change the parameters or can I simply use the defaults?**

   In most circumstances it is not necessary to change the parameters in order to run Darwin Designer, however, you may wish to modify certain values as follows:

   a. Random Seed—The Darwin Designer optimization algorithm depends on the generation of pseudo-random numbers through a random number generator. The reason the numbers are pseudo-random is that they are generated by a mathematical formula, and hence the resulting series of numbers is not actually random at all. In order to make the random numbers different the random number algorithm is initialized with what is known as a seed. For a different seed value, a different series of pseudo-random numbers will be produced. When no parameters in the Designer optimization problem change (i.e., no changes at all, including hydraulic model changes, constraint changes, etc.), running Darwin Designer twice will result in exactly the same result. Darwin Designer results are therefore repeatable in this way. One way of ensuring a different result (or at least a different progression to the same result) is by changing the random number seed. Doing this will result in different optimization results for different runs. By the nature of genetic algorithm optimization, you should not just accept the result of a single optimization run, but run several runs and make sure that all runs produce similar results. An easy way to run multiple runs and achieve different results is to change the random number seed.

   b. Penalty Factor—Penalty factor is a weighting that is used in the determination of the fitness value for an hydraulic solution. In particular the penalty factor is used to discourage the survival of designs that fail the design constraints. A higher value for penalty factor will put designs that fail the design constraints in greater disfavor, where as a lower value for penalty factor will place designs that fail the design constraints in less disfavor. A reasonable default for penalty factor has already been selected for you. However, if you find that Darwin Designer keeps settling on designs that contain constraint violation, then you may wish to increase the penalty factor value.

   c. Probabilities, Era Numbers, and Population Size—Good defaults have already been selected for you for these values, but instead of changing the random number seed when conducting multiple optimization runs of the same design, you may want to change these values. Good ranges for the values are therefore listed below for your convenience.

      Note: The upper limit values for population size, maximum era number, and era generation number are problem-dependent. For larger design models, you should use greater values than for smaller models.

      Population Size: 40 to 200

      Cut Probability: 0.5 to 2.5%

      Splice Probability: 50 to 80%

Mutation Probability: 0.5 to 2%

Maximum Era Number: 4 to 10

Era Generation Number: 50 to 200

10. **Is there a way to select design and rehab group pipes from the model drawing?**

    You cannot select pipes directly from the drawing in this first release of Darwin Designer. For this reason, we recommend you identify pipe groups and create appropriately-named selection sets before starting Darwin Designer. When you have defined the necessary selection sets, they can be used directly within Darwin Designer. Selection sets can also be used to define pressure and flow constraints, and to select boundary condition elements.

11. **Darwin Designer cannot find a feasible solution. How do I work out what is going wrong?**

    It is very likely that in using Darwin Designer, you will encounter situations where Darwin Designer cannot find a feasible solution. This happens even to those experienced in genetic-algorithm optimization and is due to the fact that the determination of which designs are feasible and which aren't is assessed by a computer subject to the information you tell it. That is, the rules are applied, with no exceptions.

    For example, if you want a minimum of 20 psi across the board, Darwin Designer will determine as infeasible any solution that does not have 20 psi at every junction. If you have a couple of junctions that are part of the detail of a tank inlet valving, for example, then maybe you don't really require 20 psi at those junctions. Perhaps what you really mean is that you want 20 psi at all service junctions. In that case, you'll find where an engineer would have said the design is feasible (because the design only fails the 20 psi requirement at non-service junctions), but Darwin Designer is unable to make that determination, since it was told 20 psi was required at all junctions. The process by which you can get around these kinds of issues is simply to identify them, correct them, and then re-run the optimization. For the case of the 20 psi junction example, the fix might be to create a selection set (in Bentley WaterCAD V8*i*) of the junctions that are service junctions, and only use those junctions as pressure constraint junctions. (The selection set can be selected from within Darwin Designer.)

    Along these same lines, you may also want to consider if any of the following things might be causing trouble, before calling technical support:

    a. Check for constraint violations in the results. Check both pressure and flow constraints for the presence of constraint violations. If any violations exist, you will need to determine why the junctions and/or pipes at which the violations occur are problematic. Maybe a minimum pressure constraint is simply impossible to meet due to the junction elevation, etc. Other things to check for are the applicability of blanket minimum and maximum pressures and veloci-

ties to modeling elements in detail models of pump stations, and the like. If you find anything, then you need either to change the model, or modify/remove the offending constraint and run the optimization again.

b. Make sure you have sufficient design options for a feasible design. That is, make sure that you have a sufficient range of pipe sizes and/or rehabilitation actions available to Darwin Designer to find a valid design.

c. Make sure that you haven't specified competing design events. While it may be possible to meet one design event or another separately, it may be impossible to meet two together if they compete with each other. For example, one design event might specify that a minimum pressure is required, and as such the corresponding pipe taking the flow to that location needs to be large, however, in the next design event with similar demands, a minimum velocity constraint means the pipe has to be sized smaller. It may be impossible to meet both design events with the single pipe size. To test this, build runs up by performing initially with only one design event, then adding more in. If all of a sudden after adding in a design event no more feasible solutions can be found, then you can try to work out what in the most recently added design event is causing the problem.

d. For multi-objective and maximum benefit optimizations, make sure you have sufficient budget specified. It may just be that you have not given Darwin Designer sufficient budget to allow a feasible design to be found. Try increasing the budget.

For more information, see Designer keeps coming up with strange results. What am I doing wrong? on page 12-1143.

Optimizing Pump Operations 13

[Energy Management and Scenario Energy Cost](#)

[Scenario Energy Cost Manager](#)

[Energy Pricing Manager](#)

[Scenario Energy Cost Analysis Calculations](#)

[Energy Cost Results](#)

Energy Management and Scenario Energy Cost

There are two levels at which energy costs can be analyzed in WaterGEMS. The tool called Scenario Energy Costs calculates energy use and cost for a single scenario while Energy Management uses the results of multiple Energy Cost scenarios to determine energy costs at a higher level of aggregation to determine the energy cost for pump stations (not just pump-by-pump) for multiple scenarios that can occur over a billing period and determine economic costs such as net present worth of pumping energy.

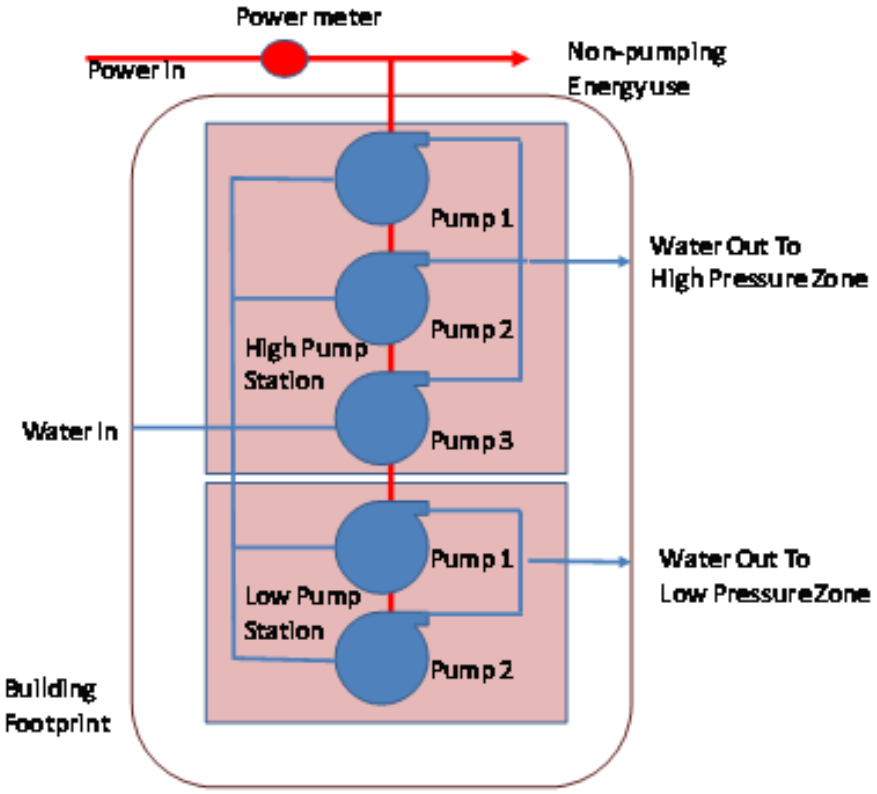
The scenario energy cost analysis determines the energy cost by pump for all pumps selected by the user. Pricing for energy cost is set up in the Pricing button in energy costing. Price functions are assigned to individual pumps in energy costing. See Scneario Energy Cost for detailed steps in running Energy Costs.

For users interested in a more complete energy analysis, running a single scenario may not be sufficient as block rate charges must be determined based on energy use over a complete billing cycle which may contain low, average and high water use periods which should be modeled as separate scenarios. In addition, the scenario corresponding to the setting of a peak demand charge is usually not an average day but some kind of peak condition that should be modeled in a separate scenario. In order to

deal with the complexities of block rates, multiple scenarios, aggregation of pumps within a station, and performing present worth calculation, the user needs to use the Energy Management analysis. Such calculations are usually required because of complex tariffs for electric power.

An important concept in energy management analysis is that of a "Power meter". A Power Meter is the basic unit that is billed by an electric utility. A Power Meter usually corresponds to a pump station. However, in WaterGEMS, a pump station is a collection of pumps serving a single pressure zone. Therefore, if a pump station building has a single electric service but has a set of Low, Medium and High service

pumps, for WaterGEMS hydraulic calculations, it is three Pump Station elements but for energy management, it corresponds to a single Power Meter. The figure below shows how a single power meter can include multiple pumps and pump stations in a single building.



Because there may be other energy uses at the pump station besides pumping, the user can specify non-pumping energy costs to account four uses such as lighting, HVAC, control systems, chemical feed equipment, etc. These costs are added in on a Power Meter basis. There may also be charges on the power bill that are not associated with individual pumping operations such as taxes, discounts, lump sum surcharges, etc. These can be added in to the overall cost and are referred to as "other costs".

The usual work flow for using the energy cost and energy management analyses may be followed as shown below:

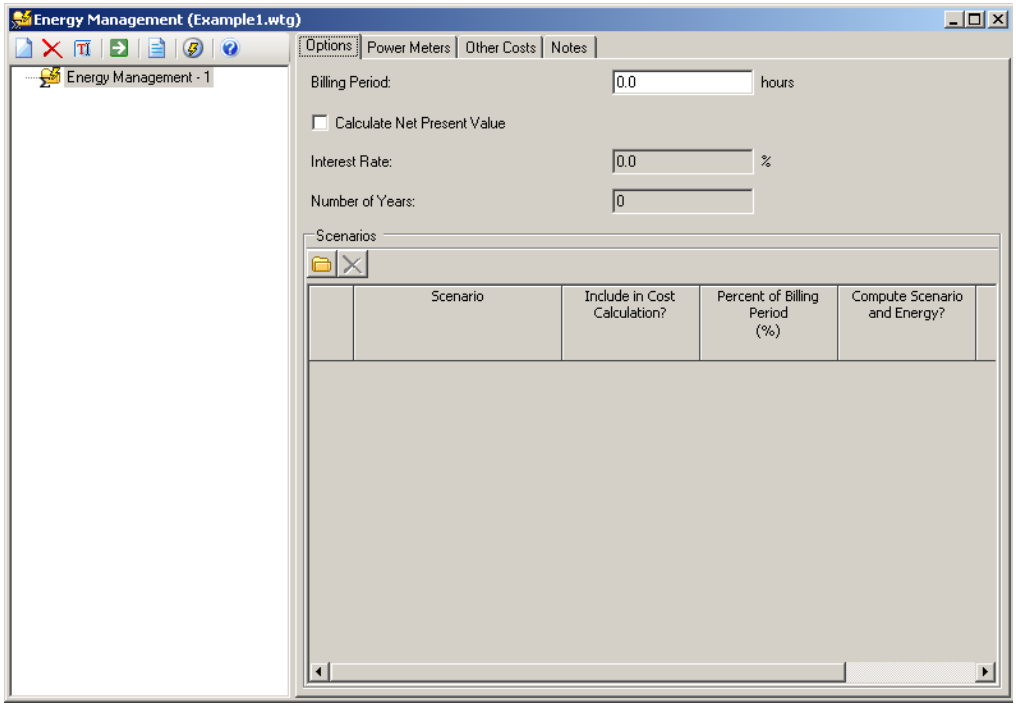
- Develop EPS scenarios to be used in energy cost
- Run scenarios
- Start scenario energy cost analysis
- Create price functions and optional carbon emission factors
- Assign price functions to pumps
- Run energy cost for each scenario of interest

- If more thorough analysis is desired, close scenario energy cost analysis and start energy management
- Create new energy management study
- Identify which pump stations/pumps are associated with each power meter
- Specify the mix of scenarios to be analyzed
- Identify interest rate and number of periods if present worth calculations needed
- Compute study
- Review results and rerun or create new studies

The energy manager analysis provides a way to combine the energy use and peak demands from multiple scenarios and multiple pumps associated with a power meter to display energy and peak demand cost based on pump, pump station, power meter, scenario or system wide. See Energy Management for detailed steps. Values reported in Energy Management Analysis are aggregated over time. To view time series energy use, it is necessary to use the Scenario Energy Cost Analysis.

Energy Management

To start Energy Management Analysis, the user selects Analysis > Energy Management or picks the  button. The first time the user enters the energy manager for a project, the Welcome dialog appears.



To create an energy management study, the user picks the **New** button  .

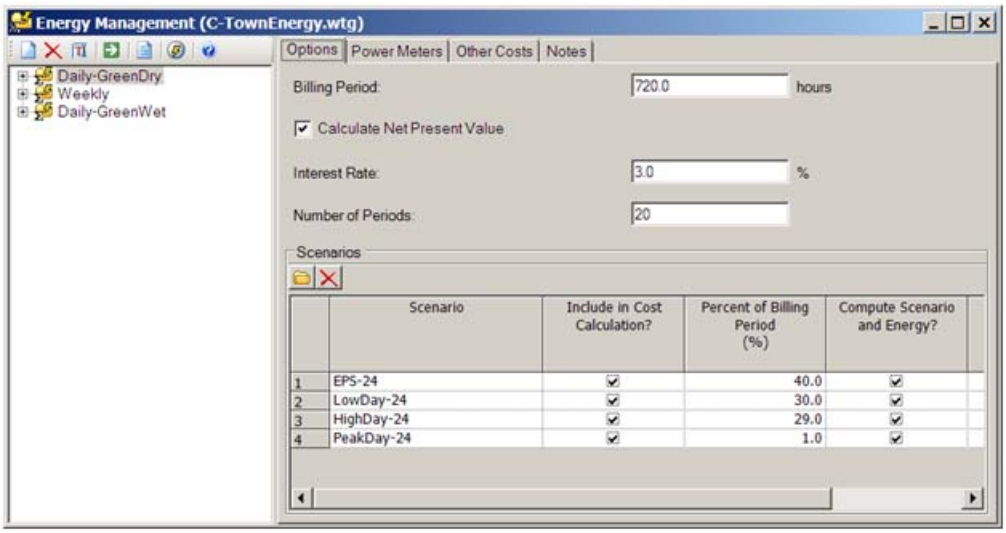
Once a study has been created, the buttons on top of the left pane enable the user to

-  **New** - create a new study
-  **Delete** - delete an existing study
-  **Rename** - change the name of a study

- **Compute** - run the energy calculations for a study
- **Report** - enter the report manager
- **Power Meter** - opens dialog for the user to associate pumps and pump stations with power meter and override some values from the scenario energy cost analysis
- **Help** - opens energy management help

The right pane of the energy management dialog contains four tabs. The function of each is described below.

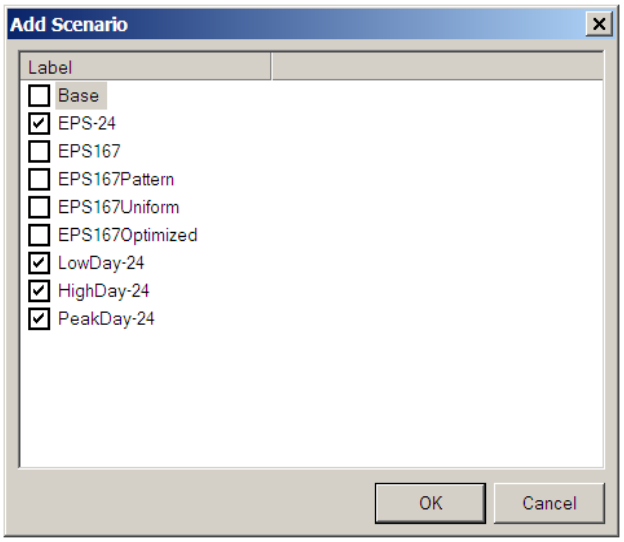
The Options tab is shown below.



The bottom portion Options tab is the place where the user selects which scenarios are to be included in the analysis, the percent of the billing period that is represented by each scenario (ideally the values would add to 100%), whether the energy management analysis should rerun the scenario (check) or use the results from the last computed scenario (unchecked) and which scenarios should be considered when determining peak demand costs. It is assumed that the time period over which the energy use is calculated is the same as the model time step. If the peak demand is based on the peak 15 minutes, the model time step (at least for that scenario should be 15 minutes).

If a scenario is included in the list but is later deleted from the model, a fatal error message will be given unless the scenario is also deleted from the Options tab list.

When the user picks the Scenario button in the bottom portion of the right pane, the following dialog appears where the user picks the scenarios to be included in the calculation. Only EPS scenarios can be used. At least one scenario must be selected.

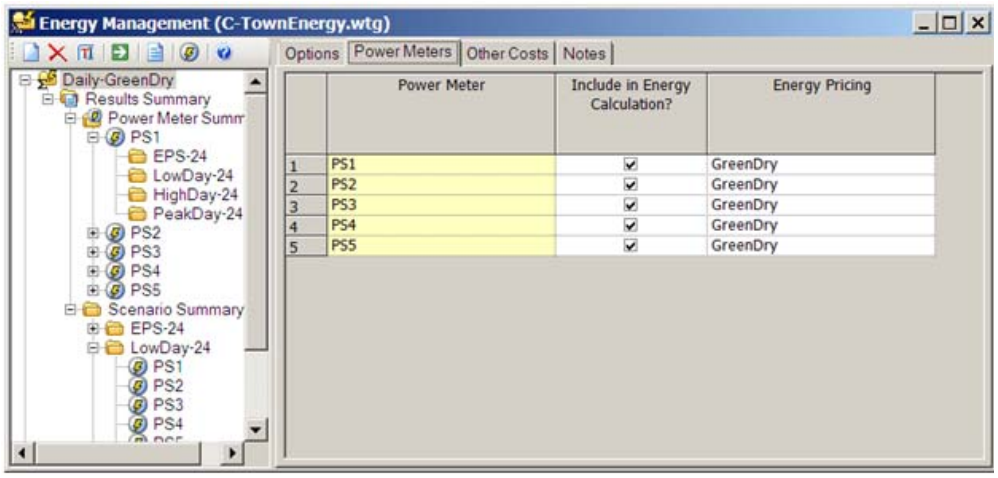


If the user un-checks "Include in cost calculation", that scenario is not used in the calculation but the scenario name is not removed from the list.

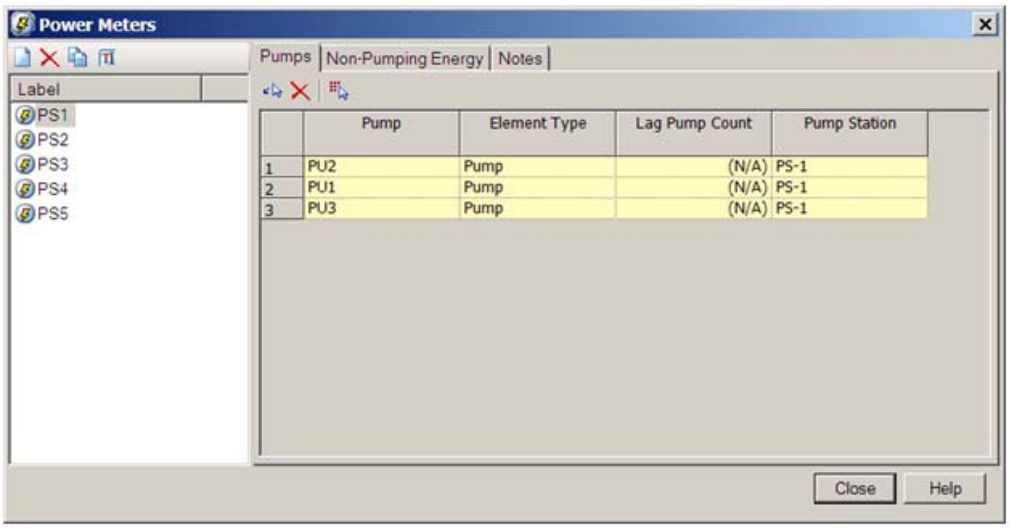
In the top portion of the right pane, the user specifies the length of the billing period over which the energy costs are to be aggregated. For example, if the billing period is 30 days, the user should specify 720 hours or 30 days.

Once the energy management analysis calculates the annual energy cost, the user can also determine the net present worth of energy cost. For this calculation, the user must check the box "Calculate Net Present Value" and enter the interest rate and number of periods. The Billing period must be greater than 0, interest rate should be between 0 and 100% and the number of periods must be greater than zero.

In the Power Meter tab for this study, the user can select which pumps or pump stations are to be included in the analysis and whether the energy price and the energy pricing to be used. The energy price selected can be different than that used in the scenario energy cost analysis. A warning will be issued if it is. At least one power meter must be selected or a fatal error message will be issued.



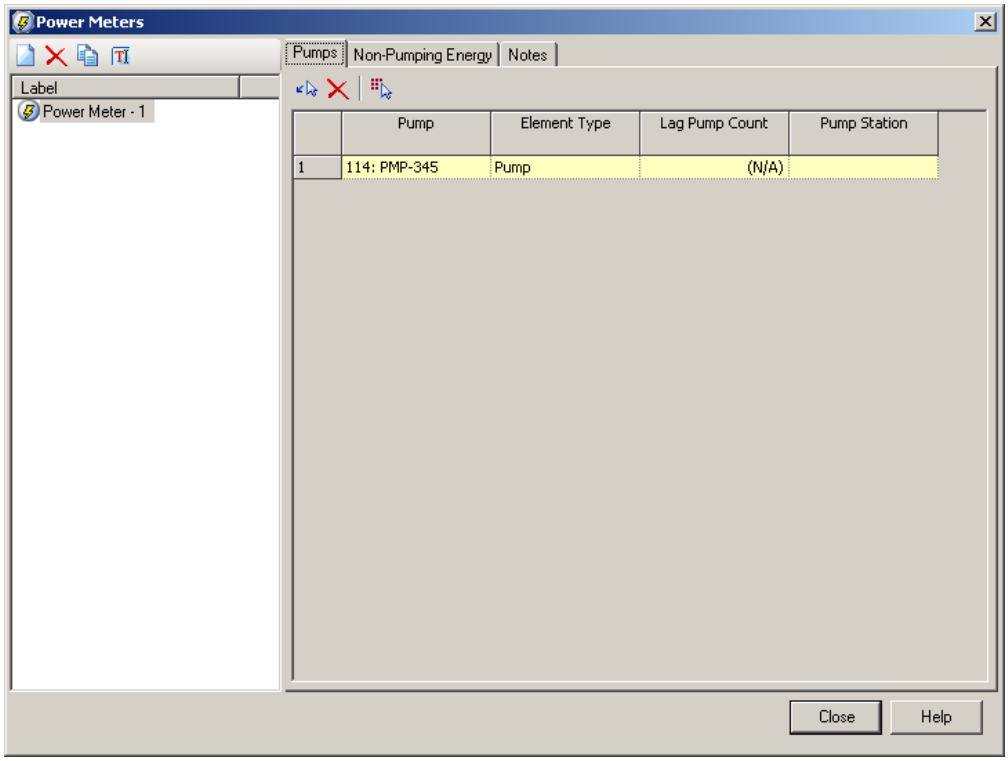
If no Power Meters have been created, the user must first pick the Power Meter button (not to be confused with the Power Meter tab) on top of the left pane. This opens the Power meter dialog where the user associates pumps and pump stations with the power meter serving them. The user should either select individual pumps or the pump station in which the pump is located. If a pump is both selected individually and the pump station it is located in is selected, then it is not double counted but treated as if it is part of the pump station.



Note: The Time of Day energy pattern billing period is for Scenario Energy Costs only, not for Energy Management calculations.

Power Meters

This dialog allows you to associates pumps and pump stations with the power meter serving them.



The dialog consists of a list pane on the left that displays all of the power meters associated with the project and a tabbed section on the right that allows you to assign pumps and other energy costs to the power meter(s).



New: Creates a new power meter.



Delete: Removes currently selected power meter.



Duplicate: Creates a copy of the currently selected power meter.

 **Rename:** Enter a new name for the currently selected power meter.

After creating a power meter, click Select From Drawing  to assign a pump/pump station to it in the Pumps Tab. Click Delete  to remove the currently highlighted pump from the list. Click Select In Drawing  to select the pump in the drawing view.

The Non-Pumping Energy tab allows you to specify additional energy costs. Enter a base power usage value and then assign a pattern that will be applied to it.

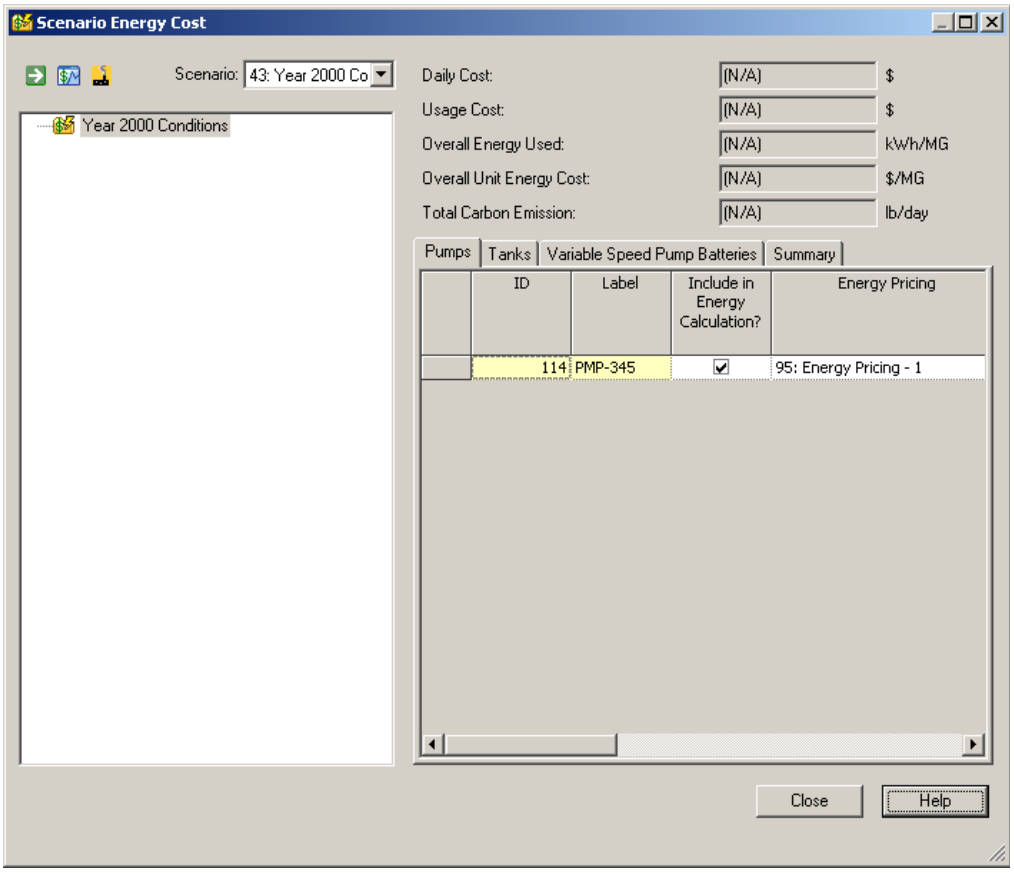
You can enter informational notes in the Notes tab.

Scenario Energy Cost Manager

The Scenario Energy Cost Manager is used to set up energy cost calculations. To calculate energy costs, the following information must be supplied:

- Specify the pumps, tanks, and variable speed pump batteries that are to be included in the energy cost calculations.
- Specify energy costs in the Energy Pricing manager.

To access the Scenario Energy Cost manager, click the Analysis menu and select the Energy Costs command, or click the Energy Costs button .



The left pane consists of a tree view that contains the name of the base scenario when it is first opened. Click the scenario icon to activate controls in the right side of the dialog that will allow you to specify the elements that will be used in the energy cost calculations.

Use the Compute button to calculate the energy costs based on the information set in the Energy Pricing Manager (accessed by using the Energy Pricing button for the currently selected scenario; select the scenario to use with the Scenario pull-down menu).

After energy costs have been computed, the tree view will also contain icons for Pump Usage, Time details, Pump details, Storage details, and Peak Demand details. Click on an icon to highlight it and view the associated results in the pane on the right.

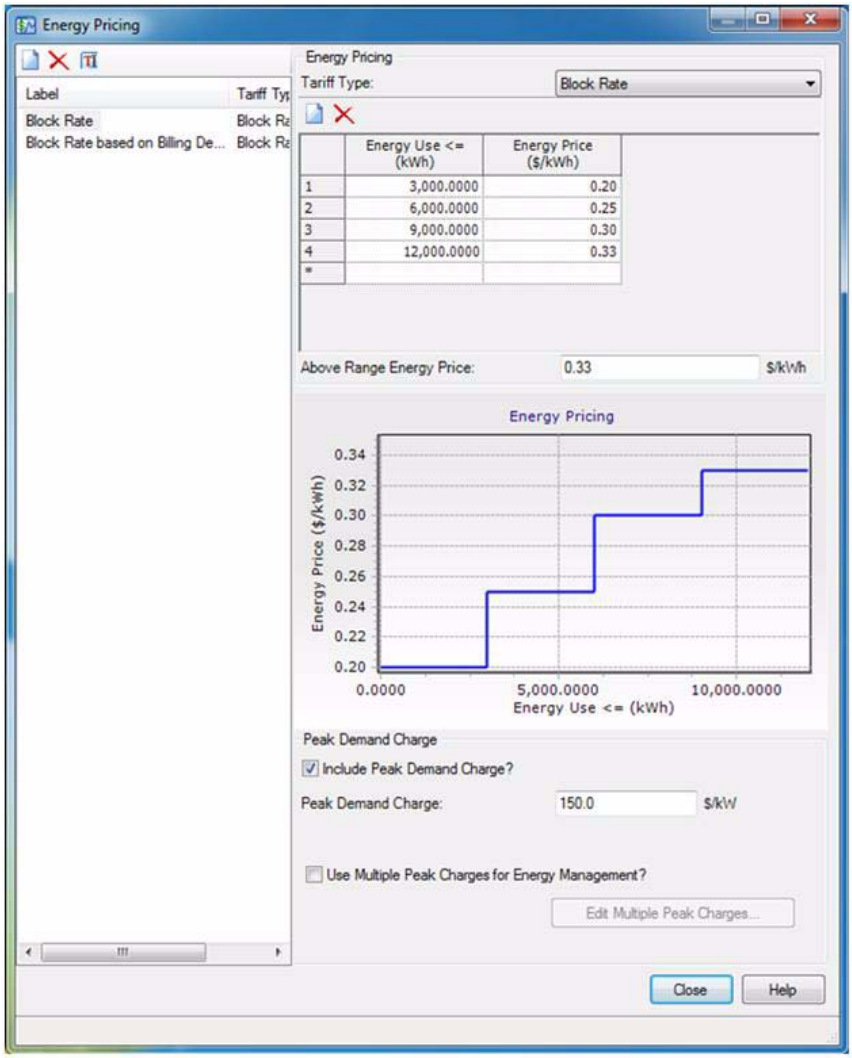
To specify the elements that will be considered in the calculation

1. Highlight the scenario icon in the tree view.
2. Click the Pumps tab. All of the pumps in the model are listed in the table. By default, all of the pumps in the model are included in the energy cost calculations. To disregard a pump during the calculation, clear the Include in Energy Calculation? check box associated with it.
3. Assign Energy Pricing to each pump that will be included in the calculation. Choose an energy price definition for each pump from the list in the Energy Pricing column. If no energy price definitions have been defined, click the ellipsis button to open the Energy Pricing Manager. See the Energy Pricing Manager topic for more details on creating a new energy pricing definition.
4. Click the Tanks tab. All of the tanks in the model are listed in the table. By default, all of the tanks in the model are included in the energy cost calculations. To disregard a tank during the calculation, clear the Include in Energy Calculation? check box associated with it.
5. If there are VSPB (variable speed pump battery) elements in your model, follow the instructions for Pumps above to specify which VSPBs are to be included in the calculation and to assign energy pricing definitions to them.

Note: VSPBs are not included in the pump station calculations.

Energy Pricing Manager

To convert energy use into energy cost, the user must enter the applicable energy price tariff. This is done by picking the second button above the left pane in the Scenario Energy Cost dialog  . This opens the energy pricing dialog. The left pane provides away for the user a way to create or delete any number of energy price functions (tariffs). Pick New to begin creating a new tariff in the right pane.



There are two general types of changes for energy: energy cost which depends on the kilowatt hours used (top part of right pane) and peak demand charges based on the peak kilowatts used (bottom part of right pane).

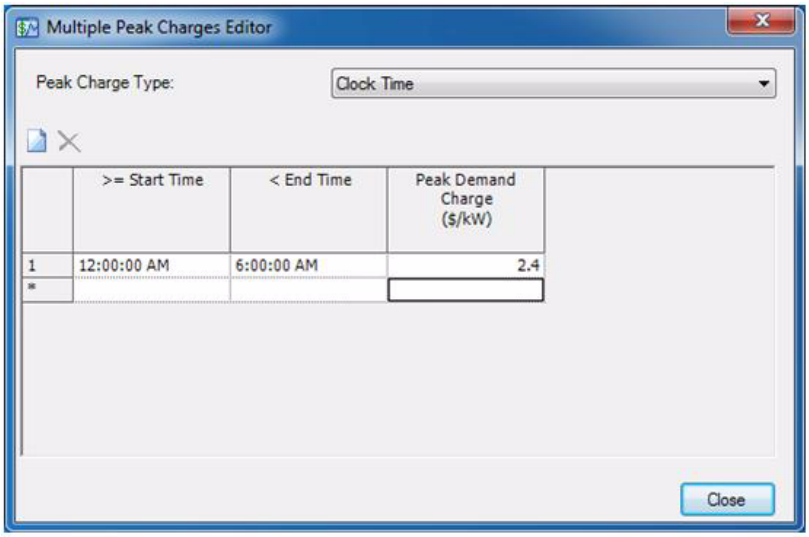
The tariff type refers to whether the energy tariff:

1. Constant - no variation over time and non-block rate
2. Time of day - energy price varies with time of day
3. Block rate - energy price depends on total energy consumed during billing period
4. Block rate based on billing demand - energy price depends on total energy consumed and the break point between the blocks depends on "billing demand" which is the peak energy use

Once the type of tariff has been selected, the data entry table corresponding to that type of tariff is displayed in the middle of the right pane. For constant price, there is a single value that must be entered. For the others, there is tabular data entry for the energy price as a function of the parameter that defines the block or the time period.

The bottom part of right pane enables the user to enter a description of any peak demand charges if they apply to this study by checking the box labeled Include Peak Demand Charge. The user enters the charge in cost units per peak demand kilowatts. The peak demand is usually taken as the peak demand over some time period and for the calculation, it is assumed that the model time step corresponds to this time period. The billing period can be entered so that this cost can be averaged and included in daily cost (but not usage cost).

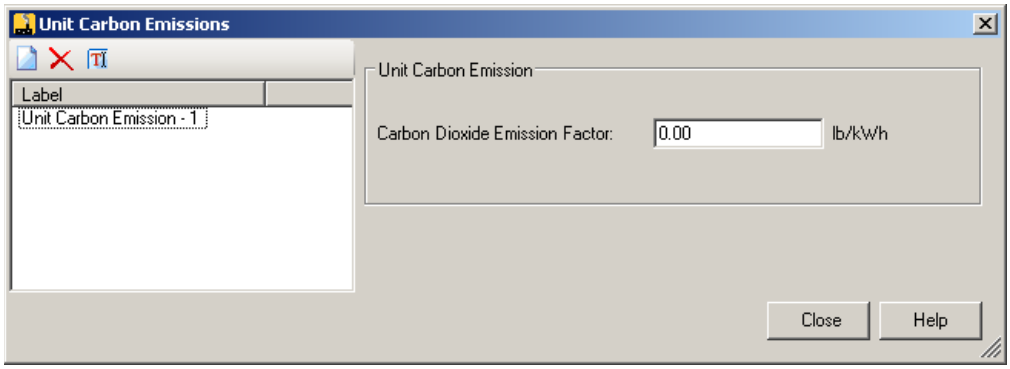
In some cases, there may be different demand charges for different times of day. The user can enter this type of tariff by picking Use Multiple Peak Charges for Energy Management. This will open the dialog below where the user can enter the time of day peak charges either as a function of clock time or simulation time.



In some cases, power is purchased from multiple energy providers each with very different tariffs. For example, energy may be purchased for an energy generation company while distribution is provided by a different company. If the tariffs are similar, then the unit prices can be added. However, if they are very different, the user should set up one tariff for each supplier and run each cost calculation separately.

Unit Carbon Emissions Dialog Box

This dialog allows you to define the amount of carbon emissions per unit of energy usage.



The dialog consists of a pane listing the Unit Carbon Emissions definitions and the the following controls:

New



Creates a new Unit Carbon Emissions definition, allowing you to define a new Carbon Dioxide Emission Factor.

Delete



Deletes the Unit Carbon Emission definition that is currently highlighted in the list pane.

Rename



Renames the Unit Carbon Emission definition that is currently highlighted in the list pane.

Wehn you highlight a Unit Carbon Emission definition in the list pane, you can edit the **Carbon Dioxide Emission Factor** associated with that definition.

Scenario Energy Cost Analysis Calculations

To run the scenario energy cost calculation:

1. Select the scenario name from the menu. The hydraulic calculations for this scenario must already have been run and the scenario must use EPS hydraulics.
2. Select the price function to use for each pump. If this is not specified you will see a warning message.
3. Click **Compute**  to run the calculation.

Energy Cost Results

Daily Cost - The energy cost divided by the number of days in the EPS run plus the demand charge divided by the days in the billing period.

Usage Cost - The total pump energy usage over the entire EPS run, not including demand charges.

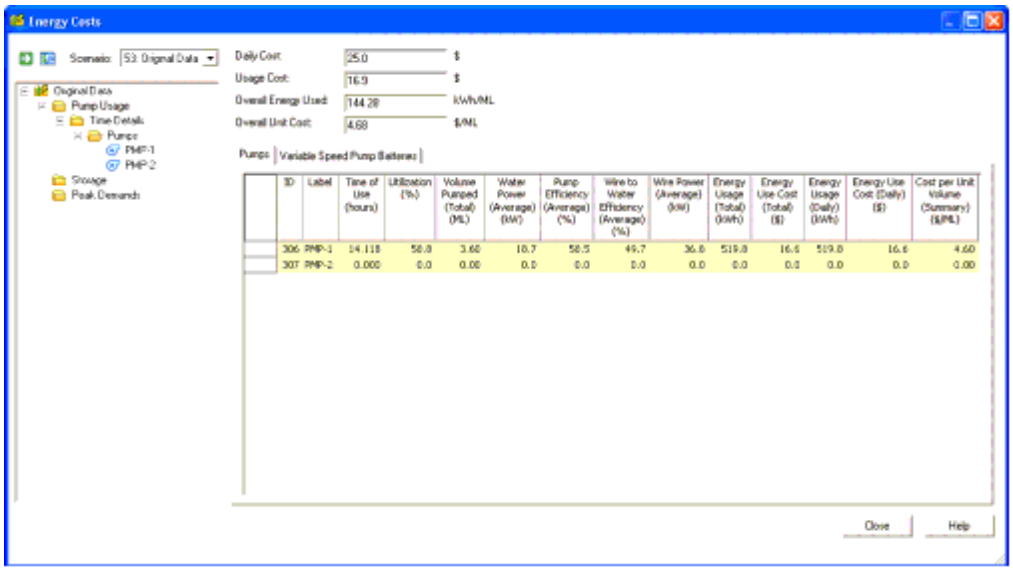
Overall Energy Used - Unit energy expended per unit of volume pumped. The formula used to arrive at this value is: (Pump Energy Used)/(Total Volume Pumped).

Overall Unit Cost - Unit cost per unit of volume pumped. The formula used to arrive at this value is: (Usage Cost)/(Total Volume Pumped).

After a successful energy cost calculation, the following results summaries appear in the tree view:

Pump Usage

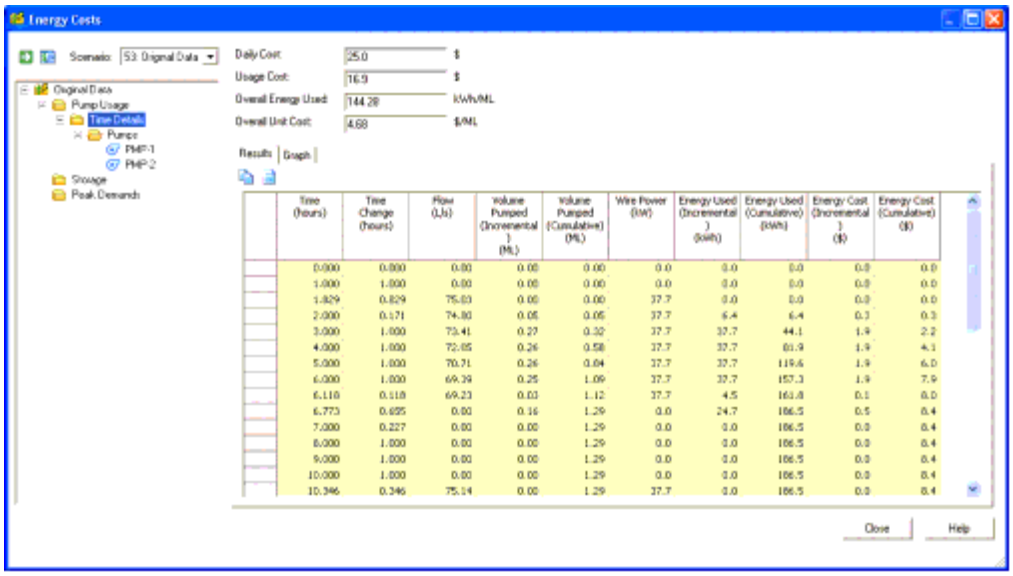
The most important results in the Pump Usage summary are the **Total Energy Use Cost** and the **Average Efficiency**, either pump or wire-to-water.



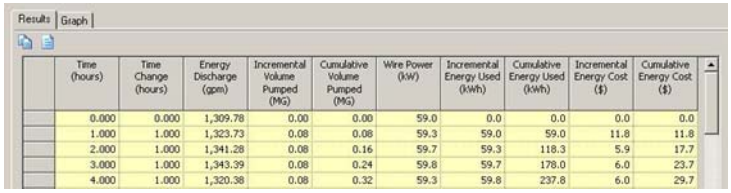
There are tabs for Pumps and Variable Speed Pump Batteries.

Time Details

The Time Details summary gives the energy usage study summed up over all the selected elements. These results can also be copied to the clipboard or displayed in a report using the Copy and Report buttons above the table.



Some values in the table are instantaneous values at that time and others are incremental values from that time to the next time. For example:

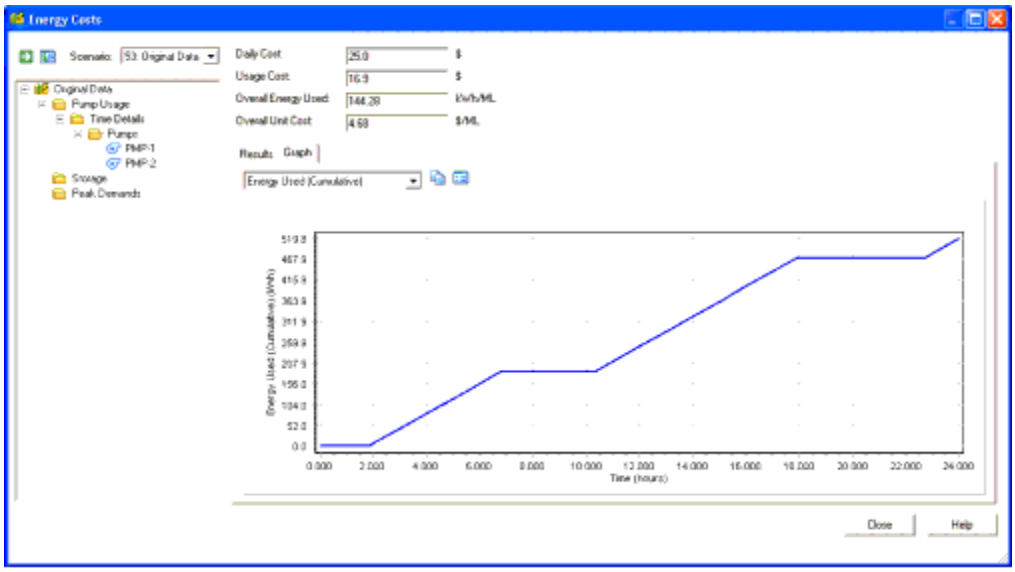


Results | Graph

	Time (hours)	Time Change (hours)	Energy Discharge (gpm)	Incremental Volume Pumped (MG)	Cumulative Volume Pumped (MG)	Wire Power (kW)	Incremental Energy Used (kWh)	Cumulative Energy Used (kWh)	Incremental Energy Cost ($)	Cumulative Energy Cost ($)
	0.000	0.000	1,309.78	0.00	0.00	59.0	0.0	0.0	0.0	0.0
	1.000	1.000	1,323.73	0.08	0.08	59.3	59.0	59.0	11.8	11.8
	2.000	1.000	1,341.28	0.08	0.16	59.7	59.3	118.3	5.9	17.7
	3.000	1.000	1,343.39	0.08	0.24	59.8	59.7	178.0	6.0	23.7
	4.000	1.000	1,320.38	0.08	0.32	59.3	59.8	237.8	6.0	29.7

The value of 1309 for discharge is the instantaneous value at time 0, while the incremental volume pumped is the volume pump from the previous time step until time equals 0. At time 3, the instantaneous value for flow is 1343 gpm but the value for Incremental volume pumped is the volume pumped between times 2 and 3, which is $(1341*60/10^6)=0.08$. Incremental values at time t(i) are the value between t(i-1) and t(i). Attributes such as wire power, efficiency, and cumulative energy used are instantaneous values corresponding to t(i).

You can also view the results in graph form by clicking on the **Graph** tab.

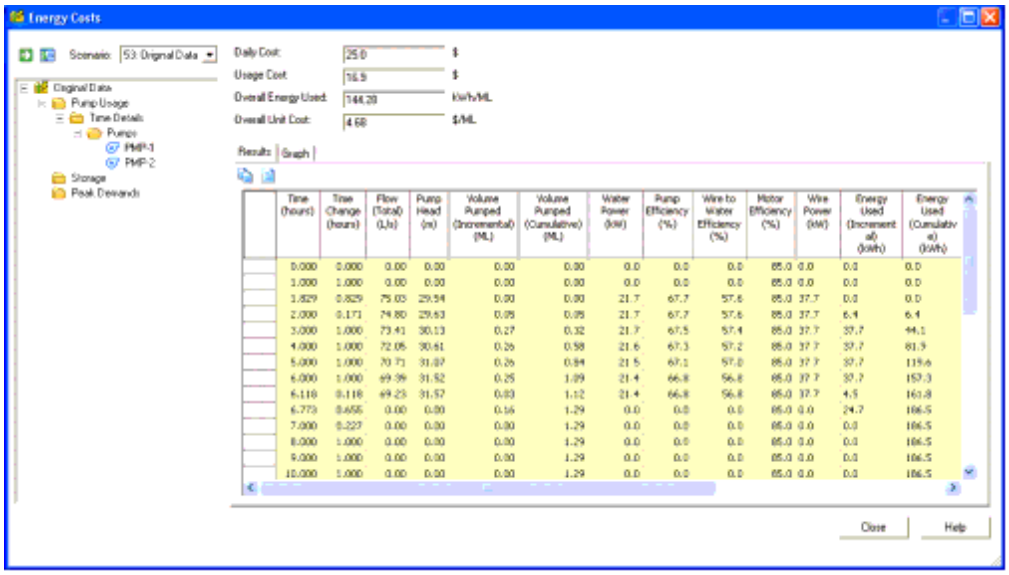


You can copy the graph to the clipboard for use in other software and you can open the **Graph Editor** to change the appearance of the graph. (See Tee Chart editor for more information.)

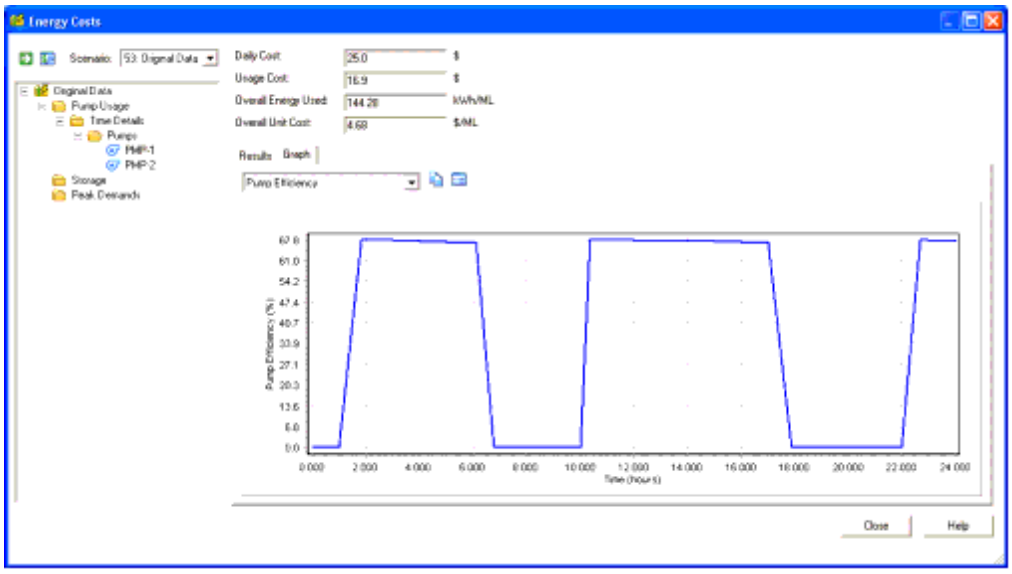
If you change the default settings for the Graph Manager, they are applied to all graphs as long as you remain in the Energy Cost Manager. Once you close the energy cost manager, the graph manager goes back to the default settings.

Pump Results

Below Time Details icon is a Pumps folder containing an icon for each individual pump. Clicking one of these pump icons will display results for that pump. It includes the information that is in the time details report, except it only includes results for one pump at a time. An additional column is shown for pump speed.



You can also view the results in graph form by clicking on the **Graph** tab.

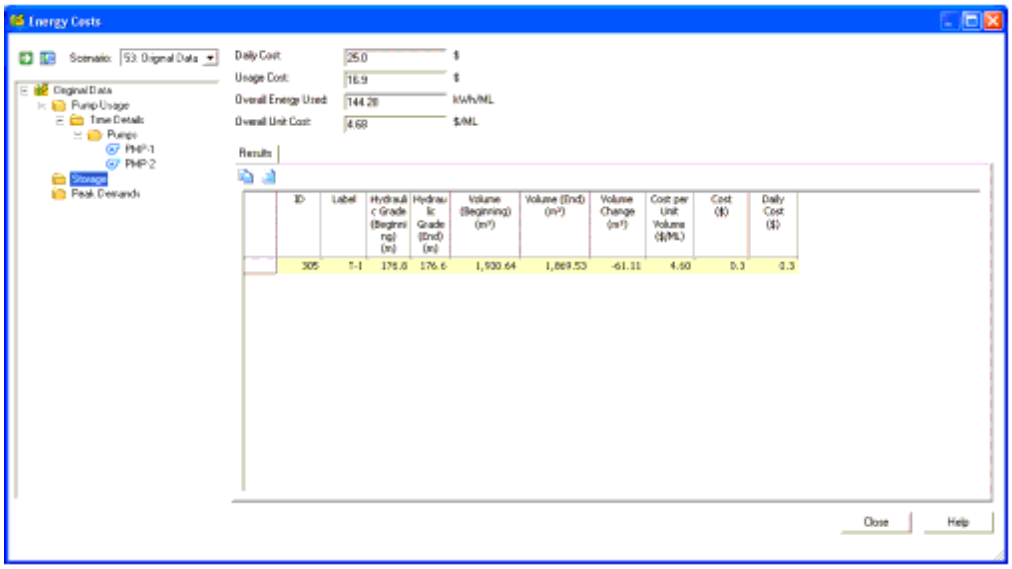


You can copy the graph to the clipboard for use in other software and you can open the **Graph Editor** to change the appearance of the graph. (See Tee Chart editor for more information.)

If you change the default settings for the Graph manager, they are applied to all graphs as long as you remain in the Energy Cost manager. Once you close the Energy Cost manager, the Graph manager goes back to the default settings.

Storage

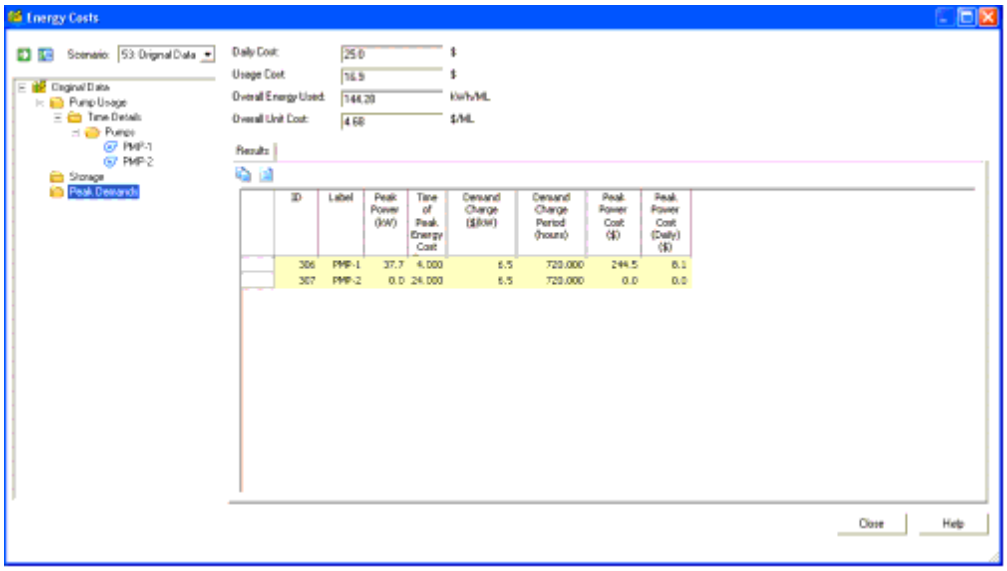
The values displayed in the storage table show the value of energy that is used by draining water from a tank or gained by storing water in a tank.



These results can also be copied to the clipboard or displayed in a report using the Copy and Report buttons above the table.

Peak Demands

The results in the Peak Demands table are used to determine the cost for capacity/demand/peaking charges that are based on peak energy use. These costs are usually applied to the energy cost as a lump sum each billing period. The table also divides the cost by the length of the billing period to determine the daily cost so that it can be added to the energy costs. Peak demand charges are usually set on a peak water use day or a day with a special event, such as a fire or large main break. Demand charges are not set on an average day.



These results can also be copied to the clipboard or displayed in a report using the Copy and Report buttons above the table.

Comparing Cost Results Across Scenarios

Within the Energy Cost manager, it is only possible to view graphs that apply to a single scenario at a time. In order to view a comparison of energy results for a single pump between multiple scenarios, it is necessary to use the Graph manager. It can be accessed when you right-click the pump and select the energy related fields and scenarios to graph in the Graph manager.

14 Optimizing Pump Schedules Using Darwin Scheduler

Darwin Scheduler is a state of the art tool for optimizing pump operation that works by using genetic algorithm optimization to control nominated pumps during an extended period simulation (EPS). The genetic algorithm optimization technique works by evolving near optimal solutions over generations of trial solutions. To reach an optimal solution it is normally expected to have to evaluate tens of thousands of solutions, sometimes more. One problem with this fact is that EPS simulations can be time consuming, especially for larger or more complicated models, and therefore run times for Darwin Scheduler can be correspondingly long. These best practices and tips offer suggestions and recommendations for using Darwin Scheduler in order to get the best performance and results out of the tool.

Best Practices and Tips

Minimize the solution space

In optimization problems one is looking for an optimal or near optimal solution from a set of possible input values. For problems with a low complexity the total number of possible permutations of valid input may be able to be completely enumerated. Consider a steady state problem where 2 pumps can be either on or off. If we represent the on state with the number 1 and the off state with the number 0, using the following notation (1, 1) we indicate that both pumps are on. One trial solution in such a problem is (1, 0). Clearly there are 4 possible permutations in this problem, the other three being (0, 1), (0, 0) and (1, 1). The set of all possible permutations of input is known as the solution space. Even if a single permutation of input or trial solution took an hour to evaluate, the entire solution space could be enumerated in 4 hours,

making it practical to do so provided that the optimal solution is not required to be known in less than that time. The solution space for this 2 pump problem is size 2^2 or 4. The solution space for an equivalent 10 pump problem is 2^10 or 1,024. What is not immediately obvious, however, is that the size of the solution space in optimization problems can quickly grow to mind boggling sizes.

For example, let us consider a pump schedule optimization problem with 10 pumps and an EPS of 24 hours duration with a hydraulic time step of 1 hour. In addition to this, let's assume the pumps are optimized as variable speed with possible settings of 0.8, 0.85, 0.9, 0.95 and 1.0. Assuming the pumps are all optimized for the entire duration of the EPS (time 0 to time 24 hours) then there are 10 x 24 = 240 speed decisions to be made for each trial solution, and each of those decisions can take on one of 5 different values. Even for this modest sounding optimization problem the size of the solution space is thus 5^240 or 5.65 x 10^167! Now let's assume that we can easily write off 99.99% of solutions as not practical or plain non-sense, then that leaves just 5.65 x 10^163 solutions for us to investigate. If we could evaluate one million trial solutions every second, it would still take 1.79 x 10^150 years to evaluate them all! One public estimate of the number of atoms in the entire observable universe is 10^80, which is virtually zero when compared to 1.79 x 10^150, so quite clearly we are talking about numbers that are so large they are difficult if not impossible to comprehend. A small increase in complexity of the problem magnifies the total number of possible solutions greatly. Conversely a small decrease in problem complexity reduces the total number of possible solutions greatly. It is therefore a very good idea to consider the following when setting up a pump scheduling optimization problem.

A. Number of pumps being optimized; keep the number of pumps being considered to the minimum possible, to the point of considering optimizing different pump stations independently if that is a reasonable thing to do hydraulically in the system being optimized.

B. Number of pump speed choices; keep the number of possible speed choices (including off setting) to the minimum possible. Consider optimizing with course speed settings to find a rough solution to the optimization problem and follow that up with an optimization that uses refined speed settings (finer, but narrower range) as a follow up optimization to the first.

C. Schedule control interval (EPS hydraulic time step); consider using a course hydraulic time step such as 2 or even 3 hours at least for initial optimization runs as this greatly reduces the size of the solution space, especially if multiple pumps are being optimized.

D. Schedule duration; consider optimizing the shortest EPS duration possible. A 24 hour duration seems to be the most reasonable choice in terms of being able to produce a repeatable schedule, whilst keeping the solution space as small as possible.

The following table shows the size of the solution space given different numbers of pumps being optimized (Pump Count), numbers of speed choices per pump (Speed Choices) and EPS time step. It is very evident the effect that increasing the number of pumps being optimized, the number of speed choices or the granularity of the EPS time step each have an exponential effect on the size of the solution space, and thus inevitably reduce the effectiveness of the optimization. When running an optimization it is wise to start out conservatively and only increase the optimization complexity to refine optimization results.

Table 14-1: The effect on optimization solution space of number of pumps to optimize, number of speed choices and EPS time step (control interval).

Pump Count	Speed Choices	Solution Space (1 hour time step)	Solution Space (2 hour time step)	Solution Space (3 hour time step)
1	6	4.7E+18	2.2E+09	1.7E+06
1	12	7.9E+25	8.9E+12	4.3E+08
1	18	1.3E+30	1.2E+15	1.1E+10
2	6	2.2E+37	4.7E+18	2.8E+12
2	12	6.3E+51	7.9E+25	1.8E+17
2	18	1.8E+60	1.3E+30	1.2E+20
3	6	1.1E+56	1.0E+28	4.7E+18
3	12	5.0E+77	7.1E+38	7.9E+25
3	18	2.4E+90	1.5E+45	1.3E+30
4	6	5.0E+74	2.2E+37	8.0E+24
4	12	4.0E+103	6.3E+51	3.4E+34
4	18	3.2E+120	1.8E+60	1.5E+40

Minimize the trial solution time

In our discussion of minimizing the solution space we consider the time required to enumerate the top 0.001% of trial solutions by assuming that we can evaluate one million trials per second. Clearly this figure is un-realistic even on today's fastest computers and for the most trivial of hydraulic models, so it's clear that the time the

model takes to solve is a significant contributor to the total time required to run Darwin Scheduler. Any improvement that can be made to the run-time of the base EPS simulation all the better for the Darwin Scheduler optimized run time. Methods to reduce run time that should be considered include:

1. **Model size**: The more hydraulic elements in a model the larger the solution matrix that needs to be solved and the longer the run-time of the solution. It is unrealistic to expect to be able to use Darwin Scheduler on a 50,000 pipe model in a few minutes if a single EPS run for such a model takes a few minutes. Strongly consider using a version or copy of the subject model that is customized for the purpose of pumping optimization. Such a model might be smaller due to excluding elements or zones etc not required for the energy optimization or it may be smaller due to skeletonization (removal) of hydraulic elements not required to be considered in the energy optimization. **In fact a skeletonized model is highly recommended for pump schedule optimization, particularly if the model is skeletonized whilst maintaining hydraulic equivalence such as is able to be performed using Skelebrator Skeletonizer. The benefit of the smaller model and quicker run time will greatly outweigh any potential or perceived side effect (if any at all) of the skeltonization process.**
2. **Model complexity**: The larger the model or more complex the model (e.g., complicated control regimes) the longer an EPS simulation will take to run due to the need to simulate additional intermediate time steps (such as times when control rules fire). Consider removing any redundant model complexity that may not be required for a pump operation simulation.
3. **Model balance**: Even a small model may take a long time to run if it is not well balanced. Examine the number of trials the model takes to solve at each time step and if it is found that it is consistently high (25-100+) then there may be time to be saved by improving this situation. A high number of trials may be indicative of a number of different symptoms such as bade control valve settings or too narrow control ranges.

Use a faster computer

These days most computers are reasonably fast, however, time is money in which case a faster computer can save both time and money. The Darwin Scheduler optimization process is computationally expensive and as such a computer with a faster CPU will produce faster results. Multi-core machines will also benefit from increased overall performance.

Carefully consider hydraulic constraints

If certain hydraulic constraints are required to be met it is a good idea to consider these carefully and only add the constraints that are essential as opposed to adding blanket constraints. Adding blanket constraints, especially for large models, is discouraged since blanket constraints are more likely to contain impossible to meet constraints (such as pressure constraints on a junction that is suction side of a pump)

and will also have a slight effect on performance (constraints have to be evaluated for every trial solution) and increase Darwin Scheduler's output file size unnecessarily. For this reason Darwin Scheduler is designed to require the user to add constraints manually.

Ensure runs are set up properly

Even for a small well balanced model run times for Darwin Scheduler will be proportional to the time a single EPS takes to run, multiplied by the number of trials required to find a near optimal solution. It is therefore a good idea to ensure that a run is progressing in an acceptable fashion in its early stages (generation 50 - 200) before leaving it to run for the remainder of the optimization. Be sure to leverage Darwin Scheduler's resume feature that allows one to stop a run, review the results (even export the solution) and then continue the run so long as no other runs have been started or no other hydraulic computation has been performed.

Plan to use the tool efficiently

One good thing about computers is that they don't need to sleep like people do. When working with larger models that may require a longer run time consider running shorter debugging optimization runs during the day, making necessary adjustments and the like, and then running the "real" runs during a lunch break or perhaps even over-night.

Allow runs sufficient time to complete

One characteristic of genetic algorithm optimization is the need for heuristic stopping criteria. In Darwin Scheduler several different criteria are available depending on the type of genetic algorithm selected. There is, however, no definitive way to determine when a run should be stopped. Running just one more generation may yield a better solution than previously found. Generally speaking, however, optimization runs should be allowed to run for at least 500 generations (preferably longer) which, depending on population size, can mean the order of 100,000+ trials. Please be patient!

Plan to do multiple runs

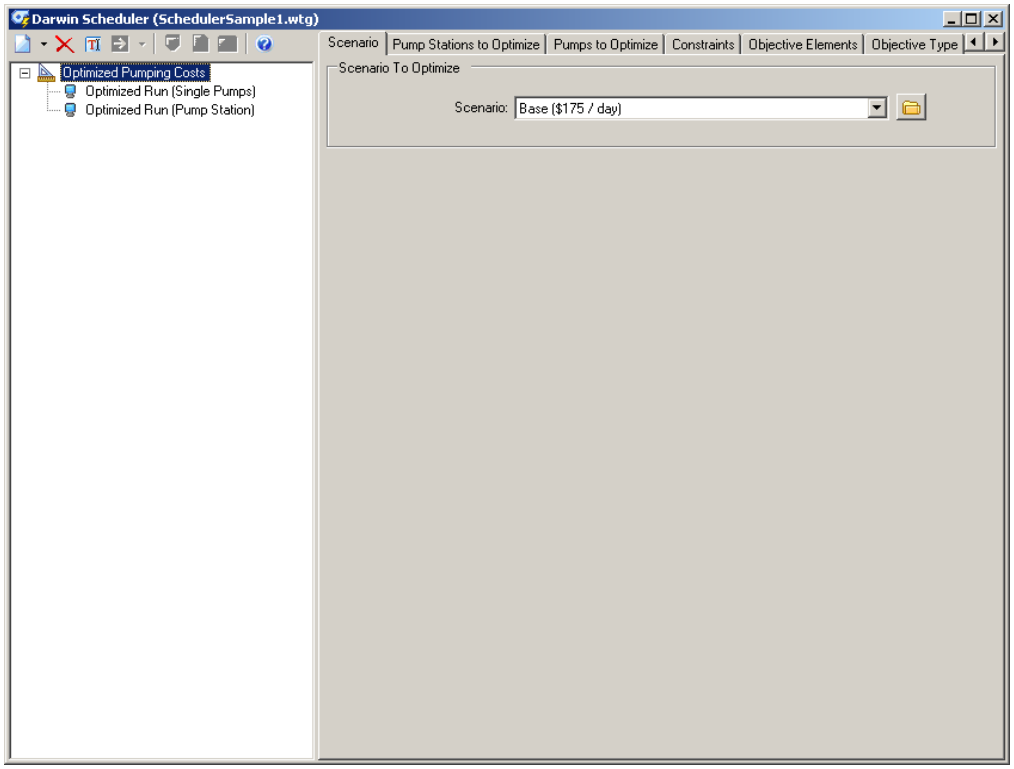
The nature of genetic algorithm optimization is such that there is a random component to the algorithm. The algorithm is driven by computationally efficient search processes; however, at the core of the algorithm random numbers are used to drive processes such as mutation, for example. Therefore, two optimization runs that are otherwise identical except for one minor change (e.g., larger population size or different random seed) will in all likelihood produce different optimized solutions. This is more likely to be the case the larger the solution space of the problem. It is therefore a good idea to run multiple optimization runs changing nothing other than one or more genetic algorithm parameters (or simply just the random seed) to ensure that the best optimized solution is really the best that can be achieved. One beneficial

characteristic of genetic algorithm optimization is its ability to find solutions that my be very close in terms of hydraulic performance, but may be themselves quite different. Engineers are therefore able to discriminate between optimized solutions based on other perhaps non hydraulic criteria.

You can also leverage an existing solution (such as the representative scenario, assuming it meets constraints) to create a Baseline Seed for scheduler to use. Export the results of a Scheduler run to a new scenario, then calculate an EPS run for the new scenario. Use this scenario as Scheduler's representative scenario to seed a new Scheduler run.

Darwin Scheduler

Darwin Scheduler allows you to optimize pump operations. By using genetic algorithm optimization to control nominated pumps during an extended period simulation (EPS), it avoids a manual trial and error approach to finding the most efficient operating schedule. Solutions and costs calculated using Darwin Scheduler can be exported back to the selected scenario.



The dialog consists of:

A toolbar.

A list pane that displays all of the Scheduler Studies Optimized Runs, and Solutions.

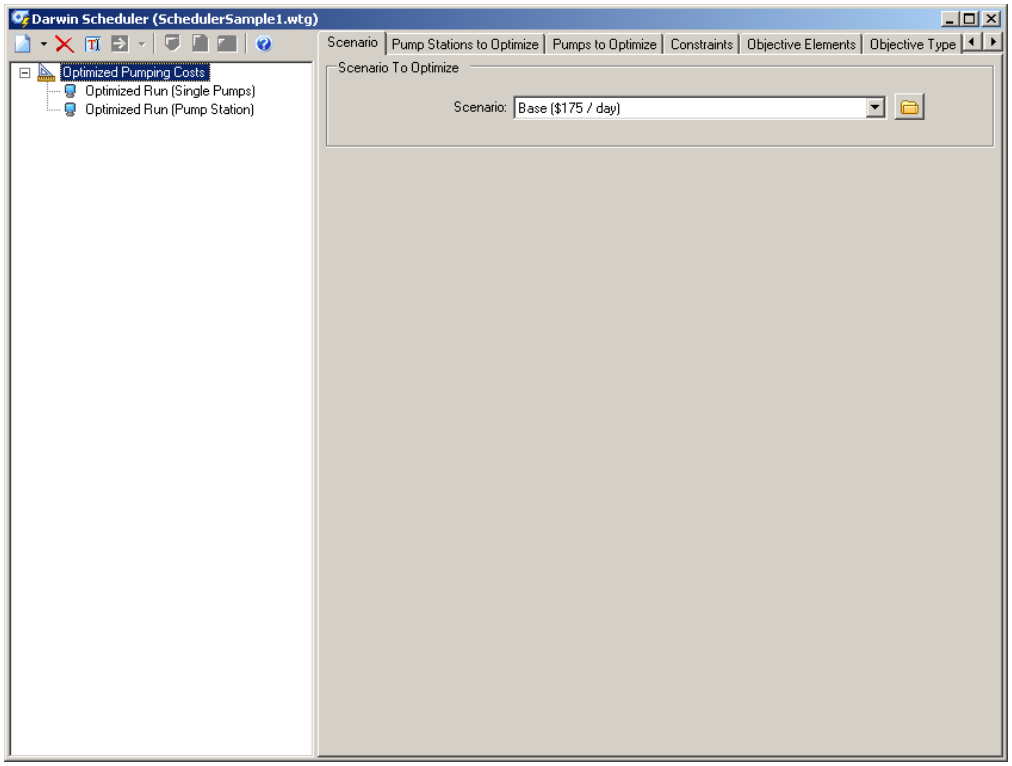
A tabbed section containing the various input data.

The toolbar consists of the following controls:

-  **New**: Opens a submenu containing the following commands:
  - **New Scheduler Study**: Creates a new Scheduler Study in the list pane.
  - **New Optimized Run**: Creates a new Optimized Run under the Scheduler Study that is currently highlighted in the list pane.
-  **Delete**: Deletes the item that is currently highlighted in the list pane.
-  **Rename**: Allows you to rename the item that is currently highlighted in the list pane.
-  **Compute**: Opens a submenu containing the following commands:
  - **Compute**: Computes the optimized run that is currently highlighted in the list pane.
  - **Resume**: Resumes the incomplete optimized run that is currently highlighted in the list pane.
-  **Export to Scenario**: Opens the Export to Scenario dialog, allowing you to define the export settings.
-  **Report**: Opens a preformatted report containing the data for the currently highlighted solution.
-  **Graph**: Opens a graph containing the data for the currently highlighted solution.
-  **Help**: Opens a the online help.

Scheduler Study

A Scheduler Study is the top-level grouping of the settings and input data related to the optimization to be performed. This includes picking a scenario to optimize, defining pump decisions, constraints and objective elements.



To start using Darwin Scheduler, you must create a Scheduler Study. All Darwin Scheduler data resides within the Scheduler Study.

A Scheduler Study includes the following:

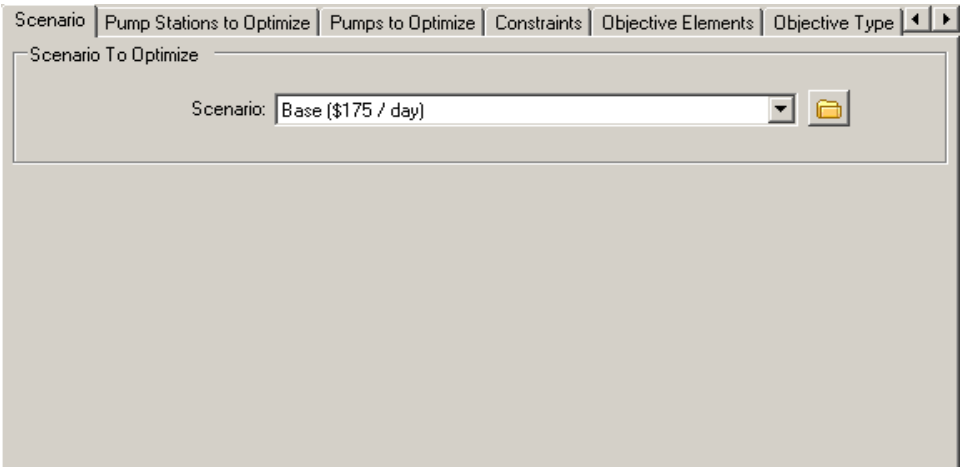
1. The scenario to optimize.
2. The set of pumps being scheduled.
3. Constraints that must be met by the solutions offered after a run.
4. Energy price data and tank definitions to be used during the optimization.
5. The type of objective.
6. Genetic algorithm options and parameters.
7. The results of optimized runs.

It is apparent that one or more of these items will be different between different scheduler studies, hence the ability to create as many scheduler studies as you need.

You can create more than one scheduler study. Each design study can include one or more optimized runs.

Scenario Tab

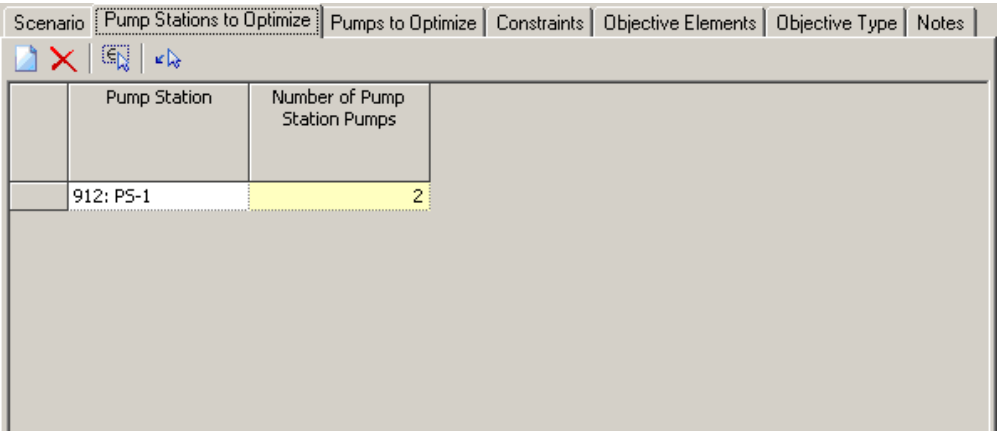
The Scenario tab allows you to select the scenario to optimize.



Select the scenario from the menu or click the Scenarios button  to open a dialog that displays the scenario hierarchy and allows you to select the desired scenario.

Pump Stations to Optimize Tab

The pump stations to optimize tab allows you to define which pump stations will be optimized by Scheduler.



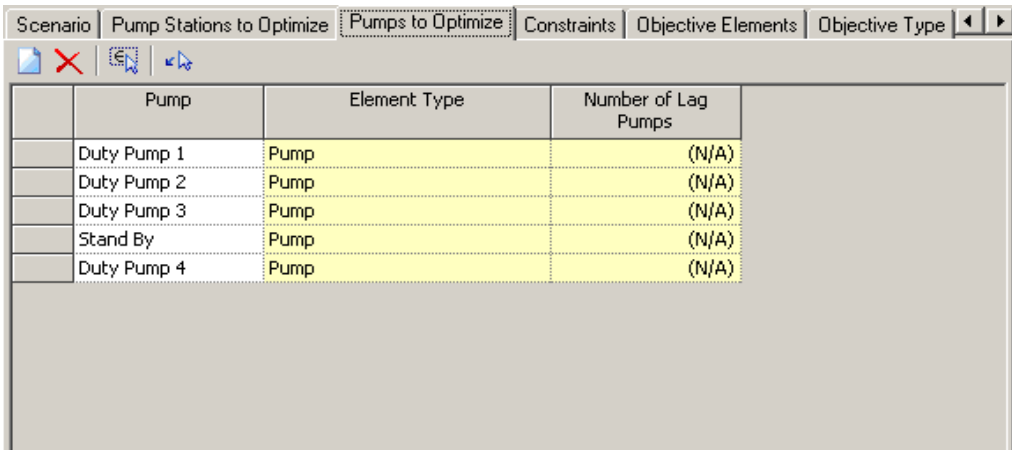
This tab consists of a table that lists the pump stations you have selected to optimize and a toolbar that consists of the following buttons:

-  **New**: Adds a row to the table.

- **Delete**: Removes the currently highlighted row from the table.
- **Initialize Table from Selection Set**: Opens the Initialize Table from Selection Set dialog, which allows you to select a predefined selection set that will be used to automatically fill in the table.
- **Select from Drawing**: Alows you to select one or more elements from the drawing.

Pumps to Optimize Tab

The pumps to optimize tab allows you to define which pumps will be optimized by Scheduler.



Pumps and pump batteries are allowable selections. For pump batteries Scheduler will also optimize the number of running lag pumps at each control time in addition to choosing the status of the main (or lead) pump.

This tab consists of a table that lists the pumps you have selected to optimize and a toolbar that consists of the following buttons:

- **New**: Adds a row to the table.
- **Delete**: Removes the currently highlighted row from the table.
- **Initialize Table from Selection Set**: Opens the Initialize Table from Selection Set dialog, which allows you to select a predefined selection set that will be used to automatically fill in the table.

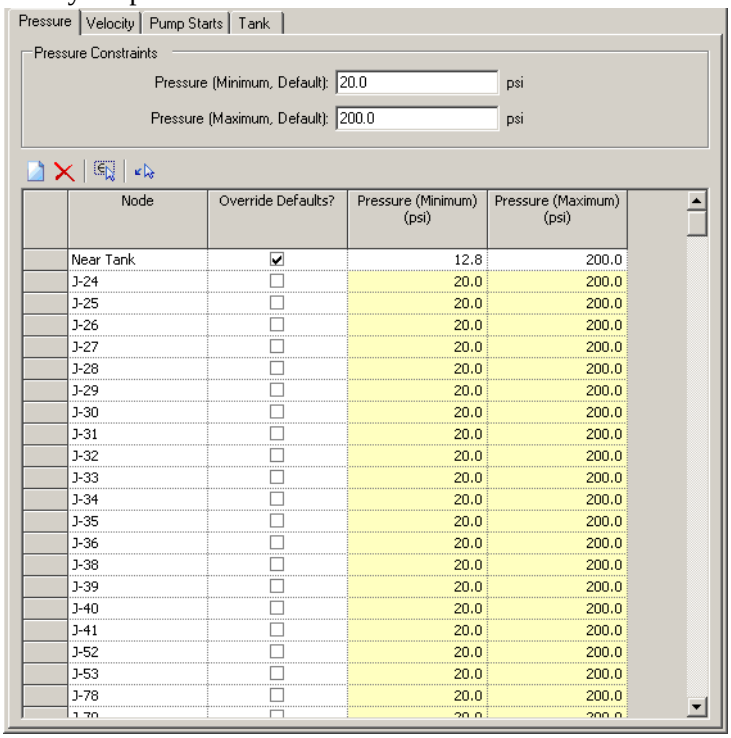
- **Select from Drawing**: Alows you to select one or more elements from the drawing.

Constraints Tab

This tab is divided into sub-tabs that allow you to define the constraints for pressure, velocity, number of pump starts, and tank levels.

Pressure Tab

This tab allows you to specify global pressure constraints, and then to override them locally at specified nodes if desired.



First, populate the table using the following toolbar buttons:

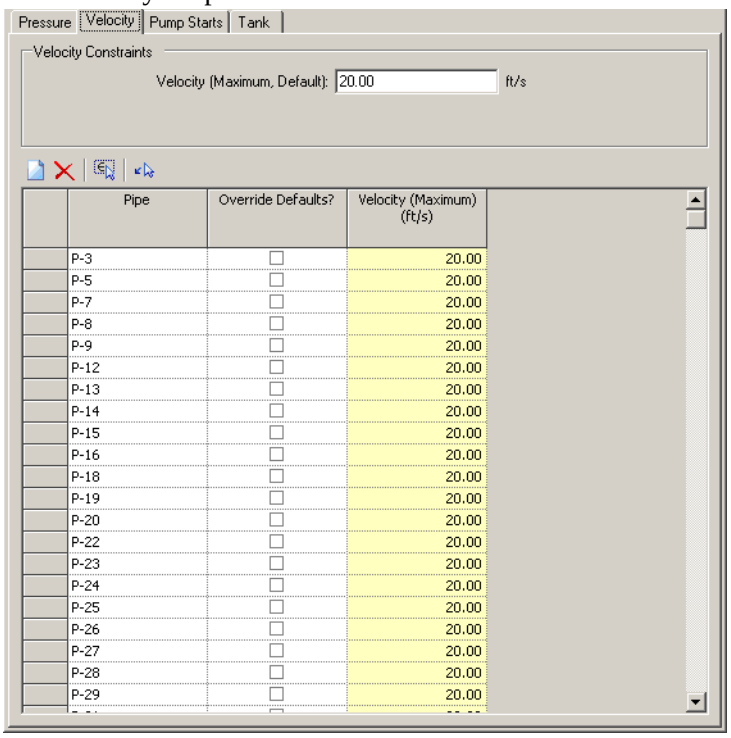
- **New**: Adds a row to the table.
- **Delete**: Removes the currently highlighted row from the table.
- **Initialize Table from Selection Set**: Opens the Initialize Table from Selection Set dialog, which allows you to select a predefined selection set that will be used to automatically fill in the table.

- **Select from Drawing**: Alows you to select one or more elements from the drawing.

Then enter the Minimum and Maximum global constraints. To override the global constraint at a node, check the corresponding Override Defaults? box and enter the values for the new minimum and maximum pressure in the corresponding fields.

Velocity Tab

This tab allows you to specify a global maximum velocity constraint, and then to override it locally at specified nodes if desired.



First, populate the table using the following toolbar buttons:

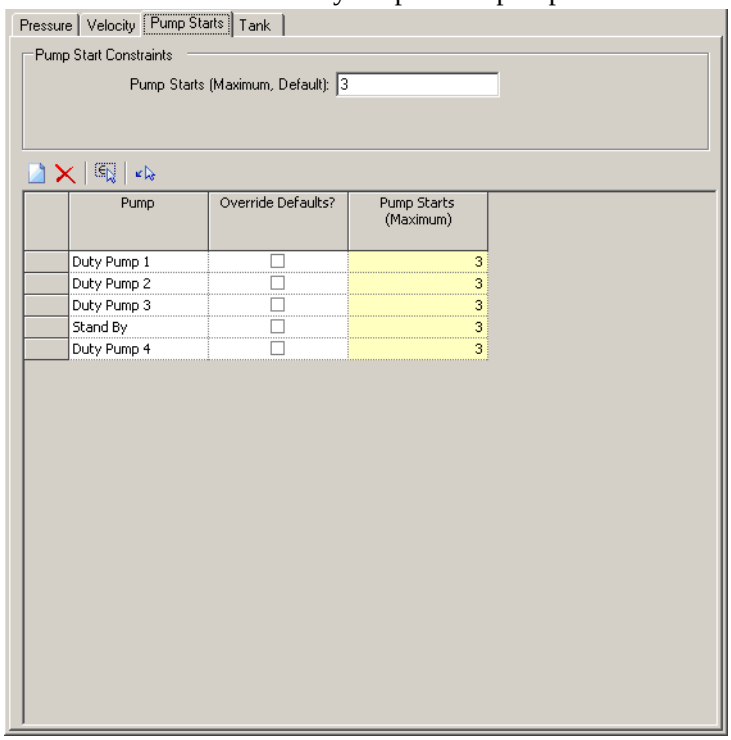
- **New**: Adds a row to the table.
- **Delete**: Removes the currently highlighted row from the table.
- **Initialize Table from Selection Set**: Opens the Initialize Table from Selection Set dialog, which allows you to select a predefined selection set that will be used to automatically fill in the table.

- **Select from Drawing**: Alows you to select one or more elements from the drawing.

Then enter the Maximum global velocity constraint. To override the global constraint at a node, check the corresponding Override Defaults? box and enter the value for the new maximum velocity in the corresponding field.

Pump Starts Tab

This tab allows you to specify the global maximum number of pump starts allowed, and then to override it locally at specified pumps if desired.



First, populate the table using the following toolbar buttons:

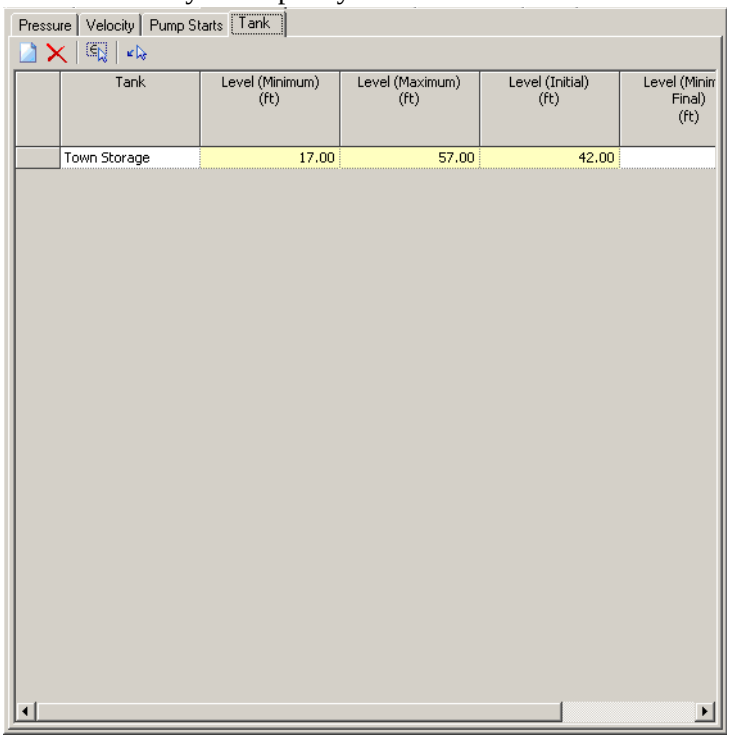
- **New**: Adds a row to the table.
- **Delete**: Removes the currently highlighted row from the table.
- **Initialize Table from Selection Set**: Opens the Initialize Table from Selection Set dialog, which allows you to select a predefined selection set that will be used to automatically fill in the table.

- **Select from Drawing**: Alows you to select one or more elements from the drawing.

Then enter the Maximum global pump starts constraint. The maximum pump starts constraint applies to the number of pump starts for the duration of the optimized schedule. To override the global constraint at a pump, check the corresponding Override Defaults? box and enter the number of maximum pump starts in the corresponding field.

Tank Tab

This tab allows you to specify the minimum final tank levels.



First, populate the table using the following toolbar buttons:

- **New**: Adds a row to the table.
- **Delete**: Removes the currently highlighted row from the table.
- **Initialize Table from Selection Set**: Opens the Initialize Table from Selection Set dialog, which allows you to select a predefined selection set that will be used to automatically fill in the table.

- **Select from Drawing**: Alows you to select one or more elements from the drawing.

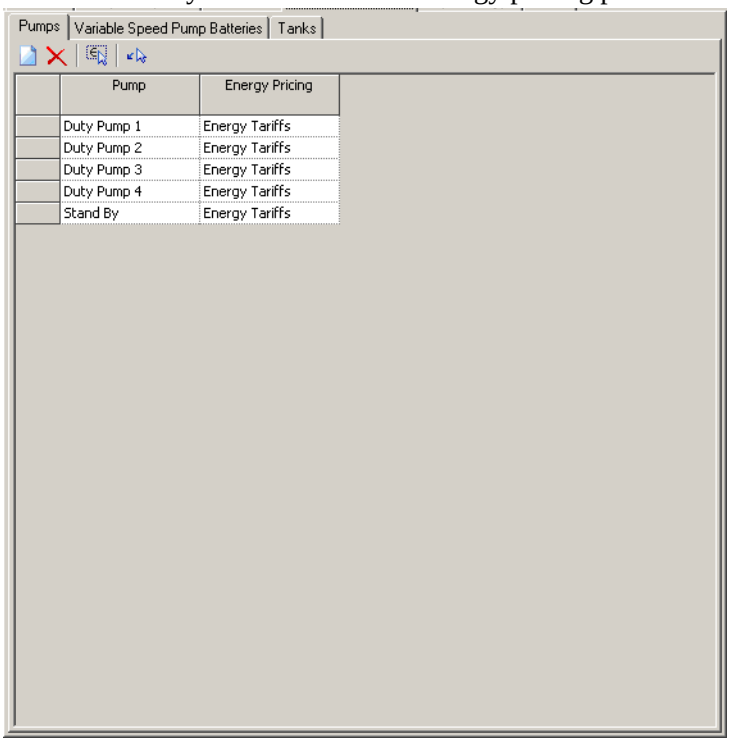
Then enter the minimum final level constraint. For each tank added to the list the current minimum, maximum and initial levels are shown to assist you in entering a correct minimum final level value.

Objective Elements Tab

This tab is divided into sub-tabs that allow you to define the energy pricing for pumps and variable speed pump batteries, as well as select the tanks that will be included.

Pumps Tab

This tab allows you to associate the energy pricing pattern with the pumps you select.



First, populate the table using the following toolbar buttons:

- **New**: Adds a row to the table.
- **Delete**: Removes the currently highlighted row from the table.

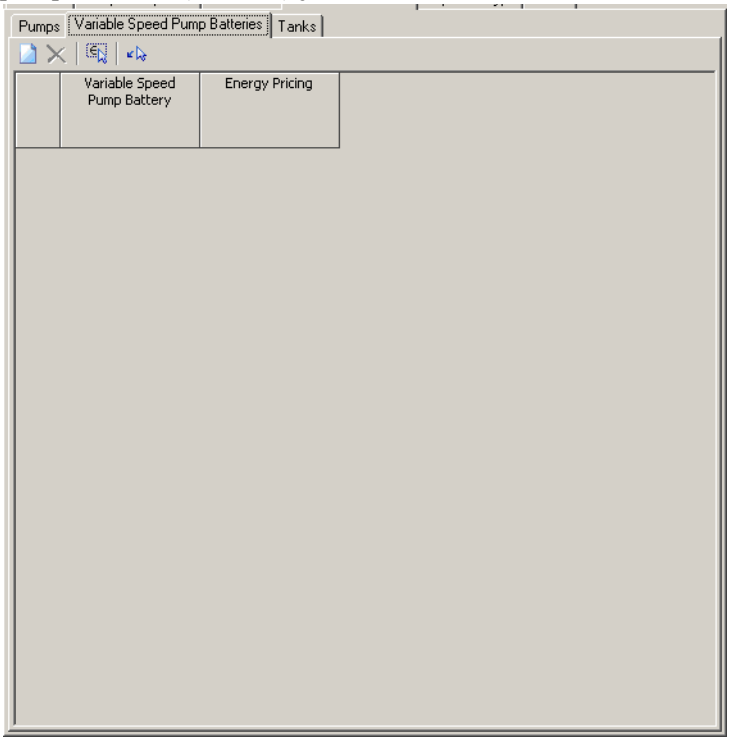
- **Initialize Table from Selection Set**: Opens the Initialize Table from Selection Set dialog, which allows you to select a predefined selection set that will be used to automatically fill in the table.

- **Select from Drawing**: Alows you to select one or more elements from the drawing.

Then select an energy pricing pattern from the menu for each pump in the table. To create a new energy pricing pattern, click the ellipsis button (...) to open the Energy Pricing manager (see Energy Pricing Manager for more information).

Variable Speed Pump Batteries Tab

This tab allows you to associate the energy pricing pattern with the variable speed pump batteries (VSPB's) you select.



First, populate the table using the following toolbar buttons:

- **New**: Adds a row to the table.

- **Delete**: Removes the currently highlighted row from the table.

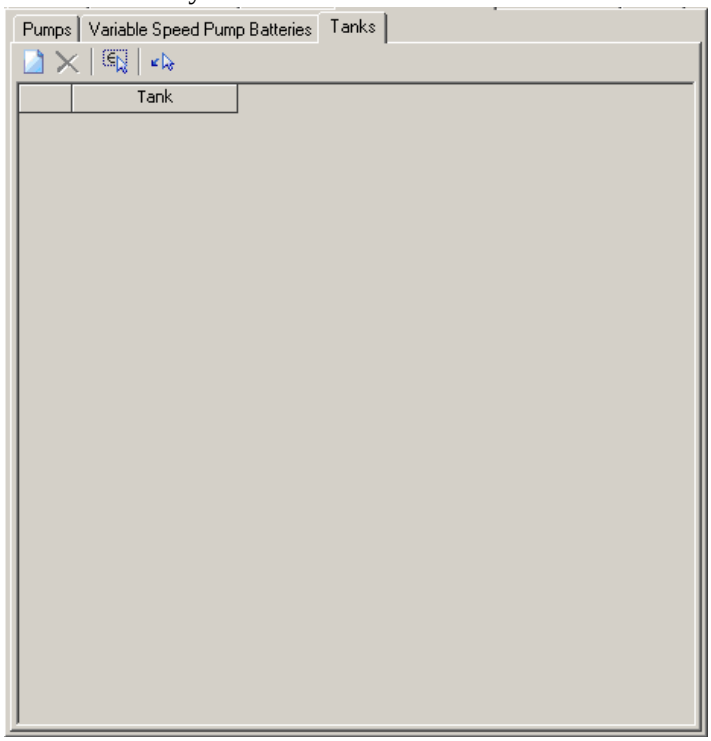
- **Initialize Table from Selection Set**: Opens the Initialize Table from Selection Set dialog, which allows you to select a predefined selection set that will be used to automatically fill in the table.

- **Select from Drawing**: Alows you to select one or more elements from the drawing.

Then select an energy pricing pattern from the menu for each VSPB in the table. To create a new energy pricing pattern, click the ellipsis button (...) to open the Energy Pricing manager (see Energy Pricing Manager for more information).

Tanks Tab

This tab allows you to select the tanks that should be used during the optimization.



Populate the table using the following toolbar buttons:

- **New**: Adds a row to the table.

- **Delete**: Removes the currently highlighted row from the table.

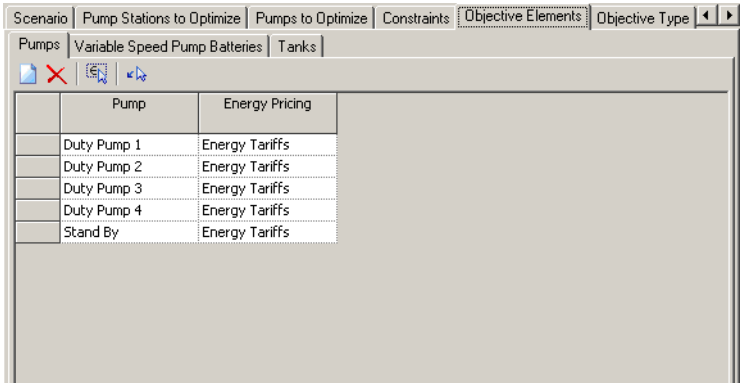
- **Initialize Table from Selection Set**: Opens the Initialize Table from Selection Set dialog, which allows you to select a predefined selection set that will be used to automatically fill in the table.

- **Select from Drawing**: Alows you to select one or more elements from the drawing.

For each row, select a tank from the menu or click the ellipsis button (...) to select one or more tanks from the drawing.

Objective Type Tab

This tab allows you to select the type of objective to optimize.



The choices include:

- **Minimize Energy Use**: This type will try to minimize the energy used. The effect of tariffs making energy cheaper at certain times is neglected in this type of optimization.
- **Minimize Energy Cost**: This type uses energy tariffs and peak demand charges to calculate the cost of energy used.

Notes Tab

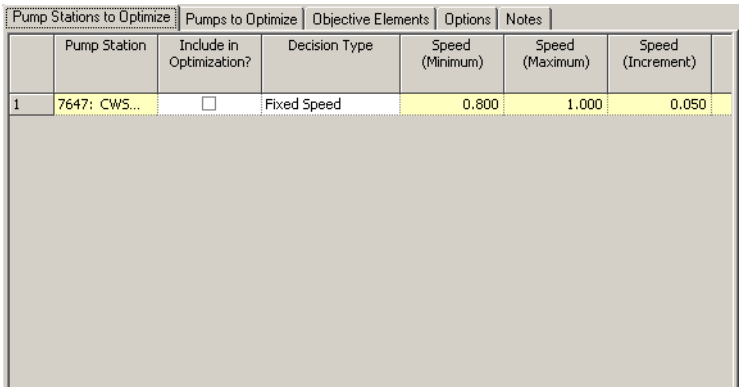
This tab allows you to enter descriptive notes that will be associated with the Scheduler Study.

Optimized Run

A Scheduler Study can contain one or more Optimized Runs. The settings for an optimized Run consist of selecting the pumps to optimize, selecting the objective elements to use, and the genetic algorithm options and parameters that will be govern the optimization.

Pump Stations to Optimize Tab

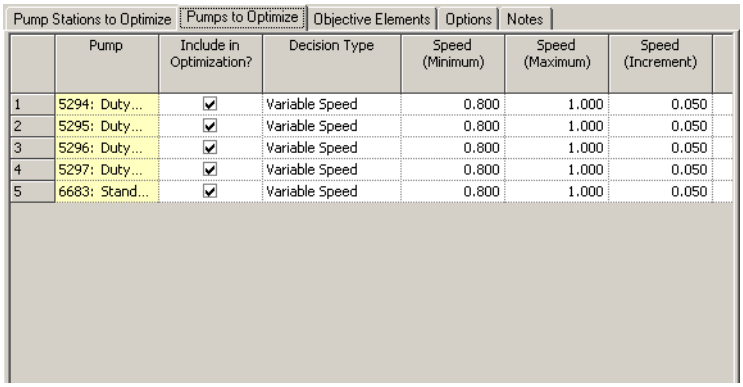
This tab allows you to define allowable pump station settings and schedule periods.



- **Include in Optimization?**: When this box is checked, the associated pump will be included in the optimization.
- **Decision Type**: This field allows you to select whether the associated pump is Fixed Speed or Variable Speed.
- **Speed (Minimum)**: The minimum speed for a variable speed pump. This field is only editable when the associated pump is a Variable Speed Decision Type.
- **Speed (Maximum)**: The maximum speed for a variable speed pump.This field is only editable when the associated pump is a Variable Speed Decision Type.
- **Speed (Increment)**: Set the increment as the lowest value that a variable speed pump's speed can be increased or decreased by. This field is only editable when the associated pump is a Variable Speed Decision Type.
- **Allow Off Setting?**:When tis box is checked, 0 speed is included in the options for variable speed pumps, in addition to the allowable choices between the minimum and maximum speed. This field is only editable when the associated pump is a Variable Speed Decision Type.
- **Time From Start**: This value, in conjunction with the Duration value, allows you to limit the scheduling period in which the associated pump may run. For instance, if the user wants to schedule one pump group only from 6am to 6pm for an EPS staring at 12am, they would enter a time from start as 6 hours, and duration as 12 hours. The scheduler engine will ensure the pumps are not running at all other times.
- **Duration**: This value, in conjunction with the Time From Start value, allows you to limit the scheduling period in which the associated pump may run. For instance, if the user wants to schedule one pump group only from 6am to 6pm for an EPS staring at 12am, they would enter a time from start as 6 hours, and duration as 12 hours. The scheduler engine will ensure the pumps are not running at all other times.

Pumps to Optimize Tab

This tab allows you to define allowable pump settings and schedule periods.



Include in Optimization?: When this box is checked, the associated pump will be included in the optimization.

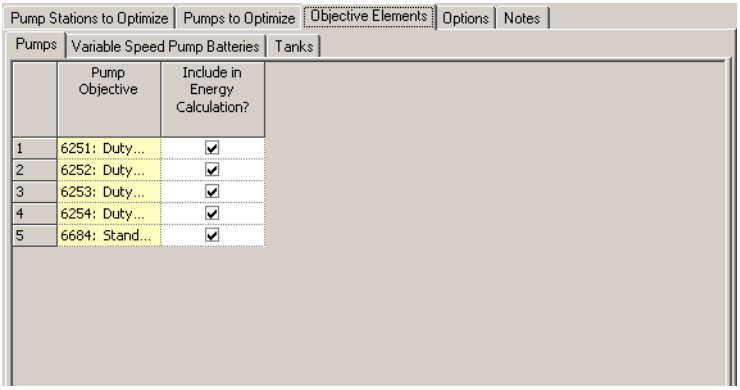
- **Decision Type**: This field allows you to select whether the associated pump is Fixed Speed or Variable Speed.
- **Speed (Minimum)**: The minimum speed for a variable speed pump. This field is only editable when the associated pump is a Variable Speed Decision Type.
- **Speed (Maximum)**: The maximum speed for a variable speed pump.This field is only editable when the associated pump is a Variable Speed Decision Type.
- **Speed (Increment)**: Set the increment as the lowest value that a variable speed pump's speed can be increased or decreased by. This field is only editable when the associated pump is a Variable Speed Decision Type.
- **Allow Off Setting?**:When tis box is checked, 0 speed is included in the options for variable speed pumps, in addition to the allowable choices between the minimum and maximum speed. This field is only editable when the associated pump is a Variable Speed Decision Type.
- **Time From Start**: This value, in conjunction with the Duration value, allows you to limit the scheduling period in which the associated pump may run. For instance, if the user wants to schedule one pump group only from 6am to 6pm for an EPS staring at 12am, they would enter a time from start as 6 hours, and duration as 12 hours. The scheduler engine will ensure the pumps are not running at all other times.
- **Duration**: This value, in conjunction with the Time From Start value, allows you to limit the scheduling period in which the associated pump may run. For instance, if the user wants to schedule one pump group only from 6am to 6pm for an EPS staring at 12am, they would enter a time from start as 6 hours, and duration as 12 hours. The scheduler engine will ensure the pumps are not running at all other times.

Objective Elements Tab

This tab is divided into sub-tabs that allow you to choose which objective elements to include in the optimization.

Pumps Tab

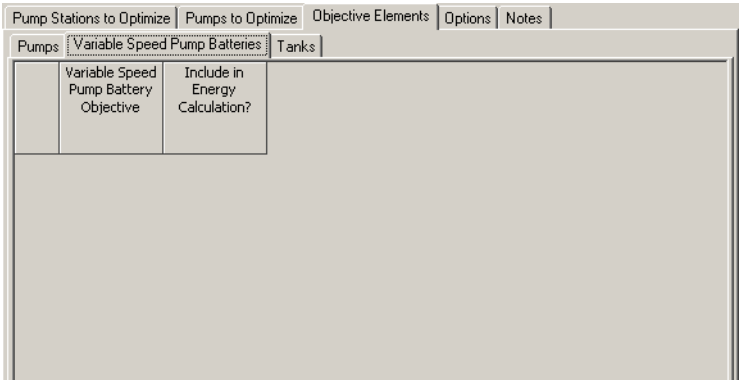
This tab allows you to define which pumps are included in the optimization.



To include a pump, check the associated Include in Energy Calculation? box.

Variable Speed Pump Batteries Tab

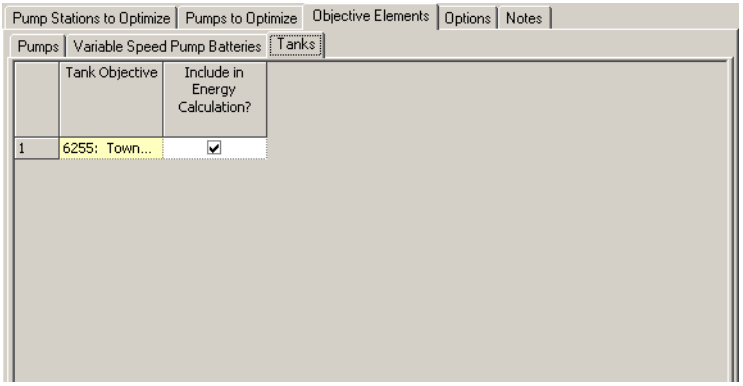
This tab allows you to define which variable speed pump batteries are included in the optimization.



To include a variable speed pump battery, check the associated Include in Energy Calculation? box.

Tanks Tab

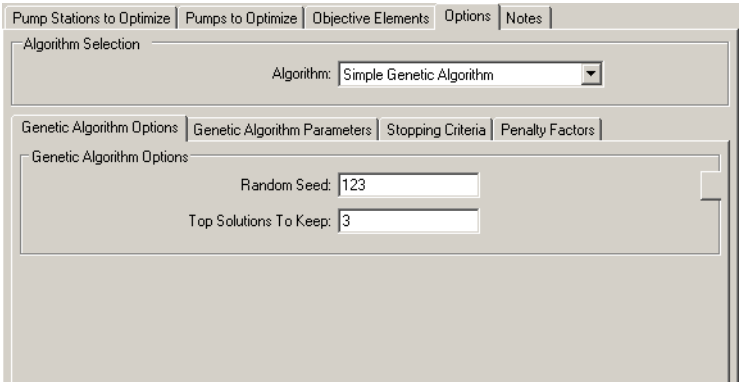
This tab allows you to define which tanks are included in the optimization.



To include a tank, check the associated Include in Energy Calculation? box.

Options Tab

This tab allows you to define the genetic algorithm options and parameters that will be govern the optimization.

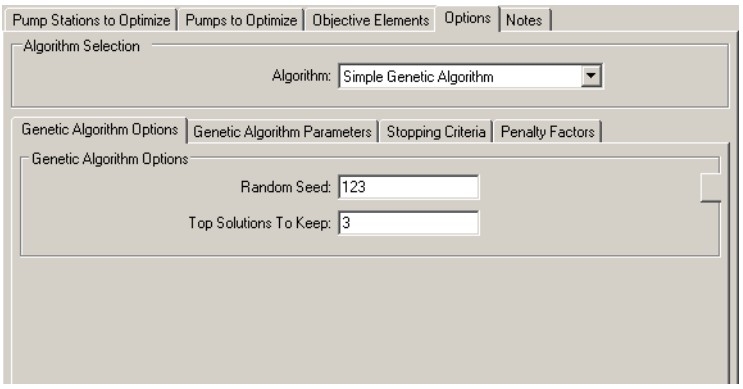


The Options tab contains an Algorithm Selection control as well as a number of subtabs. The following Algorithms are available:

- **Simple Genetic Algorithm**: An implementation of what is traditionally known as a simple genetic algorithm using well defined chromosomes and simple crossover as the primary breeding mechanism.
- **Fast Messy Genetic Algorithm**: An implementation of what is traditionally known as a messy genetic algorithm with messy or partially defined chromosomes and using splice and cut as the primary breeding mechanism.

Genetic Algorithm Options Tab

This tab allows you to define the genetic algorithm options.



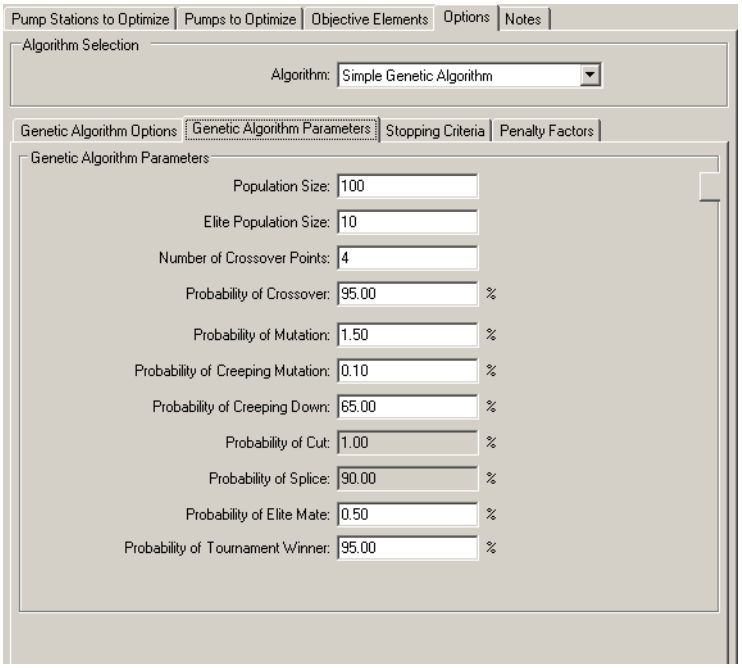
The following options are available:

- **Random Seed**: Lets you set the random number generator to a new point. Changing this value and leaving all other parameters as-is will yield a different solution set.
- **Top Solutions to Keep**: Set the number of solutions that you want to keep. Rather than presenting you with only one solution, Scheduler presents you with a customizable number of solutions, so you can review them manually.

Click the Reset button to rest all of the options on this tab to the factory defaults.

Genetic Algorithm Parameters Tab

This tab allows you to define the genetic algorithm parameters.



The following parameters are available:

- **Population Size**: Sets the number of GA solutions in each generation. Increasing Population Size results in a longer time for each generation and more solutions to be evaluated.

  The allowable range for values is from 50 to 500. We recommend you use a range of 50 to 150.

- **Elite Population Size**: Size of an elite population of chromosomes that is maintained in parallel to the main generic algorithm population.

- **Number of Crossover Points**: Defines the number of locations along each parent chromosome where the chromosome is cut in order to be crossed over with the other parent. This field is only editable when the Algorithm is set to Simple Genetic Algorithm.

- **Probability of Crossover**: The probability that a crossover operation will be performed at the point in the genetic algorithm where crossover operations are performed (during creation of the next generation). This field is only editable when the Algorithm is set to Simple Genetic Algorithm.

- **Probability of Mutation**: Sets the probability that a GA solution is randomly altered. A value closer to 100% causes the solutions to contain more randomization than values closer to 0%.

  The allowable range for values is between 0% and 100%, not inclusive. We recommend you use a value less than 10%.

- **Probability of Creeping Mutation**: The probability that a creeping mutation will occur to a new child chromosome. This field is only editable when the Algorithm is set to Simple Genetic Algorithm.

- **Probability of Creeping Down**: The probability that a gene in a child chromosome will mutate to a smaller value (e.g., lower pump speed) versus a higher value (e.g., higher pump speed). This field is only editable when the Algorithm is set to Simple Genetic Algorithm.

- **Probability of Cut**: Sets the probability that a GA solution will be split into two pieces. Setting this value closer to 100% increases the number of cuts made and reduces the average string (chromosome) length. Increasing Cut Probability causes solutions to vary more widely from one generation to the next, whereas decreasing this results in more marginal changes.

  The allowable range for values is between 0% and 100%, not inclusive. We recommend you use a value less than 10%.

  Setting the Splice probability closer to 100% increases the demand on system RAM. If you are getting out-of-memory errors when using GA Optimization, try reducing the Splice Probability closer to 0% and try increasing the Cut Probability away from 0%.

  This field is only editable when the Algorithm is set to Fast Messy Genetic Algorithm.

- **Probability of Splice**: Sets the probability that two GA solutions will be joined together. A Splice Probability set close to 100% results in long solution strings, which increases the mixing of alleles (genes) and improves the variety of solutions.

  The allowable range for values is between 0% and 100%, not inclusive. We recommend you use a range from 50% to 90%.

  This field is only editable when the Algorithm is set to Fast Messy Genetic Algorithm.

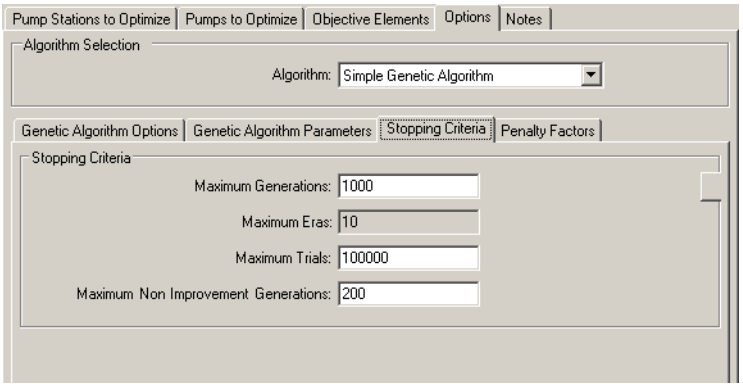
- **Probability of Elite Mate**: The probability that a chromosome from the elite population is selected as a parent for the next generation at the point in the genetic algorithm where parent selection is conducted.

- **Probability of Tournament Winner**: The probability that during parent selection the most fit chromosome is selected in a two chromosome tournament. This field is only editable when the Algorithm is set to Simple Genetic Algorithm.

Click the Reset button to rest all of the parameters on this tab to the factory defaults.

Stopping Criteria Tab

This tab allows you to define the stopping criteria at which the optimization will be considered finished.



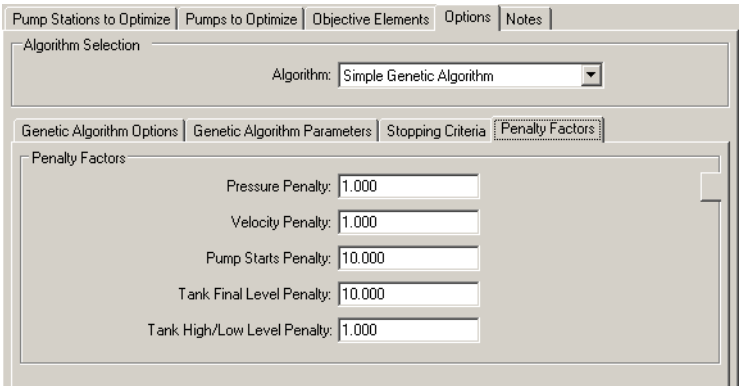
The following stopping criteria are available:

- **Maximum Generations**: The maximum number of generations to run the genetic algorithm optimization. This field is only editable when the Algorithm is set to Simple Genetic Algorithm.
- **Maximum Eras**: The maximum number of eras to run the genetic algorithm optimization. This field is only editable when the Algorithm is set to Fast Messy Genetic Algorithm.
- **Maximum Trials**: Set the maximum number of trials you want the Optimized Run to process before stopping.
- **Maximum Non Improvement Generations**: Set the number of maximum number of non-improvement generations you want the GA to process without calculating an improved fitness. If the Optimized Run makes this number of calculations without finding an improvement in fitness that is better than the defined Fitness Tolerance, the calibration will stop. Non-Improvement Generations works in conjunction with Fitness Tolerance.

Click the Reset button to rest all of the criteria on this tab to the factory defaults.

Penalty Factors Tab

This tab allows you to define the penalty factors that help narrow down the results.



Define penalty factors to help find the solution. A high penalty factor causes the GA to focus on feasible solutions, which do not violate boundaries of pressure, velocity, pump starts, or tank levels. A low penalty factor (50,000 or so) permits the GA to consider solutions that are on the boundary between feasible and infeasible solutions, possibly violating your defined boundaries by a small amount. Because the optimal solution often resides in the boundary between feasible and infeasible solutions, a high penalty factor causes the GA to find a feasible solution quickly but is less likely to find the optimal solution.

From a practical standpoint, you might consider starting with a high penalty factor and working towards a lower penalty factor as you pursue an optimal solution.

By defining penalty factors for Pressure, Velocity, Pump Starts, and Tank Final Level, you can weight these various considerations according to which is most important to you.

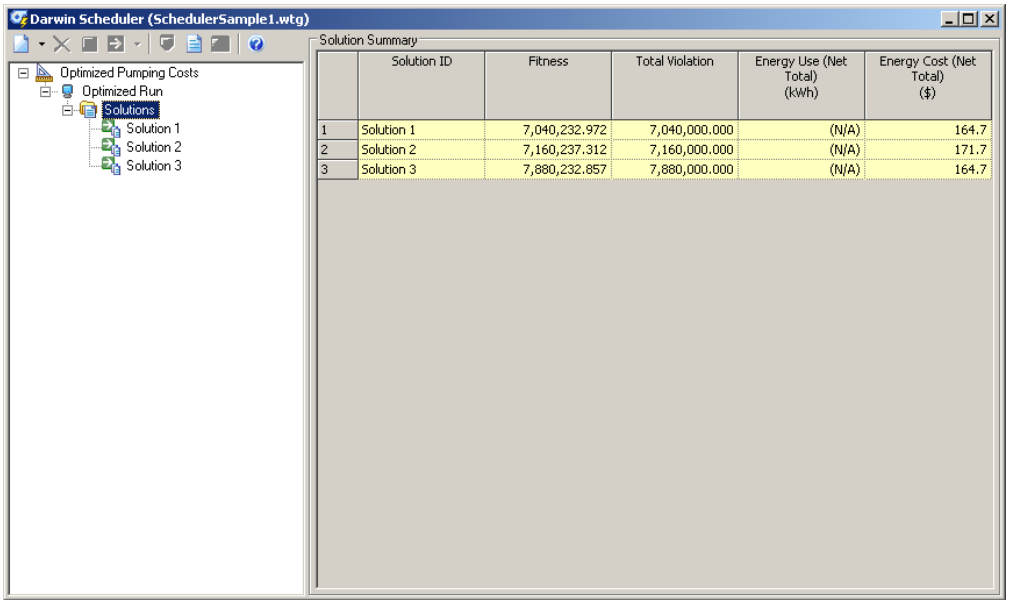
Click the Reset button to rest all of the factors on this tab to the factory defaults.

Notes Tab

This tab allows you to enter descriptive notes that will be associated with the Optimized Run.

Solutions

After an Optimized Run has been computed, a number of solutions will appear in the list pane.



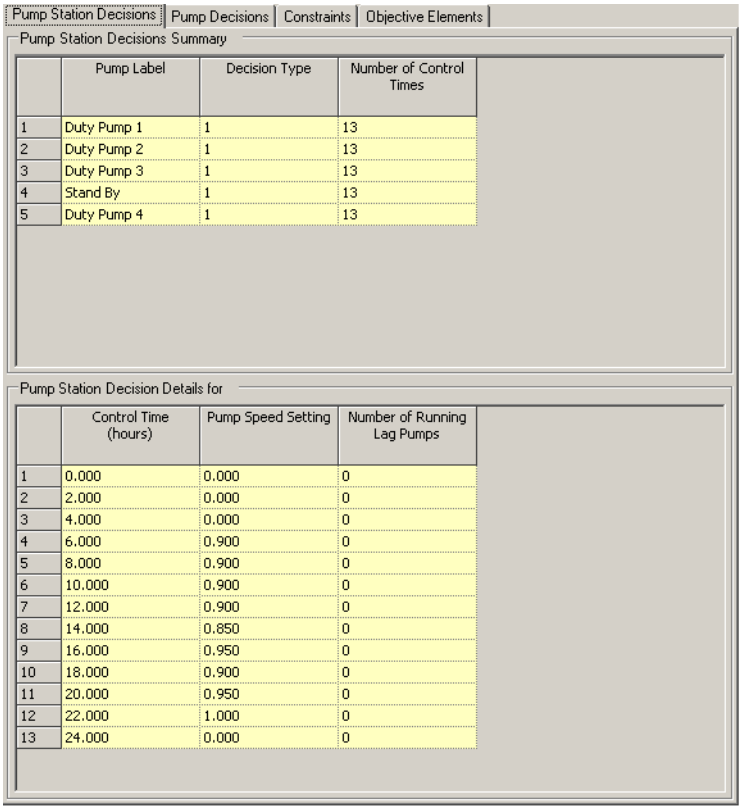
Highlighting the top-level Solutions folder will display a Solution Summary for each of the solutions generated by Scheduler.

When you highlight one of the Solutions, the tabbed area will display three tabs containing all of the solution data.

Pump Station Decisions Tab

This tab displays the pump station decisions summary and details.

The table on the top of the tabbed pane displays a summary of the results for each of the pump decisions. Click on a pump in the summary table to see the details for that pump in the Pump Decision Details table at the bottom.



Pump Station Decisions | Pump Decisions | Constraints | Objective Elements

Pump Station Decisions Summary

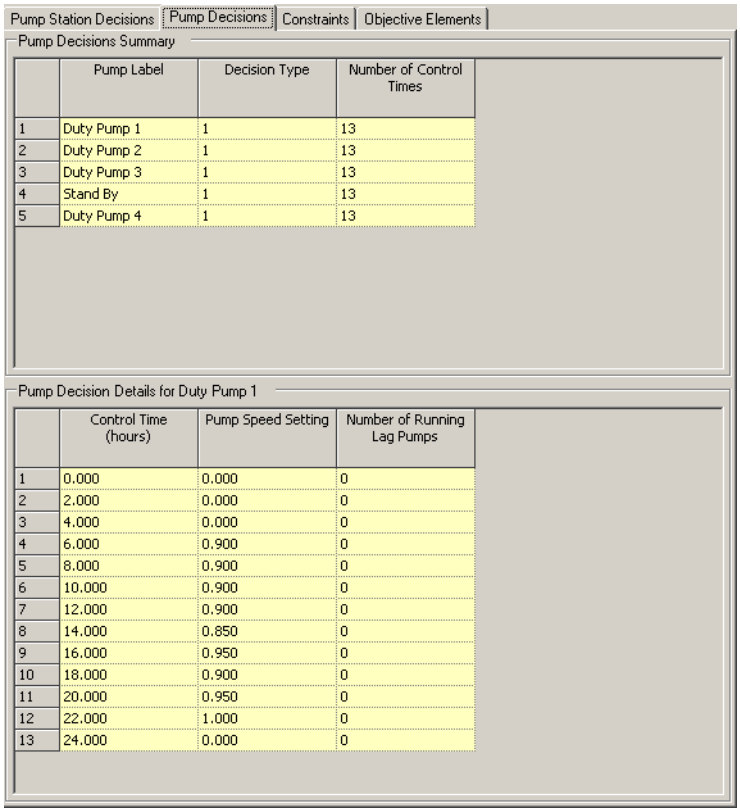
	Pump Label	Decision Type	Number of Control Times
1	Duty Pump 1	1	13
2	Duty Pump 2	1	13
3	Duty Pump 3	1	13
4	Stand By	1	13
5	Duty Pump 4	1	13

Pump Station Decision Details for

	Control Time (hours)	Pump Speed Setting	Number of Running Lag Pumps
1	0.000	0.000	0
2	2.000	0.000	0
3	4.000	0.000	0
4	6.000	0.900	0
5	8.000	0.900	0
6	10.000	0.900	0
7	12.000	0.900	0
8	14.000	0.850	0
9	16.000	0.950	0
10	18.000	0.900	0
11	20.000	0.950	0
12	22.000	1.000	0
13	24.000	0.000	0

Pump Decisions Tab

This tab displays the pump decisions summary and details.



Pump Station Decisions | Pump Decisions | Constraints | Objective Elements

Pump Decisions Summary

	Pump Label	Decision Type	Number of Control Times
1	Duty Pump 1	1	13
2	Duty Pump 2	1	13
3	Duty Pump 3	1	13
4	Stand By	1	13
5	Duty Pump 4	1	13

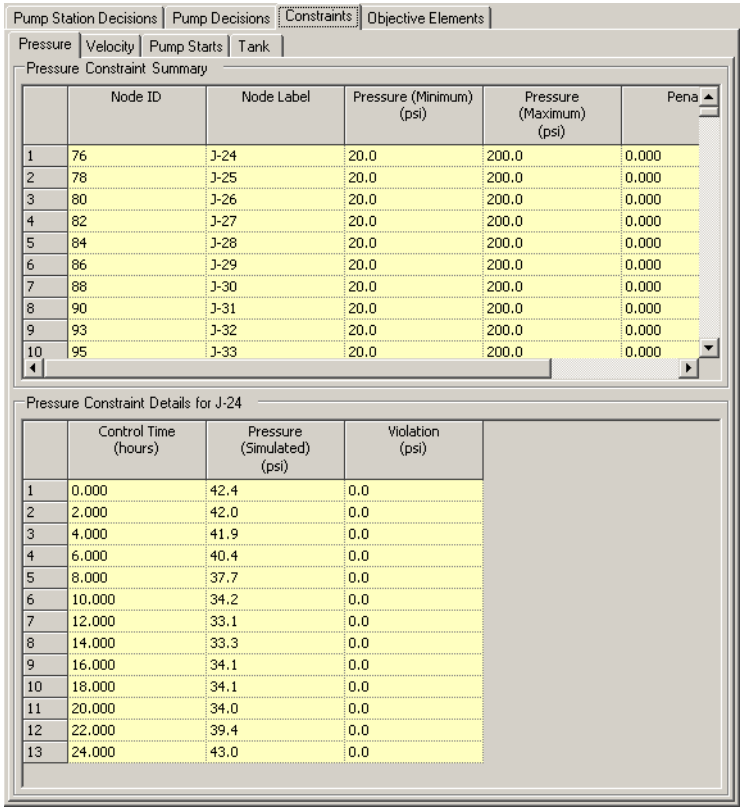
Pump Decision Details for Duty Pump 1

	Control Time (hours)	Pump Speed Setting	Number of Running Lag Pumps
1	0.000	0.000	0
2	2.000	0.000	0
3	4.000	0.000	0
4	6.000	0.900	0
5	8.000	0.900	0
6	10.000	0.900	0
7	12.000	0.900	0
8	14.000	0.850	0
9	16.000	0.950	0
10	18.000	0.900	0
11	20.000	0.950	0
12	22.000	1.000	0
13	24.000	0.000	0

The table on the top of the tabbed pane displays a summary of the results for each of the pump decisions. Click on a pump in the summary table to see the details for that pump in the Pump Decision Details table at the bottom.

Constraints Tab

This tab displays the constraints summary and details.



	Node ID	Node Label	Pressure (Minimum) (psi)	Pressure (Maximum) (psi)	Pena
1	76	J-24	20.0	200.0	0.000
2	78	J-25	20.0	200.0	0.000
3	80	J-26	20.0	200.0	0.000
4	82	J-27	20.0	200.0	0.000
5	84	J-28	20.0	200.0	0.000
6	86	J-29	20.0	200.0	0.000
7	88	J-30	20.0	200.0	0.000
8	90	J-31	20.0	200.0	0.000
9	93	J-32	20.0	200.0	0.000
10	95	J-33	20.0	200.0	0.000

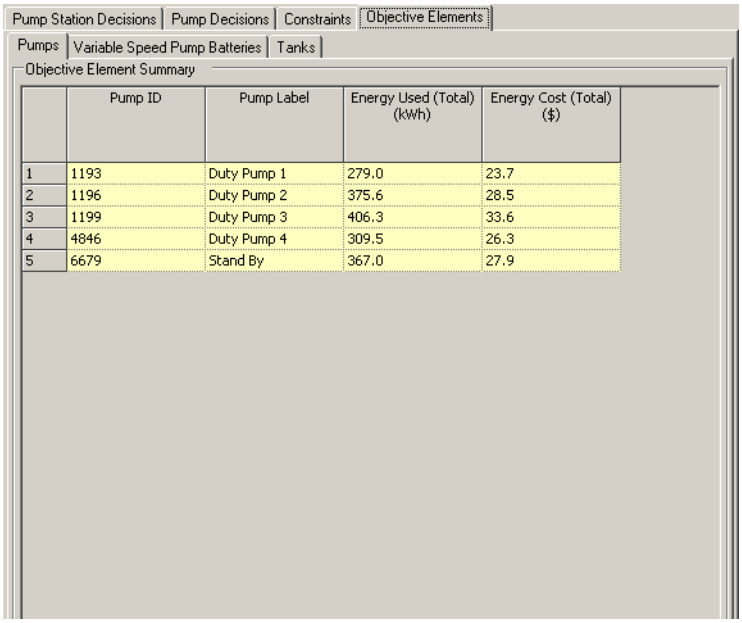
	Control Time (hours)	Pressure (Simulated) (psi)	Violation (psi)
1	0.000	42.4	0.0
2	2.000	42.0	0.0
3	4.000	41.9	0.0
4	6.000	40.4	0.0
5	8.000	37.7	0.0
6	10.000	34.2	0.0
7	12.000	33.1	0.0
8	14.000	33.3	0.0
9	16.000	34.1	0.0
10	18.000	34.1	0.0
11	20.000	34.0	0.0
12	22.000	39.4	0.0
13	24.000	43.0	0.0

The Constraints tab is further divided into subtabs for each of the constraint types: Pressure, Velocity, Pump Starts, and Tanks. For each constraint type the table lists the associated constraint values you defined, the simulated value, and the penalty assigned for violating the constraints (if any) for each element.

For the Pressure and Venlocity tabs, click on an element in the summary table to see the details for that element in the details table at the bottom.

Objective Elements Tab

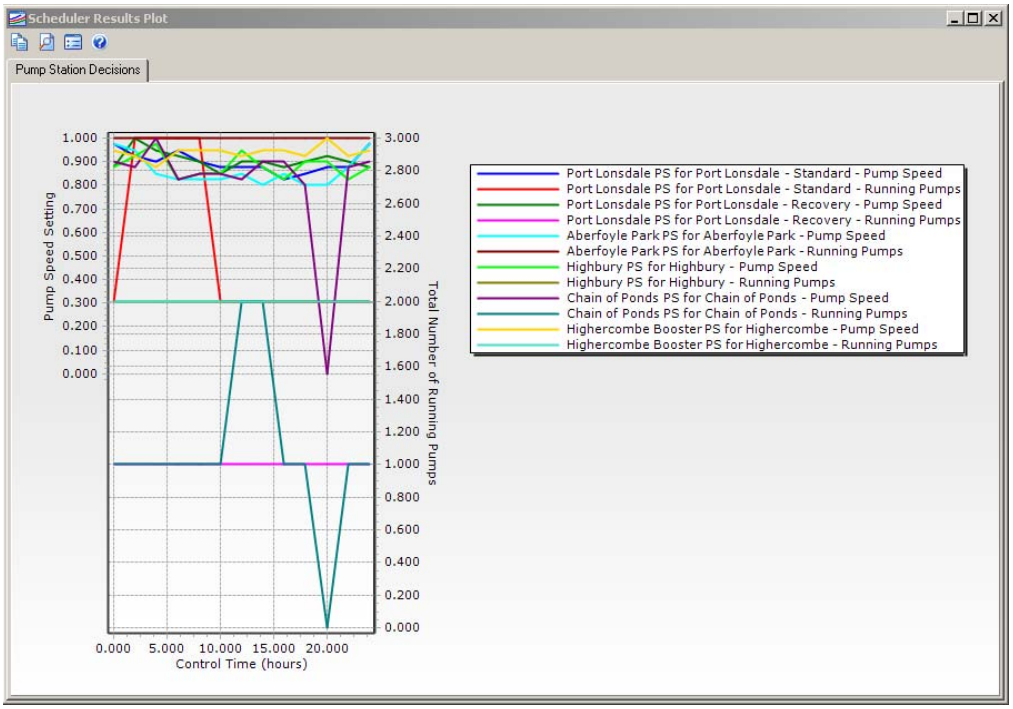
This tab displays the energy used and cost for the objective elements.



	Pump ID	Pump Label	Energy Used (Total) (kWh)	Energy Cost (Total) ($)
1	1193	Duty Pump 1	279.0	23.7
2	1196	Duty Pump 2	375.6	28.5
3	1199	Duty Pump 3	406.3	33.6
4	4846	Duty Pump 4	309.5	26.3
5	6679	Stand By	367.0	27.9

Scheduler Results Plot

Ths dialog displays a graphical plot of the pump decision results.

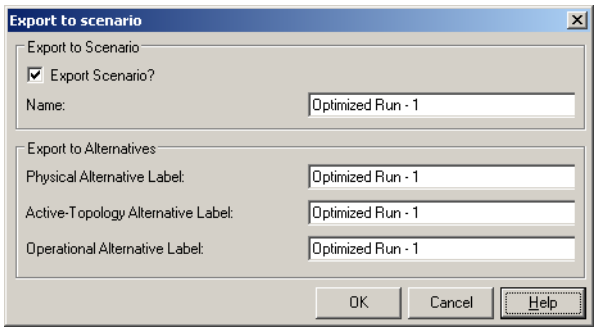


The toolbar along the top of the dialog consists of the following buttons:

- **Copy**: Copies the plot to the Windows clipboard.
- **Print Preview**: Opens a print preview window, allowing you to see how the plot will look when it is printed.
- **Options**: Opens the TeeChart Options dialog, allowing you to customize the plot settings.
- **Help**: Opens the online help.

Export to Scenario Dialog Box

Use the Export to Scenario dialog box to apply the results of your Optimized Run to your water model.



Check the Export Scenario? box to export the solution to a new scenario. You can change the default name of the new scenario by typing a different one in the Name field.

You can also change the names of the Physical, Active Topology, and Operational Alternatives that will be created by entering the new name in the approriate field.

Darwin Scheduler FAQ

1) What is the recommended work flow for using Darwin Scheduler?

The following steps provide a basic guideline for the Darwin Scheduler work flow.

a. Build and create an EPS (Extended Period Simulation) model of the hydraulic network of interest.

b. Calibrate the model.

c. Start Darwin Scheduler and create a new Scheduler Study.

d. Identify the pumps or pump stations (with a preference for pump stations) that will be optimized by Scheduler.

e. Identify the hydraulic performance criteria that must be maintained (hydraulic constraints).

f. Identify the objective elements that should be included in the calculation of the objective function (energy use or energy cost). It is possible for a pump or pump station to be included in the calculation of the objective function but not be optimized. For example, a pump that is always on need not be optimized but the costs can be included in the objective function.

g. Specify the objective type (either minimize energy use or minimize energy cost).

h. Create a new Optimized Run.

i. Select whether pumps will be optimized as fixed speed or variable speed, their allowable speed settings (if variable speed), whether pumps are allowed to be turned off (if variable speed) and also whether the pumps are optimized for the entire EPS or a portion of it. Note that if optimizing only a portion of the EPS (for any one pump decision) Scheduler turns off pumps outside of the portion of the schedule being optimized. For example, for a 24 hour EPS run a pump decision that is set for a time from start of 12 hours and duration of 12 hours will be off from time 0 to time <12, and optimized for time 12 to time <24. The pump will be off at time 24 to ensure a repeatable schedule).

j. Select which objective elements to include in the optimization objective function (by default all included in the study are selected).

k. Choose which genetic algorithm you wish to use and adjust any required parameters (see a later FAQ for information on these settings).

l. Run the optimized run.

m. Optionally stop the optimized run mid flight and review results and resume the run if results are favorable. To do this, select the Stop button in the progress dialog. After reviewing the run, the run can be restarted by picking the compute button and selecting Resume or started from the beginning by picking Compute. Closing Scheduler or performing any other function that runs a pressure computation (such as calculating another scenario) will terminate the paused scheduler run.

n. When the optimized run is complete, review summary results in Darwin Scheduler and choose solutions to export. If any results look promising, choose the solutions to export, else repeat steps k through m with different genetic algorithm parameters. Two prime examples would be population size (try larger perhaps) and random seed (try a different seed).

o. Export the chosen solutions to a new scenario by picking the Export to Scenario button on the top of the left pane.

p. Run the exported scenarios.

q. Run an energy costs analysis.

r. View and analyze the optimized schedule results.

s. Make any tweaks or adjustments to the optimized solution as appropriate, noting that due to the nature of the optimization algorithm sometimes Scheduler will turn off one pump only to turn an equivalent pump on; what the optimization is really saying in this case is that 1 pump of type x should be running.

As alluded to in step n above, it should be noted that the steps from the point of setting up the Scheduler study to exporting solutions and reviewing results can be performed in an iterative loop with adjustments made to the Darwin Scheduler input based on the results of the first set of optimization runs, aimed at improving or re-directing the next set of optimized run results. This is in fact the recommended work flow for using Darwin Scheduler.

2) What is the control interval used by Darwin Scheduler for my EPS optimization?

Darwin Scheduler uses a control interval defined by the hydraulic time-step of the EPS being optimized. This is done since to apply a change in pump setting/status a new steady state simulation is required and so it makes the most sense to align this with the hydraulic time step. To this end, you can reduce the number of pump decisions the optimization needs to make by increasing the hydraulic time step say from 1 hour to 2 hours or 2 hours to 3 hours. Any intermediate time steps that need to be simulated (such as for tanks filling or controls triggering) will still be simulated as per normal EPS protocol.

Note: If there is no reason to have a fine control interval it is strongly recommended to use a course control interval (for a 24 hour EPS consider starting as course as 3 hours) to keep the size of the solution space to a minimum. By keeping the solution space small Scheduler will produce better results. Once a course solution is yielded it is possible to run optimizations with a shorter time step, but it is recommended to do so after having reduced the number of allowable pump speed choices to be in keeping with the previous course solution. Using engineering judgment to keep the optimization solution space as small as possible will improve the Darwin Scheduler experience.

To be considered along with the above recommendation, also note that using a time increment that is too large can result in tank levels running from the current level to full or empty in a single time step. The time step should be a fraction of the time (less than 25%) it takes to fill or drain the tank. It is not

recommended to include small tanks, like hydropneumatic tanks in the same run as large tanks because they operate at much different time scales. See Best Practices and Tips **for more information.**

3) Can Darwin Scheduler be used to optimize very large models and very large numbers of pumps in a single run? If so, what is the best way to use Scheduler for such problems?

We've built no hard limits into Scheduler to prevent it from being used with very large hydraulic models, with very fine control intervals or with lots of pump decisions, however, the performance of Darwin Scheduler in terms of both run time and also optimization results is dependent on the user applying reasonable engineering judgment to minimize the complexity of the problem to be solved and also ensuring the model upon which the optimization is based runs as fast as possible.

Consider an EPS mode that takes 10 seconds to solve and factor in that for a reasonable sized solution space it may take 100,000 trial solutions to achieve a near global optimum. The computer time needed to evaluate 100,000 trial solutions is 1,000,000 seconds or 278 hours, or 11.5 days. Most people will probably not want to run optimization runs that last 11.5 days so there needs to be an appreciation that the time needed for the optimization is a function of the time to solve the model. There are a number of ways that the run time for a model can be reduced, but the main one is skeletonization, which if done correctly (such as using hydraulic equivalent skeletonization) can reduce run time significantly whilst having little or no effect on system hydraulics nor upon the results of an energy optimization.

The other side of the equation in terms of optimization performance is the number of trials required to reach a near global optimum. We've used 100,000 trials above as a reasonable number of trials, but depending on the size of the solution space (the complexity of the optimization problem) it may be more than this figure and it could also be less. The solution space is defined as the total number of combinations of possible solutions. So for the case of pump optimization it is the total possible combination of pump speed settings over the course of the optimization. See the Darwin Scheduler Best Practices and Tips topic for more information about keeping the solution space to a minimum.

4) When a Variable Speed pump is included in the pumps to optimize, Darwin Scheduler allows it to be optimized as a fixed speed pump and vice versa for regular pumps. Why?

This is a feature of Darwin Scheduler and is to allow one to assess the difference in running pumps (any pumps) as fixed speed versus variable speed without first having to modify the pump type in the model. If for example it is decided that a pump that is currently fixed speed can achieve significant operational improvements by being operated as variable speed then it may be decided to replace the existing pump with a variable speed pump.

5) What is the difference between a pump and a pump station in Darwin scheduler?

In Scheduler, a pump should be viewed in one and only one way in a given run.

a. As a single pump; each pump is treated individually and is not aware that it is part of a station and which pumps are in the station with it.

b. As part of a station; Scheduler does not consider exactly which identical pumps are running but merely keeps track of the number of identical pumps running.

Treating pumps as part of a station is win-win since it reduces the problem dimensionality and avoids un-necessary pump switches that may occur when treating pumps as individual optimization decisions. This will usually result in faster runs with better optimal solutions. However, if all the pumps in a station are different, then the results between treating the pumps individually or as part of the station will not be any different.

6) When a Variable Speed Pump Battery is included in the pumps to optimize, Darwin Scheduler sometimes has a number of running lag pumps result > 0 when the pump speed setting is 0.0. Why?

For Variable Speed Pump Battery elements, Scheduler optimizes the pump speed and number of running lag pumps as independent optimization decisions. To that end if the pump speed is 0.0 the solution considers all lag pumps to be off too, so the lead-lag relationship is maintained.

7) When should Scheduler be used to set the speed of variable speed pumps versus just setting a target head for the variable speed pump algorithm?

If the desired target head for the variable speed pump is known it can simply be set and not optimized by Scheduler. If there is a large number of pumps to optimize the problem size can be cut down by simply setting the target head of some variable speed pumps and batteries and not including those as pumps to optimize. This minimizes the solution spacer of the optimization ensuring better results for the pumps that are optimized. If, however, a reasonable target head for the system is not known, then Scheduler can assist with determining what a reasonable head setting might be as well as the pump speed.

However, there is no guarantee that the "optimal" speeds determined by Darwin Scheduler will be better than simply trying to maintain a know head or flow in a standard variable speed simulation run. This is due to the fact that Scheduler looks at discrete speeds such as 0.8, 0.85, 0.9 while the simulation run may be able to find a better solution by running at 0.86759 which Scheduler would not be able to find. The primary advantage of using Scheduler is that it can consider multiple constraints while a standard simulation only has a single set point.

8) When a Variable Speed Pump with Target Head or Flow is included in the pumps to optimize in Darwin Scheduler the pump no longer maintains the target head or flow. Why?

When a pump is selected to be optimized by Darwin Scheduler full control of that pump is given to Darwin Scheduler. The pump will ignore any VSP control properties and will not necessarily maintain target flows or heads. This is handled by setting constraints on pressures or flows. Be careful not to set the minimum and maximum constraint too close together, given the time step size and increment, or else it may not be possible to obtain a feasible solution.

9) When a pump is included in the pumps to optimize in Darwin Scheduler it no longer responds to controls. Why?

When a pump is selected to be optimized by Darwin Scheduler full control of that pump is given to Darwin Scheduler. The pump will ignore any control actions applied to it.

10) When a pump is included in the pumps to optimize in Darwin Scheduler it no longer responds to patterns. Why?

When a pump is selected to be optimized by Darwin Scheduler full control of that pump is given to Darwin Scheduler. The pump will ignore any patterns applied to it.

11) When exporting an optimized schedule that includes Variable Speed Pump Batteries, Darwin Scheduler breaks the Variable Speed Pump Battery into single pump elements. Why?

Darwin Scheduler is able to optimize the operation of Variable Speed Pump Batteries by considering them as a lead pump with the specified number of lag pumps in parallel. In order for the solution that is exported by Darwin Scheduler to match up with Darwin Scheduler's simulated hydraulics and energy cost/use it must export a scenario that is functionally equivalent to the optimized schedule. Since Variable Speed Pump Battery elements are not designed to work with pump patterns, Darwin Scheduler exports these as single pumps with a pattern applied to replicate the optimized pump schedule. Correspondingly each lag pump will have its own pattern.

12) When exporting an optimized schedule Darwin shows a higher/lower energy use value for the solution than does the energy costs tool. What is wrong?

In this case one or more tanks is included in the objective elements list in Darwin Scheduler and Scheduler is accounting for the energy deficit or credit from the tank(s) filling or draining; ensure that the energy costs tool is also accounting for the energy credit/deficit due to tanks to verify Scheduler's calculated energy costs and/or energy usage. Filling a tank is essentially storing energy for later use while draining that tank uses stored energy.

13) Why does Darwin Scheduler require "objective elements" to be specified separately to the pumps to optimize?

This is because Darwin Scheduler allows the optimization to consider any pumps or tanks in the assessment of the objective value (energy use or energy cost) as opposed to just the elements included in the optimization process as decisions or constraints. This allows selective optimization of specific pumps whilst leaving others operating according to their control rules (or VSP settings), but factoring in the cost of all (or any number) of the pumps in the model, regardless of whether they are being optimized or not.

14) Darwin Scheduler requires constraints to be entered manually. Why is there no global or blanket constraint that I can apply such as minimum pressure, for example?

Using blanket constraints is the easiest way to de-rail the optimization by inadvertently including constraints that are impossible to meet such as the suction side nodes of pumps in pressure constraints. Since constraints are entered manually (using several convenient methods) a user is encouraged to first think about the constraints that are being added. For more information please see the "Darwin Scheduler Best Practices and Tips" topic.

15) There is always a high violation number for my optimization run. Why can't Scheduler find a feasible solution (a solution that meets the constraints)?

There could be several reasons for this including:

a. The Scheduler constraints include an impossible to meet constraint such as a minimum required pressure that is on the suction side of a pump, or a required pressure near a tank with too low a level.

b. The Scheduler constraints include two or more inconsistent constraints. For example one junction may require a pressure of < 50 psi, whilst an adjacent junction might require > 50 psi. When there is high penalty associated with more than one constraint, check to see if the constraints are not mutually exclusive.

c. The schedule for optimization is not appropriate for the EPS being optimized. One example might be a 48 hour EPS run that is set up to optimize pump operation for the first 24 hours only, but requiring a high final tank level. Note that Scheduler optimized pumps are turned off outside of their optimized schedule.

d. The run has not been allowed to run sufficiently long enough for all constraints to be met by the evolved solutions.

e. If a tank is small relative to the time it takes to fill or drain it, it may consistently overshoot the maximum level or drop below the minimum. The time to fill or drain a tank should be much larger than the time step size.

16) When running a minimize energy use optimization why can't Scheduler find a solution that is better than the control based pump schedule in the scenario being optimized?

Constraints have potentially been defined that are based on the control based pump schedule and are thus affording the optimization process no flexibility in being able to change the pumping schedule. Bear in mind that an energy use optimization is more constrained than energy cost in the sense that the optimization is not able to leverage variations in energy tariffs to find a better solution. For example, if in the base pump schedule a single pump is running all day to meet hydraulic criteria, surely there is little scope for saving energy costs in that context unless there is either flexibility in hydraulic criteria or other pumps that can be utilized.

17) Darwin Scheduler is running slowly. Why?

There are a number of reasons for this, but the main reason is that in contrast to the other two Darwin tools (Calibrator and Designer) Scheduler has a higher computational overhead by virtue of the fact it simulates a full EPS run compared to just single steady state snapshots in Designer and Calibrator. For example a 24 hour EPS is a kin to running 24 Design Events in Designer or 24 Field Data Sets in Calibrator. Running a full EPS is necessary to properly evaluate a pump schedule since pump energy is used and volume changes occur over time, whereas Designer and Calibrator are more concerned with peak conditions. Then consider that for an optimization to complete, typically tens of thousands of trials are required. If a single EPS takes a full second to run, a Darwin Scheduler run will require several hours to complete. This makes running Darwin Scheduler over night on large models an attractive proposition.

For additional information on Darwin Scheduler performance and how to get the best out of Darwin Scheduler please see Best Practices and Tips.

18) How is fitness calculated?

Fitness is calculated as follows:

For an energy use optimization, fitness is calculated as the total energy use of the pump elements specified in the objective elements section for the duration of the full EPS plus the energy credit or deficit from the tanks specified in the objective elements section for the duration of the full EPS plus any penalties encountered. Tank energy credit is based on the average energy per volume pumped for the duration of the EPS. Fitness is in the units of energy (kWh).

For an energy cost optimization, fitness is calculated as the total energy cost of the pump elements specified in the objective elements section for the duration of the full EPS plus the energy cost credit or deficit from the tanks specified in the objective elements section for the duration of the full EPS, plus any penalties encountered. Tank energy cost credit is based on the average energy cost per volume pumped for the duration of the EPS. Fitness is in the units of cost ($).

For both optimization types note that a marginal value is added to the fitness of a solution based on the total number of pump starts that occur. This is applied independently and in addition to any user-defined pump start constraint and ensures that optimized solutions adopt less pump starts unless there is a significant benefit to having more pump starts.

All energy use calculations factor in pump efficiency and pump motor efficiency.

All energy cost calculations factor in specified energy tariffs.

Darwin Scheduler *does not* factor in peak demand charge.

19) What does a violation value of greater than 0.0 mean?

This simply means that the solution (or current best solution) does not meet all of the hydraulic constraints; the value itself is the penalty applied due to constraint violations. Leaving a run to execute for longer will most likely reduce violation to 0.0 meaning a feasible solution has been found. The term "feasible" is used to describe a solution that meets all the specified hydraulic constraints, however, through proper review and engineering judgment a non-feasible solution (one with violation greater than 0.0) may also be deemed to be feasible in practical terms.

20) How is violation (penalty) calculated?

The calculation of violation varies depending on the constraint type as follows:

Pressure Constraints:

$$\text{Violation} = \sum_{i=1}^{n} Pi \times PF_p \times 100000$$

Where P*i* is the average absolute pressure violation at constraint Node i, and *PFp* is the pressure penalty factor.

Velocity Constraints:

$$\text{Violation} = \sum_{i=1}^{n} Vi \times PF_v \times 100000$$

Where V*i* is the average absolute velocity violation at constraint Pipe i, and ***PFv*** is the velocity penalty factor.

Pump Start Constraints:

$$\text{Violation} = \sum_{i=1}^{n} Pi \times PF_{PS} \times 100000$$

Where P*i* is the average absolute pump start violation at constraint Pump i, and *PFps* is the pump start penalty factor. Note that violation for pump starts is calculated in a cumulative sense so that the rolling number of pump starts is used to calculate the violation at each time. This makes solutions that exceed their maximum pump starts early in the optimized schedule less desirable compared to ones that may only fail their constraint near the end of the schedule.

Tank Final Level Constraints:

$$\text{Violation} = LV \times PF_T \times 100000$$

Where *LV* is the final level violation, and *PFt* is the tank final level penalty factor.

21) What values are acceptable to use for Genetic Algorithm Parameters, Stopping Criteria and Penalty Factors?

Most users will not have to concern themselves with the adjustment of these parameters and reasonable defaults have been set as defaults for normal use. Advanced users or users that are particularly interested in optimization may wish to play with these parameters to assess their effect on the optimization process. Darwin Scheduler will not accept values for any parameter that are considered to be detrimental to the operation of the engine as a whole, however, such values still might not be recommended to use. To that end we provide some recommended ranges of values for each parameter.

Genetic Algorithm Parameters

Population Size: 50-200. Sometimes as high as 1000+

Elite Population Size: 10-20

Number of Cross Over Points: 2-10 or 2-10% of the problem length

Probability of Cross Over: 90-100%

Probability of Mutation: 1-2%

Probability of Creeping Mutation: 0-1%

Probability of Creeping Down: For this problem type higher than 50%

Probability of Cut: 1-2%

Probability of Splice: 90-95%

Probability of Elite Mate: 0-1%

Probability of Tournament Winner: 95-100%

Stopping Criteria

Maximum Generations: Typically 500 - 2000

Maximum Eras: Typically 6-12

Maximum Trials: Typically 50000 - 200000 or higher (the larger the population size used, the larger this should be)

Maximum Non Improvement Generations: 100-300

Penalty Factors

These factors are used to weight different constraint types against each other, but primarily to guide the optimization process towards areas of the solution space that contain solutions that do not violate constraints. These factors should rarely require manipulation.

Pressure Penalty: 0.5 - 2.0

Velocity Penalty: 0.5 - 2.0

Pump Starts Penalty: 5 - 20

Tank Final Level Penalty: 5 - 20

22) What is the difference between the Simple Genetic Algorithm and the Fast Messy Genetic Algorithm?

Third party research suggests that Fast Messy Genetic Algorithms are better at finding near optimal solutions to complex problems than their Simple Genetic Algorithm predecessors and as such Darwin Calibrator and Darwin Designer both employ a type of Fast Messy Genetic Algorithm. Darwin Scheduler makes use of a newly developed Genetic Algorithm component and it was little additional work for us to expose both Genetic Algorithm types to users instead of just the one so we did. This will enable those who are interested in optimization to experiment using both types of algorithm.

23) When using the Fast Messy Genetic Algorithm sometimes the number of trials on the Optimization Progress dialog pauses for an extended period of time so no trials are being evaluated. Why is this?

As part of the messy genetic algorithm process prior to the creation of a new generation of trial solutions, parents must be selected for the new generation. Owing to the nature of the messy GA solution representation suitable parent chromosomes must be compared against other chromosomes with a certain similarity measure. The process by which chromosomes are found that meet the similarity measure is called genic thresholding and sometimes this can take a little while to execute, meaning CPU time is spent for a short period on the genic thresholding process as opposed to evaluating

trial solutions. The simple genetic algorithm does not perform genic thresholding and therefore does not have this delay. Note, however, that the run-time required for genetic algorithm processes pales in significance compared to the time required to evaluate trial solutions, even for the Fast Messy Genetic Algorithm.

24) Why doesn't Darwin Scheduler stop exactly when the stop button is clicked?

The reason for this is that in order for various things to work correctly (such as the resume feature) Scheduler will complete the current generation that it is evaluating before returning control to the user. This is indicated on the Optimization Progress dialog by the Stop button becoming disabled and the Optimization Progress dialog status showing "Stopping…". Depending on the population size of the run and the time taken for a single trial this may represent several minutes, so please be patient during this process.

25) Where does Darwin Scheduler store its results?

Darwin Scheduler stores its results in a proprietary binary file format with a *.dsb (Darwin Scheduler Binary) extension. When the model is saved any Darwin Scheduler results files will be saved too.

26) Why doesn't Darwin Scheduler have more in depth results visualization features?

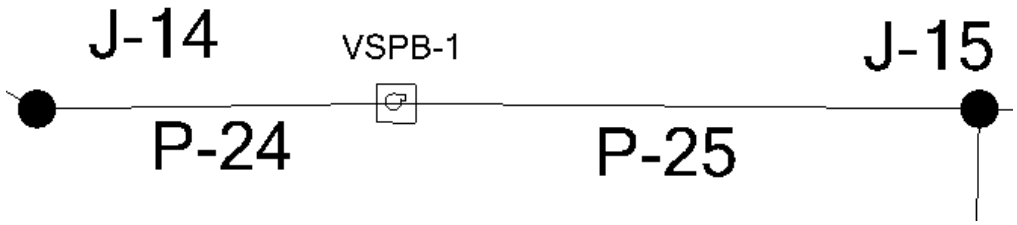
Darwin Scheduler's user interface provides summaries of the optimized pump schedules and of hydraulic performance, however, the best way to view Darwin Scheduler results is to export the optimized scenario to the model and analyze results by leveraging the full suite of results visualization tools available in the main application. Of particular value will be the Scenario Energy Cost Managerfor a detailed break down of energy use and cost.

27) Why doesn't Darwin Scheduler allow additional demands or boundary conditions to be specified like Darwin Calibrator and Darwin Designer?

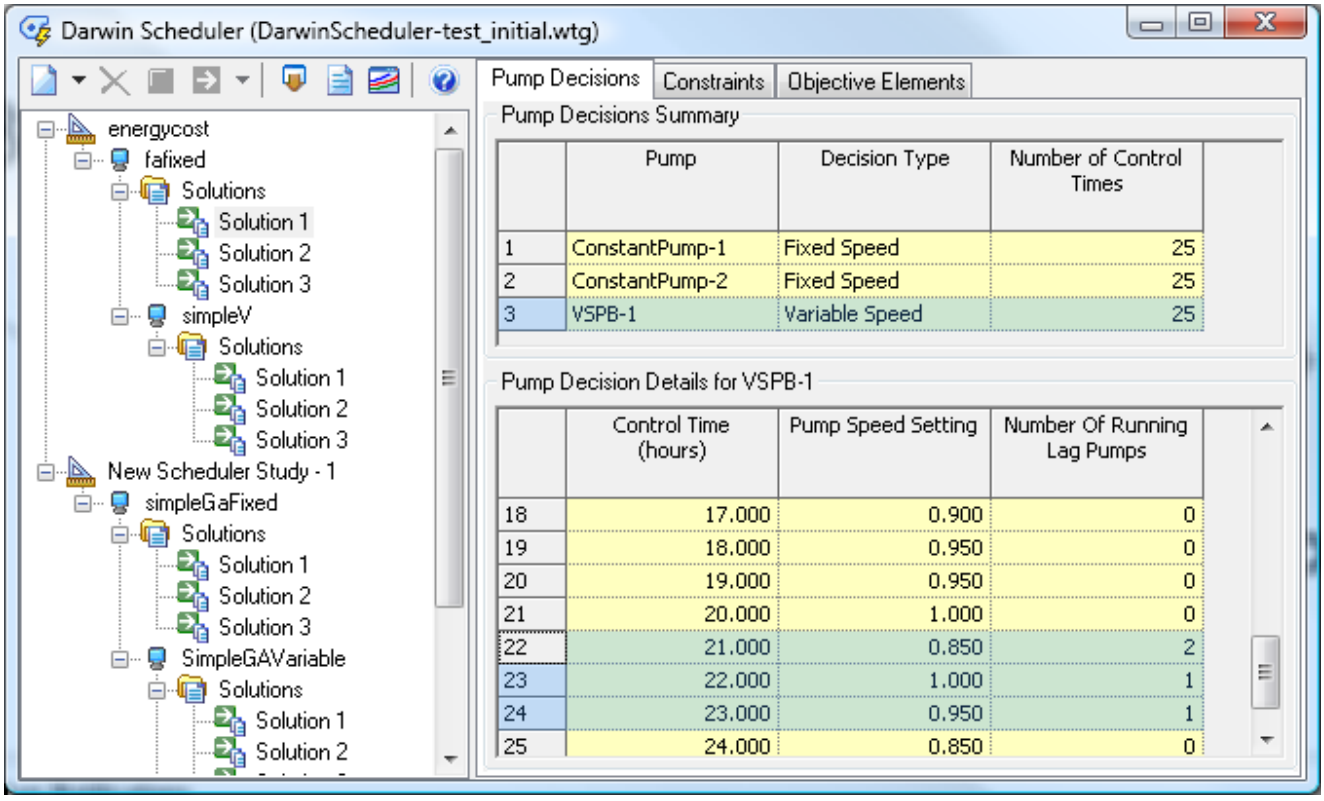
The answer to this question lies in the fact that Darwin Scheduler simulates an entire EPS run as opposed to a set of steady state snapshots like Darwin Calibrator or Darwin Designer. In those latter two tools it is necessary for a user to be able to specify boundary conditions (such as valve settings and tank levels) that define the hydraulic conditions that apply to the associated hydraulic snapshot. For example, if the snapshot is for 7am, tank levels etc will be specified for that time. This, however, is unnecessary for Darwin Scheduler since it simulates a full EPS run and therefore is able to calculate the boundary conditions at each time in the EPS run. To that end Darwin Scheduler's model input is completely acquired from the scenario being optimized. If it is necessary to consider additional demands or make other modifications to the hydraulic model before running an optimization, do so using the main application's standard scenario and alternative management tools, then select the modified scenario as the scenario to optimize in Darwin Scheduler.

28) When exporting an optimized schedule that includes Variable Speed Pump Batteries, Darwin Scheduler breaks the Variable Speed Pump Battery into single pump elements. Why?

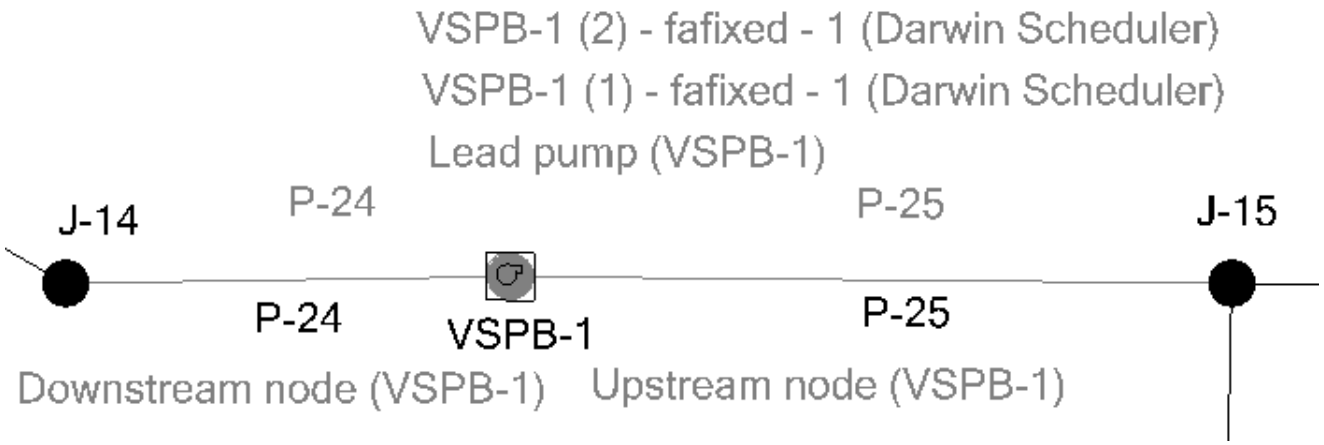
1) The initial situation: a VSPB connected to two pipes.



2)The Darwin Scheduler solution to export, showing that 2 lag pumps are needed.



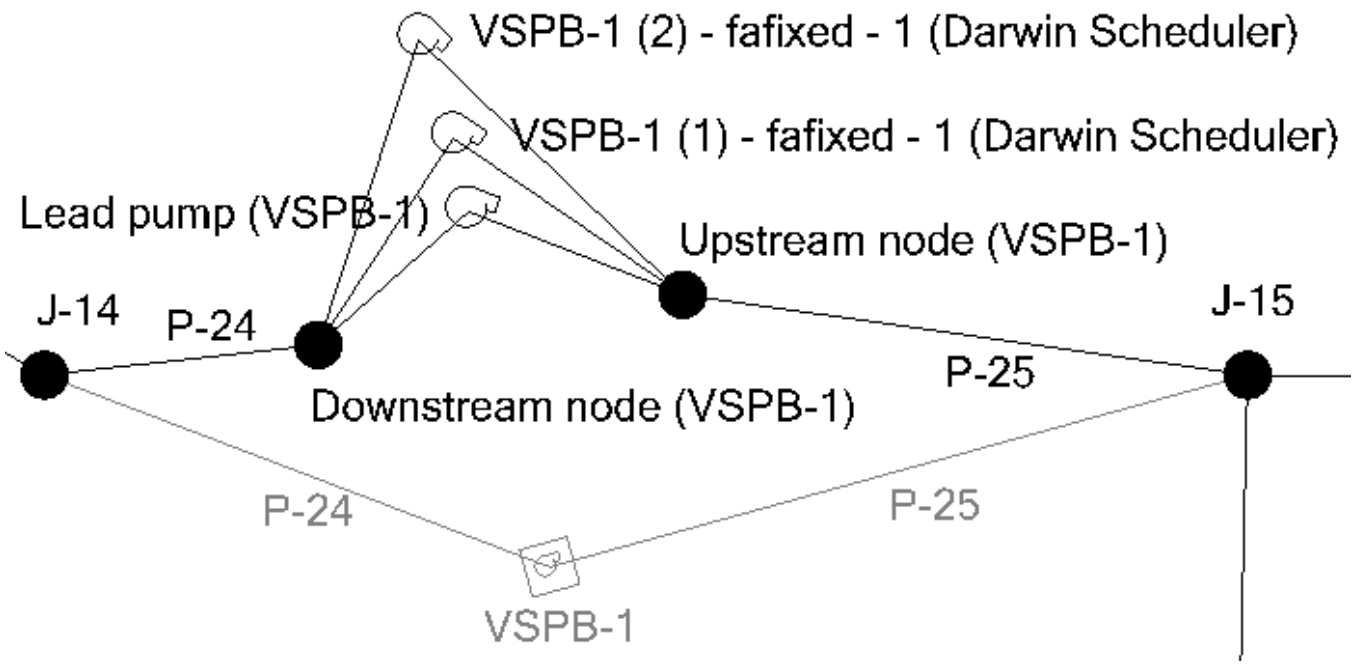
3) The situation right after exporting of solution is done (with labels re-arranged). In order to understand what elements were created, some graphical cleanup is needed. Hydraulically, the network should output the same results with (no cleanup required).



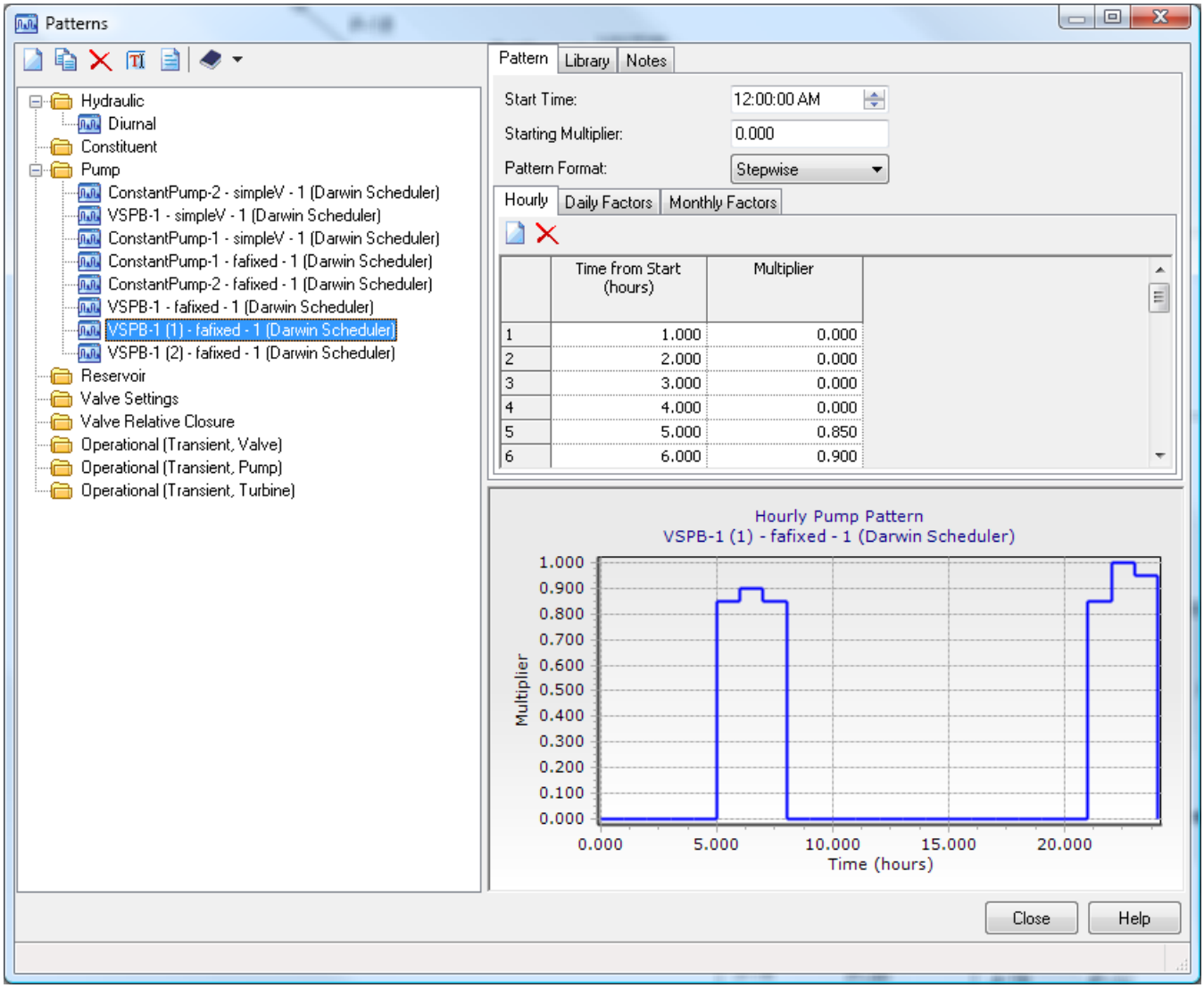
4) The situation after exporting and re-positioning the elements for a better understanding:

- The VSPB and its connecting pipes are made inactive in the new scenario created by Scheduler.
- Standard pumps are created for both the lead and each needed lag pump for the exported solution.
- Two nodes are also introduced (one upstream and one downstream of these pumps).
- Pipes connecting to the original VSPB (P-24 and P-25 in the screenshot) are duplicated and connected to those two new nodes.
- New short & large pipes (i.e. 1 ft. long, 99 in. in diameter) are setup for every standard pump in the solution, connecting them to the new upstream/downstream nodes.

- All of these new elements are only active in the exported scenario. They are left inactive in other active-topology alternatives.



5) Shows the new pump-patterns created by the export for the lead and 2 lag pumps (3 new patterns in total in the screenshot).



Presenting Your Results 15

Annotating Your Model

Color Coding A Model

Contours

Using Profiles

Viewing and Editing Data in FlexTables

Reporting

Graphs

Calculation Summary

Print Preview Window

- Node Histories Tab

Node Histories Tab

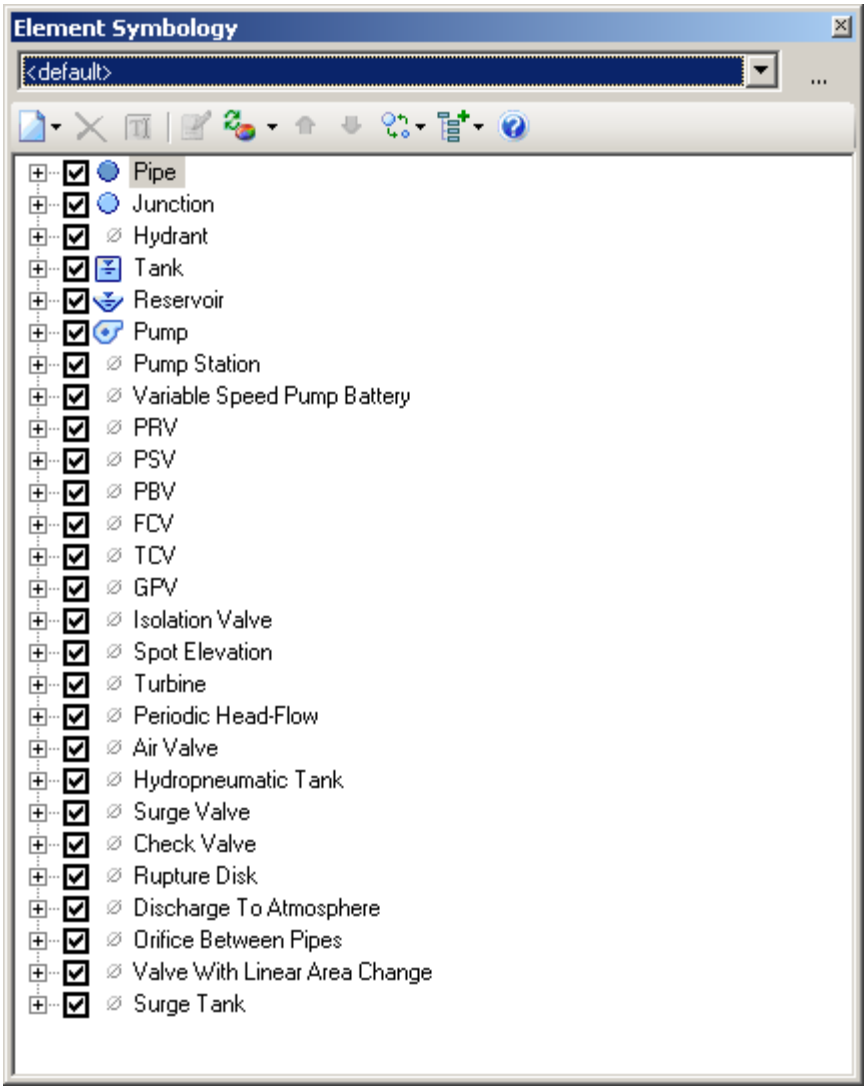
This tab consist of the following controls:

- **Working Scenario**: The current active scenario.
- **Plot**: Create a graph of the selected result attribute.
- **Additional Scenarios**: Select one or more scenarios to compare the transient node results against the results of the current scenario shown in the input field Working Scenario.
- **Node**: Displays a list of all node objects in the model with transient node results.
- **Graph Type**: Displays a list of the available transient node results available for the selected node type.

Annotating Your Model

You can annotate any of the element types in Bentley WaterCAD V8*i* using the Element Symbology manager.

To work with annotations, open the Element Symbology manager. ChooseView > Element Symbology or press <Ctrl+1> to open.



Use the Element Symbology manager to control the way that elements and their associated labels are displayed.

Note that element types that are not used in the current model are marked with an icon ⌀.

The dialog box contains a pane that lists each element type along with the following icons:

Symbology Definition	The menu lists all of the available element symbology definitions. Click the ellipsis (...) button to open the Symbology Definitions Manager.
New 	Opens a submenu containing the following commands: • **New Annotation**—Opens the Annotation Properties dialog box, allowing you to define annotation settings for the highlighted element type. • **New Color Coding**—Opens the Color Coding Properties dialog box, allowing you to define annotation settings for the highlighted element type. • **Add Folder**—Creates a folder under the currently highlighted element type, allowing you to manage the various color coding and annotation settings that are associated with an element. You can turn off all of the symbology settings contained within a folder by clearing the check box next to the folder. When a folder is deleted, all of the symbology settings contained within it are also deleted.
Delete 	Deletes the currently highlighted Color Coding or Annotation Definition or folder.
Rename 	Renames the currently highlighted object.
Edit 	Opens a Properties dialog box that corresponds with the selected background layer.

Refresh Element Symbology



Opens a shortcut menu containing the following options:

- **Refresh Element Symbology** - This can be useful if your color-coding and/or annotations are applied to a subset of elements using a query-based selection set. For performance reasons, query-based selection sets are treated as static selection sets by default. Use this option to refresh the query based selection set while refreshing element symbology.
- **Refresh Annotation** - If you change an annotation's prefix or suffix in the Property Editor, or directly in the database, selecting this command refreshes the annotation.
- **Update Annotation Offset** - If you have adjusted the Initial X or Y offsets, selecting this command resets all annotation X or Y offsets to the currently specified "initial offset" location.
- **Update Annotation Height** - If you've adjusted the height multiplier, selecting this command resets all annotation heights multipliers to the currently specified initial height multiplier

Shift Up



Moves the currently highlighted object up in the list pane.

Shift Down



Moves the currently highlighted object down in the list pane.

Drawing Style



Opens a menu containing the following commands:

- CAD Style—Displays currently highlighted element in CAD Style. Objects displayed in CAD style will appear smaller when zoomed out and larger when zoomed in.
- GIS Style—Displays currently highlighted element in GIS style. Objects displayed in GIS style will appear to remain the same size regardless of zoom level.

This button is only available in the Stand-Alone version (not in MicroStation, AutoCAD, or ArcGIS versions).

Tree



Opens a menu containing the following commands:

- Expand All—Expands each branch in the tree view pane.
- Collapse All—Collapses each branch in the tree view pane.

Help



Displays online help for the Element Symbology Manager.

The Element Symbology manager supports Copy/Paste functionality as well as Undo/Redo capability. You can copy/paste annotations, color coding definitions, and folders by right-clicking them and selecting Copy/Paste. When a folder is copied in this way all of the contents of that folder are also copied.

Using Folders in the Element Symbology Manager

Use folders in the Element Symbology manager to create a collection of color coding and/or annotation that can be turned on or off at the same time.

Adding Folders

Use element symbology folders to control whether related annotations and/or color coding displays. To create a folder in the Element Symbology manager:

1. Click **View > Element Symbology**.
2. In the Element Symbology manager, right-click an element and select **New** > **Folder**.

   Or, select the element to which you want to add the folder, click the **New** button, then select **New Folder**.
3. Name the folder.
4. You can drag and drop existing annotations and color coding into the folder you create, and you can create annotations and color coding within the folder by right-clicking the folder and selecting **New > Annotation** or **New > Color Coding**.
5. Use the folder to collectively turn on and off the annotations and color coding within the folder.

Deleting Folders

Click **View > Element Symbology**. In the Element Symbology manager, right-click the theme folder you want to delete, then select **Delete**.

Or, select the folder you want to delete, then click the **Delete** button.

Renaming Folders

Click **View > Element Symbology**. In the Element Symbology manager, right-click the theme folder you want to rename, then select **Rename**.

Or, select the folder you want to rename, then click the **Rename** button.

To add an annotation

1. Click **View > Element Symbology**.
2. In the Element Symbology manager, right-click an element and select **New** > **Annotation**.

   Or, select the element where you want to add the annotation, click the **New** button, and select **New Annotation**.
3. The Annotation Properties dialog box opens. Select the annotation you want in the Field Name menu.
4. If needed, set a Prefix or Suffix. Anything you type as a prefix is added directly to the beginning of the label and anything you type as a suffix is added to the end (you may want to include spaces as part of your prefix and suffix).

Note: If you add an annotation that uses units, you can type "%u" in the prefix or suffix field to display the units in the drawing pane.

5. Select the initial X- and Y- offset for the annotation. Offset is measured from the center of the node or polygon or midpoint of the polyline.
6. If needed, set an initial height multiplier. Use a number greater than 1 to make the annotation larger and a number between 0 and 1 to make the annotation smaller. If you use a negative number, the annotation is flipped (rotated 180 degrees).
7. If you have created selection sets, you can apply your annotation only to a particular selection set by selecting that set from the Selection Set menu. If you have not created any selection sets, then the annotation is applied to all elements of the type you are using.
8. After you finish defining your annotation, click **Apply** and then **OK** to close the Annotation Properties dialog box and create your annotation. In order to close the dialog box without creating an annotation click **Cancel**.

To delete an annotation

Click **View > Element Symbology**. In the Element Symbology manager, right-click an annotation you want to delete, then select **Delete**.

Or, select the annotation you want to delete, then click the **Delete** button.

To edit an annotation

Click **View > Element Symbology**. In the Element Symbology manager, right-click the annotation you want to edit, then select **Edit**.

Or, select the annotation you want to edit, then click the **Edit** button and the Annotation Properties dialog box will open where you can make changes.

Rename an annotation

Click **View > Element Symbology**. In the Element Symbology manager, right-click the annotation you want to rename, then select **Rename**.

Or, select the annotation you want to rename, then click the **Rename** button.

To copy an annotation

1. Click **View > Element Symbology**. In the Element Symbology manager, right-click the annotation you want to copy, then select **Copy**.
2. Right-click on the element type and folder (if applicable) under which you want the annotation to be copied and select **Paste**.

Annotation Properties

Use the Annotation Properties dialog box to define annotation settings for each element type.

Field Name — Specify the attribute that is displayed by the annotation definition.

Free Form — This field is only available when <Free Form Annotation> is selected in the Field Name list. Click the ellipsis button to open the Free Form Annotation dialog box.

Prefix — Specify a prefix that is displayed before the attribute value annotation for each element to which the definition applies.

Suffix — Specify a suffix that is displayed after the attribute value annotation for each element to which the definition applies.

Note: **If you add an annotation that uses units, you can type "%u" in the prefix or suffix field to display the units in the drawing pane.**

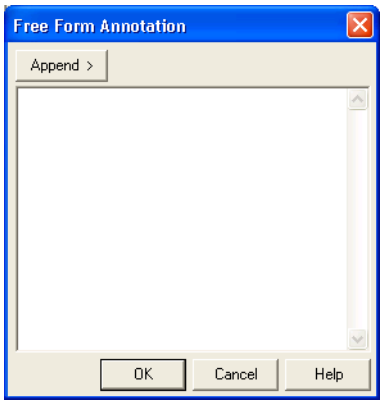
Selection Set — Specify a selection set to which the annotation settings will apply. If the annotation is to be applied to all elements, select the <All Elements> option in this field. <All Elements> is the default setting.

Initial Offset Checkbox — When this box is checked, changes made to the X and Y Offset will be applied to current and subsequently created elements. When the box is unchecked, only subsequently created elements will be affected.

Initial X Offset	Displays the initial X-axis offset of the annotation in feet. Sets the initial horizontal offset for an annotation. Set this at the time you create the annotation. Clicking OK will cause the new value to be used for all subsequent elements that you place. Clicking Apply will cause the new value to be applied to all elements.
Initial Y Offset	Displays the initial Y-axis offset of the annotation in feet. Sets the initial vertical offset for an annotation. Set this at the time you create the annotation. Clicking OK will cause the new value to be used for all subsequent elements that you place. Clicking Apply will cause the new value to be applied to all elements.
Initial Multiplier Checkbox	When this box is checked, changes made to the Height Multiplier will be applied to current and subsequently created elements. When the box is unchecked, only subsequently created elements will be affected.
Initial Height Multiplier	Sets the initial size of the annotation text. Set this at the time you create the annotation. Clicking OK will cause the new value to be used for all subsequent elements that you place. Clicking Apply will cause the new value to be applied to all elements.

Free Form Annotation Dialog Box

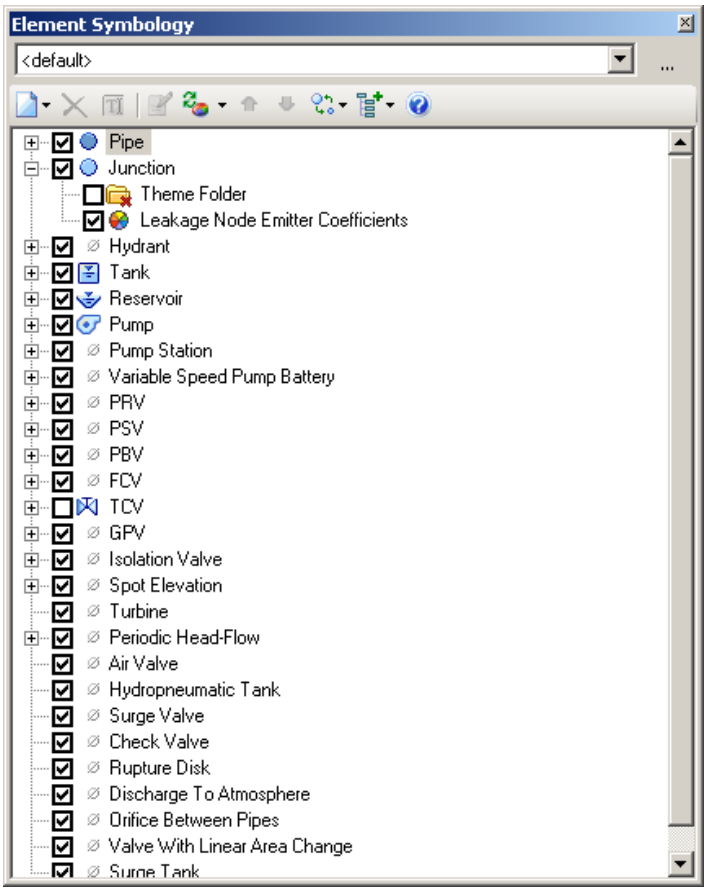
The Free Form Annotation dialog box allows you to type custom annotations for an element type.



To create an annotation, type the text as you want it to appear in the drawing. You can add element attributes to the text string by clicking the Append button and selecting the attribute from the categorized list.

Symbology Definitions Manager

The Symbology Definitions manager lets you add, edit, and remove and manage the symbology definitions that are associated with the project.



The dialog box contains a list pane that displays each of the definitions currently contained within the project, a display pane that details the settings for the currently highlighted definition, along with a toolbar. The toolbar consists of the following buttons:

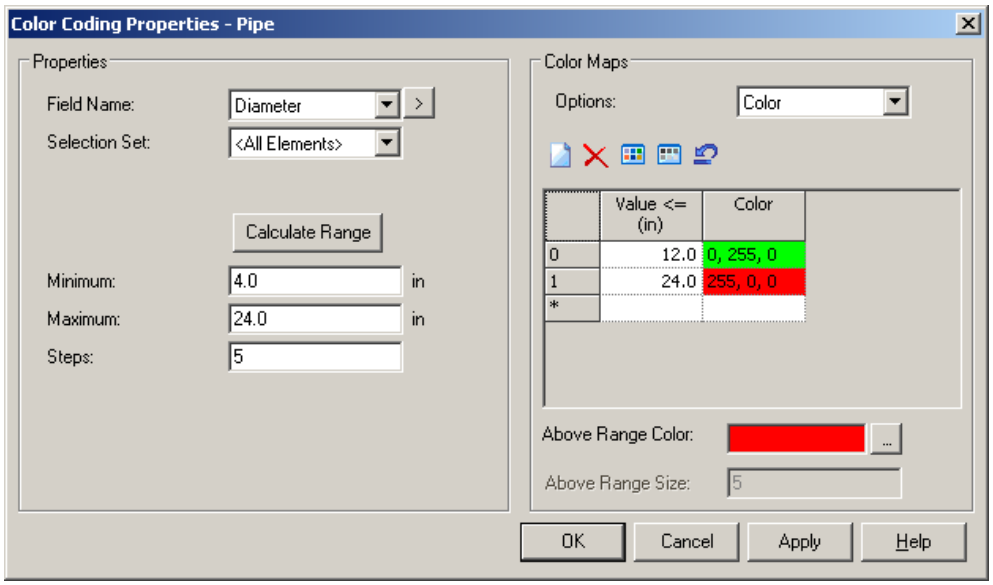
	**New**	Creates a new symbology definition in the list pane.
	**Import**	Allows you to import a previously exported symbology definition.
	**Export**	Exports the currently highlighted symbology definition as an **.sde** file that can be imported into other projects.
	**Delete**	Removes the currently highlighted symbology definition.
	**Duplicate**	Creates a copy of the currently highlighted symbology definition.
	**Rename**	Lets you rename the currently highlighted symbology definition.
	**Help**	Displays online help for the Symbology Definitions manager.

When you create a new definition, all of the annotation and color settings will be turned off. To change the settings for a definition, change the current symbology definition to the one you want to edit in the Element Symbology Manager and make the desired changes there (i.e. turn on/off the desired elements, create new annotations and color coding and turn them on or off, etc.).

Color Coding A Model

Use color coding to help you quickly see what's going on in your model or to change the color and/or size of elements based on the value of data that you select, such as flow or element size.

To work with color coding, go to View > Element Symbology > New Color Coding to open the Color Coding Properties dialog box.



The dialog box consists of the following controls:

Properties

Field Name — Select the attribute by which the color coding is applied.

Selection Set — Apply a color coding to a previously defined selection set.

Calculate Range — Automatically finds the minimum and maximum values for the selected attribute and enters them in the appropriate Min. and Max fields.

Minimum — Define the minimum value of the attribute to be color coded.

Maximum — Define the maximum value of the attribute to be color coded.

Steps — Specify how many rows are created in the color maps table when you click Initialize. When you click Initialize, a number of values equal to the number of Steps are created in the color maps table. The low and high values are set by the Min and Max values you set.

Color Map

Options

Select whether you want to use color coding, sizing, or both to code and display your elements.

Map colors to value ranges for the attribute being color coded. The following buttons are found along the top of the table:

- **New**—Creates a new row in the Color Maps table. 
- **Delete**—Deletes the currently highlighted row from the Color Maps table. 
- **Initialize**—Finds the range of values for the specified attribute, divides it into equal ranges based on the number of Steps you have set, and assigns a color to each range. 
- **Ramp**—Generates a gradient range between two colors that you specify. Pick the color for the first and last values in the list, then Bentley WaterCAD V8*i* automatically sets intermediate colors for the other values. For example, picking red as the first color and blue as the last color produces varying shades of purple for the other values. 
- **Invert**—Reverse the order of the colors/sizes used in the Color Map table. 

Above Range Color

Displays the color that is applied to elements whose value for the specified attribute fall outside the range defined in the color maps table. This selection is available if you choose Color or Color and Size from the Options list.

Above Range Size

Displays the size that is applied to elements whose value for the specified attribute fall outside the range defined in the color maps table. This selection is available if you choose Size or Color and Size from the Options list.

To add color coding, including element sizing

1. Click **View > Element Symbology**.
2. In the Element Symbology manager, right-click an element and select **New** > **Color Coding**.

Or, select the element you want to add the color coding, click the **New** button, and select **New Color Coding**.

3. The Color Coding Properties dialog box opens. Select the properties you want to color code from the Field Name and Selection Set menus. Once you've selected the Field Name, more information opens.
4. In the Color Maps Options menu, select whether you want to apply color, size, or both to the elements you are coding.
   a. Click **Calculate Range**. This automatically sets the maximum and minimum values for your coding. These values can be set manually.
   b. Click **Initialize**. This automatically creates values and colors in the Color Map. These values can be set manually.
5. After you finish defining your color coding, click Apply and then **OK** to close the Color Coding Properties dialog box and create your color coding, or **Cancel** to close the dialog box without creating a color coding.
6. Click **Compute** to compute your network.
7. To see the network color coding and/or sizing change over time:
   a. Click **Analysis > Time Browser**, if needed, to open the Time Browser dialog box.
   b. Click **Play** to use the Time Browser to review your color coding over time.

To delete a color coding definition

Click **View > Element Symbology**. In the Element Symbology manager, right-click the color coding you want to delete, then select **Delete**.

Or, select the color coding you want to delete, then click the **Delete** button.

To edit a color coding definition

Click **View > Element Symbology**. In the Element Symbology manager, right-click the color coding you want to edit, then select **Edit**.

Or, select the color coding you want to edit, then click the **Edit** button.

To rename a color coding definition

Click **View > Element Symbology**. In the Element Symbology manager, right-click the color coding you want to rename, then select **Rename**.

Or, select the color coding you want to rename, then click the **Rename** button.

To copy a color coding definition

1. Click **View > Element Symbology**. In the Element Symbology manager, right-click the color coding you want to copy, then select **Copy**.
2. Right-click on the folder under which you want the defintion to be copied and select **Paste**.

Color Coding Legends

You can add color coding legends to the drawing view. A legend displays a list of the colors and the values associated with them for a particular color coding definition.

To add a color coding legend

Right-click the color coding definition in the Element Symbology dialog and select the Insert Legend command.

To move a color coding legend

1. Click the legend in the drawing view to highlight it.
2. Click and hold onto the legend grip (the square in the center of the legend), then drag the legend to the new location.

To resize a color coding legend

1. Right-click the legend in the drawing view and select the Scale command.
2. Move the mouse to resize the legend and click the left mouse button to accept the new size.

To remove a color coding legend

Right-click the color coding definition in the Element Symbology dialog and select the Remove Legend command.

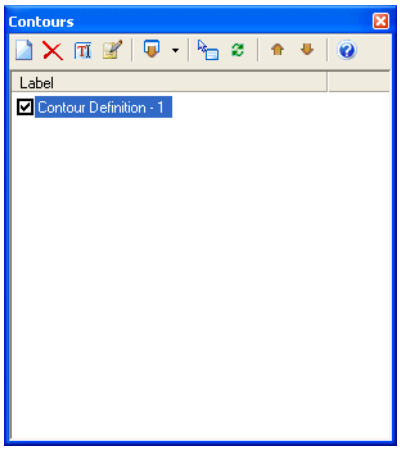
To refresh a color coding legend

Right-click the color coding definition in the Element Symbology dialog and select the Refresh Legend command.

Contours

Using WaterCAD V8*i* you can visually display calculated results for many attributes using contour plots.

The Contours dialog box is where all of the contour definitions associated with a project are stored. Choose View > Contours to open the Contours dialog box.



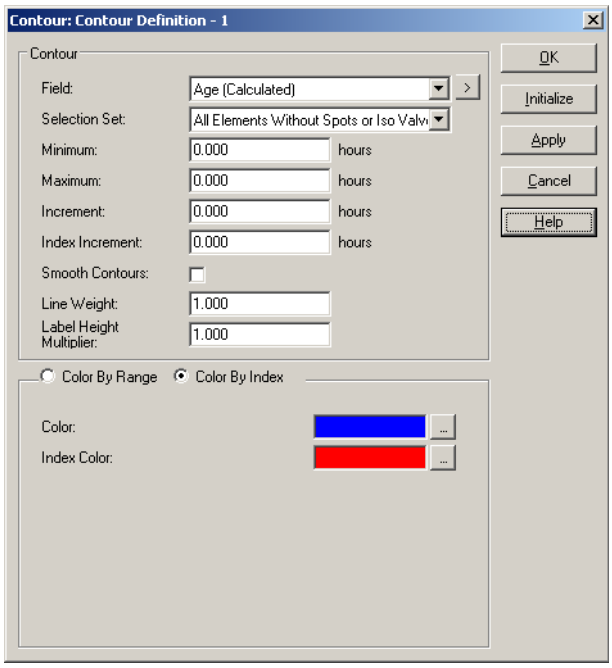
The dialog box contains a list pane that displays all of the contours currently contained within the project, along with a toolbar.

New		Opens the Contour Definition dialog box, allowing you to create a new contour.
Delete		Deletes the currently selected contour. You can hold down the Ctrl key while clicking on items in the list to select multiple entries at once.
Rename		Renames the currently selected contour.
Edit		Opens the Contour Definition dialog box, where you can modify the settings of the currently selected contour.

Export		Clicking this button opens a submenu containing the following commands: • **Export to Shapefile** - Exports the contour to a shapefile, opening the Export to File Manager to select the shapefile. • **Export to DXF** - Exports the contour as a .dxf drawing. • **Export to Native Format** - Opens the DXF Properties dialog box, allowing you to add it to the Background Layers Manager.
View Contour Browser		Opens the Contour Browser dialog, allowing you to display detailed contour results for points in the drawing view.
Refresh		Regenerates the contour.
Shift Up		Moves the currently selected contour up in the list pane.
Shift Down		Moves the currently selected contour down in the list pane.
Help		Displays online help for the Contours.

Contour Definition

The Contour Definition dialog box contains the information required to generate contours for a calculated network.



Contour

Field — Select the attribute to apply the contour.

Selection Set — Apply an attribute to a previously defined selection set or to one of the following predefined options:

- **All Elements** - Calculates the contour based on all elements in the model, including spot elevations.
- **All Elements Without Spots** - Calculates the contour based on all elements in the model, except for spot elevations.

Minimum	Lowest value to be included in the contour map. It may be desirable to use a minimum that is above the absolute minimum value in the system to avoid creating excessive lines near a pump or other high-differential portions of the system.
Maximum	Highest value for which contours will be generated.
Increment	Step by which the contours increase. The contours created will be evenly divisible by the increment and are not directly related to the minimum and maximum values. For example, a contour set with 10 minimum, 20 maximum, and an increment of 3 would result in the following set: [12, 15, 18] not [10, 13, 16, 19].
Index Increment	Value for which contours will be highlighted and labeled. The index increment should be an even multiple of the standard increment.
Smooth Contours	The Contour Smoothing option displays the results of a contour map specification as smooth, curved contours.
Line Weight	The thickness of contour lines in the drawing view.
Label Height Multiplier	When contours are created, there are labels (text) placed on the end of the index contours. This text has a default size. The Label Height Multiplier field allows you to scale the text size for these labels up/down.

Color by Range

Contours are colored based on attribute ranges. Use the Initialize button to create five evenly spaced ranges and associated colors.



Initialize—This button, located to the right of the Contour section, will initialize the Minimum, Maximum, Increment, and Index Increment values based on the actual values observed for the elements in the selection set.

Tip: **Initialization can be accomplished by clicking the Initialize button to automatically generate values for the minimum, maximum, increment, and index increment to create an evenly spaced contour set.**

Ramp—Automatically generate a gradient range between two colors that you specify. Pick the color for the first and last values in the list and the program will select colors for the other values.

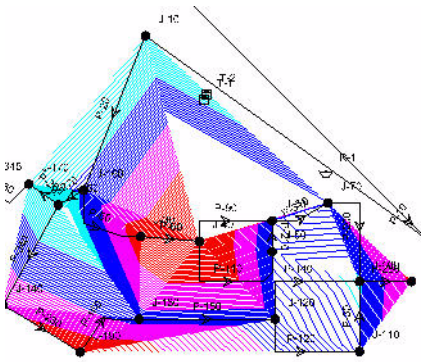
Color by Index

The standard contours and index contours have separately controlled colors that you can make the contours more apparent.

Contour Plot

The Contour Plot window displays the results of a contour map specification as accurate, straight-line contours.

View the changes in the mapped attribute over time by using the animation feature. Choose Analysis > Time Browser and click the **Play** button to automatically advance through the time step increments selected in the Increment bar.



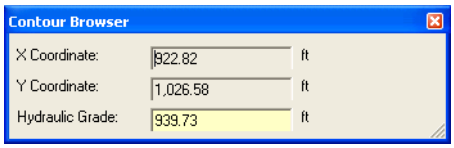
The plot can be printed or exported as a .DXF file. Choose File > Export > DXF to export the plot.

Tip: **Although the straight-line contours generated by this program are accurate, smooth contours are often more desirable for presentation purposes. You can smooth the contours by clicking Options and selecting Smooth Contours.**

Note: **Contour line index labels can be manually repositioned in this view before sending the plot to the printer. The Contour Plot Status pane displays the Z coordinate at the mouse cursor.**

Contour Browser Dialog Box

The Contour Browser dialog box displays the X and Y coordinates and the calculated value for the contour attribute at the location of the mouse cursor in the drawing view.



Enhanced Pressure Contours

Normal contouring routines only include model nodes, such as junctions, tanks and reservoirs. When spot elevations are added to the drawing, however, you can create more detailed elevation contours and enhanced pressure contours.

These enhanced contours include not only the model nodes but also the interpolated and calculated results for the spot elevations. Enhanced pressure contours can help the modeler to understand the behavior of the system even in areas that have not been included directly in the model.

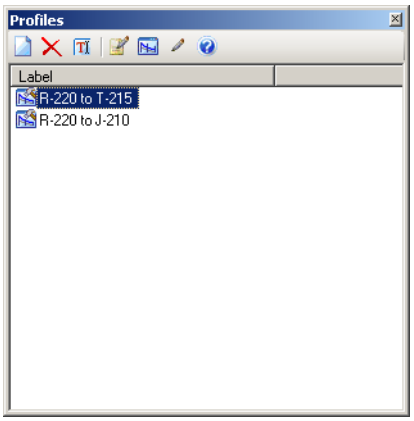
Using Profiles

A profile is a graph that plots a particular attribute across a distance, such as ground elevation along a section of piping. As well as these side or sectional views of the ground elevation, profiles can be used to show other characteristics, such as hydraulic grade, pressure, and constituent concentration.

You define profiles by selecting a series of adjacent elements. To create or use a profile, you must first open the Profiles manager. The Profiles manager is a dockable window where you can add, delete, rename, edit, and view profiles.

The Profiles dialog box is where you can create, view, and edit profile views of elements in the network.

The dialog box contains a list pane that displays all of the profiles currently contained within the project, along with a toolbar.



New 	Opens the Profile Setup dialog box, where you can select the elements to be included in the new profile from the drawing view.
Delete 	Deletes the currently selected profile. You can hold down the Ctrl key while clicking on items in the list to select multiple entries at once.
Rename 	Renames the currently selected profile.
Edit 	Opens the Profile Setup dialog box, where you can modify the settings of the currently selected profile.
View Profile 	Opens the Profile viewer, allowing you to view the currently selected profile.
Highlight Profile 	When this toggle button is on, elements contained within the currently highlighted profile will be highlighted in the drawing pane to increase their visibility.
Help 	Displays online help for Profiles.

By default, all profiles are created as Transient Report Paths. A Transient Report Path is denoted by a small hammer icon. 

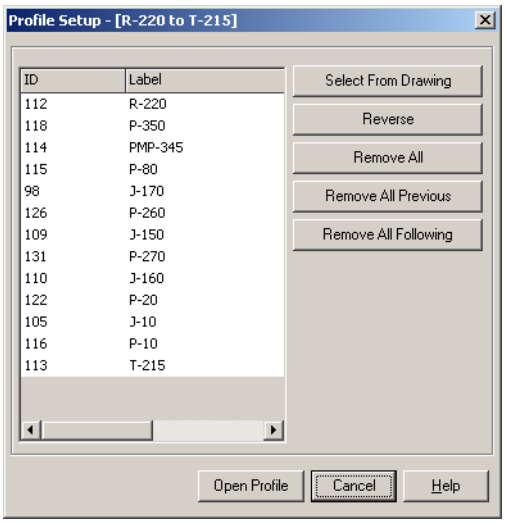
When a transient analysis is completed in HAMMER, profile results will only be stored for those elements along a previously defined Transient Report Path.

You can right-click a profile in the Profile Manager and uncheck the Transient Report Path toggle command in the context menu. When unchecked, transient analysis results will not be saved for that profile. Reducing the number of Transient Report Paths can reduce output file sizes and improve calculation times.

Transient Report Paths are not used directly in WaterGEMS/WaterCAD - in those products results from all profiles are always available. However the Transient Report Path toggle and hammer icon are included in WaterGEMS/WaterCAD so that projects created within any of the three programs will be compatible.

Profile Setup

Setting up a profile is a matter of selecting the adjacent elements on which the profile is based. When you click on New in the Profiles dialog box the following dialog box opens.



The Profile Setup dialog box includes the following options:

ID	The element ID of the corresponding profile element.
Label	The label of the corresponding profile element.
Select From Drawing	Selects and clears elements for the profile.
Reverse	Reverses the profile, so the first node in the list becomes the last and the last node becomes the first.
Remove All	Removes all elements from the profile.
Remove All Previous	Removes all elements that appear before the selected element in the list. If the selected element is a pipe, the associated node is not removed.

Remove All Following	Removes all elements that appear after the selected element in the list. If the selected element is a pipe, the associated node is not removed.
Open Profile	Closes the Profile Setup dialog box and opens the Profile Series Options dialog box.

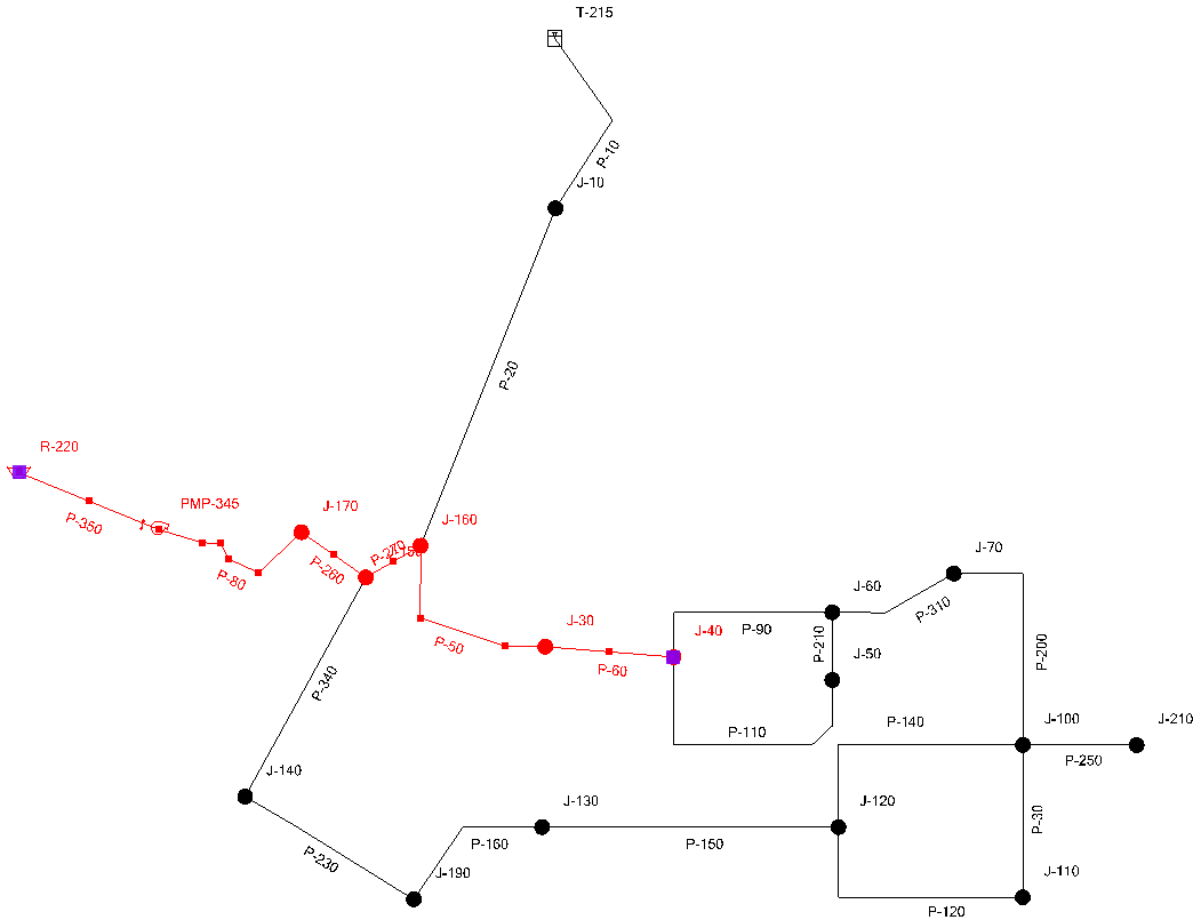
You can edit your list of profile elements at any time and compute your network with the Profile Viewer dialog box open, but you must click Refresh to update the display of that dialog box if you do make changes.

Note that certain changes made to the network (morphing one element into another, reconnecting pipes) can break existing profiles that include the modified element(s). If this happens, delete the last node before the break (where the modified element is) in the profile setup dialog and edit it accordingly to add the modified elements.

Note: In AutoCAD mode, you cannot use the shortcut menu, you must re-open the Profile Setup dialog box.

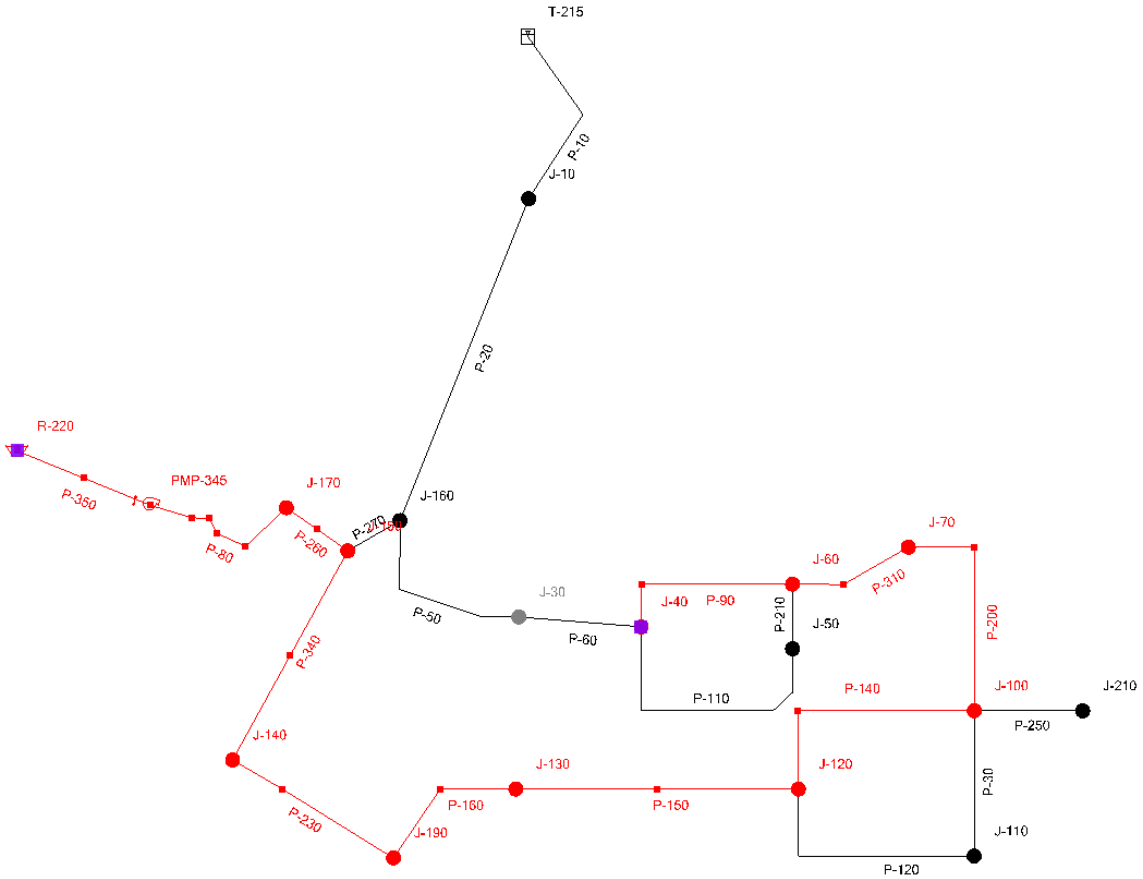
Profile Selection with Inactive Elements

Normally, WaterCAD V8*i* will select the shortest path between two elements when setting up a profile, as shown below:



The user has selected R-220 and J-40; the profile is the shortest path between the selected elements

If one or more elements along the shortest path is Inactive, WaterCAD V8*i* will select the shortest path that avoids the inactive elements, as shown below:

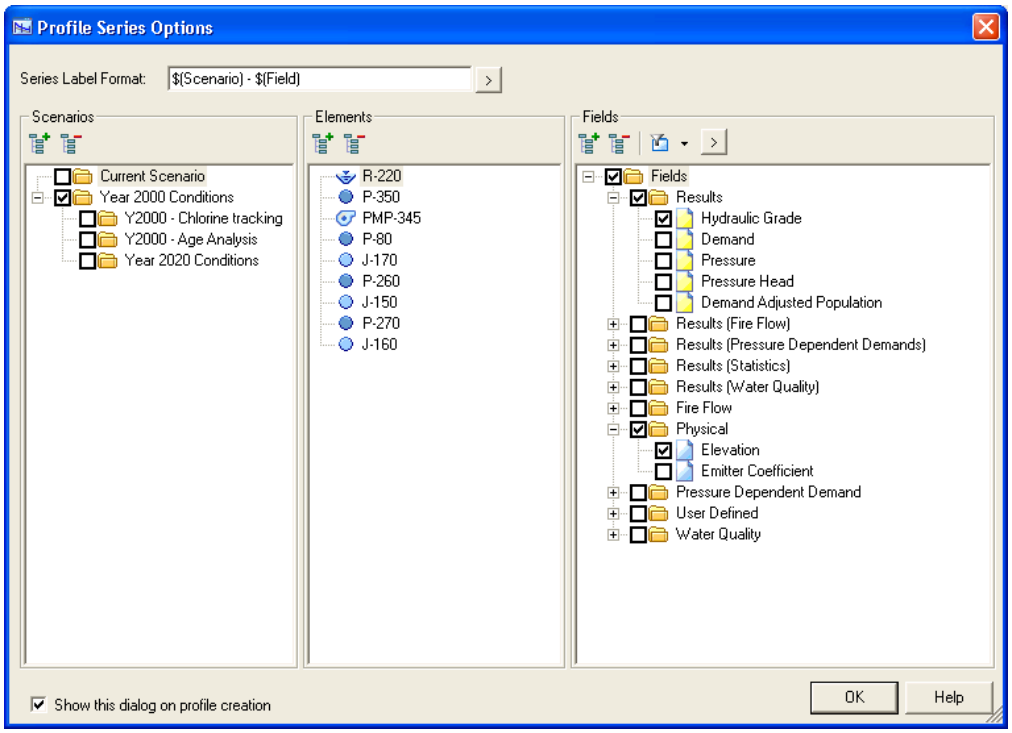


The user has again selected R-220 and J-40 but J-30 is Inactive. The profile is the shortest path around the inactive element

You can include inactive elements in a profile; to do so, create a profile along the desired path up to the first inactive element. Then click on each inactive pipe that you wish to include in the profile until the profile path is complete, or your path returns to the active elements again.

Profile Series Options Dialog Box

The Profile Series Options dialog box allows you to adjust the display settings for the profile view. You can define the legend labels, the scenario (or scenarios), and the attribute (or attributes) that are displayed in the profile plot.



The Series Label Format field allows you to define how the series will be labeled in the legend of the profile view. Clicking the [>] button allows you to choose from predefined variables such as Field name and Element label.

The Scenarios pane lists all of the available scenarios. Check the box next to a scenario to display the data for that scenario in the profile view. The Expand All button opens all of the folders so that all scenarios are visible; the Collapse button closes the folders.

The Elements pane lists all of the elements that will be displayed in the profile view. The Expand All button expands the list tree so that all elements are visible; the Collapse button collapses the tree.

The Fields pane lists all of the available input and output fields. Check the box next to a field to display the data for that field type in the profile view. The Expand All button opens all of the folders so that all fields are visible; the Collapse button closes the folders. The Filter by Field Type button allows you to display only Input or Output fields in the list. Clicking the [>] button opens a submenu that contains all of the available fields grouped categorically.

Note that profiles don't show any results for the intermediate points along a pipe. To see the results of transient calculations for these intermediate points, you will need to use the Transient Results Viewer.

The Show this dialog on profile creation check box is enabled by default; uncheck this box to skip this dialog when a new profile is created.

Profile Viewer

When you complete setting up your profile a Profile viewer will open which contains the profile in graph or data format.



It consists of the profile display pane and the following controls:

Profile Series Setting Opens the Profile Series Options box.



Chart Settings



Opens the Chart Options dialog box to view and modify the display settings for the current profile plot.

Note: Never delete or rename any of the series entries on the Series Tab of the Chart Options dialog box. These series were specifically designed to enable the display of the Profile Plots.

Print



Prints the current view of the profile to your default printer. If you want to use a printer other than your default, use Print Preview to change the printer and print the profile.

Print Preview



Opens a print preview window containing the current view of the profile. You can use the Print Preview dialog box to select a printer and preview the output before you print it.

Note: Do not change the print preview to grayscale, as doing so might hide some elements of the display.

Copy



Copies the contents of the Profile viewer dialog box as an image to the Windows clipboard from where you can paste it into another application, such as Microsoft® Word or Adobe® Photoshop®.

Zoom Extents



Magnifies the profile so that the entire graph is displayed.

Zoom



Magnify or reduce the display of a section of the graph. To zoom or magnify an area, select the Zoom Window tool, click to the left of the area you want to magnify, then drag the mouse to the right, across the area you want to magnify, so that the area you want to magnify is contained within the marquee that the Zoom Window tool draws. After you have selected the area you want to magnify, release the mouse button to stop dragging.
To zoom out, or reduce the magnification, drag the mouse from right to left across the magnified image.

Animation Controls



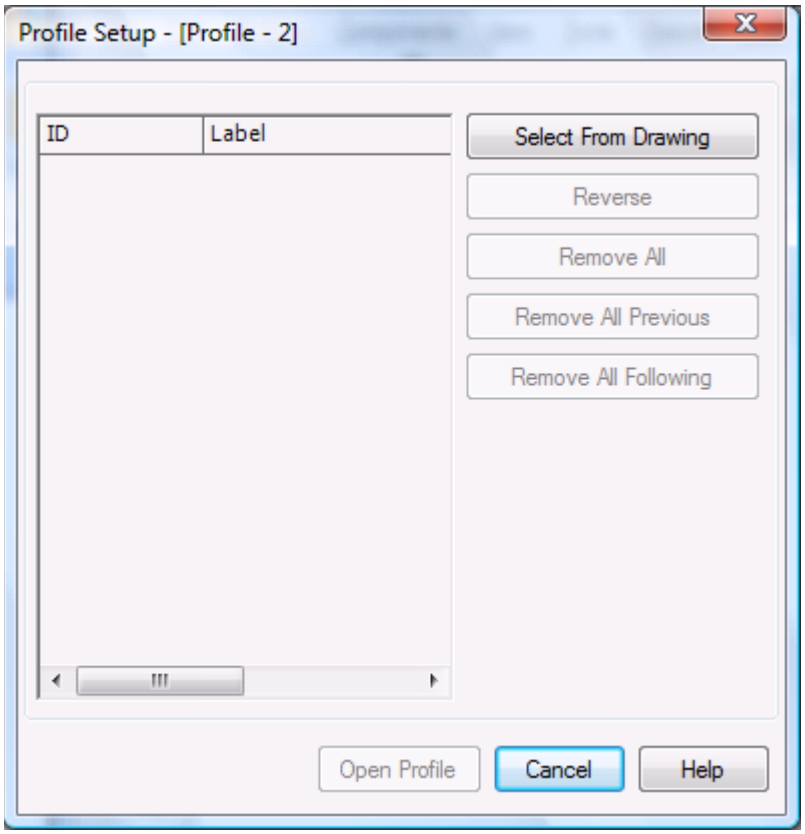
- **Go to start**—Sets the currently displayed time step to the beginning of the simulation.
- **Pause/Stop**—Stops the animation. Restarts it again with another click.
- **Play**—Advances the currently displayed time step from beginning to end.
- **Time**—Shows the current time step that is displayed in the drawing pane.
- **Time Slider**—Manually move the slider representing the currently displayed time step along the bar, which represents the full length of time that the scenario encompasses.

To create a new profile

1. Choose **View** > **Profiles** or click the Profiles Manager icon on the View toolbar to open the Profiles manager.

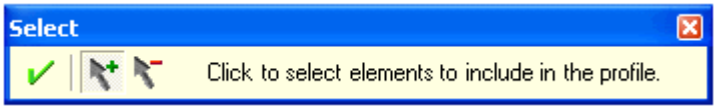
2. Click **New**  .

3. The Profile Setup dialog box opens.



4. Select the Elements you want to use:

   a. Click **Select from Drawing**. The Select dialog box opens:



   To create a profile, the user can select the beginning and ending element of the profile and then pick the green check. The shortest path between those elements will be used to draw the profile. If the user wants to create a profile along a path other than the shortest path, the user should initially draw the path through the first element that the profile will be forced through and then add elements as described below. The profile will display in the drawing in red and the node elements that the user selected along the profile will be in purple.

   b. To add elements to the profile, click elements in the drawing pane. (By default, the **Add** button  is active in the Select dialog box.) You can only add elements to either end of your selection.

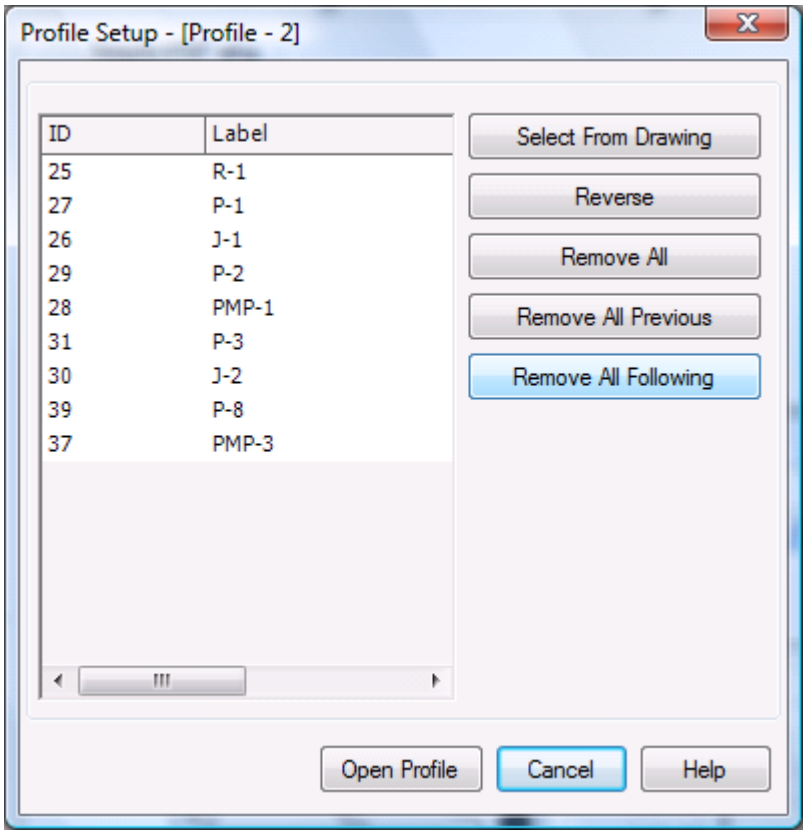
When the Add button is toggled on, you can select elements to add to the profile; elements that you successfully select are highlighted in red.

c. To remove elements from the profile, click the **Remove** button in the Select dialog box. Thereafter, elements you select in the drawing pane are removed from the profile. You can only remove elements from either end of your selection.

When the Remove button is toggled on, you can remove elements from the profile; unselected elements are not highlighted.

d. When you are finished adding elements to your profile, click the **Done** button in the Select dialog box.

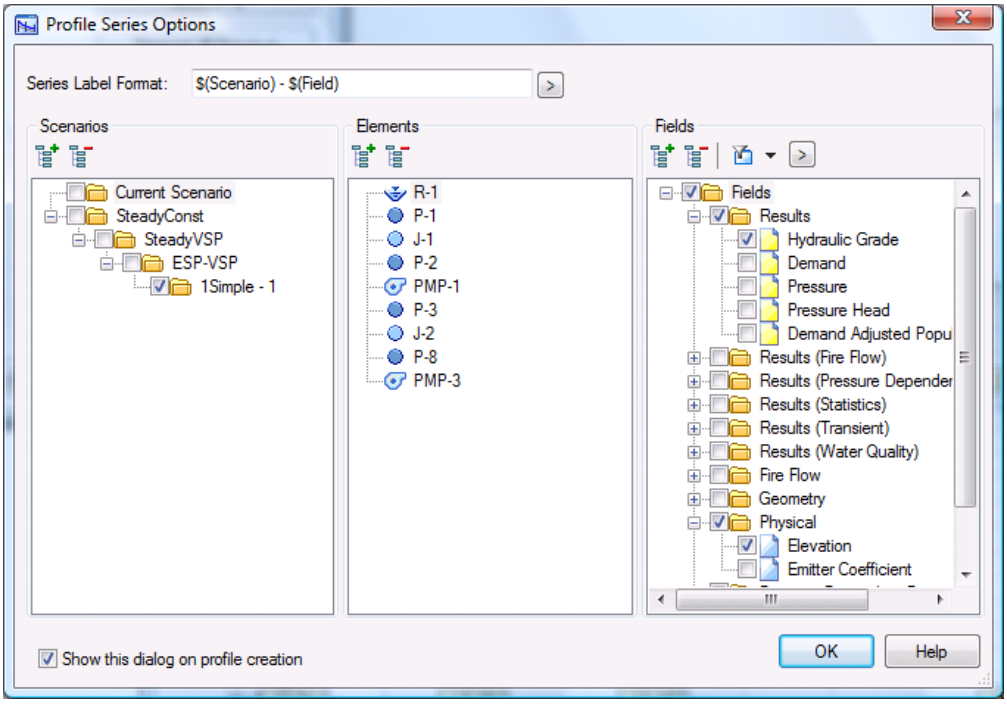
5. The Profile Setup dialog box opens and displays a list of the elements you selected.



6. Click **Open Profile** to close the Profile Setup dialog box and open the Profile Series Options box.

Note: If you want to close the Profile Setup box without saving your changes, click Cancel or close the dialog.

7. Select the **Scenarios, Elements,** and **Fields** to be included in the Profile. Then click **OK**. By default the Elevation and Hydraulic Grade fields are selected for the current scenario.



8. The Profile viewer opens.
9. Once you have created a profile you can open it by double clicking on the name of the profile or by right clicking and selecting **Open** from the menu.

To edit a profile

You can edit a profile to change the elements that it uses or the order in which those elements are used.

1. Choose **View > Profiles** to open the Profiles manager.
2. In the Profiles manager, right-click the profile you want to edit, then select **Edit**

    .

   Or, select the profile you want to edit, then click **Edit** .
3. The Profile Setup dialog box opens. Modify the profile as needed and click **Open Profile** to save your changes or **Cancel** to exit without saving your changes.

To delete a profile

Click **View > Profiles** to open the Profiles manager. In the Profiles manager, right-click the profile you want to delete, then select **Delete**  .

Or, select the profile you want to delete, then click **Delete**.

To rename a profile

Click **View > Profiles** to open the Profiles manager. In the Profiles manager, right-click the profile you want to rename, then select **Rename**  .

Or, select the profile you want to rename, then click **Rename**.

To highlight the profile path in the drawing

Click **View > Profile** to open the Profiles Manager, the click the **Highlight** button  .

Or, select the profile, then right click the Highlight command.

There is an additional right click option, "Transient Report Path". This is used when a WaterGEMS/CAD model is imported into HAMMER for transient analysis. A report on transients is prepared for any path for which this option is checked.

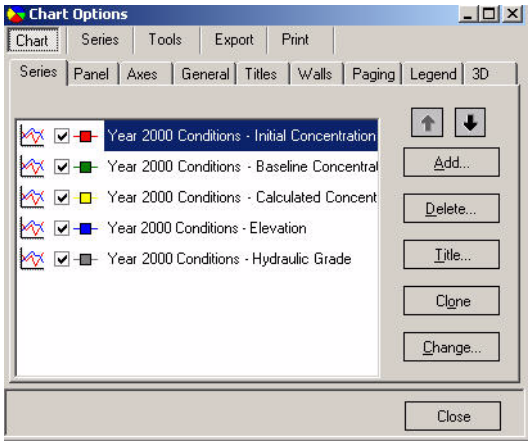
To view a profile

1. Click **Compute**  to calculate flows.
2. Click **View > Profiles** to open the Profile manager.
3. In the Profile manager, select the profile you want to view, and right click **Open** or double-click the profile to be viewed.

Note: You can edit your list of profile elements at any time and compute your network with the Profile Viewer dialog box open, but you must click Refresh to update the display of that dialog box if you do make changes.

4. The Profile dialog box opens.

5. In order to change the look of the profile click **Chart Settings** .



6. If you want to print you can use **Print Preview** to see what it will look like and then **Print**.

To animate a profile

1. Click **Compute** to calculate flows.
2. Click **View > Profiles** to open the Profiles manager.
3. In the Profiles manager, select the profile you want to view and click the **Profile** button to open the profile in Profile Viewer.
4. In the Profile dialog box, move the **Time** slider or click one of the animation controls and watch the profile change over time in the Profile Viewer. As needed, click the **Pause** button in the Scenario Animation dialog box to study the profile at a given time.

Viewing and Editing Data in FlexTables

Using FlexTables you can view input data and results for all elements of a specific type in a tabular format. You can use the standard set of FlexTables or create customized FlexTables to compare data and create reports.

You can view all elements in the project, all elements of a specific type, or any subset of elements. Additionally, to ease data input and present output data for specific elements, FlexTables can be:

- Filtered
- Globally edited
- Sorted.

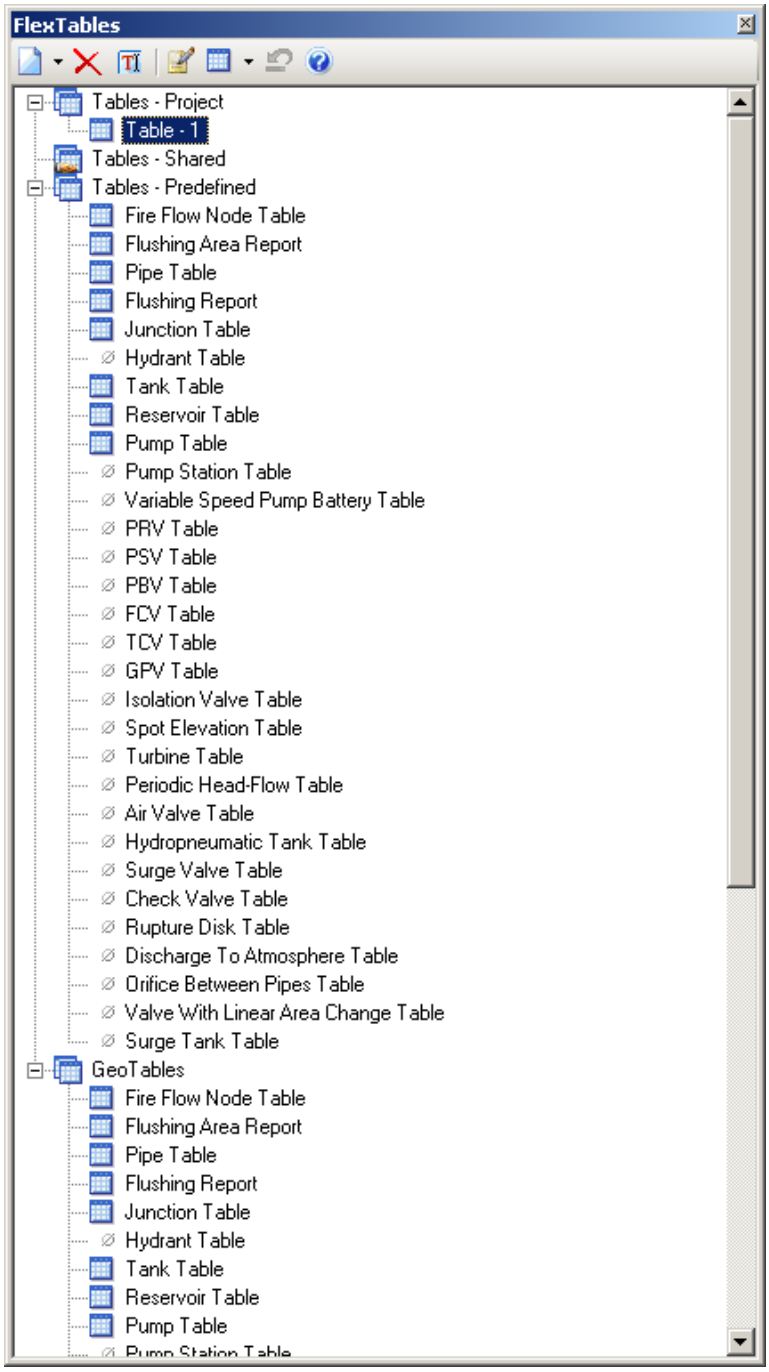
If you need to edit a set of properties for all elements of a certain type in your network, you might consider creating a FlexTable and making your changes there rather than editing each element one at a time in sequence.

FlexTables can also be used to create results reports that you can print, save as a file, or copy to the Windows clipboard for copying into word processing or spreadsheet software.

To work with FlexTables, select the FlexTables manager or go to **View > FlexTables** <Ctrl+7> to open the FlexTables manager if it is closed.

FlexTables

Using the FlexTables manager you can create, manage, and delete custom tabular reports. The dialog box contains a list pane that displays all of the custom FlexTables currently contained within the project, along with a toolbar.



Note that element types that are not used in the current model are marked with an icon ⌀.

The toolbar contains the following icons:

New



Opens a menu containing the following commands:

- **FlexTable**—Creates a new tabular report and opens the FlexTable Setup dialog box, where you can define the element type that the FlexTable displays and the columns that are contained in the table.
- **Folder**—Creates a folder in the list pane in order to group custom FlexTables.

Delete



Deletes the currently selected FlexTable.

Rename



Renames the currently selected FlexTable.

Edit



Opens the FlexTable Setup dialog box, allowing you to make changes to the format of the currently selected table.

Open



Opens a menu containing the following commands:

- Open-Opens the currently selected FlexTable.
- Open On Selection-Opens the FlexTable for the element that is highlighted in the drawing.

Reset to Factory Defaults



When a Predefined table is highlighted in the list, this button allows you to reset the highlighted table to the factory default.

Help



Displays online help for the FlexTable manager.

Working with FlexTable Folders

You can add, delete, and rename folders in the FlexTable manager to organize your FlexTables into groups that can be turned off as one entity. You can also create folders within folders. When you start a new project, Bentley WaterCAD V8*i* displays two items in the FlexTable manager: Tables - Project (for project-level FlexTables) and Tables - Shared (for FlexTables shared by more than one Bentley WaterCAD V8*i* project). You can add new FlexTables and FlexTable folders to either item or to existing folders.

To add a FlexTable folder

1. Click **View** > **FlexTables** or  to open the FlexTables manager.
2. In the FlexTable manager, select either Tables - Project or Tables - Shared, then click the New button.
   - If you are creating a new folder within an existing folder, select the folder, then click the New button.
3. Click **New Folder** from the menu.
4. Right-click the new folder and click **Rename** or click  .
5. Type the name of the folder, then press <**Enter**>.

To delete a FlexTable folder

1. Click **View** > **FlexTables** to open the FlexTables manager.
2. In the FlexTables manager, select the folder you want to delete, then click the Delete button.
   - You can also right-click a folder to delete, then select **Delete** from the shortcut menu.

To rename a FlexTable folder

1. Click **View** > **FlexTables** to open the FlexTables manager.
2. In the FlexTables manager, select the folder you want to rename, then click the Rename button.
   - You can also right-click a folder to rename, then select **Rename** from the shortcut menu.
3. Type the new name of the folder, then press **Enter**.
   - You can also rename a FlexTable folder by selecting the folder, then modifying its label in the Properties Editor.

FlexTable Dialog Box

FlexTables are displayed in the FlexTable dialog box. The dialog box contains a toolbar, the rows and columns of data in the FlexTable, and a status bar.

The toolbar contains the following buttons:



Export 
Export to a Tab Delimited file .txt or a Comma Delimited File .csv.

Copy 
Copy the contents of the selected table cell, rows, and/or columns for the purpose of pasting into a different row or column or into a text editing program such as Notepad.

Paste 
Paste the contents of the Windows clipboard into the selected table cell, row, or column. Use this with the Copy button.

Edit 
Opens the FlexTable Setup dialog box, so you can make changes to the format of the currently selected table.

Zoom To



Centers the drawing view on the element that is currently highlighted in the table.

Report



Report Current Time Step, Report All Time Steps, or Report in XML.

Selection Set



Opens a submenu containing the following commands:

- **Create Selection Set**—Creates a new static selection set (a selection set based on selection) containing the currently selected elements in the FlexTable.
- **Add to Selection Set**—Adds the currently selected elements in the FlexTable to an existing selection set.
- Remove from Selection Set—Removes the currently selected element in the FlexTable from an existing Selection Set.
- **Relabel**-Opens an Element Relabeling box where you can Replace, Append, or Renumber.

Select in Drawing 	Opens a submenu containing the following commands: • **Select In Drawing**—Selects the currently highlighted element(s) in the drawing pane. • **Add to Current Selection** —Adds the currently selected elements to the group of elements currently selected in the drawing pane. • **Remove from Current Selection** — Removes the currently selected elements from the group of elements currently selected in the drawing pane. • **Select Within Current Selection**— Selects the element or elements that are both currently highlighted in the FlexTable and are already selected in the Drawing Pane.

Opening FlexTables

You open FlexTables from within the FlexTable manager.

To open FlexTables

1. Click **View > FlexTables** or click the FlexTables button on the View toolbar to open the FlexTables manager.
2. Perform one of the following steps:
   - Right-click the FlexTable you want to open, then select **Open**.
   - Select the FlexTable you want to open, then click the **Open** button.
   - Double-click the FlexTable you want to open.

Creating a New FlexTable

You can create project-level or shared FlexTables.

- Project-level FlexTables are available only for the project in which you create them.
- Shared tables are available in all projects.

To create a new FlexTable

Project-level and shared FlexTables are created the same way:

1. Click **View > FlexTables** or click the FlexTables button on the View toolbar to open the FlexTables manager.
2. In the FlexTables manager, right-click **Tables - Project** or **Tables - Shared**, then select **New > FlexTable**.

   Or, select **Tables - Project** or **Tables - Shared**, click the **New** button, then select **FlexTable**.
3. The Table Setup dialog box opens.
4. Select the Table Type to be created.
5. Filter the table by element type.
6. Select the items to be included by double-clicking on the item or select the item and click the Add arrow to move to the Selected Columns pane.
7. Click **OK**.
8. The table displays in the FlexTables manager; you can type to rename the table or accept the default name.

Deleting FlexTables

Click **View > FlexTables** to open the FlexTables manager. In the FlexTables manager, right-click the FlexTable you want to delete, then select **Delete**.

Or, select the FlexTable you want to delete, then click the **Delete** button. You cannot delete predefined FlexTables.

Note: You cannot delete predefined FlexTables.

Naming and Renaming FlexTables

You name and rename FlexTables in the FlexTable manager.

To rename FlexTables

1. Click **View > FlexTables** or click the FlexTables button on the View toolbar to open the FlexTables manager.
2. Perform one of the following steps:
   - Right-click the FlexTable you want to rename, then select **Rename**.
   - Select the FlexTable you want to rename, then click the **Rename** button.
   - Click the FlexTable you want to rename, to select it, then click the name of the FlexTable.

Note: You cannot rename predefined FlexTables.

Editing FlexTables

You can edit a FlexTable to change the columns of data it contains or the values in some of those columns.

Editable columns:	Columns that contain data you can edit are displayed with a white background. You can change these columns directly in the FlexTable and your changes are applied to your model when you click **OK**. The content in the FlexTable columns can be changed in other areas, such as in a Property Editor or managers. If you make a change that affects a FlexTable outside the FlexTable, the FlexTable is updated automatically to reflect the change.
Non-editable columns:	Columns that contain data you cannot edit are displayed with a yellow background and correspond to model results calculated by the program and composite values. The content in these columns can be changed in other areas, for example a Property Editor or by running a computation. If you make a change that affects a FlexTable outside the FlexTable, the FlexTable is updated automatically to reflect the change.

To edit a FlexTable

1. Click **View > FlexTables** to open the FlexTables manager, then you can:
   - Right-click the FlexTable, then select **Edit**.
   - Double-click the FlexTable to open it, then click **Edit**.
   - Click the FlexTable to select it, then click the **Edit** button.
2. The Table dialog box opens. .
3. Use the Table dialog box to include and exclude columns and change the order in which the columns appear in the table.
4. Click **OK** after you finish making changes to save your changes and close the dialog box; or click **Cancel** to close the dialog box without making changes.

Editing Column-Heading Text

To change the text of a column heading:

1. Click **View > FlexTables** to open the FlexTables manager.
2. In the FlexTables manager, open the FlexTable you want to edit.
3. Right-click the column heading and select **Edit Column Label**.
4. Type the new name for the label and click **OK** to save those changes and close the dialog box or **Cancel** to exit without making any changes.

Changing Units, Format, and Precision in FlexTables

To change the units, format, or precision in a column of a FlexTable:

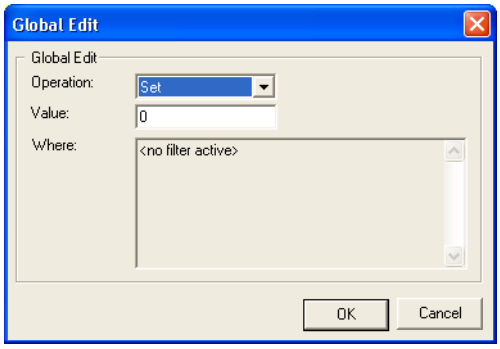
1. Click **View > FlexTables** to open the FlexTables manager.
1. In the FlexTables manager, open the FlexTable you want to edit.
2. Right-click the column heading and select **Units**.
3. Make the changes you want and click **OK** to save those changes or **Cancel** to exit without making any changes.

Navigating in Tables

The arrow keys, <Ctrl+End>, <Page Up>, <Page Down>, and <Ctrl+arrow> keys navigate to different cells in a table.

Globally Editing Data

Using FlexTables, you can globally edit all of the values in an entire editable column. Globally editing a FlexTable column can be more efficient for editing properties of an element than using the Properties Editor or managers to edit each element in your model individually.



Operation	Select the type of edit to perform: • Set: Changes each of the entries in the column to the value in the Value box. • Add: Adds the value in the Value box to each of the entries in the column. • Divide: Divides each of the entries in the column by the value in the Value box. • Multiply: Multiplies each of the entries in the column by the value in the Value box. • Subtract: Subtracts the value in the Value box from each of the entries in the column.
Value	Type the value that will be used in the chosen Operation to edit the entries of the column.
Where	When the Table has an active filter, the SQL Query used by the filter is displayed in this pane.

To globally edit the values in a FlexTable column

1. Click **View** > **FlexTables** to open the FlexTables manager.
2. In the FlexTables manager, open the FlexTable you want to edit and find the column of data you want to change.

If necessary, you might need to first create a FlexTable or edit an existing one to make sure it contains the column you want to change.

3. Right-click the column heading and select **Global Edit**.
4. In the **Operation** field, select what you want to do to data in the column: Add, Divide, Multiply, Set, or Subtract.

   Note: The Operation field is only available for numeric data.

5. In the **Global Edit** field, type or select the value.

Sorting and Filtering FlexTable Data

You can sort and filter your FlexTables to focus on specific data or present your data in one of the following ways:

To sort the order of columns in a FlexTable

You can sort the order of columns in a FlexTable in two ways:

- Edit the FlexTable; open the Table dialog box and change the order of the selected tables using the up and down arrow buttons.

  The top-most item in the Selected Columns pane appears furthest to the left in the resulting FlexTable.
- Open the FlexTable, click the heading of the column you want to move, then click again and drag the column to the new position. You can only move one column at a time.

To sort the contents of a FlexTable

1. Open the FlexTable to be edited.
2. Right-click a column heading to rank the contents of the column.
3. Select **Sort** then choose.
   - **Sort Ascending**—Sorts alphabetically from A to Z, from top to bottom. Sorts numerically from negative to positive, from top to bottom. Sorts selected check boxes to the top and cleared ones to the bottom.
   - **Sort Descending**—Sorts alphabetically from Z to A, from top to bottom. Sorts numerically from positive to negative, from top to bottom. Sorts cleared check boxes to the top and selected ones to the bottom.
   - **Custom**—Select one or more sort keys
   - **Reset**—Back to the original sorting order

To filter a FlexTable

Filter a FlexTable by creating a query.

1. Open the FlexTable to be filtered.
2. Right-click the column heading to filter and select **Filter**.

   Select **Custom** to open the Query Builder dialog box.
3. All input and results fields for the selected element type appear in the Fields list pane, available SQL operators and keywords are represented by buttons, and available values for the selected field are listed in the Unique Values list pane. Perform the following steps to construct your query:

   a. Double-click the field to include in your query. The database column name of the selected field appears in the preview pane.

   b. Click the desired operator or keyword button. The SQL operator or keyword is added to the SQL expression in the preview pane.

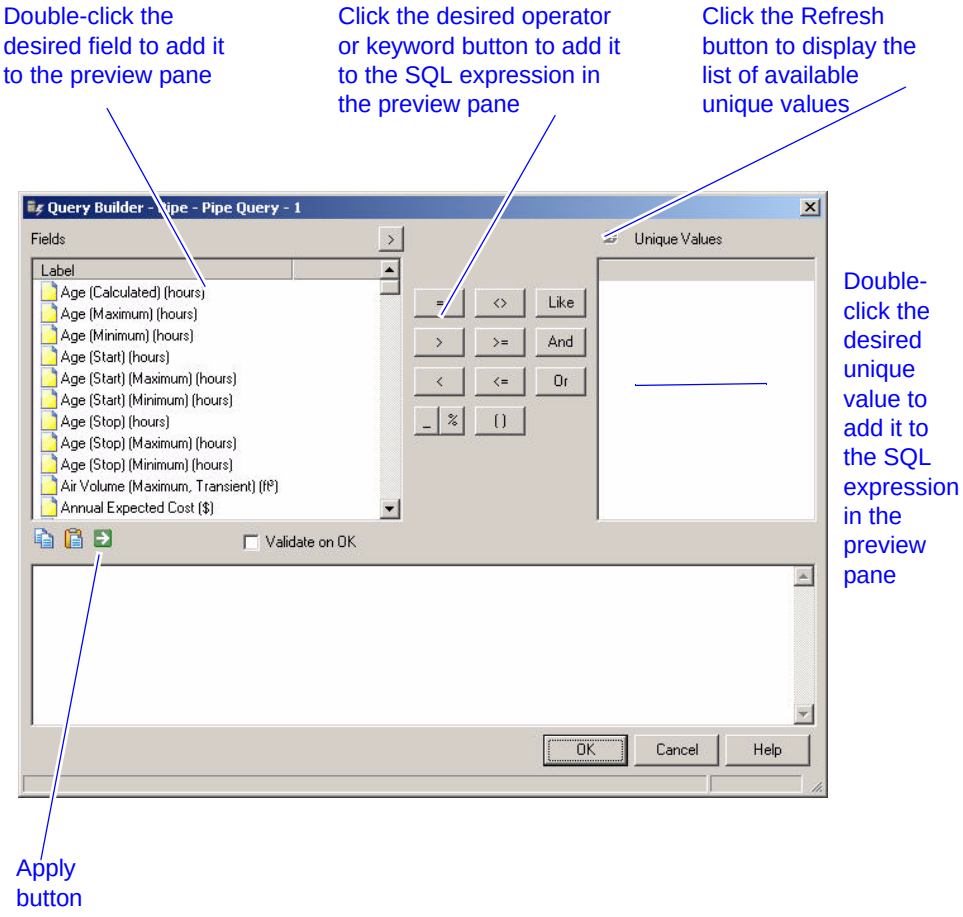
   c. Click the **Refresh** button above the Unique Values list pane to see a list of unique values available for the selected field. The **Refresh** button becomes disabled after you use it for a particular field.

   d. Double-click the unique value you want to add to the query. The value is added to the SQL expression in the preview pane.

   e. Click **Apply**  above the preview pane to validate your SQL expression. If the expression is valid, the window “Query Successful" opens. Click **OK**. The word **VALIDATED** will be at the bottom of the window.



f. Click **OK**.



The FlexTable displays columns of data for all elements returned by the query and the word “FILTERED” is displayed in the FlexTable status bar.

The status pane at the bottom of the Table window always shows the number of rows displayed and the total number of rows available (for example, 10 of 20 elements displayed).

If you change the values for an attribute that is being sorted or filtered, the sort or filter operation needs to be reapplied. To do this, use the Apply Sort/Filter command accessible from the right-click context menu.

To reset a filter

1. Right-click the column heading you want to filter.
2. Select **Filter**.

3. Click Reset.
4. Click **Yes** to reset the active filter.

To reapply a sort or filter operation

1. Right-click the column heading for the sort or filter operation you want reapplied.
2. Select Apply Sort/Filter.

Custom Sort Dialog Box

You can sort elements in the table based on one or more columns in ascending or descending order. For example, the following table is given:

Slope (ft./ ft.)	Depth (ft.)	Discharge (cfs)
0.001	1	4.11
0.002	1	5.81
0.003	1	7.12
0.001	2	13.43
0.002	2	19.00
0.003	2	23.27

A custom sort is set up to sort first by Slope, then by Depth, in ascending order. The resulting table would appear in the following order:

Slope (ft./ ft.)	Depth (ft.)	Discharge (cfs)
0.001	1	4.11
0.001	2	13.43
0.002	1	5.81
0.002	2	19.00
0.003	1	7.12
0.003	2	23.27

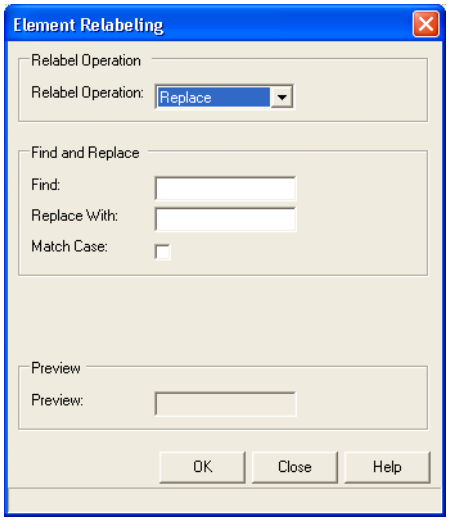
Customizing Your FlexTable

There are several ways to customize tables to meet a variety of output requirements:

- **Changing the Report Title**—When you print a table, the table name is used as the title for the printed report. You can change the title that appears on your printed report by renaming the table.
- **Adding/Removing Columns**—You can add, remove, and change the order of columns from the Table Setup dialog box.
- **Drag/Drop Column Placement**—With the Table window open, select the column heading of the column that you would like to move and drag the column to its new location.
- **Resizing Columns**—With the Table open, click the vertical separator line between column headings. Notice that the cursor changes shape to indicate that you can resize the column. Drag the column separator to the left or right to stretch the column to its new size.
- **Changing Column Headings**—With the Table window open, right-click the column heading that you wish to change and select **Edit Column Label**.

Element Relabeling Dialog

This dialog is where you perform global element relabeling operations for the Label column of the FlexTable.



The element relabeling tool allows you to perform three types of operations on a set of element labels: Replace, Renumber, and Append. The active relabel operation is chosen from the list box in the Relabel Operations section of the Relabel Elements dialog box. The entry fields for entering the information appropriate for the active relabel operation appear below the Relabel Operations section. The following list presents a description of the available element relabel operations.

- **Replace**—This operation allows you to replace all instances of a character or series of characters in the selected element labels with another piece of text. For instance, if you selected elements with labels P-1, P-2, P-12, and J-5, you could replace all the Ps with the word Pipe by entering P in the Find field, Pipe in the Replace With field, and clicking the Apply button. The resulting labels are Pipe-1, Pipe-2, Pipe-12, and J-5. You can also use this operation to delete portions of a label. Suppose you now want to go back to the original labels. You can enter Pipe in the Find field and leave the Replace With field blank to reproduce the labels P-1, P-2, P-12, and J-5. There is also the option to match the case of the characters when searching for the characters to replace. This option can be activated by checking the box next to the Match Case field.
- **Renumber**—This operation allows you to generate a new label, including suffix, prefix, and ID number for each selected element. For example, if you had the labels P-1, P-4, P-10, and Pipe-12, you could use this feature to renumber the elements in increments of five, starting at five, with a minimum number of two digits for the ID number field. You could specify a prefix P- and a suffix -Z1 in the Prefix and Suffix fields, respectively. The prefix and suffix are appended to the front and back of the automatically generated ID number. The value of the new ID

for the first element to be relabeled, 5, is entered in the Next field. The value by which the numeric base of each consecutive element is in increments, 5, is entered in the Increment field. The minimum number of digits in the ID number, 2, is entered in the Digits field. If the number of digits in the ID number is less then this value, zeros are placed in front of it. Click the Apply button to produce the following labels: P-05-Z1, P-10-Z1, P-15-Z1, and P-20-Z1.

- **Append**—This operation allows you to append a prefix, suffix, or both to the selected element labels. Suppose that you have selected the labels 5, 10, 15, and 20, and you wish to signify that these elements are actually pipes in Zone 1 of your system. You can use the append operation to add an appropriate prefix and suffix, such as P- and -Z1, by specifying these values in the Prefix and Suffix fields and clicking the Apply button. Performing this operation yields the labels P-5-Z1, P-10-Z1, P-15-Z1 and P-20-Z1. You can append only a prefix or suffix by leaving the other entry field empty. However, for the operation to be valid, one of the entry fields must be filled in.

The Preview field displays an example of the new label using the currently defined settings.

FlexTable Setup Dialog Box

The Table Setup dialog box is where you can customize tables through the following options:

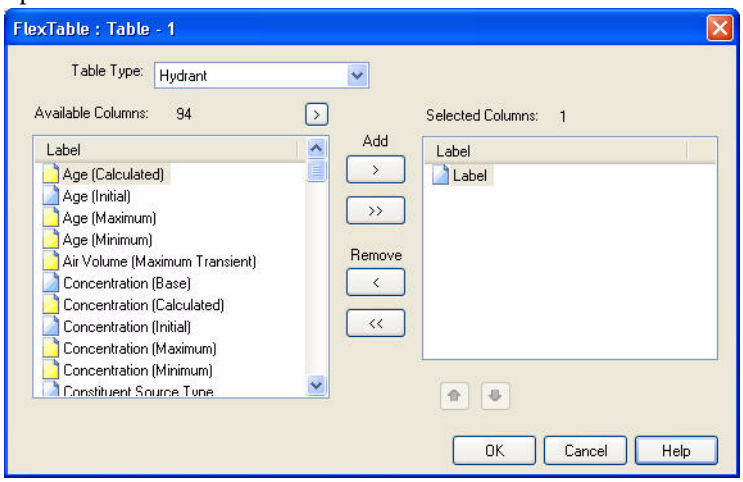


Table Type Specifies the type of elements that appear in the table. It also provides a filter for the attributes that appear in the Available Columns list. When you choose a table type, the available list only contains attributes that can be used for that table type. For example, only manhole attributes are available for a manhole table.

Available Columns

Contains all the attributes that are available for your table design. The Available Columns list is located on the left side of the Table Setup dialog box. This list contains all of the attributes that are available for the type of table you are creating. The attributes displayed in yellow represent non-editable attributes, while those displayed in white represent editable attributes.
Click the Arrow button [>] to open a submenu that contains all of the available fields grouped categorically.

Selected Columns

Contains attributes that appear in your custom designed FlexTable. When you open the table, the selected attributes appear as columns in the table in the same order that they appear in the list. You can drag and drop or use the up and down buttons to change the order of the attributes in the table. The Selected Columns list is located on the right-hand side of the Table Setup dialog box. To add columns to the Selected Columns list, select one or more attributes in the Available Columns list, then click the **Add** button [>].

Add and Remove Buttons



Select or clear columns to be used in the table and arrange the order the columns appear.
The Add and Remove buttons are located in the center of the Table Setup dialog box.

- [>] Adds the selected items from the Available Columns list to the Selected Columns list.
- [>>] Adds all of the items in the Available Columns list to the Selected Columns list.
- [<] Removes the selected items from the Selected Columns list.
- [<<] Removes all items from the Selected Columns list.

To rearrange the order of the attributes in the Selected Columns list, select the item to be moved, then click the up or down button  .

Copying, Exporting, and Printing FlexTable Data

You can output your FlexTable several ways:

- Copy FlexTable data using the clipboard
- Export FlexTable data as a text file
- Create a FlexTable report.

To copy FlexTable data using the clipboard

You can copy your FlexTable data using the clipboard and paste it into another Windows application, such as a word-processing application as tab-delimited text.

1. Click **View > FlexTables** to open the FlexTables manager.
2. In the FlexTables manager, open the FlexTable you want to use.
3. Click **Copy**. The contents of the FlexTable are copied to the Windows clipboard.

   Caution: Make sure you paste the data you copied before you copy anything else to the Windows clipboard. If you copy something else to the clipboard before you paste your FlexTable data, your FlexTable data will be lost from the clipboard.

4. Paste <Ctrl+v> the data into other Windows software, such as your word-processing application.

To export FlexTable data as a text file

You can export the data in a FlexTable as tab- or comma-delimited ASCII text for use in other applications, such as Notepad, spreadsheet, or word processing software.

1. Click **View > FlexTables** to open the FlexTables manager.
2. In the FlexTables manager, open the FlexTable you want to use.
3. Click **Export to File**  **.**
4. Select either **Tab Delimited** or **Comma Delimited**.
5. When prompted, set the path and name of the .txt file you want to create.

To create a FlexTable report

Create a FlexTable Report if you want to print a copy of your FlexTable and its values.

1. Click **View > FlexTables** to open the FlexTables manager.
2. In the FlexTables manager, open the FlexTable you want to use.

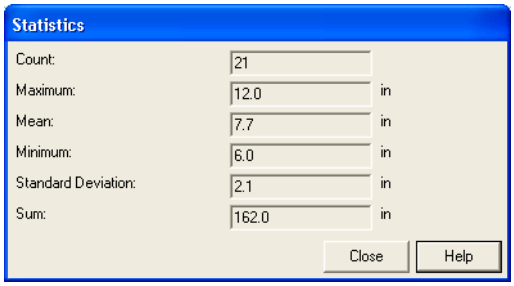
   Note: Instead of Print Preview, you can click Print to print the report without previewing it.

3. Click **Report** and select one of the options. A print preview of the report displays to show what your report will look like.

Note: You cannot edit the format of the report.

Statistics Dialog Box

The Statistics dialog box displays statistics for the elements in a FlexTable. You can right-click any unitized input or output column and choose the Statistics command to view the count, maximum value, mean value, minimum value, standard deviation, and sum for that column.



Using Sparklines

In FlexTable reports, the result columns only show the result value at the current time step. To visualize how the results vary over time, the graphing feature can be used to draw the results; while this method works for individual elements, there is no easy way to see the results over time for all elements at the same time. To address this, the Sparkline feature has been added. When Sparklines are turned on, a results column is added to the FlexTable that displays a miniature graph of the result values over time.

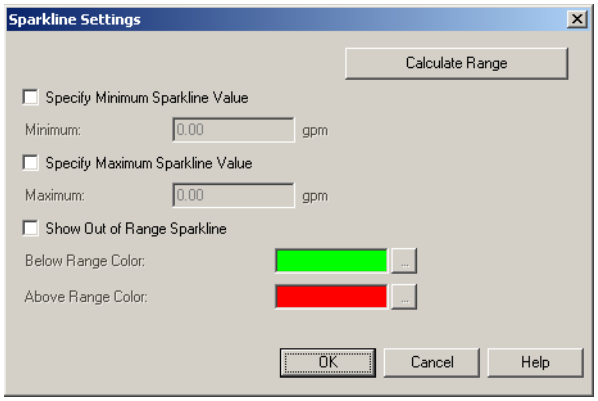
To turn on Sparklines for a result attribute, create your FlexTable as usual, then right click the column heading for the desired result attribute and select **Show Sparklines** from the context menu.

When there is a currently active Sparklines column, you can right click the column heading and select **Sparkline Settings** to change the display settings for the graphs. See Sparkline Settings.

To turn Sparklines off, right click the attribute heading and select **Hide Sparklines**.

Sparkline Settings

This dialog alloows you to specify the settings used for the Sparklines feature.



The dialog consist of the following controls:

- **Calculate Range**: This button allows you to automatically determine the minimum and maximum values. Clicking this button opens a submenu with the following options
  - **Full Range**: When this option is selected, a precise values are used to calculate the range.
  - **Quick Range**: When this option is selected, a rough estimate of the range of values is used.
- **Specify Minimum Sparkline Value**: When this box is checked, you may specify the minimum value for the range in the Minimum field.
- **Specify Maximum Sparkline Value**: When this box is checked, you may specify the maximum value for the range in the Maximum field.
- **Show Out of Range Sparklines**: When this box is checked, sparklines that fall outside the specified range will still be displayed; values that fall below the specified range will be displayed in the selected Below Range Color and values that fall above the specified range will be displayed in the selected Above Range Color.

Reporting

Use reporting to create printable content based on some aspect of your model, such as element properties or results.

You need to compute your model before you can create reports about results, such as the movement of water in your network. You can also create reports about input data without computing your model, such as conduit diameters. (To compute your model, after you set up your elements and their properties, click **Compute**.)

You can access reports by:

- Clicking the **Report** menu.
- Right-clicking any element, then selecting **Report.**

Using Standard Reports

There are several standard reports available. To access the standard reports, click the **Report** menu, then select the report.

Reports for Individual Elements

You can create reports for specific elements in your network by computing the network, right-clicking the element, then selecting **Report**. You cannot format the report, but you can print it by clicking the **Print** icon.

Creating a Scenario Summary Report

To create a report that summarizes your scenario, click **Report** > **Scenario Summary**. The report dialog box opens and displays your report. You cannot format the report, but you can print it by clicking the **Print** button.

Creating a Project Inventory Report

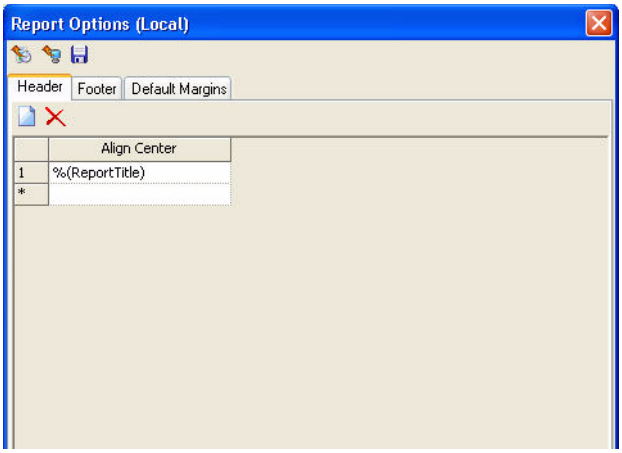
To create a report that provides an overview of your network, click **Report > Project Inventory**. The report dialog box opens and displays your report. You cannot format the report, but you can print it by clicking the **Print** button.

Creating a Pressure Pipe Inventory Report

To create a report that lists the total lengths of pipe by diameter, material type, and volume, click Report > Pressure Pipe Inventory. The report dialog opens and displays the Pressure Pipe Inventory report. You can copy rows, columns, or the entire table to the clipboard by highlighting the desired rows and/or columns and clicking **Ctrl+C**.

Report Options

The Report Options dialog box offers control over how a report is displayed.



Load factory default settings to current view  . Click to restore the default settings to the current view.

Load global default settings to current view  . Click to view the stored global settings as local settings.

Save current view settings to global settings  . Click to set the current report options as the global default.

The header and footer can be fully customized and you can edit text to be displayed in the cells or select a pre-defined dynamic variable from the cell's menu.

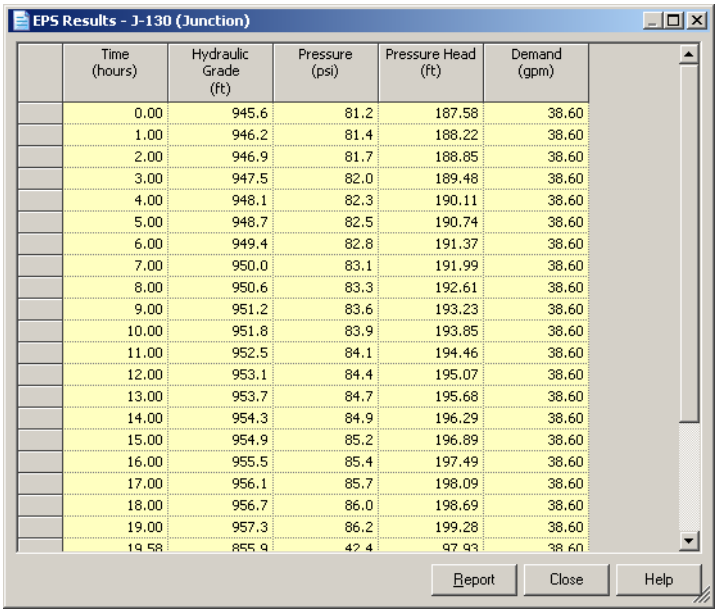
- %(Company) - The name specified in the project properties.
- % (DateTime) - The current system date and time.
- % (BentleyInfo) - The standard Bentley company information.
- % (BentleyName) - The standard Bentley company name information.
- % (Pagination) - The report page out of the maximum pages.
- % (ProductInfo) - The current product and its build number.
- % (ProjDirectory) - The directory path where the project file is stored.
- % (ProjEngineer) - The engineer specified in the project properties.
- % (ProjFileName) - The full file path of the current project.
- % (ProjStoreFileName) - The full file path of the project.

- % (ProjTitle) - The name of the project specified in the project properties.
- % (ReportTitle) - The name of the report.
- %(Image) - Allows you to browse to and attach an image to the report header.
- % (AcademicLicense) - Adds text string: **Licensed for Academic Use Only**.
- % (HomeUseLicense) - Adds text string: **Licensed for Home Use Only**.
- % (ActiveScenarioLabel) - The label of the currently active scenario.

You can also select fonts, text sizes, and customize spacing, as well as change the default margins in the Default Margins tab.

Results Table Dialog Box

This dialog is accessed by right-clicking any element in the drawing pane and selecting the Results Table command. It displays a summary of a standard selection of results related to that element type.



Click the Report button to generate a preformatted report containing the data in the table.

Graphs

Use graphs to visualize your model or parts of your model, such as element properties or results. The model needs to be computed before you can create graphs. After you set up your elements and their properties, click the **Compute** button. 

After the model has been calculated, you can graph elements directly from the drawing view.

To graph a single element

Right-click an element in the drawing view and select the Graph command.

To graph a group of elements

1. Select a group of elements by drawing a selection box around them or by holding down the Ctrl key and then clicking a series of elements.
2. Right-click one of the selected elements and select the Graph command.

To Graph the elements contained in a selection set

1. Click the View menu and choose the Selection Sets command.
2. In the Selection Sets dialog, highlight the selection set to be graphed and click the Select In Drawing button.
3. Right-click one of the selected elements and select the Graph command.

Graph Manager

The Graph manager contains any graph you have created and saved in the current session or in a previous session. Graphs listed in the Graph manager retain any customizations you have applied. You can graph computed values, such as flow and velocity.

To use the Graph Manager

1. Compute your model and resolve any errors.
2. Open the Graph manager, click **View** > **Graphs**.
3. To Create a Graph select the elements that you want included from the drawing. Once you have selected the element you can either Right-click an element and select **Graph** or select the type of graph from the New button menu.

4. The Graph manager contains a toolbar with the following icons:

New 	Select a line-series, bar chart, or scatter plot graph using the currently selected elements in your model. If no elements are selected, you are prompted to select one or more elements to graph.
Delete 	Deletes the currently highlighted graph. You can hold down the Ctrl key while clicking on items in the list to select multiple entries at once.
Rename 	Renames the currently highlighted graph.
View 	Opens the Graph dialog box to view the currently highlighted graph.
Add to Graph 	Opens the Select toolbar, allowing you to add or remove elements to the currently highlighted graph.
Help 	Displays online help for the Graph manager.

5. Bentley WaterCAD V8*i* assumes initial flow—flow at time 0—in all networks to be 0; thus, graphs of flow begin at 0 for time 0.
6. If needed, click **Chart Settings** to change the display of the graph.

   Tip: If you want your graph to display over more time (for example, it displays a 24-hour time period and you want to display a 72-hour period), click Analysis > Calculation Options and change Total Simulation Time in the Property Editor.

7. After you create a graph, it is available in the Graph manager. You can select it by double-clicking it. Also, you can right-click a graph listed in Graph manager to:
   - Delete it

- Rename the graph's label
- Open it, by selecting **Properties**.

Note: Graphs are not saved in Graph manager after you close the program.

Add to Graph Dialog Box

This dialog appears after you initiate an Add to Graph command and allows you to choose a previously defined graph to add the element to.

Select the desired graph from the Add to: menu, then click OK. To cancel the command, click the Cancel button.

Printing a Graph

To print a graph click  , or click **Print Preview**  to view your graph then click print.

Working with Graph Data: Viewing and Copying

You can view the data that your graphs are based on. To view your data, create a graph, then, after the Graph dialog box opens, click the **Data** tab.

You can copy this data to the Windows clipboard for use in other applications, such as word-processing software.

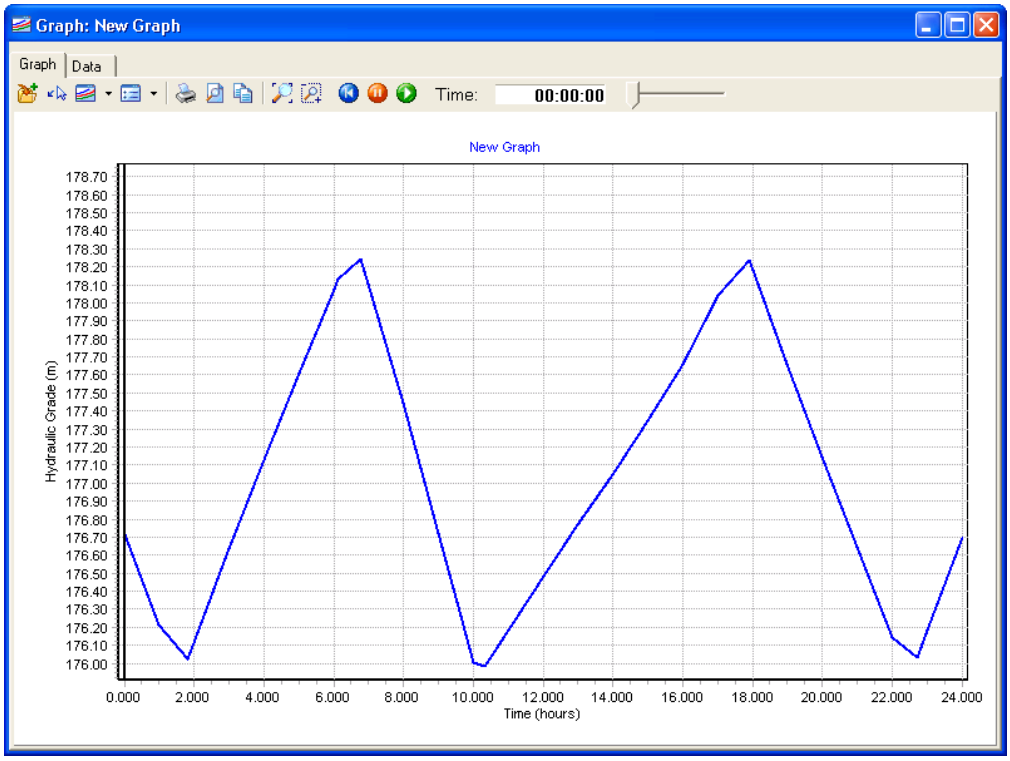
To copy this data

1. Click in the top-most cell of the left-most column to select the entire table, click a column heading to select an entire column, or click a row heading to select an entire row.
2. Press <**Ctrl+C**> to copy the selected data to the clipboard.
3. As needed, press <**Ctrl+V**> to paste the data as tab-delimited text into other software.

To print out the data for a graph, copy and paste it into another application, such as word-processing software or Notepad, and print the pasted content.

Graph Dialog Box

Using the Graph dialog box you can view and modify graph settings. After you create a graph, you view it in the Graph dialog box.



The following controls are available:

Graph Tab

Add to Graph Manager



Saves the Graph to the Graph manager. When you click this button, the graph options (i.e., attributes to graph for a specific scenario) and the graph settings (i.e., line color, font size) are saved with the graph. If you want to view a different set of data (for example, a different scenario), you must change the scenario in the Graph Series Options dialog box. Graphs that you add to the Graph manager are saved when you save your model, so that you can use the graph after you close and reopen Bentley WaterCAD V8*i* .

Add to Graph



Adds new elements to the graph using the current graph series options. Clicking this button returns you to the drawing view and opens a Select toolbar, allowing you to change which elements are included in the graph.

Graph Series Options



Selects Graph Series Options to control what the graph displays.
Select Observed Data to
display user-defined attribute values alongside calculated results in the graph display dialog.

Chart Settings



Opens a submenu containing the following commands:

- Chart Options— Change graph display settings.
- Detailed Labels—Click to view more information on the graph.
- Legend-Click to view a legend for the graph.
- Save Chart Options As Default—Saves the current chart options as the new default settings for future graphs.
- Apply Default Chart Options—Applies the default chart options to the current graph.
- Restore Factory Default Chart Options—Deletes the currently saved default chart options and replaces them with the default settings that were originally installed with WaterCAD V8*i*.

Print



Prints the current view in the graph display pane.

Print Preview



Opens the Print Preview dialog box to view the current image and change the print information.

Copy



Copies the current view in the graph display pane to the Windows Clipboard.

Zoom Extents



Zooms out so that the entire graph is displayed.

Zoom



Zooms in on a section of the graph. When the tool is toggled on, you can zoom in on any area of the graph by clicking on the chart to the left of the area to be zoomed, holding the mouse button, then dragging the mouse to the right (or the opposite extent of the area to be magnified) and releasing the mouse button when the area to be zoomed has been defined.
To zoom back out, click and hold the mouse button, drag the mouse in the opposite direction (right to left), and release the mouse button.

Time (VCR) Controls

Evaluate plots over time.



- If you click Go to start, the Time resets to zero and the vertical line that marks time resets to the left edge of the Graph display.



- If you click Pause, the vertical line that moves across the graph to mark time pauses, as does the Time field.



- If you click Play, a vertical line moves across the graph and the Time field increments.

The following controls are also available:

- **Time**—Displays the time location of the vertical black bar in the graph display. This is a read-only field; to set a specific time, use the slider button.
- **Slider**—Set a specific time for the graph. A vertical line moves in the graph display and intersects your plots to show the value of the plot at a specific time. Use the slider to set a specific time value.

Graph Display Pane

Displays the graph.

Data Tab

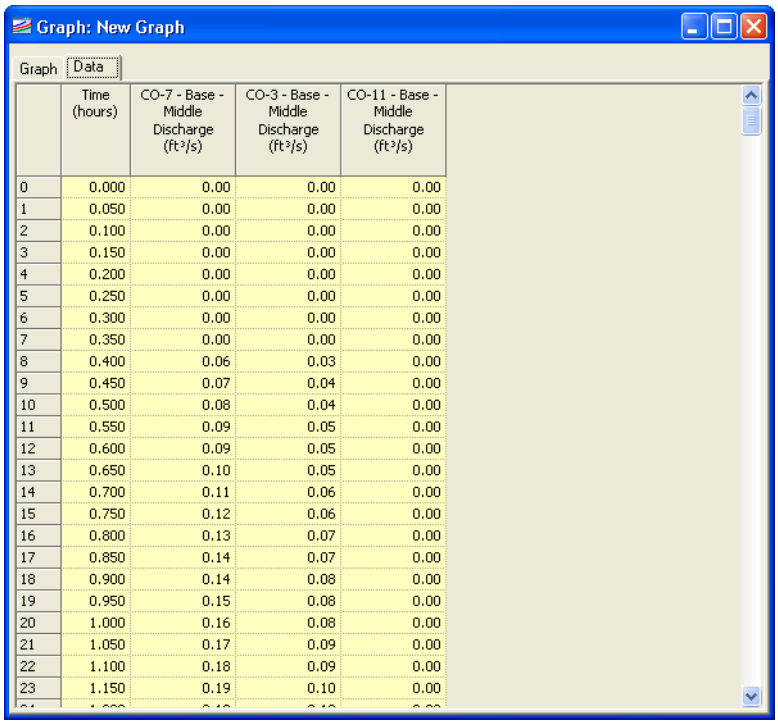
Data Table

The Data tab displays the data that make up the graphs. If there is more than one item plotted, the data for each plot is provided.

You can copy and paste the data from this tab to the clipboard for use in other applications, such as Microsoft Excel.

To select an entire column or row, click the column or row heading. To select the entire contents of the Data tab, click the heading cell in the top-left corner of the tab. Use <Ctrl+C> and <Ctrl+V> to paste your data. The column and row headings are not copied.

The Data tab is shown below.



	Time (hours)	CO-7 - Base - Middle Discharge (ft³/s)	CO-3 - Base - Middle Discharge (ft³/s)	CO-11 - Base - Middle Discharge (ft³/s)
0	0.000	0.00	0.00	0.00
1	0.050	0.00	0.00	0.00
2	0.100	0.00	0.00	0.00
3	0.150	0.00	0.00	0.00
4	0.200	0.00	0.00	0.00
5	0.250	0.00	0.00	0.00
6	0.300	0.00	0.00	0.00
7	0.350	0.00	0.00	0.00
8	0.400	0.06	0.03	0.00
9	0.450	0.07	0.04	0.00
10	0.500	0.08	0.04	0.00
11	0.550	0.09	0.05	0.00
12	0.600	0.09	0.05	0.00
13	0.650	0.10	0.05	0.00
14	0.700	0.11	0.06	0.00
15	0.750	0.12	0.06	0.00
16	0.800	0.13	0.07	0.00
17	0.850	0.14	0.07	0.00
18	0.900	0.14	0.08	0.00
19	0.950	0.15	0.08	0.00
20	1.000	0.16	0.08	0.00
21	1.050	0.17	0.09	0.00
22	1.100	0.18	0.09	0.00
23	1.150	0.19	0.10	0.00

Saving Graph Settings

You can use the **Chart Options > Save Chart Options as Default** command to save the current graph settings as the template that will be applied to new graphs in this and future projects.

Graph settings are saved to the **DefaultGraphOptions.xml.bin** file and is stored in the in **C:\Users\<user>\AppData\Roaming\Bentley\WaterGEMS\8** directory (in Windows Vista and Windows 7).

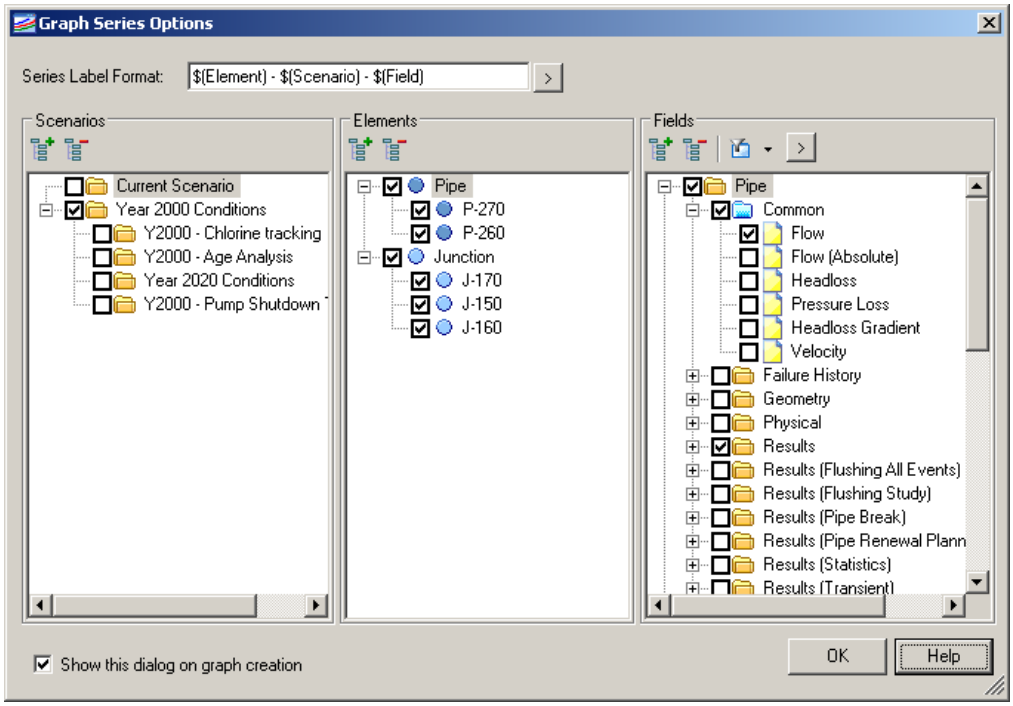
For Windows XP the location is **C:\Documents and Settings\User.Name\Application Data\Bentley\WaterGEMS\8**.

Note: These settings are on a per-user basis.

To reset the options to the factory default , click **Chart Options > Restore Factory Default Chart Options**, then use the **Chart Options > Save Chart Options as Default** command.

Graph Series Options Dialog Box

The Graph Series Options dialog box allows you to adjust the display settings for the graph. You can define the legend labels, the scenario (or scenarios), and the attribute (or attributes) that are displayed in the graph.



The Series Label Format field allows you to define how the series will be labeled in the legend of the graph. Clicking the [>] button allows you to choose from predefined variables such as Field name and Element label.

The Scenarios pane lists all of the available scenarios. Check the box next to a scenario to display the data for that scenario in the graph. The Expand All button opens all of the folders so that all scenarios are visible; the Collapse button closes the folders.

The Elements pane lists all of the elements that will be displayed in the graph. The Expand All button expands the list tree so that all elements are visible; the Collapse button collapses the tree.

The Fields pane lists all of the available input and output fields. Check the box next to a field to display the data for that field type in the graph. The Expand All button opens all of the folders so that all fields are visible; the Collapse button closes the folders. The Filter by Field Type button allows you to display only Input or Output fields in the list. Clicking the [>] button opens a submenu that contains all of the available fields grouped categorically.

The Show this dialog on profile creation check box is enabled by default; uncheck this box to skip this dialog when a new profile is created.

For any given element, the most commonly used fields are displayed underneath a Common folder, colored blue (see screenshot above). To graph all of these attributes you can simply check the Common box.

Observed Data Dialog Box

Use this feature to display user-supplied time variant data values alongside calculated results in the graph display dialog. Model competency can sometimes be determined by a quick side by side visual comparison of calculated results with those observed and collection in the field.

- **Get familiar with your data** - If you obtained your observed data from an outside source, you should take the time to get acquainted with it. Be sure to identify units of time and measurement for the data. Be sure to identify what the data points represent in the model; this helps in naming your line or bar series as it will appear in the graph.
- **Preparing your data** - Typically, observed data can be organized as a collection of points in a table. In this case, the time series data can simply be copied to the clipboard directly from the source and pasted right into the observed data input table. Ensure that your collection of data points is complete. That is, every value must have an associated time value. Oftentimes data points are stored in tab or comma delimited text files; these two import options are available as well. See the Sample Observed Data Source topic for an example of the observed data source file format.
- **Specifying the characteristics of your data** - The following charecteristics must be defined:
  - **Time from Start** - An offset of the start time for an EPS scenario.
  - **Y Dimension** - Unit class for the observed data point(s).
  - **Numeric Formatter** - Group of units that correspond to the selected value.
  - **Y Unit** - A preview of the current displayed unit for the selected format.

  Note: Go to Tools > Options > Units for a complete list of formats.

  Caution: Observed data can only be saved if the graph is saved.

To create Observed Data

1. Click New  .
2. Set hours, dimension, and formatter.

3. Add hours and Y information (or import a .txt or .csv file ).

   Note: Note that the when importing data, the times in the data file must be valid time-of-day values, like 9:00 or 23:00. They cannot span multiple days. Therefore values greater than 24 hours, like 25:00, are invalid.

4. Click Graph  to view the Observed data.
5. Click Close.

Sample Observed Data Source

Below is an example of an Observed Data source for import and graph comparison. The following table contains a flow meter data collection retreived in the field for a given pipe. We will bring this observed data into the model for a quick visual inspection against our model's calculated pipe flows.

Table 15-1: Observed Flow Meter Data (Time in Hours)

Time (hrs)	Flow (gpm)
0.00	125
0.60	120
3.00	110
9.00	130
13.75	100
18.20	125
21.85	110

With data tabulated as in the table above, we could simply copy and paste these rows directly into the table in the Observed Data dialog. However if we had too many points to manage, natively exporting our data to a comma delimited text file may be a better import option. Text file import is also a better option when our time values are not formatted in units of time such as hours, as in the table below.

Table 15-2: Observed Flow Meter Data (24-Hr Clock)

Time (24-hr clock)	Flow (gpm)
00:00	125
00:36	120
03:00	110
09:00	130
13:45	100
18:12	125
21:51	110

Below is a sample of what a comma-delimited (*.csv) file would look like:

```
0:00,125

0:36,120

3:00,110

9:00,130

13:45,100

18:12,125

21:51,110
```

Note: Database formats (such as MS Access) are preferable to simple spreadsheet data sources. The sample described above is intended only to illustrate the importance of using expected data formats.

To import the comma delimited data points:

1. Click the Import toolbar button from the Observed Data dialog.
2. Pick the source .csv file.
3. Choose the Time Format that applies, in this case, HH:mm:ss, and click OK.

Chart Options Dialog Box

Use the Chart Options dialog box to format a graph.

Note: Changes you make to graph settings are not retained for use with other graphs.

To open Chart Options dialog box:

1. Open your project and click **Compute**.
2. Select one or more elements, right-click, then select **Graph**.
3. Click the **Chart Settings** button.



Click one of the following links to learn more about Chart Options dialog box:

- Chart Options Dialog Box - Chart Tab on page 15-1298
- Chart Options Dialog Box - Series Tab on page 15-1323
- Chart Options Dialog Box - Tools Tab on page 15-1331
- Chart Options Dialog Box - Export Tab on page 15-1332
- Chart Options Dialog Box - Print Tab on page 15-1334
- Border Editor Dialog Box on page 15-1335
- Gradient Editor Dialog Box on page 15-1336
- Color Editor Dialog Box on page 15-1337
- Color Dialog Box on page 15-1337
- Hatch Brush Editor Dialog Box on page 15-1338
- Pointer Dialog Box on page 15-1341
- Change Series Title Dialog Box on page 15-1342

- Chart Tools Gallery Dialog Box on page 15-1342
- TeeChart Gallery Dialog Box on page 15-1354

Chart Options Dialog Box - Chart Tab

The Chart tab lets you define overall chart display parameters. This tab is subdivided into second-level sub-tabs:

- Series Tab
- Panel Tab
- Axes Tab
- General Tab
- Titles Tab
- Walls Tab
- Paging Tab
- Legend Tab
- 3D Tab

Series Tab

Use the Series tab to display the series that are associated with the current graph. To show a series, select the check box next to the series' name. To hide a series, clear its check box. The Series tab contains the following controls:

Up/Down arrows	Lets you select the printer you want to use.
Add	Adds a new series to the current graph. The TeeChart Gallery opens, see TeeChart Gallery Dialog Box.
Delete	Lets you remove the currently selected series.
Title	Lets you rename the currently selected series.
Clone	Creates a duplicate of the currently selected series.
Change	Lets you edit the currently selected series. The TeeChart Gallery opens, see TeeChart Gallery Dialog Box.

Panel Tab

Use the Panel tab to set how your graph appears in the Graph dialog box. The Panel tab includes the following sub-tabs:

Borders Tab

Use the Borders tab to set up a border around your graph. The Borders tab contains the following controls:

Border	Lets you set the border of the graph. The Border Editor opens, see Border Editor Dialog Box.
Bevel Outer	Lets you set a raised or lowered bevel effect, or no bevel effect, for the outside of the chart border.
Color	Lets you set the color for the bevel effect that you use; inner and outer bevels can use different color values.
Bevel Inner	Lets you set a raised or lowered bevel effect, or no bevel effect, for the inside of the chart border.
Size	Lets you set a thickness for the bevel effect that you use; inner and outer bevels use the same size value.

Background Tab

Use the Background tab to set a color or image background for your graph. The Background tab contains the following controls:

Color	Lets you set a color for the background of your graph. The Color Editor opens, see Color Editor Dialog Box.
Pattern	Lets you set a pattern for the background of your graph. The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box.
Transparent	Makes the background of the graph transparent.

Background Image

Lets you set an existing image as the background of the graph. Click **Browse**, then select the image (including .bmp, .tif, .jpg, .png,. and .gif). After you have set a background image, you can remove the image from the graph by clicking **Clear**.
You can control the Style of the background image:

- **Stretch**—Resizes the background image to fill the entire background of the graph.
- **Tile**—Repeats the background image as many times as needed to fill the entire background of the graph.
- **Center**—Puts the background image in the horizontal and vertical center of the graph.
- **Normal**—Puts the background image in the top-left corner of the graph.

Gradient Tab

Use the Gradient tab to create a gradient color background for your graph. The Gradient tab contains the following subtabs and controls:

Format Tab

Visible

Determines whether a gradient displays or not. Select this check box to display a gradient you have set up, clear this check box to hide the gradient.

Direction

Sets the direction of the gradient. Vertical causes the gradient to display from top to bottom, Horizontal displays a gradient from right to left, and Backward/Forward diagonal display gradients from the left and right bottom corners to the opposite corner.

Angle

Lets you customize the direction of the gradient beyond the Direction selections.

Colors Tab

Start

Lets you set the starting color for your gradient. Opens the Color Editor dialog box.

Middle	Lets you select a middle color for your gradient. The Color Editor opens. Select the **No Middle Color** check box if you want a two-color gradient. Opens the Color Editor dialog box.
End	Lets you select the final color for your gradient. Opens the Color Editor dialog box.
Gamma Correction	Lets you control the brightness with which the background displays to your screen; select or clear this check box to change the brightness of the background on-screen. This does not affect printed output.
Transparency	Lets you set transparency for your gradient, where 100 is completely transparent and 0 is completely opaque.

Options Tab

Sigma	Lets you set the location on the chart background of the gradient's end color.
Sigma Focus	Lets you use the options controls. Select this check box to use the controls in the Options tab.
Sigma Scale	Lets you control how much of the gradient's end color is used by the gradient background.

Shadow Tab

Use the Shadow tab to create a shadow for your graph. The Shadow tab contains the following controls:

Visible	Lets you display a shadow for your graph. Select this check box to display the shadow, clear this check box to turn off the shadow effect.
Size	Set the size of the shadow by increasing or decreasing the numbers for Horizontal and/or Vertical Size.
Color	Lets you set a color for the shadow of your graph. You might set this to gray but can set it to any other color.

Pattern	Lets you set a pattern for the shadow of your graph. The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box.
Transparency	Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Axes Tab

Use the Axes tab set how your axes display. It includes the following controls and subtabs:

Visible	When checked, displays all of your graph's axes; clear it to hide all of the graph's axes.
Behind	When checked, displays all of your graph's axes behind the series display; clear it to display the axes in front of the series display.
Axes	Select the axis you want to edit. The Scales, Labels, Ticks, Title, Minor, and Position tabs and their controls pertain only to the selected axis.

Caution: Do not delete the axes called Custom 0 and Custom 1, as these are reserved axes that are needed by Bentley WaterCAD V8*i* .

Scales Tab

Use the Scales tab to define your axes scales. The Scales tab contains the following controls:

Automatic	Lets you automatically or manually set the minimum and maximum axis values. Select this check box if you want TeeChart to automatically set both minimum and maximum, or clear this check box if you want to manually set either or both.
Visible	Displays the axis if selected, hides the axis if cleared.

Inverted — Reverses the order in which the axis scale increments. If the minimum value is at the origin, then selecting Inverted puts the maximum value at the origin.

Change — Lets you change the increment of the axis.

Increment — Displays the increment value you set for the axis.

Logarithmic — Lets you use a logarithmic scale for the axis.

Log Base — If you select a logarithmic scale, set the base you want to use in the text box.

Minimum Tab

Auto — Lets you automatically or manually set the minimum axis value.

Change — Lets you enter a value for the axis minimum.

Offset — Lets you adjust the axis scale to change the location of the minimum or maximum axis value with respect to the origin.

Maximum Tab

Auto — Lets you automatically or manually set the maximum axis value.

Change — Lets you enter a value for the axis maximum.

Offset — Lets you adjust the axis scale to change the location of the minimum or maximum axis value with respect to the origin.

Labels Tab

Use the Labels tab to define your axes text. The Labels tab contains the following subtabs and controls:

Style Tab

Visible — Lets you show or hide the axis text.

Multi-line — Lets you split labels or values into more than one line if the text contains a space. Select this check box to enable multi-line text.

Round first — Controls whether axis labels are automatically rounded to the nearest magnitude.

Label on axis — Controls whether Labels just at Axis Minimum and Maximum positions are shown. This applies only if the maximum value for the axis matches the label for extreme value on the chart.

Size — Determines distance between the margin of the graph and the placement of the labels.

Angle — Sets the angle of the axis labels. In addition to using the up and down arrows to set the angle in 90° increments, you can type an angle you want to use.

Min. Separation % — Sets the minimum distance between axis labels.

Style — Lets you set the label style.

- **Auto**—Lets TeeChart automatically set the label style.
- **Value**—Sets axis labeling based on minimum and maximum axis values.
- **Text**—Uses text for labels. Since Bentley WaterCAD V8*i* uses numeric values, this is not implemented; don't use it.
- **None**—Turns off axis labels.
- **Mark**—Uses SeriesMarks style for labels. Since Bentley WaterCAD V8*i* uses numeric values, this is not implemented; don't use it.

Format Tab

Exponential — Displays the axis label using an exponent, if appropriate.

Values Format — Lets you set the numbering format for the axis labels.

Default Alignment — Lets you select and clear the default TeeChart alignment for the right or left axes only.

Text Tab

Font — Lets you set the font properties for axis labels. This opens the Windows Font dialog box.

Color — Lets you select the color for the axis label font. Double-click the colored square between Font and Fill to open the Color Editor dialog box (see Color Editor Dialog Box).

Fill — Lets you set a pattern the axis label font. The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box.

Shadow —Lets you set a shadow for the axis labels.

- **Visible**—Lets you display a shadow for the axis labels. Select this check box to display the axis label shadow.
- **Size**—Lets you set the location of the shadow. Use larger numbers to offset the shadow by a large amount.
- **Color**—Lets you set a color for the shadow. You might set this to gray but can set it to any other color. The Color Editor opens.
- **Pattern**—Lets you set a pattern for the shadow. The Hatch Brush Editor opens.
- **Transparency**—Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Ticks Tab

Use the Ticks tab to define the major ticks and their grid lines. The Ticks tab contains the following controls:

Axis — Lets you set the properties of the selected axis. Opens the Border Editor dialog box.

Grid — Lets you set the properties of the graph's grid lines that intersect the selected axis. Opens the Border Editor dialog box.

Ticks	Lets you set the properties of the tick marks that are next to the labels on the label-side of the selected axis. Opens the Border Editor dialog box.
Len	Sets the length of the Ticks or Inner ticks.
Inner	Lets you set the properties of the tick marks that are next to the labels on the graph-side of the selected axis. Opens the Border Editor dialog box.
Centered	Lets you align between the grid labels the graph's grid lines that intersect the selected axis.
At Labels Only	Sets the axis ticks and axis grid to be drawn at labels only. Otherwise, they are drawn at all axis increment positions.

Title Tab

Use the Title tab to set the axis titles. The Title tab contains the following subtabs and controls:

Style Tab

Title	Lets you type a new axis title.
Angle	Sets the angle of the axis title. In addition to using the up and down arrows to set the angle in 90° increments, you can type an angle you want to use.
Size	Determines distance between the margin of the graph and the placement of the labels.
Visible	Check box that lets you display or hide the axis title.

Text Tab

Font	Lets you set the font properties for axis title. This opens the Windows Font dialog box.
Color	Lets you select the color for the axis title font. Double-click the colored square between Font and Fill to open the Color Editor dialog box (see Color Editor Dialog Box).

Fill	Lets you set a pattern the axis title font. The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box
Shadow	Lets you set a shadow for the axis title. • **Visible**—Lets you display a shadow for the axis title. Select this check box to display the axis label shadow. • **Size**—Lets you set the location of the shadow. Use larger numbers to offset the shadow by a large amount. • **Color**—Lets you set a color for the shadow. You might set this to gray but can set it to any other color. The Color Editor opens. • **Pattern**—Lets you set a pattern for the shadow. The Hatch Brush Editor opens. • **Transparency**—Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Minor Tab

Use the Minor tab to define those graph ticks that are neither major ticks. The Minor tab contains the following controls and tabs:

Ticks	Lets you set the properties of the minor tick marks. The Border Editor opens, see Border Editor Dialog Box.
Length	Sets the length of the minor tick marks.
Grid	Lets you set the properties of grid lines that align with the minor ticks. The Border Editor opens, see Border Editor Dialog Box.
Count	Sets the number of minor tick marks.

Position Tab

Use the Position tab to set the axes position for your graph. The Position tab contains the following controls:

Position %	Sets the position of the axis on the graph in pixels or as a percentage of the graph's dimensions.

Start %	Sets the start of the axis as percentage of width (horizontal axis) and height (vertical axis) of the graph. The original axis scale is fitted to new axis height/width.
End %	Sets the end of the axis as percentage of width (horizontal axis) and height (vertical axis) of the graph. The original axis scale is fitted to new axis height/width.
Units	Lets you select pixels or percentage as the unit for the axis position.
Z %	Sets the Z dimension as a percentage of the graph's dimensions. This is unused by Bentley WaterCAD V8*i* .

General Tab

Use the General tab to preview a graph before you print it and set up scrolling and zooming for a graph. It includes the following controls:

Print Preview	Lets you see the current view of the document as it will be printed and lets you define the print settings, such as selecting a printer to use. Opens the Print Preview dialog box.
Margins	Lets you specify margins for your graph. There are four boxes, each corresponding with the top, bottom, left, and right margins, into which you enter a value that you want to use for a margin.
Units	Lets you set pixels or percentage as the units for your margins. Percentage is a percentage of the original graph size.
Cursor	Lets you specify what your cursor looks like. Select a cursor type from the drop-down list, then click **Close** to close the TeeChart editor, and the new cursor style displays when the cursor is over the graph.

Zoom Tab

Use the Zoom tab to set up zooming on, magnifying, and reducing the display of a graph. The Zoom tab contains the following controls:

Allow	Lets you magnify the graph by clicking and dragging with the mouse.
Animated	Lets you set a stepped series of zooms.
Steps	Lets you set the number of steps used for successive zooms if you selected the Animated check box.
Pen	Lets you set the thickness of the border for the zoom window that surrounds the magnified area when you click and drag. The Border Editor opens, see Border Editor Dialog Box.
Pattern	The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box.
Minimum pixels	Lets you set the number of pixels that you have to click and drag before the zoom feature is activated.
Direction	Lets you zoom in the vertical or horizontal planes only, as well as both planes.
Mouse Button	Lets you set the mouse button that you use to click and drag when activating the zoom feature.

Scroll Tab

Use the Scroll tab to set up scrolling and panning across a graph. The Scroll tab contains the following controls:

Allow Scroll	Lets you scroll and pan over the graph. Select this check box to turn on scrolling, clear the check box to turn it off.
Mouse Button	Lets you set the mouse button that you click to use the scroll feature.

Titles Tab

The Titles tab lets you define titles to use for your graph. It includes the following controls and tabs:

Title	Lets you set the location of the titles you want to use. The Titles sub tabs apply to the Title that is currently selected in the Title drop-down list.

Style Tab

Use the Style tab to display and create a selected title. Type the text of the title in the text box on the Style tab. The Style tab contains the following controls:

Visible	Lets you display the selected title.
Adjust Frame	Lets you wrap the frame behind the selected title to the size of the title text. Each title can have a frame behind it (see Format Tab). By default, this frame is transparent. If you turn off transparency to see the frame, the frame can be sized to the width of the graph or set to snap to the width of the title text. Select the **Adjust Frame** check box to set the width of the frame to the width of the title text; clear this check box to set the width of the frame to the width of the graph.
Alignment	Lets you set the alignment of the selected title.

Position Tab

Use the Position tab to set the placement of the selected title. The Position tab contains the following controls:

Custom	Lets you set a custom position for the selected title. Select this check box to set a custom position.
Left/Top	Lets you set the location of the selected title relative to the left and top of the graph. If you select the Custom check box, use these settings to position the selected title.

Format Tab

Use the Format tab to set and format a background shape behind the selected title. The Format tab contains the following controls:

Color	Lets you set a color for the fill of the shape you create behind the selected title. The Color Editor opens, see Color Editor Dialog Box.
Frame	Lets you define the outline of the shape you create behind the selected title. The Border Editor opens, see Border Editor Dialog Box.
Pattern	Lets you set a pattern for the fill of the shape you create behind the selected title. The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box.
Round Frame	Lets you round the corners of the rectangular shape you create behind the selected title. Select this check box to round the corners of the shape.
Transparent	Lets you set the fill of the shape you create behind the selected title as transparent. If the shape is completely transparent, you cannot see it, so clear this check box if you cannot see a shape that you expect to see.
Transparency	Lets you set transparency for the shape, where 100 is completely transparent and 0 is completely opaque.

Text Tab

Use the Text tab to format the text used in the selected title. The Text tab contains the following controls:

Font	Lets you set the font properties for the text. This opens the Windows Font dialog box.
Color	Lets you select the color for the text. Double-click the colored square between Font and Fill to open the Color Editor dialog box (see Color Editor Dialog Box).
Fill	Lets you set a pattern for the text. The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box.

Shadow — Lets you set a shadow for the text.

- **Visible**—Lets you display a shadow for the text. Select this check box to display the axis label shadow.
- **Size**—Lets you set the location of the shadow. Use larger numbers to offset the shadow by a large amount.
- **Color**—Lets you set a color for the shadow. You might set this to gray but can set it to any other color. The Color Editor opens.
- **Pattern**—Lets you set a pattern for the shadow. The Hatch Brush Editor opens.
- **Transparency**—Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Gradient Tab

Note: To use the Gradient tab, clear the Transparent check box in the Chart > Titles > Format tab.

Use the Gradient tab to create a gradient color background for your axis title. The Gradient tab contains the following controls:

Format Tab

Visible — Sets whether a gradient displays or not. Select this check box to display a gradient you have set up, clear this check box to hide the gradient.

Direction — Sets the direction of the gradient. Vertical causes the gradient to display from top to bottom, Horizontal displays a gradient from right to left, and Backward/Forward diagonal display gradients from the left and right bottom corners to the opposite corner.

Angle — Lets you customize the direction of the gradient beyond the Direction selections.

Colors Tab

Start — Lets you set the starting color for your gradient.

Middle	Lets you select a middle color for your gradient. The Color Editor opens. Select the **No Middle Color** check box if you want a two-color gradient.
End	Lets you select the final color for your gradient.
Gamma Correction	Lets you control the brightness with which the background displays to your screen; select or clear this check box to change the brightness of the background on-screen. This does not affect printed output.
Transparency	Lets you set transparency for your gradient, where 100 is completely transparent and 0 is completely opaque.

Options Tab

Sigma	Lets you use the options controls. Select this check box to use the controls in the Options tab.
Sigma Focus	Lets you set the location on the chart background of the gradient's end color.
Sigma Scale	Lets you control how much of the gradient's end color is used by the gradient background.

Shadow Tab

Use the Shadow tab to create a shadow for the background for the selected title. The Shadow tab contains the following controls:

Visible	Lets you display a shadow. Select this check box to display the shadow, clear this check box to turn off the shadow effect.
Size	Set the size of the shadow by increasing or decreasing the numbers for Horizontal and/or Vertical Size.
Color	Lets you set a color for the shadow. You might set this to gray but can set it to any other color. The Color Editor opens, see Color Editor Dialog Box.

Pattern	Lets you set a pattern for the shadow. The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box.
Transparency	Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Bevels Tab

Note: To use the Gradient tab, clear the Transparent check box in the Chart > Titles > Format tab.

Use the Bevels tab to create rounded effects for the background for the selected title. The Bevels tab contains the following controls:

Bevel Outer	Lets you set a raised or lowered bevel effect, or no bevel effect, for the background for the selected title.
Color	Lets you set the color for the bevel effect that you use; inner and outer bevels can use different color values.
Bevel Inner	Lets you set a raised or lowered bevel effect, or no bevel effect, for the inside of the background for the selected title.
Size	Lets you set a thickness for the bevel effect that you use; inner and outer bevels use the same size value.

Walls Tab

Use the Walls tab to set and format the edges of your graph. The Walls tab contains the following subtabs:

Left/Right/Back/Bottom Tabs

Use the Left, Right, Back, and Bottom tabs to select the walls that you want to edit. You might have to turn off the axes lines to see the effects (see Axes Tab on page 15-1302) for the back wall and turn on 3D display to see the effects for the left, right, and bottom walls (see 3D Tab on page 15-1322).

The Left, Right, Back, and Bottom tabs contain the following controls:

Color	The Color Editor opens, see Color Editor Dialog Box.
Border	The Border Editor opens, see Border Editor Dialog Box.
Pattern	The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box.
Gradient	Lets you set a color gradient for your walls. The Gradient Editor opens, see Gradient Editor Dialog Box.
Visible	Lets you display the walls you set up.
Dark 3D	Lets you automatically darken the depth dimension for visual effect. Select a Size 3D larger than 0 to enable this check box.
Size 3D	Lets you increase the size of the wall in the direction perpendicular to it's length (the graph resizes automatically as a result).
Transparent	Lets you set transparency for your background, where 100 is completely transparent and 0 is completely opaque.

Paging Tab

Use the Paging tab to display your graph over several pages. The Paging tab contains the following controls:

Points per Page	Lets you scale the graph to fit on one or many pages. Set the number of points you want to display on a single page of the graph, up to a maximum of 100.
Scale Last Page	Scales the end of the graph to fit the last page.
Current Page Legend	Shows only the current page items when the chart is divided into multiple pages.
Show Page Number	Lets you display the current page number on the graph.

Arrows	Lets you navigate through a multi-page graph. Click the single arrows to navigate one page at a time. Click the double arrows to navigate directly to the last or first pages of the graph.

Legend Tab

Use the Legend tab to display and format a legend for your graph. The Legend tab includes the following controls:

Style Tab

Use the Style tab to set up and display a legend for your graph. The Style tab contains the following controls:

Visible	Lets you show or hide the legend for your graph.
Inverted	Lets you draw legend items in the reverse direction. Legend strings are displayed starting at top for Left and Right Alignment and starting at left for Top and Bottom Legend orientations.
Check boxes	Activates/deactivates check boxes associated with each series in the Legend. When these boxes are unchecked in the legend, the associated series are invisible.
Font Series Color	Sets text in the legend to the same color as the graph element to which it applies.
Legend Style	Lets you select what appears in the legend.
Text Style	Lets you select how the text in the legend is aligned and what data it contains.
Vert. Spacing	Controls the space between rows in the legend.
Dividing Lines	Lets you use and define lines that separate columns in the legend. The Border Editor opens, see Border Editor Dialog Box.

Position Tab

Use the Position tab to control the placement of the legend. The Position tab contains the following controls:

Position	Lets you place the legend on the left, top, right, or bottom of the chart.
Resize Chart	Lets you resize your graph to accommodate the legend. If you do not select this check box, the graph and legend might overlap.
Margin	Lets you set the amount of space between the graph and the legend.
Position Offset %	Determines the vertical size of the Legend. Lower values place the Legend higher up in the display
Custom	Lets you use the Left and Top settings to control the placement of the legend.
Left/Top	Lets you enter a value for custom placement of the legend.

Symbols Tab

Use the Symbols tab to add to the legend symbols that represent the series in the graph. The Symbols tab contains the following controls:

Visible	Lets you display the series symbol next to the text in the legend.
Width	Lets you resize the symbol that displays in the legend. You must clear **Squared** to use this control.
Width Units	Lets you set the units that are used to size the width of the symbol.
Default border	Lets you use the default TeeChart format for the symbol. If you clear this check box, you can set a custom border using the Border button.
Border	Lets you set a custom border for the symbols. You must clear Default Border to use this option. The Border Editor opens, see Border Editor Dialog Box.
Position	Lets you put the symbol to the left or right of its text.

Continuous	Lets you attach or detach legend symbols. If you select this check box, the color rectangles of the different items are attached to each other with no vertical spacing. If you clear this check box, the legend symbols are drawn as separate rectangles.
Squared	Lets you override the width of the symbol, so you can make the symbol square shaped.

Format Tab

Use the Format tab to set and format the box that contains the legend. The Format tab contains the following controls:

Color	Lets you set a color for the fill of the legend's box. The Color Editor opens, see Color Editor Dialog Box.
Frame	Lets you define the outline of the legend's box. The Border Editor opens, see Border Editor Dialog Box.
Pattern	Lets you set a pattern for the fill of the legend's box. The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box.
Round Frame	Lets you round the corners of the legend's box. Select this check box to round the corners of the shape.
Transparent	Lets you set the fill of the legend's box as transparent. If the shape is completely transparent, you cannot see it, so clear this check box if you cannot see a shape that you expect to see.
Transparency	Lets you set transparency for the legend's box, where 100 is completely transparent and 0 is completely opaque.

Text Tab

Use the Text tab to format the text used in the legend. The Text tab contains the following controls:

Font — Lets you set the font properties for the text. This opens the Windows Font dialog box.

Color — Lets you select the color for the text. Double-click the colored square between Font and Fill to open the Color Editor dialog box (see Color Editor Dialog Box).

Fill — Lets you set a pattern for the text. The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box.

Shadow — Lets you set a shadow for the text.

- **Visible**—Lets you display a shadow for the text. Select this check box to display the axis label shadow.
- **Size**—Lets you set the location of the shadow. Use larger numbers to offset the shadow by a large amount.
- **Color**—Lets you set a color for the shadow. You might set this to gray but can set it to any other color. The Color Editor opens.
- **Pattern**—Lets you set a pattern for the shadow. The Hatch Brush Editor opens.
- **Transparency**—Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Gradient Tab

Use the Gradient tab to create a gradient color background for your legend. The Gradient tab contains the following controls:

Format Tab

Visible — Sets whether a gradient displays or not. Select this check box to display a gradient you have set up, clear this check box to hide the gradient.

Direction — Sets the direction of the gradient. Vertical causes the gradient to display from top to bottom, Horizontal displays a gradient from right to left, and Backward/Forward diagonal display gradients from the left and right bottom corners to the opposite corner.

Angle — Lets you customize the direction of the gradient beyond the Direction selections.

Colors Tab

Start — Lets you set the starting color for your gradient.

Middle — Lets you select a middle color for your gradient. The Color Editor opens. Select the **No Middle Color** check box if you want a two-color gradient.

End — Lets you select the final color for your gradient.

Gamma Correction — Lets you control the brightness with which the background displays to your screen; select or clear this check box to change the brightness of the background on-screen. This does not affect printed output.

Transparency — Lets you set transparency for your gradient, where 100 is completely transparent and 0 is completely opaque.

Options Tab

Sigma — Lets you use the options controls. Select this check box to use the controls in the Options tab.

Sigma Focus — Lets you set the location on the chart background of the gradient's end color.

Sigma Scale — Lets you control how much of the gradient's end color is used by the gradient background.

Shadow Tab

Use the Shadow tab to create a shadow for the legend. The Shadow tab contains the following controls:

Visible	Lets you display a shadow. Select this check box to display the shadow, clear this check box to turn off the shadow effect.
Size	Set the size of the shadow by increasing or decreasing the numbers for Horizontal and/or Vertical Size.
Color	Lets you set a color for the shadow. You might set this to gray but can set it to any other color. The Color Editor opens, see Color Editor Dialog Box.
Pattern	Lets you set a pattern for the shadow. The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box.
Transparency	Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Bevels Tab

Use the Bevels tab to create a rounded effects for the legend. The Bevels tab contains the following controls:

Bevel Outer	Lets you set a raised or lowered bevel effect, or no bevel effect, for the background for the selected title.
Color	Lets you set the color for the bevel effect that you use; inner and outer bevels can use different color values.
Bevel Inner	Lets you set a raised or lowered bevel effect, or no bevel effect, for the inside of the background for the selected title.
Size	Lets you set a thickness for the bevel effect that you use; inner and outer bevels use the same size value.

3D Tab

Use the 3D tab to add a three-dimensional effect to your graph. The 3D tab contains the following controls:

3 Dimensions	Lets you display the chart in three dimensions. Select this check box to turn on three-dimensional display.
3D %	Lets you increase or decrease the three-dimensional effect. Set a larger percentage for more three-dimensional effect, or a smaller percentage for less effect.
Orthogonal	Lets you fix the graph in the two-dimensional work plane or, if you clear this check box, lets you use the Rotation and Elevation controls to rotate the graph freely.
Zoom Text	Lets you magnify and reduce the size of the text in a graph when using the zoom tool. clear this check box if you want text, such as labels, to remain the same size when you use the zoom tool.
Quality	Lets you select how the graph displays as you manipulate and zoom on it.
Clip Points	Trims the view of a series to the walls of your graph's boundaries, to enhance the three-dimensional effect. Turn this on to trim the graph. You only see this effect when the graph is in certain rotated positions.
Zoom	Lets you magnify and reduce the display of the graph in the Graph dialog box.
Rotation	Lets you rotate the graph. You must clear **Orthogonal** to use this control.
Elevation	Lets you rotate the graph. You must clear **Orthogonal** to use this control.
Horiz. Offset	Lets you adjust the left-right position of the graph.
Vert. Offset	Lets you adjust the up-down position of the graph.

Perspective	Lets you rotate the graph. You must clear **Orthogonal** to use this control.

Chart Options Dialog Box - Series Tab

Use the Series tab to set up how the series in your graph display. Select the series you want to edit from the drop-down list at the top of the Series tab.

The Series tab is organized into second-level sub-tabs:

- Format Tab
- Point Tab
- General Tab
- Data Source Tab
- Marks Tab

Format Tab

Use the Format tab to set up how the selected series appears. The Format tab contains the following controls:

Border	Lets you format the graph of the selected series. The Border Editor opens, see Border Editor Dialog Box.
Color	Lets you set a color for the graph of the selected series. The Color Editor opens, see Color Editor Dialog Box.
Pattern	Lets you set a pattern for the graph of the selected series. This might only be visible on a three-dimensional graph (see 3D Tab). The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box.
Dark 3D	Lets you automatically darken the depth dimension for visual effect.
Color Each	Assigns a different color to each series indicator.
Clickable	This is unused by Bentley WaterCAD V8*i* .

Color Each line	Lets you enable or disable the coloring of connecting lines in a series. This is unused by Bentley WaterCAD V8*i* .
Height 3D	Lets you set a thickness for the three-dimensional effect in three-dimensional graphs.
Stack	Lets you control how multiple series display in the Graph dialog box. • **None**—Draws the series one behind the other. • **Overlap**—Arranges multiple series with the same origin using the same space on the graph such that they might overlap several times. • **Stack**—Lets you arrange multiple series so that they are additive. • **Stack 100%**—Lets you review the area under the graph curves.
Transparency	Lets you set transparency for your series, where 100 is completely transparent and 0 is completely opaque.
Stairs	Lets you display a step effect between points on your graph.
Inverted	Inverts the direction of the stairs effect
Outline	Displays an outline around the selected series. The Border Editor opens.

Point Tab

Use the Point tab to set up how the points that make up the selected series appear. The Point tab contains the following controls:

Visible	Lets you display the points used to create your graph.
3D	Lets you display the points in three dimensions.
Dark 3D	Lets you automatically darken the depth dimension for visual effect.

Inflate Margins	Adjusts the margins of the points to display points that are close to the edge of the graph. If you clear this option, points near the edge of the graph might only partly display.
Pattern	Lets you set a pattern for the points in your series. The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box. You must clear **Default** to use this option.
Default	Lets you select the default format for the points in your series. This overrides any pattern selection.
Color Each	Assigns a different color to each series indicator.
Style	Lets you select the shape used to represent the points in the selected series.
Width/Height	Lets you set a size for the points in the selected series.
Border	Lets you set the outline of the shapes that represent the points in the selected series. The Border Editor opens, see Border Editor Dialog Box.
Transparency	Lets you set transparency for the points in the selected series, where 100 is completely transparent and 0 is completely opaque.

General Tab

Use the General tab to modify basic formatting and relationships with axes for series in a graph. The General tab contains the following controls:

Show in Legend	Lets you show the series title in the legend. To use this feature, the legend style has to be Series or LastValues (see Style Tab).
Cursor	Lets you specify what your cursor looks like. Select a cursor type from the drop-down list, then click **Close** to close the TeeChart editor, and the new cursor style displays when the cursor is over the graph.

Depth	Lets you set the depth of the three-dimensional effect (see 3D Tab).
Auto	Lets you automatically size the three-dimensional effect. clear and then select this check box to reset the depth of the three-dimensional effect.
Values	Controls the format of the values displayed when marks are on and they contain actual numeric values
Percents	Controls the format of the values displayed when marks are on and they contain actual numeric values.
Horizontal Axis	Lets you define which axis belongs to a given series, since you can have multiple axes in a chart.
Vertical Axis	Lets you define which axis belongs to a given series, since you can have multiple axes in a chart.
Date Time	This is unused by Bentley WaterCAD V8*i* .
Sort	Sorts the points in the series using the labels list.

Data Source Tab

Use this tab to connect a TeeChart series to another chart, table, query, dataset, or Delphi database dataset.

This lets you set the number of random points to generate and overrides the points passed by Bentley WaterCAD V8*i* to the chart control. The Data Source feature can be useful in letting you set its sources as functions and do calculations between the series created by Bentley WaterCAD V8*i* .

- **Random**—xxxx not sure
- **Number of sample values**—xxxx not sure
- **Default**—xxxx not sure
- **Apply**—xxxx not sure

Marks Tab

Use the Marks tab to display labels for points in the selected series. Series-point labels are called marks. The Marks tab contains the following tabs and controls:

Style Tab

Use the Style tab to set how the marks display. The Style tab contains the following controls:

Visible	Lets you display marks.
Clipped	Lets you display marks outside the graph border. clear this check box to let marks display outside the graph border, or select it to clip the marks to the graph border.
Multi-line	Lets you display marks on more than one line. Select this check box to enable multi-line marks.
All Series Visible	Lets you display marks for all series.
Style	Lets you set the content of the marks.
Draw every	Sets the interval of the marks that are displayed. Selecting 2 would display every second mark, and 3 would display every third, etc.
Angle	Lets you rotate the marks for the selected series.

Arrow Tab

Use the Arrow tab to display a leader line on the series graph to indicate where the mark applies. The Arrow tab contains the following controls:

Border	Lets you set up the leader line. The Border Editor opens, see Border Editor Dialog Box.
Pointer	Lets you set up the arrow head (if any) used by the leader line. The Pointer dialog box opens, see Pointer Dialog Box.
Arrow head	Lets you select the kind of arrow head you want to add to the leader line.
Size	Lets you set the size of the arrow head.

Length	Lets you set the size of the leader line and arrow head, or just the leader line if there is no arrow head.
Distance	Lets you set the distance between the leader line and the graph of the selected series.

Format Tab

Use the Format tab to set and format the boxes that contains the marks. The Format tab contains the following controls:

Color	Lets you set a color for the fill of the boxes. The Color Editor opens, see Color Editor Dialog Box.
Frame	Lets you define the outline of the boxes. The Border Editor opens, see Border Editor Dialog Box.
Pattern	Lets you set a pattern for the fill of the boxes. The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box.
Round Frame	Lets you round the corners of the boxes. Select this check box to round the corners of the shape.
Transparent	Lets you set the fill of the boxes as transparent. If the shape is completely transparent, you cannot see it, so clear this check box if you cannot see a shape that you expect to see.
Transparency	Lets you set transparency for the boxes, where 100 is completely transparent and 0 is completely opaque.

Text Tab

Use the Text tab to format the text used in the marks. The Text tab contains the following controls:

Font	Lets you set the font properties for the text. This opens the Windows Font dialog box.

Color — Lets you select the color for the text. Double-click the colored square between Font and Fill to open the Color Editor dialog box (see Color Editor Dialog Box).

Fill — Lets you set a pattern for the text. The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box.

Shadow — Lets you set a shadow for the text.

- **Visible**—Lets you display a shadow for the text. Select this check box to display the axis label shadow.
- **Size**—Lets you set the location of the shadow. Use larger numbers to offset the shadow by a large amount.
- **Color**—Lets you set a color for the shadow. You might set this to gray but can set it to any other color. The Color Editor opens.
- **Pattern**—Lets you set a pattern for the shadow. The Hatch Brush Editor opens.
- **Transparency**—Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Gradient Tab

Use the Gradient tab to create a gradient color background for your marks. The Gradient tab contains the following subtabs and controls:

Format Tab

Visible — Sets whether a gradient displays or not. Select this check box to display a gradient you have set up, clear this check box to hide the gradient.

Direction — Sets the direction of the gradient. Vertical causes the gradient to display from top to bottom, Horizontal displays a gradient from right to left, and Backward/Forward diagonal display gradients from the left and right bottom corners to the opposite corner.

Angle	Lets you customize the direction of the gradient beyond the Direction selections.

Colors Tab

Start	Lets you set the starting color for your gradient.
Middle	Lets you select a middle color for your gradient. The Color Editor opens. Select the **No Middle Color** check box if you want a two-color gradient.
End	Lets you select the final color for your gradient.
Gamma Correction	Lets you control the brightness with which the background displays to your screen; select or clear this check box to change the brightness of the background on-screen. This does not affect printed output.
Transparency	Lets you set transparency for your gradient, where 100 is completely transparent and 0 is completely opaque.

Options Tab

Sigma	Lets you use the options controls. Select this check box to use the controls in the Options tab.
Sigma Focus	Lets you set the location on the chart background of the gradient's end color.
Sigma Scale	Lets you control how much of the gradient's end color is used by the gradient background.

Shadow Tab

Use the Shadow tab to create a shadow for the marks. The Shadow tab contains the following controls:

Visible	Lets you display a shadow. Select this check box to display the shadow, clear this check box to turn off the shadow effect.
Size	Set the size of the shadow by increasing or decreasing the numbers for Horizontal and/or Vertical Size.

Color	Lets you set a color for the shadow. You might set this to gray but can set it to any other color. The Color Editor opens, see Color Editor Dialog Box.
Pattern	Lets you set a pattern for the shadow. The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box.
Transparency	Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Bevels Tab

Use the Bevels tab to create a rounded effects for your marks. The Bevels tab contains the following controls:

Bevel Outer	Lets you set a raised or lowered bevel effect, or no bevel effect, for the background for the selected title.
Color	Lets you set the color for the bevel effect that you use; inner and outer bevels can use different color values.
Bevel Inner	Lets you set a raised or lowered bevel effect, or no bevel effect, for the inside of the background for the selected title.
Size	Lets you set a thickness for the bevel effect that you use; inner and outer bevels use the same size value.

Chart Options Dialog Box - Tools Tab

Use the Tools tab to add special figures in order to highlight particular facts on a given chart. For more information, see Chart Tools Gallery Dialog Box on page 15-1342. The Tools tab contains the following controls:

Add	Lets you add a tool from the Chart Tools Gallery. To be usable in the current graph, a tool needs to be added and set to Active.

Delete	Deletes the selected tool from the list of those available in the current graph.
Active	Activates a selected tool for the current graph. To be usable in the current graph, a tool needs to be added and set to Active.
Up/Down arrow	These are unused by Bentley WaterCAD V8*i* .

Note: **Each tool has its own parameters, see** Chart Tools Gallery Dialog Box**.**

Chart Options Dialog Box - Export Tab

Use the Export tab to save your graph for use in another application. The Export tab contains the following controls:

Copy	Lets you copy the contents of the graph to the Windows clipboard, so you can paste it into another application. You must consider the type of data you have copied when choosing where to paste it. For example, if you copy a picture, you cannot paste it into a text editor, you must paste it into a photo editor or a word processor that accepts pictures. Similarly, if you copy data, you cannot paste it into an image editor, you must paste it into a text editor or word processor.
Save	Lets you create a new file from the contents of the graph.

Picture Tab

Use the Picture tab to save your graph as a raster image or to copy the graph as an image to the clipboard. The Picture tab contains the following controls and subtabs:

Format	Lets you select the format of the picture you want to save. GIF, PNG, and JPEG are supported by the Worldwide Web, a metafile is a more easily scalable format. A Bitmap is a Microsoft BMP file that is widely supported on Windows operating systems, whereas TIFF pictures are supported on a variety of Microsoft and non-Microsoft operating systems.

Options Tab

Colors — Lets you use the default colors used by your graph or to convert the picture to use grayscale. This feature is used when you save the picture as a file, not by the copy option.

Size Tab

Width/Height — Lets you change the width and height of the picture. These values are measured in pixels and are used by both the Save and Copy options

Keep aspect ratio — Lets you keep the relationship between the height and width of the picture the same when you change the image size. If you clear this check box, you can distort the picture by setting height or width sizes that are not proportional to the original graph.

Note: Changing the size of a graph using these controls might cause some loss of quality in the image. Instead, try saving the graph as a metafile and resizing the metafile after you paste or insert it into its destination.

Native Tab

The Native tab contains the following controls:

Include Series Data — This is unused by Bentley WaterCAD V8*i* .

File Size — Displays the size of an ASCII file containing the data from the current graph.

Data Tab

The Data tab contains the following controls:

Series — Lets you select the series from which you copy data.

Format — Lets you select a file type to which you can save the data. This is not used by the Copy function.

Include — Select the data you want to copy.

Text separator	Lets you specify how you want rows of data separated. This is supported by the Save function and only by the Copy function if you first saved using the text separator you have selected, before you copy.

Chart Options Dialog Box - Print Tab

Use the Print tab to preview and print your graph. The Print tab contains the following controls and subtabs:

Printer	Lets you select the printer you want to use.
Setup	Lets you configure the printer you want to use. For example, if the selected printer supports printing on both sides of a page, you might want to turn on this feature.
Print	Prints the displayed graph to the selected printer.

Page Tab

Orientation	Lets you set up the horizontal and vertical axes of the graph. Many graphs print better in Landscape orientation because of their width:height ratio.
Zoom	Lets you magnify the graph as displayed in the print preview window. Use the scrollbars to inspect the graph if it doesn't fit within the preview window after you zoom. Changing the zoom does not affect the size of the printed output.
Margins	Lets you set up top, bottom, left, and right margins that are used when you print.
Margin Units	Lets you set the units used by the Margins controls: percent or hundredths of an inch.

Format Tab

Print Background	When checked, prints the background of the graph.

Quality	You do not need to change this setting. The box is cleared by default.
Proportional	Lets you change the graph from proportional to non-proportional. When you change this setting, the preview pane is automatically updated to reflect the change. This box is checked by default.
Grayscale	Prints the graph in grayscale, converting colors into shades of gray.
Detail Resolution	Lets you adjust the detail resolution of the printout. Move the slider to adjust the resolution.
Preview Pane	Displays a small preview of the graph printout.

Border Editor Dialog Box

The Border Editor dialog box lets you define border properties for your graph. The Border Editor dialog box contains the following controls:

Visible	Displays or hides the border. Select this check box to display the border.
Color	Lets you select a color for the border. The Color Editor dialog box opens, see Color Editor Dialog Box.
Ending	Lets you set the ending style of the border.
Dash	Lets you select the dash style, if you have a selection other than Solid set for the border style.
Width	Lets you set the width of the border.
Style	Lets you set the style for the border. Solid is an uninterrupted line.
Transparency	Lets you set transparency for your border, where 100 is completely transparent and 0 is completely opaque.

Gradient Editor Dialog Box

Use the Gradient Editor dialog box to set a blend of two or three colors as the fill. Click **OK** to apply the selection. The Gradient Editor contains the following controls and tabs:

Format Tab

Visible — Sets whether a gradient displays or not. Select this check box to display a gradient you have set up, clear this check box to hide the gradient.

Direction — Sets the direction of the gradient. Vertical causes the gradient to display from top to bottom, Horizontal displays a gradient from right to left, and Backward/Forward diagonal display gradients from the left and right bottom corners to the opposite corner.

Angle — Lets you customize the direction of the gradient beyond the Direction selections.

Colors Tab

Start — Lets you set the starting color for your gradient.

Middle — Lets you select a middle color for your gradient. The Color Editor opens. Select the **No Middle Color** check box if you want a two-color gradient.

End — Lets you select the final color for your gradient.

Gamma Correction — Lets you control the brightness with which the background displays to your screen; select or clear this check box to change the brightness of the background on-screen. This does not affect printed output.

Transparency — Lets you set transparency for your gradient, where 100 is completely transparent and 0 is completely opaque.

Options Tab

Sigma — Lets you use the options controls. Select this check box to use the controls in the Options tab.

Sigma Focus	Lets you set the location on the chart background of the gradient's end color.
Sigma Scale	Lets you control how much of the gradient's end color is used by the gradient background.

To access the Gradient Editor dialog box, click Chart Settings in the Graph dialog box, then click the Tools tab. Select the Axis tab and Color Band tool, then click the Gradient button.

Color Editor Dialog Box

Use the Color Editor dialog box to select a color. Click the basic color you want to use then click **OK** to apply the selection. The Color Editor dialog box contains the following controls:

Transparency	Lets you set transparency for your color, where 100 is completely transparent and 0 is completely opaque.
Custom	Lets you define a custom color to use. The Color dialog box opens, see Color Dialog Box.
OK/Cancel	Click **OK** to use the selection. Click **Cancel** to close the dialog box without making a selection.

To access the Color Editor dialog box, click a Color button in the Chart Options dialog box.

Color Dialog Box

Use the Color dialog box to select a basic color or to define a custom color. After you select the color you want to use, click **OK** to apply the selection.

Basic colors	Lets you click a color to select it.
Custom colors	Displays colors you have created and selected for use.
Color matrix	Lets you use the mouse to select a color from a range of colors displayed.
Color|Solid	Displays the currently defined custom color.

Hue/Sat/Lum	Lets you define a color by entering values for hue, saturation, and luminosity.
Red/Green/Blue	Lets you define a color by entering values of red, green, and blue colors.
Add to Custom Colors	Adds the current custom color to the Custom colors area.

To access the Color dialog box, click the Custom button in the Color Editor dialog box.

Hatch Brush Editor Dialog Box

Use the Hatch Brush Editor dialog box to set a fill. The Hatch Brush Editor dialog box contains the following controls and tabs:

Visible	Displays or hides the pattern. Select this check box to display the selected pattern.

- Hatch Brush Editor Dialog Box - Solid Tab
- Hatch Brush Editor Dialog Box - Hatch Tab
- Hatch Brush Editor Dialog Box - Gradient Tab
- Hatch Brush Editor Dialog Box - Image Tab

Hatch Brush Editor Dialog Box - Solid Tab

Use the Solid tab to set a solid color as the fill. The Solid tab contains the following controls:

Transparency	Lets you set transparency for your color, where 100 is completely transparent and 0 is completely opaque.
Custom	Lets you define a custom color to use. The Color dialog box opens, see Color Dialog Box.
OK/Cancel	Click **OK** to use the selection. Click **Cancel** to close the dialog box without making a selection.

Hatch Brush Editor Dialog Box - Hatch Tab

Use the Hatch tab to set a pattern as the fill. Click **OK** to apply the selection. The Hatch tab contains the following controls:

Hatch Style	Select the pattern you want to use. These display using the currently selected background and foreground colors.
Background/ Foreground	Select the color you want to use for the background and foreground of the pattern. This opens the Color Editor, see Color Editor Dialog Box.
%	Lets you set transparency for your color, where 100 is completely transparent and 0 is completely opaque.

Hatch Brush Editor Dialog Box - Gradient Tab

Use the Gradient tab to set a blend of two or three colors as the fill. Click **OK** to apply the selection. The Gradient tab contains the following controls:

Format Tab

Visible	Sets whether a gradient displays or not. Select this check box to display a gradient you have set up, clear this check box to hide the gradient.
Direction	Sets the direction of the gradient. Vertical causes the gradient to display from top to bottom, Horizontal displays a gradient from right to left, and Backward/Forward diagonal display gradients from the left and right bottom corners to the opposite corner.
Angle	Lets you customize the direction of the gradient beyond the Direction selections.

Colors Tab

Start	Lets you set the starting color for your gradient.
Middle	Lets you select a middle color for your gradient. The Color Editor opens. Select the **No Middle Color** check box if you want a two-color gradient.

End	Lets you select the final color for your gradient.
Gamma Correction	Lets you control the brightness with which the background displays to your screen; select or clear this check box to change the brightness of the background on-screen. This does not affect printed output.
Transparency	Lets you set transparency for your gradient, where 100 is completely transparent and 0 is completely opaque.

Options Tab

Sigma	Lets you use the options controls. Select this check box to use the controls in the Options tab.
Sigma Focus	Lets you set the location on the chart background of the gradient's end color.
Sigma Scale	Lets you control how much of the gradient's end color is used by the gradient background.

Hatch Brush Editor Dialog Box - Image Tab

Use the Image tab to select an existing graphic file or picture to use as the fill. Click **OK** to apply the selection. The Image tab contains the following controls:

Browse	Lets you navigate to then select the graphic file you want to use. When selected, the graphic displays in the tab.
Style	Lets you define how the graphic is used in the fill. • **Stretch**—Resizes the image to fill the usable space. • **Tile**—Repeats the image to fill the usable space. • **Center**—Puts the image in the horizontal and vertical center. • **Normal**—Puts the image in the top-left corner

Pointer Dialog Box

Use the Pointer dialog box to set up a pointers for use with leader lines. The Pointer dialog box contains the following controls:

Visible	Sets whether a pointer displays or not.
3D	Lets you display the pointer in three dimensions.
Dark 3D	Lets you automatically darken the depth dimension for visual effect.
Inflate Margins	Adjusts the margins of the pointers to display pointers that are close to the edge of the graph. If you clear this option, pointers near the edge of the graph might only partly display.
Pattern	Lets you set a pattern for the pointers. The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box. You must clear **Default** to use this option.
Default	Lets you select the default format for the pointers. This overrides any pattern selection.
Color Each	Assigns a different color to each pointer.
Style	Lets you select the shape used to represent the pointers.
Width/Height	Lets you set a size for the pointers.
Border	Lets you set the outline of the shapes that represent the pointers. The Border Editor opens, see Border Editor Dialog Box.
Transparency	Lets you set transparency for the pointers, where 100 is completely transparent and 0 is completely opaque.

To access the Pointer dialog box, click Chart Settings in the Graph dialog box, then click Series > Marks > Arrow.

Change Series Title Dialog Box

Use the Change Series Title dialog box to change the title of a selected series. Type the new series title, then click **OK** to apply the new name or **Cancel** to close the dialog box without making a change.

To access the Change Series title dialog box, click **Chart Settings** in the Graph dialog box, then click the Series tab, then the **Title** button.

Chart Tools Gallery Dialog Box

Use the Chart Tools Gallery dialog box to add tools to your graph. For more information, see Chart Options Dialog Box - Tools Tab on page 15-1331.

Click one of the following links to learn more about the Chart Tools Gallery dialog box:

- Chart Tools Gallery Dialog Box - Series Tab
- Chart Tools Gallery Dialog Box - Axis Tab
- Chart Tools Gallery Dialog Box - Other Tab

Chart Tools Gallery Dialog Box - Series Tab

Use the Series tab to add tools related to the series in your chart. The Series tab contains the following tools:

Cursor

Displays a draggable cursor line on top of the series. After you have added the Cursor tool to your graph, you can modify the following settings:

Series	Lets you select the series to which you want to apply the tool.
Style	Lets you select a horizontal line, vertical line, or both as the format of the tool.
Snap	Causes the cursor tool to adhere to the selected series.
Follow Mouse	Causes the cursor tool to follow your movements of the mouse.

Pen	Lets you define the cursor tool. The Border Editor opens, see Border Editor Dialog Box.

Drag Marks

Lets you drag series marks. To use this tool, you must display the marks for a selected series, see Marks Tab. After you have added the Drag Marks tool to your graph, you can modify the following settings:

Series	Lets you select the series to which you want to apply the tool.
Reset Positions	Moves any marks you have dragged back to their original position.

Drag Point

Lets you drag a series point. After you have added the Drag Point tool to your graph, you can modify the following settings:

Series	Lets you select the series to which you want to apply the tool.
Style	Lets you constrain the movement of the series point to one axis or both (no constraint).
Mouse Button	Lets you select the mouse button you click to drag.
Cursor	Lets you select the appearance of the cursor when using the tool.

Draw Line

Lets you draw a line on the graph by dragging. After you have added the Draw Line tool to your graph, you can modify the following settings:

Series	Lets you select the series to which you want to apply the tool.
Pen	Lets you define the line. The Border Editor opens, see Border Editor Dialog Box.
Button	Lets you select the mouse button you click to drag.

Enable Draw	Enables the Draw Line tool. Select this check box to let you draw lines, clear it to prevent you from drawing lines.
Enable Select	Lets you select and move lines that you have drawn. Select this check box, then click and drag the line you want to move. clear this check box if you want to prevent lines from being moved.
Remove All	Removes all lines you have drawn.

Gantt Drag

Lets you move and resize Gantt bars by dragging. This is unused by Bentley WaterCAD V8*i* .

Image

Displays a picture using the selected series axes as boundaries. After you have added the Image tool to your graph, you can modify the following settings:

Series	Lets you select the series to which you want to apply the tool.
Browse	Lets you navigate to and select the image you want to use. Browse is unavailable when there is a selected image. To select a new image, first clear the existing one.
Clear	Lets you remove a selected image. Clear is unavailable when there is no selected image.
Mode	Lets you set up the image you select. • **Normal**—Puts the background image in the top-left corner of the graph. • **Stretch**—Resizes the background image to fill the entire background of the graph. The image you select conforms to the series to which you apply it. • **Center**—Puts the background image in the horizontal and vertical center of the graph. • **Tile**—Repeats the background image as many times as needed to fill the entire background of the graph.

Mark Tips

Displays data in tooltips when you move the cursor over the graph. After you have added the Mark Tips tool to your graph, you can modify the following settings:

Series	Lets you select the series to which you want to apply the tool
Style	Lets you select what data the tooltips display.
Action	Sets when the tooltips display. Select **Click** if you want the tooltips to display when you click, or select **Move** if you want the tooltips to display when you move the mouse.
Delay	Lets you delay how quickly the tooltip displays.

Nearest Point

Lets you define and display an indicator when you are near a point in the selected series. After you have added the Nearest Point tool to your graph, you can modify the following settings:

Series	Lets you select the series to which you want to apply the tool.
Fill	Lets you set the fill for the nearest-point indicator. The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box.
Border	Lets you set the outline of the nearest-point indicator. The Border Editor opens, see Border Editor Dialog Box.
Draw Line	Creates a line from the tip of the cursor to the series point.
Style	Sets the shape for the indicator
Size	Sizes the indicator.

Pie Slices

Outlines or expands slices of pie charts when you move the cursor or click them. This is unused by Bentley WaterCAD V8*i* .

Series Animation

Animates series points. After you have added the Series Animation tool to your graph, you can modify the following settings:xxxx seems broken.

Series	Lets you select the series to which you want to apply the tool.
Steps	Lets you select the steps used in the animation. Set this control towards 100 for smoother animation and away from 100 for quicker, but less smooth animation.
Start at min. value	Lets you start the animation at the series' minimum value. clear this check box to set your own start value.
Start value	Sets the value at which the animation starts. To use this control, you must clear **Start at min. value**.
Execute!	Starts the animation.

Chart Tools Gallery Dialog Box - Axis Tab

Use the Axis tab to add tools related to the axes in your chart. The Axis tab contains the following tools:

Axis Arrows

Lets you add arrows to the axes. The arrows permit you to scroll along the axes. After you have added the Axis Arrows tool to your graph, you can modify the following settings:

Axis	Select the axis to which you want to add arrows.
Border	Lets you set the outline of the arrows. The Border Editor opens, see Border Editor Dialog Box.
Fill	Lets you set the fill for the arrows. The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box.
Length	Lets you set the length of the arrows.
Inverted Scroll	Lets you change the direction in which the arrows let you scroll.

Scroll	Changes the magnitude of the scroll. Set a smaller percentage to reduce the amount of scroll caused by one click of an axis arrow, or set a larger percentage to increase the amount of scroll caused by a click.
Position	Lets you set an axis arrow at the start, end, or both positions of the axis.

Color Band

Lets you apply a color band to your graph for a range of values you select from an axis. After you have added the Color Band tool to your graph, you can modify the following settings:

Axis	Select the axis that you want to use to define the range for the color band.
Border	Lets you set the outline of the color band. The Border Editor opens, see Border Editor Dialog Box.
Pattern	Lets you set the fill of the color band. The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box.
Gradient	Lets you set a gradient for the color band. A gradient overrides any solid color fill you might have set. The Gradient Editor opens, see Gradient Editor Dialog Box.
Color	Lets you set a solid color for the color band. The Color Editor opens, see Color Editor Dialog Box.
Start Value	Sets where the color band begins. Specify a value on the selected axis.
End Value	Sets where the color band ends. Specify a vale on the selected axis.
Transparency	Lets you set transparency for your color, where 100 is completely transparent and 0 is completely opaque.

Draw Behind	Lets you position the color band behind the graphs. If you clear this check box, the color band appears in front of your graphs and hides them, unless you have transparency set.

Color Line

Lets you apply a color line, or plane in three dimensions, at a point you set at a value on an axis. After you have added the Color Line tool to your graph, you can modify the following settings:

Axis	Select the axis that you want to use to define the location for the line.
Border	Lets you set the outline of the color line. The Border Editor opens, see Border Editor Dialog Box.
Value	Sets where the color line is. Specify a value on the selected axis.
Allow Drag	Lets you drag the line or lock the line in place. Select this check box if you want to permit dragging. clear this check box if you want the line to be fixed in one location.
Drag Repaint	Lets you smooth the appearance of the line as you drag it.
No Limit Drag	Lets you drag the line beyond the axes of the graph, or constrain the line to boundaries defined by those axes. Select this check box to permit unconstrained dragging.
Draw Behind	Lets you position the color line behind the graphs. If you clear this check box, the color band appears in front of your graphs. This is more noticeable in 3D graphs.
Draw 3D	Lets you display the line as a 2D image in a 3D chart. If you have a 3D chart (see 3D Tab), clear this check box to display the line as a line rather than a plane.

Chart Tools Gallery Dialog Box - Other Tab

Use the Other tab to add tools to your chart, including annotations. The Other tab contains the following tools:

3D Grid Transpose

Swaps the X and Z coordinates to rotate the series through 90 degrees. This is unused by Bentley WaterCAD V8*i* .

Annotation

Lets you add text to the chart. After you have added the Annotation tool to your graph, you can modify the following settings:

Options Tab

Text	Lets you enter the text you want for your annotation.
Text alignment	Sets the alignment of the text inside the annotation box.
Cursor	Lets you set the style of the cursor when you move it over the annotation.

Position Tab

Auto	Lets you select a standard annotation position.
Custom	Lets you select a custom position for the annotation. Select this check box to override the Auto setting and enable the Left and Top controls.
Left/Top	Lets you set a position from the Left and Top edges of the graph tab for the annotation.

Callout Tab

Border	Lets you set up the leader line. The Border Editor opens, see Border Editor Dialog Box.
Pointer	Lets you set up the arrow head (if any) used by the leader line. The Pointer dialog box opens, see Pointer Dialog Box.
Position	Sets the position of the callout.

Distance — Lets you set the distance between the leader line and the graph of the selected series.

Arrow head — Lets you select the kind of arrow head you want to add to the leader line.

Size — Lets you set the size of the arrow head.

Format Tab

Color — Lets you set a color for the fill of the boxes. The Color Editor opens, see Color Editor Dialog Box.

Frame — Lets you define the outline of the boxes. The Border Editor opens.

Pattern — Lets you set a pattern for the fill of the boxes. The Hatch Brush Editor opens, see Hatch Brush Editor Dialog Box.

Round Frame — Lets you round the corners of the boxes. Select this check box to round the corners of the shape.

Transparent — Lets you set the fill of the boxes as transparent. If the shape is completely transparent, you cannot see it, so clear this check box if you cannot see a shape that you expect to see

Transparency — Lets you set transparency for the boxes, where 100 is completely transparent and 0 is completely opaque.

Text Tab

Font — Lets you set the font properties for text. This opens the Windows Font dialog box.

Color — Lets you select the color for the text font. Double-click the colored square between Font and Fill to open the Color Editor dialog box.

Fill — Lets you set a pattern for the text font. The Hatch Brush Editor opens.

Shadow

Lets you set a shadow for the text.

- **Visible**—Lets you display a shadow for the text. Select this check box to display the shadow.
- **Size**—Lets you set the location of the shadow. Use larger numbers to offset the shadow by a large amount.
- **Color**—Lets you set a color for the shadow. You might set this to gray but can set it to any other color. The Color Editor opens.
- **Pattern**—Lets you set a pattern for the shadow. The Hatch Brush Editor opens.
- **Transparency**—Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Gradient Tab

Format

Format—Lets you set up the gradient's properties.

- **Visible**—Sets whether a gradient displays or not. Select this check box to display a gradient you have set up, clear this check box to hide the gradient.
- **Direction**—Sets the direction of the gradient. Vertical causes the gradient to display from top to bottom, Horizontal displays a gradient from right to left, and Backward/Forward diagonal display gradients from the left and right bottom corners to the opposite corner.
- **Angle**—Lets you customize the direction of the gradient beyond the Direction selections.

Colors

Lets you set the colors used for your gradients. The Start, Middle, and End selections open the Color Editor, see Color Editor Dialog Box.

- **Start**—Lets you set the starting color for your gradient.
- **Middle**—Lets you select a middle color for your gradient. The Color Editor opens. Select the **No Middle Color** check box if you want a two-color gradient.
- **End**—Lets you select the final color for your gradient.
- **Gamma Correction**—Lets you control the brightness with which the background displays to your screen; select or clear this check box to change the brightness of the background on-screen. This does not affect printed output.
- **Transparency**—Lets you set transparency for your gradient, where 100 is completely transparent and 0 is completely opaque.

Options

Lets you control the affect of the start and end colors on the gradient, the middle color is not used.

- **Sigma**—Lets you use the options controls. Select this check box to use the controls in the Options tab.
- **Sigma Focus**—Lets you set the location on the chart background of the gradient's end color.
- **Sigma Scale**—Lets you control how much of the gradient's end color is used by the gradient background.

Shadow Tab

Visible

Lets you display a shadow. Select this check box to display the shadow, clear this check box to turn off the shadow effect.

Size

Set the size of the shadow by increasing or decreasing the numbers for Horizontal and/or Vertical Size.

Color	Lets you set a color for the shadow. You might set this to gray but can set it to any other color. The Color Editor opens.
Pattern	Lets you set a pattern for the shadow. The Hatch Brush Editor opens.
Transparency	Lets you set transparency for your shadow, where 100 is completely transparent and 0 is completely opaque.

Bevels Tab

Bevel Outer	Lets you set a raised or lowered bevel effect, or no bevel effect, for the outside of the legend.
Color	Lets you set the color for the bevel effect that you use; inner and outer bevels can use different color values.
Bevel Inner	Lets you set a raised or lowered bevel effect, or no bevel effect, for the inside of the legend.
Size	Lets you set a thickness for the bevel effect that you use; inner and outer bevels use the same size value.

Page Number

Lets you add a page number annotation. For more information, see Annotation.

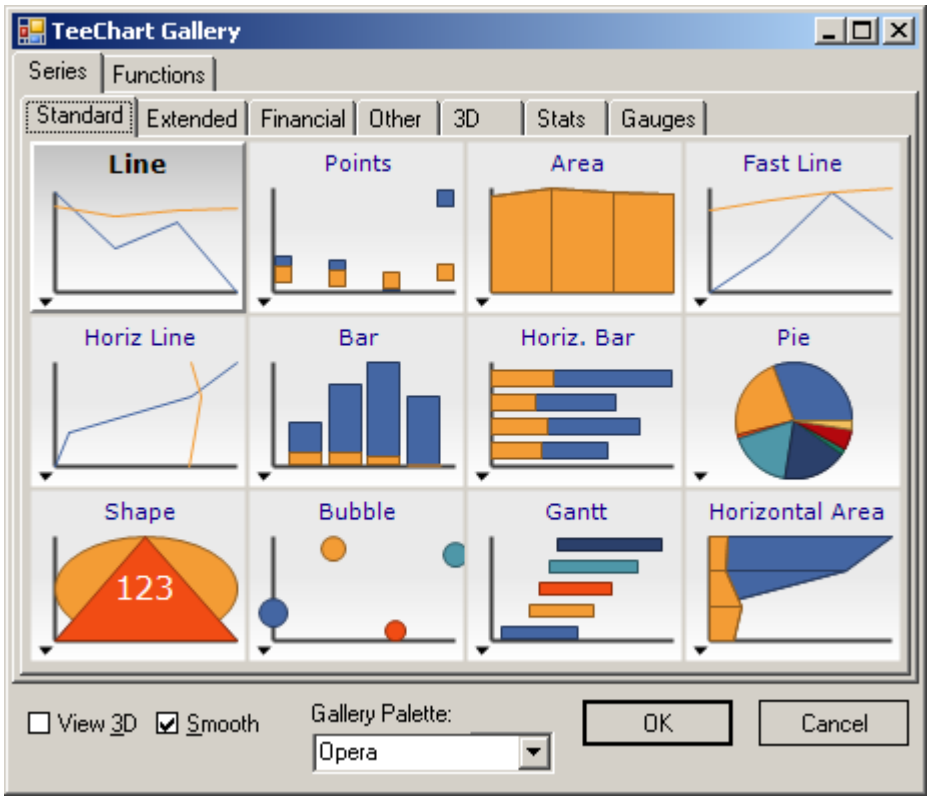
Rotate

Lets you rotate the chart by dragging. After you have added the Rotate tool to your graph, you can modify the following settings:

Inverted	Reverses the direction of the rotation with respect to the direction you move the mouse.
Style	Lets you rotate horizontally, vertically, or both. Rotation is horizontal rotation about a vertical axis, whereas elevation is vertical rotation about a horizontal axis.
Outline	Lets you set the outline. The Border Editor opens, see Border Editor Dialog Box.

TeeChart Gallery Dialog Box

Use the TeeChart Gallery dialog box to change the appearance of a series.



Series

The available series chart designs include:

- **Standard**
- **Stats**
- **Financial**
- **Extended**
- **3D**
- **Other**
- **View 3D**—Lets you view the chart design in two or three dimensions. Select this check box to view the charts in 3D, clear it to view them in 2D.
- **Smooth**—Smooths the display of the charts. Select this check box to smooth the display, clear it to turn off smoothing.

Functions

The available function chart designs include:

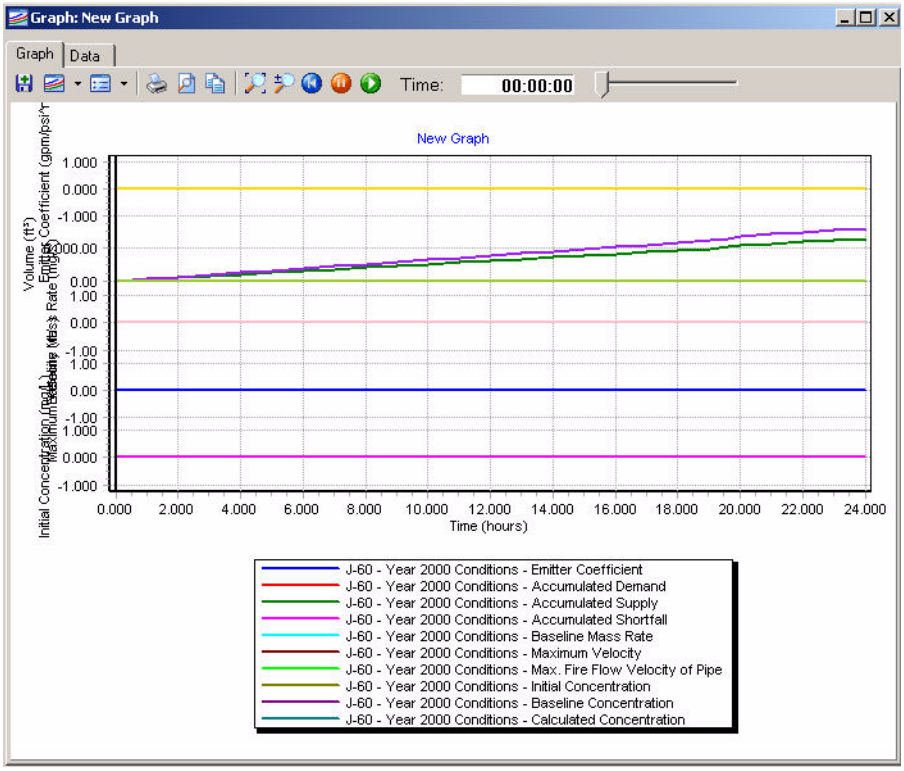
- **Standard**
- **Financial**
- **Stats**
- **Extended**
- **View 3D**—Lets you view the chart design in two or three dimensions. Select this check box to view the charts in 3D, clear it to view them in 2D.
- **Smooth**—Smooths the display of the charts. Select this check box to smooth the display, clear it to turn off smoothing.

Customizing a Graph

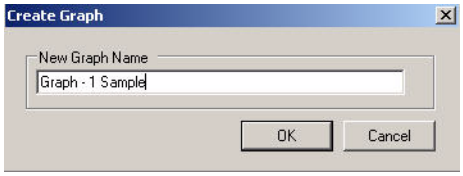
To customize a graph

1. If you do not have your own model, open one of the example files.
2. Create a graph.
   a. Click **Compute**.
   b. Close the Calculation Summary.
   c. Save your model.

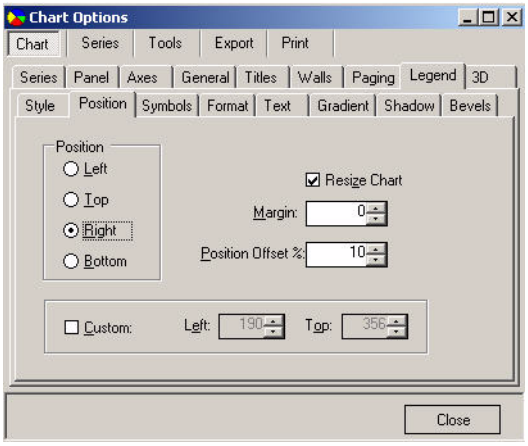
d. Right click an element. To add more than one element press <Shift+click>, then right-click and select **Graph**.



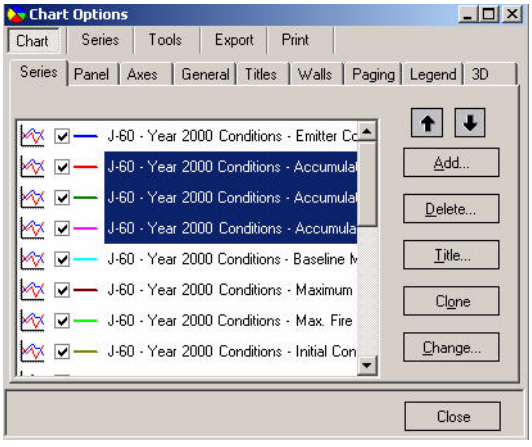
e. Click **Add to Graph Manager**  to save to the Graph manager.



3. Move the legend.
   a. Click **Chart Settings**, to open the Chart Options dialog box.
   b. Click the **Chart** icon, **Legend** tab, and **Position** subtab.
   c. Click **Right** in the Position area to set the legend to the right side of the graph. You can use other controls on this subtab to move the legend.



4. Change the line colors and weights.
   a. Click **Chart Settings** to open the Chart Options dialog box.
   b. In the Chart > Series tab click the series to edit, then select and highlight it. You can select more than one series by pressing <Ctrl> or <Shift> + click.



   c. Click **Series** and select the **Format** tab.
   d. Click **Color** to open the Color Editor and select a new color.

e. Click **OK** after you click the color you want to use. The series that are changed are those that you highlighted in the Chart > Series tab.

f. Click **Outline** to open the Border Editor to change the thickness of a line.

g. Select **Visible**.

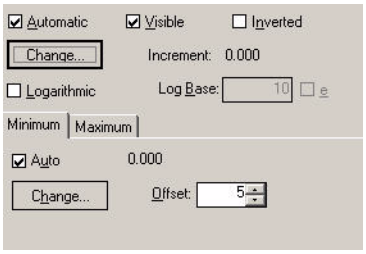
h. Change the **Width**.

i. Make sure the Transparency is set to **0** if you want the line to appear opaque.

j. Click **OK** after you define the line width and attributes. The series that are changed are those that you highlighted in the Chart > Series tab.

5. Change the interval between labels, grid, and ticks.

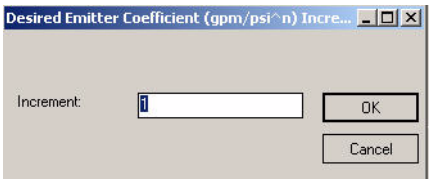
a. Click **Chart > Axes > Scales > Change** to change the interval between labels on the axes.



b. Select the Axis you want to change from the list of axes in the Axes area.



c. In the Increment dialog box, type the new value and click **OK**. This also changes the distance between major and minor ticks.



d. If needed, change the axis you have selected for changes.

e. Click **Chart > Axes > Minor** and change the **Count** to change the interval between minor ticks on the axes.

6. You can show and hide a grid associated with the major ticks.
   a. Click **Chart > Axes > Ticks**.
   b. Select the axis to change the grid, then click **Grid**.
   c. In the Border Editor dialog box, select or clear **Visible** to show or hide the grid.

7. You can show and hide a grid associated with the minor ticks.
   a. Click **Chart > Axes > Minor**.
   b. Select the axis to change the grid, then click **Grid**.
   c. In the Border Editor dialog box, select or clear **Visible** to show or hide the grid.

8. You can set the minimum and maximum range for an axis.
   a. Click **Chart > Axes > Scales**.
   b. Select the axis to change the grid, then click **Grid**.
   c. Use the Minimum tab to change the minimum value for an axis. Clear the **Auto** check box.
   d. Click **Change**.
   e. Set the minimum value for the axis.
   f. Use the Maximum tab to change the maximum value for an axis. Clear the **Auto** check box.
   g. Click **Change**.
   h. Set the maximum value for the axis.

9. Change the background colors.
   a. Click **Chart > Panel >** and select **Background**.
   b. Use the **Color** and **Pattern** buttons to set a background color and/or pattern for the graph.

10. Change the number of decimal places used in axis labels.
    a. Click **Chart > Axes > Labels > Format**.
    b. Select the axis you want to change.
    c. Change the number of decimal places by making a selection from the **Values Format** menu.

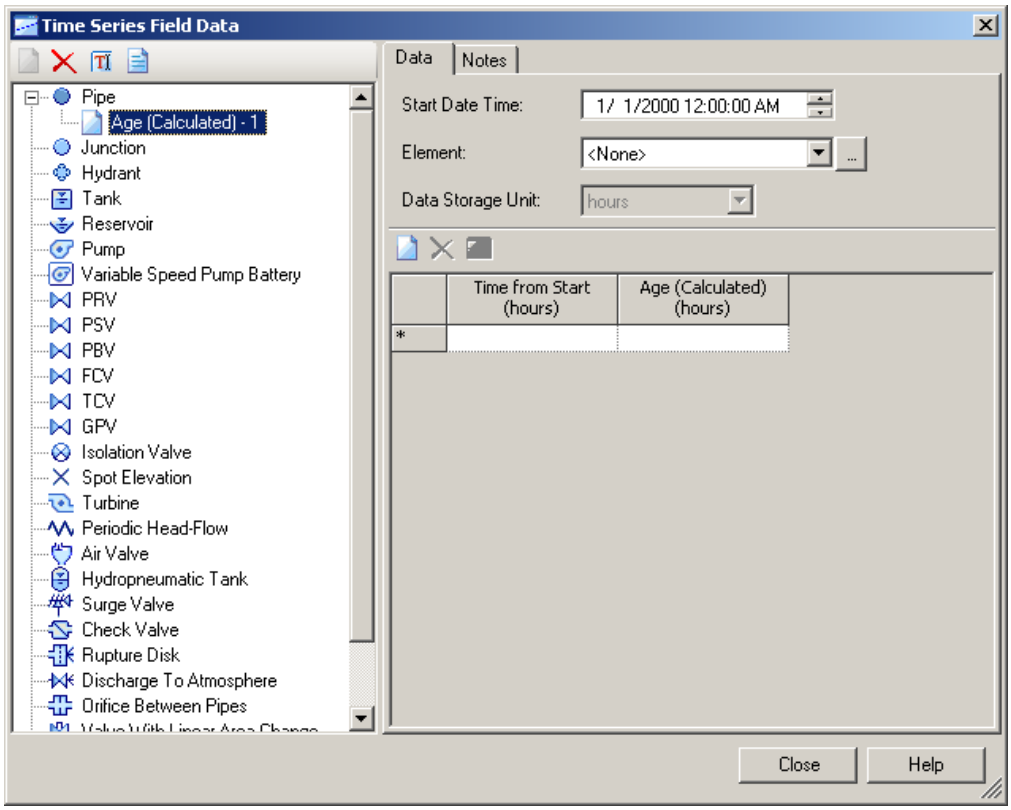
11. Change the fonts used by the axes and titles.
    a. Click **Chart > Axes > Labels** > **Text**.
    b. Select the axis you want to change.
    c. Click **Font** to open the Font dialog box and change the format of the fonts used by the axis labels.
    d. Click **OK**.
12. Add a text box to the graph.
    a. Click **Tools > Add > Other > Annotation**.
    b. In the Text pane, type the text you want in your annotation.

Note: There are some limitations to user modifications to the graphs in Bentley WaterCAD V8*i* . For example, changes to the format of the axis ticks (the values shown on the axis) are overridden and use the proper formatter. You can change the format via the Tools->Options, Units tab or by right-clicking the axis in question and click on the <Formatter> Properties... menu item. This will open the Set Field Options Dialog Box. In this dialog you can change the unit, display precision and format.

Time Series Field Data

The Time Series Field Data dialog allows you to enter your observed field data and compare it to the calculated results from the model in graph format. This is especially useful in comparing time series data for model calibration.

Use this feature to display user-supplied time variant data values alongside calculated results in the graph display dialog. Model competency can sometimes be determined by a quick side by side visual comparison of calculated results with those observed in the field



- **Get familiar with your data** - If you obtained your observed data from an outside source, you should take the time to get acquainted with it. Be sure to identify units of time and measurement for the data. Be sure to identify what the data points represent in the model; this helps in naming your line or bar series as it will appear in the graph. Each property should be in a separate column in your data source file.
- **Preparing your data** - Typically, observed data can be organized as a collection of points in a table. In this case, the time series data can simply be copied to the clipboard directly from the source and pasted right into the observed data input table. Ensure that your collection of data points is complete. That is, every value must have an associated time value. Oftentimes data points are stored in tab or comma delimited text files; these two import options are available as well.
- **Starting time series data entry** - To create a time series data set, click the Component menu and select Time Series Field Data. Pick the element type (e.g. Pipe, Junction) and select the New button on the top row of the dialog. (You may also right click on the Element Type Name and click the Add button) You will then see the Select Associated Modeling Attribute dialog where you select the

property (attribute) to be imported. Choose the attribute and click OK. You may import any number of data sets for any Property and Element. The data set will have the default name of Property-N (e.g. Flow - 1). To change the name, click the Rename button (third button along the top of the table).

- **Specifying the characteristics of your data** - The following charecteristics must be defined:
  - Start Date Time - Specify the date and time the field data was collected. It is important to ensure that your data shows correctly on the plot compared to the simulated data. For example, if the calculation Base Date and Start Time differ from the field data, they will not overlay properly on any graphs of the corresponding data.
  - Element - Choose the element that represents the field data measurement location. Click the ellipsis button to select the element from the drawing.
  - Data Storage Unit - The storage unit doesn't generally need to be changed, however it becomes a consideration when the user wants to import/export time-series data using ModelBuilder. ModelBuilder sets the value using the underlying (unitless) time-series data field, so (unlike most fields), there is no conversion of values to storage units when working directly with the field.

    To address this issue, you can specify the storage unit associated with the time series. Note that if the user changes the storage unit, existing values will be interpreted differently. The user can retain their values by copying them from the table, changing the unit, and pasting the values back in.
  - Time From Start - Specify an offset of the start time and date for an EPS scenario.
  - Attribute Value - Enter the value for the specified attribute at the specified Time from Start.

You can perform a quick graphical check on the data import by clicking the Graph button at the top of the data table.

If the number of observations is large, it is best to use the Copy/Paste commands. Copy the data from the original source to the clipboard, then go to the top of the Time from Start or Property (e.g. Flow) column and hit CTRL-V to paste the values into the appropriate column.

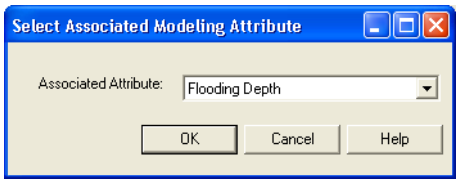
Click the Close button when done.

The data is saved with the model file. If you modify the source data file, the changes will not appear until time series data is imported again.

To add the time series field data to a graph, first create the graph of the property from an EPS model run (e.g. right click on element and pick Graph). In the Graph options dialog, select Time Series Field Data and then the name of the time series (in the Field pane (right pane). The field data will appear in the graph as points (by default) while the model results will appear as a continuous line. This can be changed using the Chart Settings button at the top of the graph (third from left).

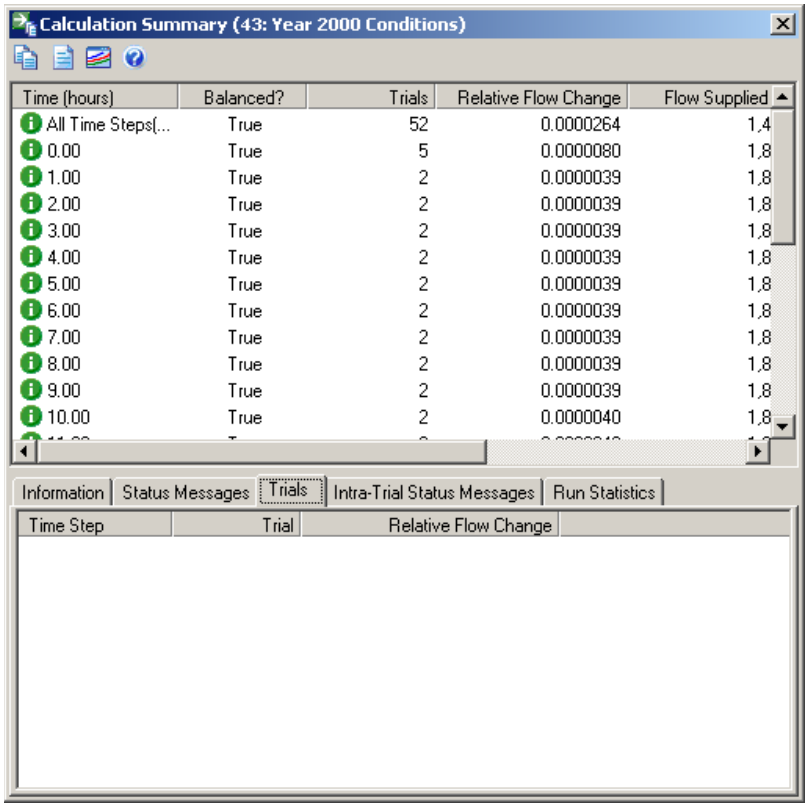
Select Associated Modeling Attribute Dialog Box

This dialog appears when you create a new field data set in the Time Series Field Data dialog. Choose the attribute represented in the time series data source. The available attributes will vary depending on the element type chosen.



Calculation Summary

The calculation summary gathers useful information related to the state of the calculation (e.g. success/failure), status messages for elements (e.g. pump on/off, tank full/empty), and the system flow results (e.g. flow demanded, flow stored).



The following controls are available in the Calculation Summary dialog box:

- **Copy** - Copies the calculation summary to the Windows clipboard.
- **Report** - Opens the Calculation Summary report.
- **Graph** - Opens the Calculation Summary Graph.
- **Help** - Opens the online help for this dialog.

The tabs below the time step table contain the following information:

- **Information Tab**: This tab displays any element messages for the currently selected time step.
- **Status Messages Tab**: This tab displays any status messages for the currently selected time step.
- **Trials Tab**: This tab displays the relative flow change for each of the trials for the currently selected time step.

- **Intra-Trial Status Messages**: This tab displays the status changes between trials for the currently selected time step.
- **Run Statistics Tab**: This tab displays calculation statistics such as the time the calculation was completed, how long the calculation took to load and run, and the number of time steps, links, and nodes that were calculated.

Note: The stats displayed under this tab pertain only to Steady State and EPS runs. For fire flow and flushing analysis the run times reported do not include the times for all the nodes to run, just the base Steady State run.

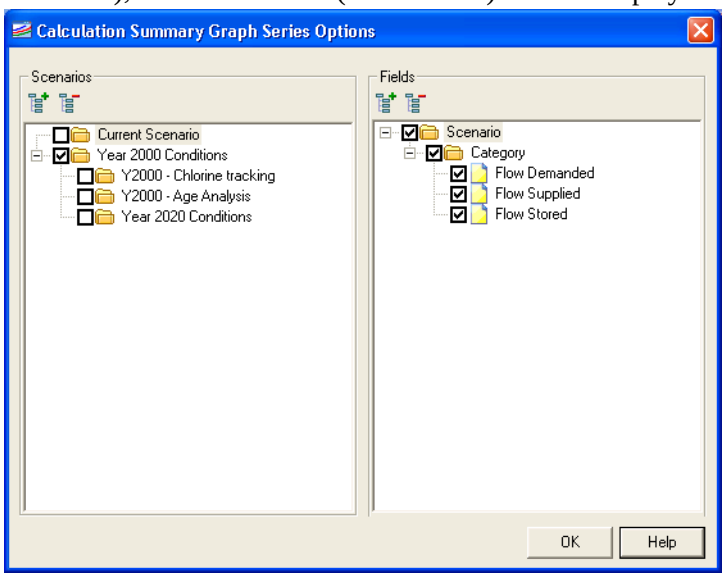
To obtain a Calculation Summary

1. Click Compute and the Calculation Summary box will open.
   or
2. From the Analysis Menu click Calculation Detailed Summary.



Calculation Summary Graph Series Options Dialog Box

The Calculation Summary Graph Series Options dialog box allows you to adjust the display settings for the calculation summary graph. You can define the scenario (or scenarios), and the attribute (or attributes) that are displayed in the graph.

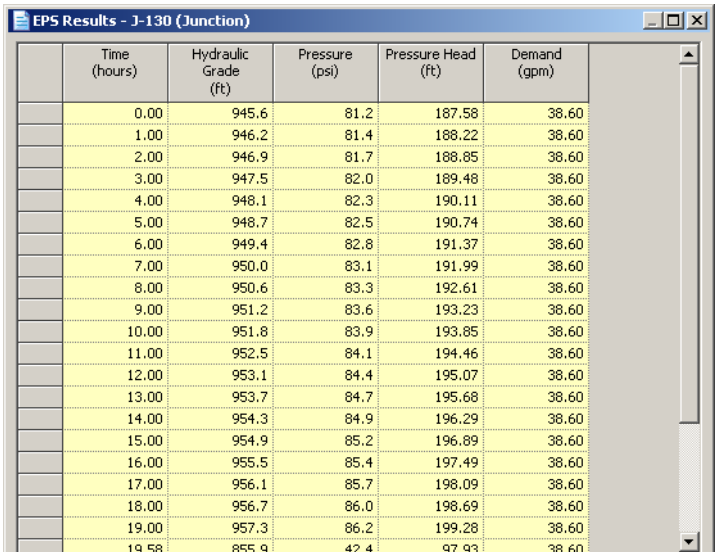


The Scenarios pane lists all of the available scenarios. Check the box next to a scenario to display the data for that scenario in the graph. The Expand All button opens all of the folders so that all scenarios are visible; the Collapse button closes the folders.

The Fields pane lists all of the available output fields. Check the box next to a field to display the data for that field type in the graph. The Expand All button opens all of the folders so that all fields are visible; the Collapse button closes the folders.

Results Table Dialog Box

The Results Table displays calculated results for each time step at the currently selected element.



Time (hours)	Hydraulic Grade (ft)	Pressure (psi)	Pressure Head (ft)	Demand (gpm)
0.00	945.6	81.2	187.58	38.60
1.00	946.2	81.4	188.22	38.60
2.00	946.9	81.7	188.85	38.60
3.00	947.5	82.0	189.48	38.60
4.00	948.1	82.3	190.11	38.60
5.00	948.7	82.5	190.74	38.60
6.00	949.4	82.8	191.37	38.60
7.00	950.0	83.1	191.99	38.60
8.00	950.6	83.3	192.61	38.60
9.00	951.2	83.6	193.23	38.60
10.00	951.8	83.9	193.85	38.60
11.00	952.5	84.1	194.46	38.60
12.00	953.1	84.4	195.07	38.60
13.00	953.7	84.7	195.68	38.60
14.00	954.3	84.9	196.29	38.60
15.00	954.9	85.2	196.89	38.60
16.00	955.5	85.4	197.49	38.60
17.00	956.1	85.7	198.09	38.60
18.00	956.7	86.0	198.69	38.60
19.00	957.3	86.2	199.28	38.60
19.58	855.9	42.4	97.93	38.60

Print Preview Window

The Print Preview window can be used to print documents, such as reports and graphs. You can see the current view of the document as it will be printed and define the print settings.

The following controls are available in the Print Preview window:

Search	Opens a Find dialog, allowing you to search for specified terms in the document.
Open	Opens a previously saved Preview Document File (.prnx).
Save	Saves the current prview as a Preview Document File
Print	Opens a Print dialog, allowing you to choose the printer, pages to be printed, and number of copies.

 **Quick Print**	Prints the document using the default printer.
 **Page Setup**	Opens the Page Seuip dialog, allowing you to specify the page setup settings, including page size, orientation, and margins.
 **Scale**	Opens a submenu that allows you to set the document scale.
 **Hand Tool**	Clicking this button toggles the Hand tool, which allows you to move the page around.
 **Magnifier**	Clicking this button toggles the Magnifier tool, which allows you to zoom the document view.
 **Zoom Out**	Zooms the page out.
Zoom	Displays the current zoom; also allows you choose the current zoom level.
 **Zoom In**	Zooms the page in.
 **First Page**	Sets the view to the first page of the document.
Previous Page	Sets the view to the previous page of the document.
Next Page	Sets the view to the next page of the document.
 **Last Page**	Sets the view to the last page of the document.
 **Multiple Pages**	Opens a submenu that allows you to define the number of pages that are viewed at once.
 **Color**	Opens a submenu that allows you to choose the background color of the document.
 **Watermark**	Opens the Watermark dialog, allowing you to define the watermark settings.

 **Export Document**	Opens the Export dialog, which allows you to define the export settings and export the document as one of the following document types: • PDF (.pdf) • HTML (.html) • MHT (.mht) • RTF (.rtf) • Excel (.xls) • CSV (.csv) • Text (.txt) • Image (.bmp, .gif, .jpg, .png, .tiff, .emf, .wmf)
 **Send via Email**	Opens the Export dialog, which allows you to define the export settings and export the document as one of the following document types: • PDF (.pdf) • HTML (.html) • MHT (.mht) • RTF (.rtf) • Excel (.xls) • CSV (.csv) • Text (.txt) • Image (.bmp, .gif, .jpg, .png, .tiff, .emf, .wmf) After the file is exported it is attached to an email, which you can then send using the specified email address and other settings.
Exit	Closes the Print Preview dialog.

Print Preparation

Detailed help for the Print Preparation feature can be found in the PrintPreparation.chm found in the Bentley/HAMMER folder.

Also note the following considerations

- **For Admins**: To set up a template, create the Legend rectangle by placing a **Viewport Area** and choosing the **Legend** mode.
- **For Users**: When creating a print model, it's important to note that you must perform an **Insert Legend from Element Symbology** command before the legend will show up in the print model. All the legends that you have inserted will show up in the viewport area that was set up in the template.

Importing and Exporting Data 16

Moving Data and Images between Model(s) and other Files

Importing a WaterCAD V8i Database

Exporting a HAMMER v7 Model

Importing and Exporting EPANET Files

Importing and Exporting Submodel Files

Exporting a DXF File

File Upgrade Wizard

Moving Data and Images between Model(s) and other Files

WaterCAD V8*i* offers numerous ways of moving data and images between models and to/from models and external files. Selecting the best approach can make the process easy. An overview of the different approaches and their suitability for various tasks is presented below. Each of these items is covered in greater detail elsewhere in the documentation.

1. **Copy/paste**:This is the easiest way to move tabular data to and from models. Simply highlight the data to be copied (or an entire table). Select Copy or CTRL-C. Move to where the data are to be placed. Select Paste or CTRL-V.
2. **ModelBuilder** (see Using ModelBuilder to Transfer Existing Data): This is best for moving data from GIS/CAD/database/spreadsheet sources to and from the model. Importing to the model is called "Synching in" (Build Model) and exporting from the model is called "Synching out". To move data between

models, first copy out to an intermediate file (e.g. shape file for element data, spreadsheet for component data). Two overall types of data can be moved to and from the model.

a. Element data consists of the actual pipes, nodes, etc that make up the model. ModelBuilder preserves the correct x-y coordinates and properties of the elements. This is useful for GIS/CAD data.

b. Component data and collections (e.g. pump definitions, patterns, unit demands) do not have spatial coordinates. These are written to a spreadsheet/ database file and then imported into another model.

3. **Import/Export Submodels** (see Importing and Exporting Submodel Files): This is used to create new models from subsets of another model, or to merge one model into another, or to create a new model from multiple existing models.
4. **Libraries** (see Engineering Libraries): These files can also be used to store component data (e.g. pump definitions, patterns) for use by other models. These are usually stored as XML files. For components that have libraries, it is usually easier to move data with the libraries instead of with ModelBuilder.
5. **LoadBuilder** (see Using LoadBuilder to Assign Loading Data): LoadBuilder is used to convert spatial demand/load data from a variety of source files into nodal load/demand values.
6. **TRex** (see Applying Elevation Data with TRex): Terrain extraction is used to convert a variety of digital elevation data into nodal elevation data.
7. **Flex Table to Shapefile** (see Viewing and Editing Data in FlexTables): From within a flex table, it is possible to create a shapefile for that type of element.
8. **Time series field data**: This is used to import field observations of element properties into the model for comparison with model results, especially in graphs. Copy/paste can be used as part of creation of time series field data.
9. **Import/Export EPANET** (see Importing and Exporting EPANET Files):This is used to move model data to or from EPANET. Because EPANET does not support as many features and properties as Bentley models, some data are lost.
10. **Import model data base** (see Importing a WaterCAD V8i Database): This is used to create a new model from a WaterGEMS, WaterCAD, or Hammer *.wtg.sqlite file. It differs from submodel import in that is creates a new project instead of appending the model to an existing model.
11. **DXF export** (see Exporting a DXF File): This creates a dxf file of the model which can be opened in CAD software like MicroStation.)
12. **Hyperlinks** (see Hyperlinks): These are used to attach external files (e.g. doc, jpg) to model elements.

13. **Background layers** (see Using Background Layers): These are used in the stand alone version to display a variety of raster and vector images behind the model. In other platforms, the display of background layers is controlled by the platform specific native software functions.
14. **Copy images to clipboard**: To move an image from the model to the clipboard for use in other applications (e.g. Word. PowerPoint), click on the dialog/image to get focus, select Alt-PrtSreen. Then paste from clipboard.
15. **Exporting Graphs and Profiles** (see Graphs and Using Profiles): Graphs and profiles created with the model can be exported to a variety of formats including BMP, JPG, PNG, and GIF from the Chart Options dialog.
16. **Shared tables** (see Viewing and Editing Data in FlexTables): Shared tables are used to store the format of flex tables so that they can be used by other models. These are stored in C:\Documents and Settings\<User Name>\Local Settings\Application Data\Bentley\<Product Name>\8 (under Windows 2003 Server/XP) or C:\Users\<User Directory>\AppData\Local\Bentley\<Product Name>\8 (under Windows Vista, Windows 7, and Server 2008). Highlight the flex table, right click, and select Duplicate > As shared flex table.

Importing a WaterCAD V8*i* Database

You can import a WaterCAD V8*i* database file, which will create a new model using the data in the database.

To import a WaterCAD V8*i* Database

1. Click the File menu, select Import, then choose WaterCAD V8*i* Database from the submenu.
2. Browse to and highlight the wtg.sqlite file to import.
3. Click Open.

Exporting a HAMMER v7 Model

You can export your model as a HAMMER v7 input file, which can then be opened in HAMMER v7.

To export a HAMMER v7 Input File

1. Click the File menu, select Export, then choose HAMMER 7.
2. Choose a file name and location for the HAMMER input file and click the Save button.
3. Click OK in the HAMMER Export prompt.

Importing and Exporting EPANET Files

You can import and export EPANET input files.

To import an EPANET file

1. Click the File menu, select Import, then choose EPANET from the submenu.
2. Browse to and highlight the .inp input file to import.
3. Click Open.

To export an EPANET file

1. Click the File menu, select Export, then choose EPANET from the submenu.
2. Type a name for the input file.
3. Click Save.

Importing and Exporting Submodel Files

Using the Submodel Import feature, you can import another model, or any portion thereof, into your project. Input data stored in the Alternatives as well as any supporting data (i.e. Patterns, Pump Definitions, Constituents, etc) will also be imported. It is important to notice that existing elements in the model you want to import the submodel into (i.e. the target model) will be matched with incoming elements by using their label. Incoming input data will override existing data in the target model for any element matched by its label. That also applies to scenarios, alternatives, calculation options and supporting data. Furthermore, any element in the incoming submodel that could not be matched with any existing element by their label, will be created in the target model.

For example, the submodel you want to import contains input data that you would like to transfer in two Physical Alternatives named “Smaller Pipes” and “Larger Pipes”. The target model contains only one Physical Alternative named “Larger Pipes”. In that case, the input data in the alternative labeled "Larger Pipes" in the submodel will replace the alternative with the same name in the target model. Moreover, the alternative labeled "Smaller Pipes" as well as its input data will be added to the target model without replacing any existing data on it because there is no existing alternative with the same label. Notice that imported elements will be assigned default values in those existing alternatives in the target model that could not be matched.

Notice that regular models can be imported as a submodel of a larger model as their file format and extension are the same.

For more information about input data transfer, see Exporting a Submodel.

Note: The label-matching strategy used during submodel import will be applied to any set of alternatives, including Active Topology alternatives. Therefore, if no Active Topology alternative stored in the submodel matches the existing ones in the target model, the imported elements will preserve their active topology values in the alternatives created from the submodel, but they will be left as "Inactive" in those previously existing alternatives in the target model. That is because the default value for the "Is Active" attribute in active topology alternatives other than the one that is current is "False".

To import a submodel

1. Click the File menu and select Import…Submodel.
2. In the Select Submodel File to Import dialog box, select the submodel file to be imported. Click the Open button.

Exporting a Submodel

You can export any portion of a model as a submodel for import into other projects. Input data is also stored in the file that is created in the process of Exporting a Submodel. This input data will be imported following a label-matching strategy for any element, alternative, scenario, calculation option or supporting data in the submodel. For more information about input data transfer, see Importing and Exporting Submodel Files.

To export a submodel

1. In the drawing view, highlight the elements to be exported as a submodel. To highlight multiple elements, hold down the Shift key while clicking elements.
2. Click the File menu and select Export…Submodel.
3. In the Select Submodel File to Export dialog box, specify the directory to which the file should be saved, enter a name for the submodel and click the Save button.

Importing a Bentley Water Model

For Bentley Water versions newer than the 2004 , please see the Bentley Water documentation regarding the Export to WaterCAD V8*i* command.

To import a Bentley Water 2004 Model

Click the File menu and select Import, then choose the Bentley Water 2004 Model command.

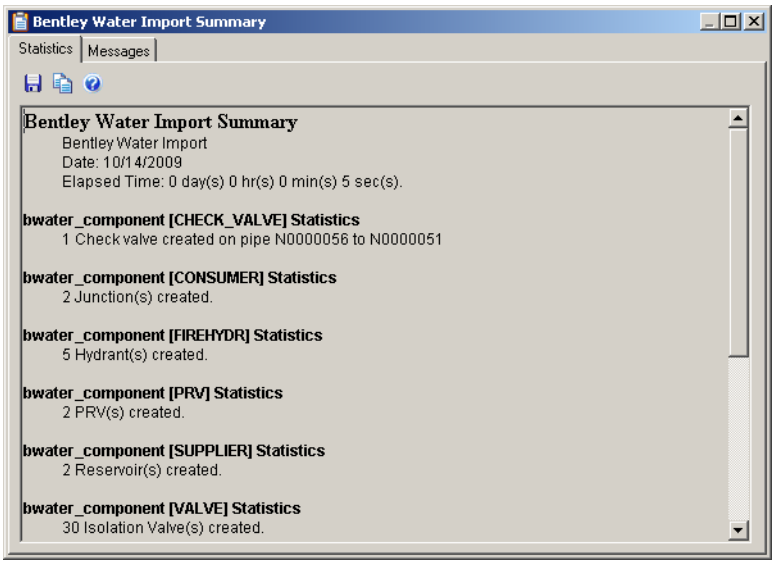
The Bentley Water Import wizard Opens. .

Specify the input data source by selecting a data source type, a data source, and a geometry data file (*.dat). If you want to update only those elements specified in the geometry data file, check the associated checkbox. Click Next.

Specify the node, pipe, component, adn elevation table names. When finished, click Next.

Specify the unit options for the model. When finished, click Finish.

Progress indicator runs. When completed, a Bentley Water Import Summary opens.



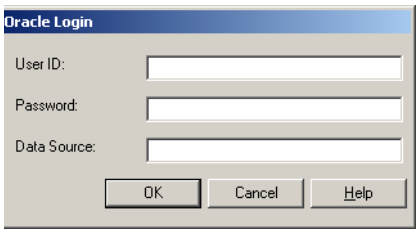
The Save button allows you to save the statistics to a Rich Text file (*.rtf). The Copy button copies the statistics to the Windows clipboard.

Close the Import Summary.

When prompted with “Do you wish to synchronize the drawing now?”, click “Yes” to synchronize immediately or “No” to synchronize later.

Oracle Login

This dialog appears when you choose an Oracle Spatial Data source.

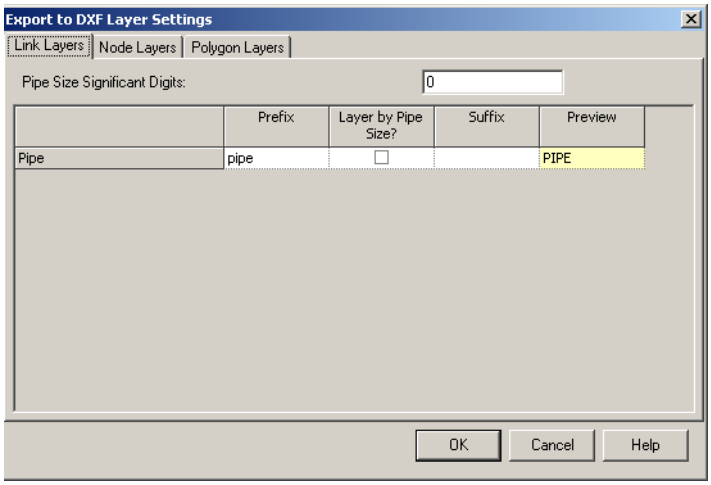


Enter the oracle User ID, Password, and Data Source, then click OK.

Exporting a DXF File

A project can be saved in .dxf format for use by AutoCAD and other CAD-based applications. When you use the Export command, you first specify the drive, directory, and file name of the .DXF file to be saved; then the Export to DXF Layer Settings window opens, allowing you specify the names of the .dxf layers on a per-element type basis.

The Export to DXF Layer Settings dialog is divided into tabs for Link Layers, Node Layers, and Polygon Layers.

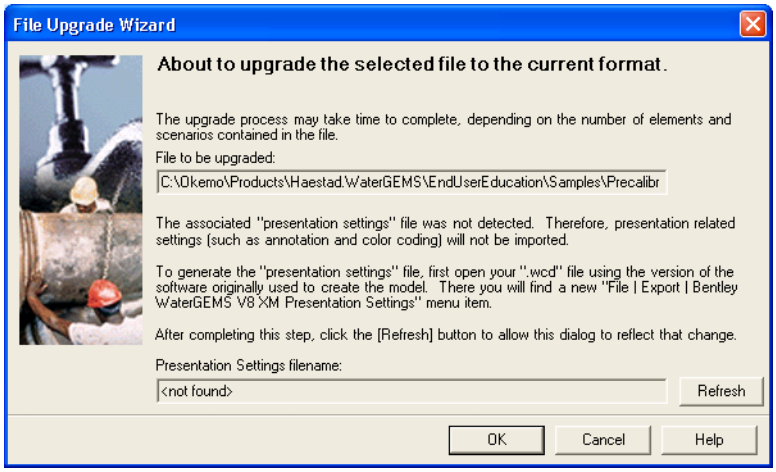


Each tab contains a table that allows you to specify a prefix and suffix for the associated dxf layer. The Preview field displays how the label will appear.

The Link Layers tab has additional controls: Entering a value in the Pipe Size Significant Digits field allows you to organize the pipe layer into multiple layers taking the pipe sizes into account using the Layer by Pipe Size checkbox.

File Upgrade Wizard

The File Upgrade Wizard allows you to allows you to upgrade older WaterCAD V8*i* database files to the most current format.



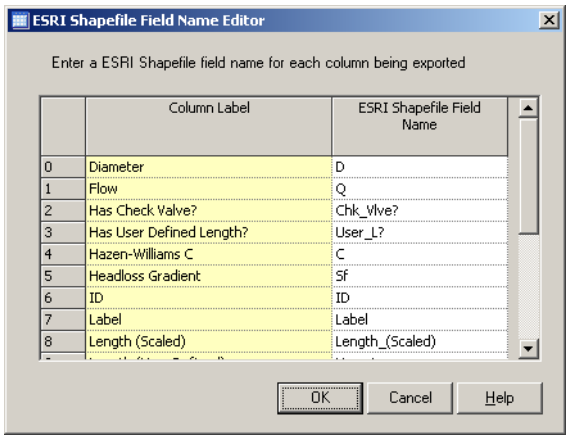
If you have v3 installed, installing v8 will add a new command to your v3 File>Export menu. Open the model to be upgraded in v3 and perform the **File>Export>Bentley** WaterCAD V8*i* **Presentation Settings** command to obtain a presentation settings file that can be used when upgrading the model file.

Export to Shapefile

It is possible to export model elements and data to create a shapefile. Unlike the other export features in Bentley WaterCAD V8*i* , the export to shapefile operation occurs in a FlexTable as opposed to the File > Export menu. Shapefiles must be created one element type at a time. That means there will be a separate shapefile to junctions, pipes, tanks, etc.

To create a shapefile, open the FlexTable for the type of element. Use selection sets or filtering to reduce the size of the FlexTable to what is desired in the shapefile. Use the table edit feature to eliminate any columns that are not desired.

When FlexTable is in correct form, pick the first button at the top left of the table which is the Export button. A Specify File Name to Export dialog ill open, allowing you to specify the file name and path for the shapefile. When the user names the file and clicks Save, the dialog below appears.



It is important to insure that any shapefile field names are less than or equal to 10 characters. The default name for shapefile field is the name of the column in the FlexTable. (If the user changes the name to something different from the FlexTable column name, the editor remembers it when other shapefiles are created from this table.) Once the names are acceptable, hit OK to create the shapefile. A shapefile consisting of .dbf, .shx and .shp files are created.

17 Menus

File Menu

Edit Menu

Analysis Menu

Components Menu

View Menu

Tools Menu

Report Menu

Help Menu

File Menu

The File menu contains the following commands:

New	Creates a new project. When you select this command, a new untitled project is created.
Open	Opens an existing project. When you select this command, the Open dialog box opens, so you can choose which program to open.
Close	Closes the current project without exiting the program.
Close All	Closes all currently open projects.
Save	Saves the current project.

Save As — Saves the current project under a new project name and/or to a different directory location.

Save All — Saves all currently open projects.

Update Server Copy — Updates the ProjectWise server copy using the current project.

Import — Opens a menu containing the following commands:

- WaterGEMS V8i/HAMMER **Database**—Opens a Select WaterCAD V8*i* Database File to Import window where you can choose the file to import (*.sqlite).
- **EPANET**—Opens a Select EPANET File to Import window where you can choose the file to import (*.inp).
- **Submodels**—Opens a Select Submodel File to Import window where you can choose the file to import (*.sqlite).

Bentley Water 2004 Edition Model—Opens a Bentley Water Import window where you can specify the output water model file.

Export — Opens a menu containing the following commands:

- DXF—Export the current network layout as a DXF drawing.
- EPANET—Opens a Select EPANET File to export window where you can choose the file to export (*.inp).
- **Submodels**—Export the current project to a Submodel file (*.sqlite).
- **HAMMER 7**—Export the current project to a WaterCAD V8*i* input file (.inp).
- **Publish i-model**—Opens the Publish to i-model dialog.

Seed

Seed files allow you to save project settings and data as a template (the seed file has an .sts extension). You can then reuse these settings/data while creating new projects using the data from the previously saved seed file. Selecting the Seed command opens a submenu containing the following commands:

- **New from Seed**: Allows you to create a new project using the previously saved seed file you specify.
- **Save to Seed**: Saves the current project settings and data as a seed file for reuse in future projects.

Page Setup

Opens the Page Setup dialog box where the print settings can be set up.

Print Preview

Opens a submenu containing the following commands:

- **Fit to Page**—Opens the Print Preview window, displaying the current view as it will be printed. The view will be zoomed in or out so that the current view fits to a single page of the default page size.
- **Scaled**—Opens the Print Preview window, displaying the current view as it will be printed. The view will be scaled so that it matches the user-defined drawing scale (this is defined on the Drawing Tab of the Options dialog: Tools > Options).

Print

Opens a submenu containing the following commands:

- **Fit to Page**—Prints the current view. The view will be zoomed in or out so that the current view fits to a single page of the default page size.
- **Scaled**—Prints the current view. The view will be scaled so that it matches the user-defined drawing scale (this is defined on the Drawing Tab of the Options dialog: Tools > Options).

Project Properties

Opens the Project Properties dialog box where Title, File Name, Engineer, Company, Date, and Notes can be added.

Recent Files	When the Recent Files Visible option is selected in the Options dialog box, the most recently opened files will appear in the File menu.
Exit	Closes the program.

Edit Menu

The Edit menu contains the following commands:

Undo	Cancels the last data input action on the currently active dialog box. Clicking Undo again cancels the second-to-last data input action, and so on.
Redo	Cancels the last undo command.
Delete	Deletes the currently highlighted element.
Select by Polygon	Selects elements by Polygon.
Select All	Selects all of the elements in the network.
Invert Selection	Selects all of the currently unselected elements in the drawing pane and deselects all of the currently selected elements.
Select by Element	Opens a menu listing all available element types. Select one of the element types from the submenu to select all elements of that type in the model.
Select by Attribute	Opens a menu listing all available attribute types. Select one of the attribute types from the menu and the Query Builder dialog box opens.
Clear Selection	Deselects the currently selected element(s).
Clear Highlight	Removes Network Navigator highlighting for all elements.
Find Element	Finds a specific element by entering the element's label.

Analysis Menu

The Analysis menu contains the following commands:

Scenarios	Opens the Scenario Manager, which allows you to create, view, and manage project scenarios.
Alternatives	Opens the Alternative Manager, which allows you to create, view, and manage alternatives.
Calculation Options	Opens the Calculation Options Manager, which allows you to create, view, and manage calculation settings for the project.
Post Calculation Processor	Opens the Post Calculation Processor dialog.
Totalizing Flow Meters	Opens the Totalizing Flow Meters manager where you can create new meters.
Hydrant Flow Curves	Opens the Hydrant Flow Curves dialog box, which allows you to view, edit, and create hydrant flow definitions.
System Head Curves	Opens the System Head Curves manager.
Energy Costs	Opens the Scenario Energy Cost Managerwhere you can view and compute energy costs.
Darwin Calibrator	Opens the Darwin Calibrator where you can create, edit, and run calibration studies.
Darwin Designer	Opens the Darwin Designer where you can create, edit, and run designer studies and design runs.
Darwin Scheduler	Opens the Darwin Scheduler where you can create, edit, and run scheduler studies and design runs.
Criticality	Opens the Segmentation and Criticality Manager where you can create new criticality scenarios.
Pressure Zone	Opens the Pressure Zone manager where you can identify elements that are located in a pressure zone based on the boundaries of the zone.
Time Browser	Opens the Time Browser dialog box, where you can manipulate the currently displayed time step and animate the drawing pane.

Fire Flow Results Browser	Opens the Fire Flow Results Browser, which allows you to quickly jump to fire flow nodes and display the results of fire flow analysis at the highlighted node.
Flushing Results Browser	Opens the Flushing Results Browser, allowing you to display the results of the flushing analysis at various locations.
Calculation Summary	Opens the Calculation Summary to view results.
User Notifications	Opens User Notifications allowing you to view warnings and errors uncovered by the validation process.
Validate	Runs a diagnostic check on the network data to alert you to possible problems that may be encountered during calculation. This is the manual validation command, and it checks for input data errors. It differs in this respect from the automatic validation that WaterCAD V8*i* runs when the compute command is initiated, which checks for network connectivity errors as well as many other things beyond what the manual validation checks.
Compute	Calculates the network. Prior to calculating, an automatic validation routine is triggered, which checks the model for network connectivity errors and performs other validation.

Components Menu

The Components menu contains the following commands:

Controls	Opens the Controls manager where you can set controls, conditions, actions, and logical control sets.
Zones	Opens the Zones manager where you can create, edit, duplicate, or delete zones.
Patterns	Opens the Patterns manager where you can create and edit patterns.

Pressure Dependent Demand Functions	Opens the Pressure Dependent Demand Functions manager where you can create and edit pressure dependent demands.
Unit Demands	Opens the Unit Demands manager where you can create and edit unit demands based on area, count and population.
Pump Definitions	Opens the Pump Definitions manager where you can create and edit pump definitions.
Minor Loss Coefficients	Opens the Minor Loss Coefficients Manager dialog.
GPV Headloss Curves	Opens the GPV Headloss Curves manager where you can create and edit headloss curves for General Purpose Valves.
Constituents	Opens the Constituents manager where you can create, edit, duplicate, or delete constituents.
Valve Characteristics	Opens the Valve Characteristics dialog.
Air Flow Curves	Opens the Air Flow Curves dialog.
Time Series Field Data	Opens the Time Series Field Data dialog.
Engineering Libraries	Opens the Engineering Libraries Manager.

View Menu

The View menu contains the following commands:

Element Symbology	Opens the Element Symbology Manager, which allows you to create, view, and manage annotation and color-coding in your project.
Background Layers	Opens the Background Layers Manager, which allows you to create, view, and manage the background layers associated with the project.
Network Navigator	Opens the Network Navigator.
Selection Sets	Opens the Selection Sets Manager, which allows you to create, view, and manage selection sets associated with the project.

Queries	Opens the Query Manager, where you can create SQL expressions for use with selection sets and FlexTables.
Prototypes	Opens the Prototypes Manager, where you can enter default values for elements in your model. Prototypes can reduce data entry requirements if a group of network elements share common data.
FlexTables	Opens the FlexTables Manager, where you can create, view, and manage the tabular reports for the project.
Graphs	Opens the Graph Manager, where you can create, view, and manage graphs for the project.
Profiles	Opens the Profile Manager, where you can create, view, and manage the profiles for the project.
Contours	Opens the Contours manager where you can create and edit contour definitions.
Named Views	Opens the Named Views manager where you can create, edit, and use Named Views.
Aerial View	Opens the Aerial View navigation window.
Properties	Turns the Properties Editor display on or off.
Property Grid Customizations	Opens the Property Grid Customizations Manager.
Auto-Refresh	Turns automatic updates to the main window view on or off whenever changes are made to the Bentley WaterCAD V8*i* datastore. When selected, a check mark indicates that automatic updates are turned on.
Refresh Drawing	Updates the main window view according to the latest information contained in the Bentley WaterCAD V8*i* datastore.

Zoom

Opens a menu containing the following commands:

- **Zoom Extents**—Sets the view so that the entire network is visible in the drawing pane.
- **Zoom Window**—Activates the manual zoom tool, which lets you specify a portion of the drawing to enlarge.
- **Zoom In**—Enlarges the size of the model in the drawing pane.
- **Zoom Out**—Reduces the size of the model in the drawing pane.
- **Zoom Realtime**—Enables the realtime zoom tool, which allows you to zoom in and out by moving the mouse while holding down the left mouse button.
- **Zoom Center**—Opens the Zoom Center dialog box, which allows you to enter drawing coordinates that will be centered in the drawing pane.
- **Zoom to Selection**—Enables you to zoom to specific elements in the drawing. You must select the elements to zoom to before you select the tool.
- **Zoom Previous**—Resets the zoom level to the last setting.
- **Zoom Next**—Resets the zoom level to the setting that was active before a Zoom Previous command was executed.

Pan

Activates the Pan tool, which allows you to move the model within the drawing pane. When you select this command, the cursor changes to a hand, indicating that you can click and hold the left mouse button and move the mouse to move the drawing.

Toolbars

Opens a menu that lists each of the available toolbars. Select one of the toolbars in the menu to turn that toolbar on or off.

Reset Workspace

Resets the Bentley WaterCAD V8*i* workspace so that the dockable managers appear in their default factory-set positions.

Tools Menu

The Tools menu contains the following commands:

Active Topology Selection	Opens a Select dialog to select elements in the drawing to make them Inactive or Active.
ModelBuilder	Opens the ModelBuilder Connections Manager, where you can create, edit, and manage ModelBuilder connections to be used in the model-building/model-synchronizing process.
TRex	Opens the TRex wizard where you can assign elevation to model nodes using data from outside sources.
SCADAConnect	Opens the SCADAConnect manager where you can add or edit SCADA connections.
Skelebrator Skeletonizer	Opens the Skelebrator manager, where you can define and perform skeletonization operations.
LoadBuilder	Opens the LoadBuilder manager where you can assign demands to model nodes using data from outside sources.
Thiessen Polygon	Opens the Wizard used to create Thiessen polygons for use with LoadBuilder.
Demand Control Center	Opens the Demand Control Center manager where you can add new demands, delete existing demands, or modify existing demands.
Unit Demand Control Center	Opens the Unit Demand Control Center manager where you can add new unit demands, delete existing unit demands, or modify existing unit demands.
Scenario Comparison	The scenario comparison tool enables you to compare input values between any two scenarios to identify differences quickly.
Hyperlinks	Associate external files, such as pictures or movie files, with elements in the model.

User Data Extensions	Opens the User Data Extension dialog box, which allows you to add and define custom data fields. For example, you can add new fields such as the pipe installation date.
Assign Isolation Valves to Pipes	Opens the Assign Isolation Valves to Pipes where you can find and assign isolation valves to their closest pipes according to user-defined tolerances.
Batch Pipe Split	Opens the Batch Pipe Split dialog.
Batch Morph	Opens the Batch Morph dialog.

Database Utilities Opens a menu containing the following commands:

- **Compact Database**—When you delete data from a Bentley WaterCAD V8*i* project, such as elements or alternatives, the database store that Bentley WaterCAD V8*i* uses can become fragmented, causing unnecessarily large data files, which impact performance substantially. Compacting the database eliminates the empty data records, thereby defragmenting the datastore and improving the performance of the file.

  Note: Every tenth time a file is saved, Bentley WaterCAD V8*i* will automatically prompt you to compact the database. If you open a file without saving it, the count does not go up. If you open and save a file multiple times in the same session, the count only goes up on the first save. If you open, save, and close the file, the count goes up. Click Yes to compact the database, or no to close the prompt dialog box without compacting. Since compacting the database can take time, especially for larger models, you may want to postpone the compact procedure until a later time. You can modify how Bentley WaterCAD V8*i* compacts the database in the Options dialog box.

- **Synchronize Drawing**—Synchronizes the current model drawing with the project database.
- **Update Database Cache**—Updates the current model to reflect any changes made in the database.
- **Update Results From Project Directory**—This command copies the model result files (if any) from the project directory (the directory where the project .sqlite file is saved) to the custom result file directory. The custom result directory is specified in Tools>Options>Project tab. This allows you to make a copy of the results that may exist in the model's save directory and replace the current results being worked on with them.
- **Copy Results to Project Directory**—This command copies the result files that are currently being used by the model to the project directory (where the project .sqlite is stored).

Layout	Opens a menu that lists each of the available element types. Select one of the element types to place that element in your model.
External Tools	Run an existing external tool or create a new one by opening up the External Tools manager.
Options	Opens the Options dialog box, which allows you to change Global settings, Drawing, Units, Labeling, and ProjectWise.

Report Menu

The Report menu contains the following commands:

Element Tables	Opens a menu that allows you to display FlexTables for any link or node element. These predefined FlexTables contain most of the input data and results for each instance of the selected element in the model.
Scenario Summary	Opens the Scenario Summary Report.
Project Inventory	Opens the Project Inventory Report, which contains the number of each of the various element types that are in the network.
Pressure Pipe Inventory	Opens the Pressure Pipe Inventory report.
Report Options	Opens the Report Options box where you can set Headers and Footers for the predefined reports.

Help Menu

The Help menu contains the following commands:

Bentley WaterCAD V8*i* Help	Opens the online help Table of Contents.
Quick Start Lessons	Opens the online help to the Quick Start Lessons Overview topic.
Welcome Dialog	Opens the Welcome dialog box.

Check for SELECT Updates	Opens your Web browser to the Bentley Web site, where you can check for Bentley WaterCAD V8*i* updates.
Bentley Institute Training	Opens your browser to the Bentley Institute Training web site.
Bentley Professional Services	Opens your browser to the Bentley Professional Services web site.
Bentley SELECT Support	Opens your browser to SELECTservices area of the Bentley web site.
Bentley Communities	Opens your browser to the BentleyCommunities section of the website.
Bentley.com	Opens the home page on the Bentley web site.
About Bentley WaterCAD V8*i*	Opens the About Bentley Bentley WaterCAD V8*i* dialog box, which displays copyright information about the product, registration information, and the current version number of the release.

Technical Reference 18

Pressure Network Hydraulics

Friction and Minor Loss Methods

Engineer's Reference

Variable Speed Pump Theory

Hydraulic Equivalency Theory

Thiessen Polygon Generation Theory

Method for Modeling Pressure Dependent Demand

References

Pressure Network Hydraulics

In practice, pipe networks consist not only of pipes but of miscellaneous fittings, services, storage tanks and reservoirs, meters, regulating valves, pumps, and electronic and mechanical controls.

Network Hydraulics Theory

For modeling purposes, these system elements are organized into the following categories:

- **Pipes**—Transport water from one location (or node) to another.
- **Junctions/Nodes**—Specific points, or nodes, in the system at which an event of interest is occurring. This includes points where pipes intersect, where there are major demands on the system such as a large industry, a cluster of houses, or a fire hydrant, or critical points in the system where pressures are important for analysis purposes.

- **Reservoirs and Tanks**—Boundary nodes with a known hydraulic grade that define the initial hydraulic grades for any computational cycle. They form the baseline hydraulic constraints used to determine the condition of all other nodes during system operation. Boundary nodes are elements such as tanks, reservoirs, and pressure sources.
- **Pumps**—Represented as nodes. Their purpose is to provide energy to the system and raise the water pressure.
- **Valves**—Mechanical devices used to stop or control the flow through a pipe, or to control the pressure in the pipe upstream or downstream of the valve. They result in a loss of energy in the system.

An event or condition at one point in the system can affect all other parts of the system. While this complicates the approach that the engineer must take to find a solution, there are some governing principles that drive the behavior of the network, including the Conservation of Mass and Energy Principle, and the Energy Principle.

The two modes of analysis are Steady-State Network Hydraulics and Extended Period Simulation. This program solves for the distributions of flows and hydraulic grades using the Gradient Algorithm.

The Energy Principle

The first law of thermodynamics states that for any given system, the change in energy is equal to the difference between the heat transferred to the system and the work done by the system on its surroundings during a given time interval.

The energy referred to in this principle represents the total energy of the system minus the sum of the potential, kinetic, and internal (molecular) forms of energy, such as electrical and chemical energy. The internal energy changes are commonly disregarded in water distribution analysis because of their relatively small magnitude.

In hydraulic applications, energy is often represented as energy per unit weight, resulting in units of length. Using these length equivalents gives engineers a better feel for the resulting behavior of the system. When using these length equivalents, the state of the system is expressed in terms of head. The energy at any point within a hydraulic system is often represented in three parts:

Pressure Head:	p/γ
Elevation Head:	z
Velocity Head:	$V^2/2g$

Where:	p	=	Pressure (N/m^2, $lb./ft.^2$)
	γ	=	Specific weight (N/m^3, $lb./ft.^3$)
	z	=	Elevation (m, ft.)
	V	=	Velocity (m/s, ft./sec.)
	g	=	Gravitational acceleration constant (m/s^2, $ft./sec.^2$)

These quantities can be used to express the headloss or head gain between two locations using the energy equation.

The Energy Equation

In addition to pressure head, elevation head, and velocity head, there may also be head added to the system, by a pump for instance, and head removed from the system due to friction. These changes in head are referred to as head gains and headlosses, respectively. Balancing the energy across two points in the system, you then obtain the energy equation:

$$\frac{p_1}{\gamma} + z_1 + \frac{V_1^2}{2g} + h_p = \frac{p_2}{\gamma} + z_2 + \frac{V_2^2}{2g} + h_L$$

Where:

p = Pressure (N/m2, lb./ft.2)

γ = Specific weight (N/m3, lb./ft.3)

z = Elevation at the centroid (m, ft.)

V = Velocity (m/s, ft./sec.)

g = Gravitational acceleration constant (m/s2, ft./sec.2)

hp = Head gain from a pump (m, ft.)

hL = Combined headloss (m, ft.)

The components of the energy equation can be combined to express two useful quantities, which are the hydraulic grade and the energy grade.

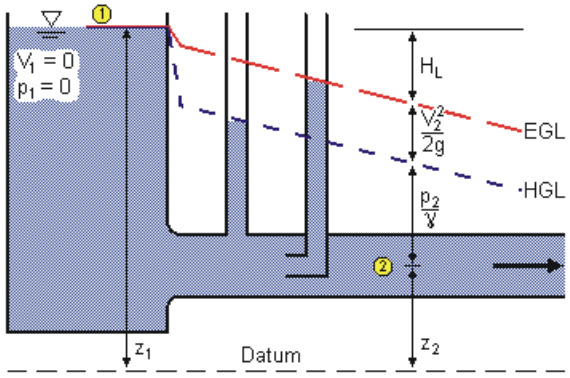
Hydraulic and Energy Grades

Hydraulic Grade

The hydraulic grade is the sum of the pressure head (p/γ) and elevation head (z). The hydraulic head represents the height to which a water column would rise in a piezometer. The plot of the hydraulic grade in a profile is often referred to as the hydraulic grade line, or HGL.

Energy Grade

The energy grade is the sum of the hydraulic grade and the velocity head ($V^2/2g$). This is the height to which a column of water would rise in a pitot tube. The plot of the energy grade in a profile is often referred to as the energy grade line, or EGL. At a lake or reservoir, where the velocity is essentially zero, the EGL is equal to the HGL, as can be seen in the following diagram.



EGL and HGL

Conservation of Mass and Energy

Conservation of Mass

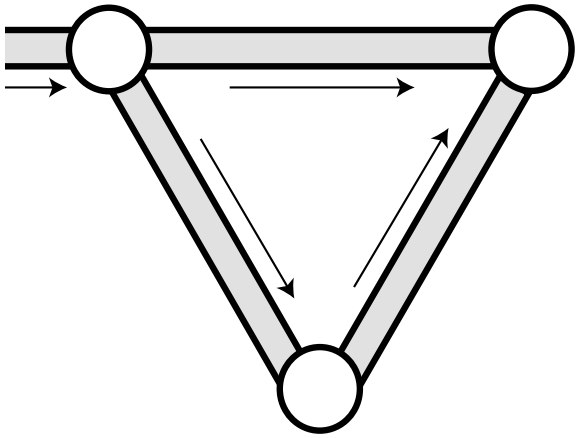
At any node in a system containing incompressible fluid, the total volumetric or mass flows in must equal the flows out, less the change in storage. Separating these into flows from connecting pipes, demands, and storage, you obtain:

$$\sum Q_{IN}\Delta t = \sum Q_{OUT}\Delta t + \Delta V_S$$

Where:	Q_{IN}	=	Total flow into the node (m^3/s, cfs)
	Q_{OUT}	=	Total demand at the node (m^3/s, cfs)
	ΔV_S	=	Change in storage volume (m^3, $ft.^3$)
	Δt	=	Change in time (s)

Conservation of Energy

The conservation of energy principle states that the headlosses through the system must balance at each point. For pressure networks, this means that the total headloss between any two nodes in the system must be the same regardless of what path is taken between the two points. The headloss must be sign consistent with the assumed flow direction (i.e., gain head when proceeding opposite the flow direction and lose head when proceeding in the flow direction).



Conservation of Energy

The same basic principle can be applied to any path between two points. As shown in the figure above, the combined headloss around a loop must equal zero in order to achieve the same hydraulic grade as at the beginning.

The Gradient Algorithm

The gradient algorithm for the solution of pipe networks is formulated upon the full set of system equations that model both heads and flows. Since both continuity and energy are balanced and solved with each iteration, the method is theoretically guaranteed to deliver the same level of accuracy observed and expected in other well-known algorithms such as the Simultaneous Path Adjustment Method (Fowler) and the Linear Theory Method (Wood).

In addition, there are a number of other advantages that this method has over other algorithms for the solution of pipe network systems:

- The method can directly solve both looped and partly branched networks. This gives it a computational advantage over some loop-based algorithms, such as Simultaneous Path, which require the reformulation of the network into equivalent looped networks or pseudo-loops.
- Using the method avoids the post-computation step of loop and path definition, which adds significantly to the overhead of system computation.
- The method is numerically stable when the system becomes disconnected by check valves, pressure regulating valves, or modeler's error. The loop and path methods fail in these situations.
- The structure of the generated system of equations allows the use of extremely fast and reliable sparse matrix solvers.

The derivation of the Gradient Algorithm starts with two matrices and ends as a working system of equations.

Derivation of the Gradient Algorithm

Given a network defined by N unknown head nodes, P links of unknown flow, and B boundary or fixed head nodes, the network topology can be expressed in two incidence matrices:

$A_{12} = A_{21}^T$ (P x N) Unknown head nodes incidence matrix

and

$A_{10} = A_{01}^T$ (P x B) Fixed head nodes incidence matrix

The following convention is used to assign matrix values:

$A_{12}^{(i,j)} = 1, 0, \text{ or } -1$ (PxN) Unknown head nodes incidence matrix

Assigned nodal demands are given by:

$q^T = [q_1, q_{2,\ldots}, q_N]$ (1 x N) Nodal demand vector

Assigned boundary nodal heads are given by:

$H_f^T = [H_{f1}, H_{f2},\ldots, H_{fB}]$ (1 x B) Fixed nodal head vector

The headloss or gain transform is expressed in the matrix:

$F^T(Q) = [f_1, f_2\ldots, f_p]$ (1 x P) Non-linear laws expressing headlosses in links

$f_i = f_i(Q_i)$

These matrix elements that define known or iterative network state can be used to compute the final steady-state network represented by the matrix quantities for unknown flow and unknown nodal head.

Unknown link flow quantities are defined by:

$Q^T = [Q_1,Q_2\ldots, Q_p]$ (1 x P) Unknown link flow rate vector

Unknown nodal heads are defined by:

$H^T = [H_1, H_2 \ldots, H_N]$ (1 x N) Unknown nodal head vector

These topology and quantity matrices can be formulated into the generalized matrix expression using the laws of energy and mass conservation:

$$A_{12}H + F(Q) = -A_{10}H_f$$

$$A_{12}Q = q$$

A second diagonal matrix that implements the vectorized head change coefficients is introduced. It is generalized for Hazen-Williams friction losses in this case:

$$A_{11} = \begin{bmatrix} R_1|Q_1|^{n_1-1} & & & & \\ & R_2|Q_2|^{n_2-1} & & & \\ & & \ldots & & \\ & & & \ldots & \\ & & & & R_P|Q_P|^{n_P-1} \end{bmatrix}$$

This yields the full expression of the network response in matrix form:

$$\begin{bmatrix} A_{11} & A_{12} \\ A_{21} & 0 \end{bmatrix} \begin{bmatrix} Q \\ H \end{bmatrix} = \begin{bmatrix} -A_{10}H_f \\ q \end{bmatrix}$$

To solve the system of non-linear equations, the Newton-Raphson iterative scheme can be obtained by differentiating both sides of the equation with respect to Q and H to get:

$$\begin{bmatrix} NA_{11} & A_{12} \\ A_{21} & 0 \end{bmatrix} \begin{bmatrix} dQ \\ dH \end{bmatrix} = \begin{bmatrix} -dE \\ dq \end{bmatrix}$$

with

$$N = \begin{bmatrix} n_1 & & & \\ & n_2 & & \\ & & \ldots & \\ & & & n_P \end{bmatrix}$$

The final recursive form of the Newton-Raphson algorithm can now be derived after matrix inversion and various algebraic manipulations and substitutions (not presented here). The working system of equations for each solution iteration, k, is given by:

$$H^{k+1} = -(A_{21}N^{-1}A_{11}{}^{-1}A_{12})^{-1}\left\{A_{21}N^{-1}(Q^k + A_{11}{}^{-1}A_{10}H_f) + (q - A_{21}Q^k)\right\}$$

$$Q^{k+1} = (1 - N^{-1})Q^k - N^{-1}A_{11}{}^{-1}(A_{12}H^{k+1} + A_{10}H_f)$$

The solution for each unknown nodal head for each time iteration is computationally intensive. This high-speed solution utilizes a highly optimized sparse matrix solver that is specifically tailored to the structure of this matrix system of equations.

Sources:

Todini, E. and S. Pilati, “A gradient Algorithm for the Analysis of Pipe Networks,” Computer Applications in Water Supply, Vol. 1—Systems Analysis and Simulation, ed. By Bryan Callback and Chin-Hour Or, Research Studies Press LTD, Watchword, Hertfordshire, England.

The Linear System Equation Solver

The Conjugate Gradient method is one method that, in theory, converges to an exact solution in a limited number of steps. The Gradient working equation can be expressed for the pressure network system of equations as:

$$Ax = b$$

where:

$$x = H^{k+1}$$

$$b = -\left\{A_{21}N^{-1}(Q^k + A_{11}^{-1}A_{10}H_f) + (q - A_{21}Q^k)\right\}$$

The structure of the system matrix A at the point of solution is:

$$A = A_{21}(NA_{11})^{-1}A_{12} = A_{21}DA_{12}$$

and it can be seen that the nature of the topological matrix components yield a total working matrix A that is:

- Symmetric
- Positive definite
- Stieltjes type.

Because of the symmetry, the number of non-zero elements to be retained in the matrix equals the number of nodes plus the number of links. This results in a low density, highly sparse matrix form. It follows that an iterative solution scheme would be preferred over direct matrix inversion in order to avoid matrix fill-in, which serves to increase the computational effort.

Because the system is symmetric and positive definite, a Cholesky factorization can be performed to give:

$$A = LL^T$$

where L is lower triangular with positive diagonal elements. Making the Cholesky factorization allows the system to be solved in two steps:

$$y = L^{-1}b$$

$$x = (L^{T})^{-1}y$$

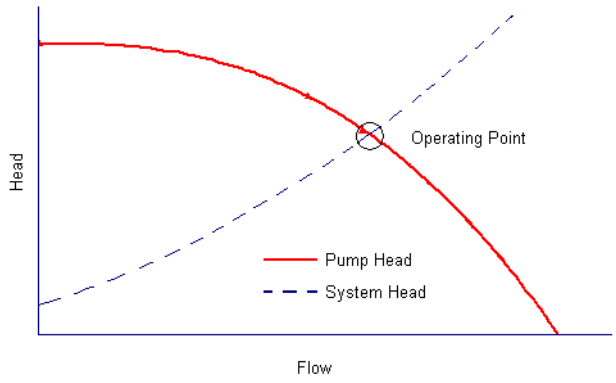
The use of this approach over more general sparse matrix solvers that implement traditional Gaussian elimination methods without consideration to matrix symmetry is preferred since performance gains are considerable. The algorithm utilized in this software solves the system of equations using a variant of Cholesky's method which has been optimized to reduce fill-in of the factorization matrix, thus minimizing storage and reducing overall computational effort.

Pump Theory

Pumps are an integral part of many pressure systems. Pumps add energy, or head gains, to the flow to counteract headlosses and hydraulic grade differences within the system.

A pump is defined by its characteristic curve, which relates the pump head, or the head added to the system, to the flow rate. This curve is indicative of the ability of the pump to add head at different flow rates. To model behavior of the pump system, additional information is needed to ascertain the actual point at which the pump will be operating.

The system operating point is based on the point at which the pump curve crosses the system curve representing the static lift and headlosses due to friction and minor losses. When these curves are superimposed, the operating point can easily be found. This is shown in the figure below.



System Operating Point

As water surface elevations and demands throughout the system change, the static head (Hs) and headlosses (HL) vary. This changes the location of the system curve, while the pump characteristic curve remains constant. These shifts in the system curve result in a shifting operating point over time.

Variable Speed Pumps

A pump's characteristic

curve is fixed for a given motor speed and impeller diameter, but can be determined for any speed and any diameter by applying the affinity laws. For variable speed pumps, these affinity laws are presented as:

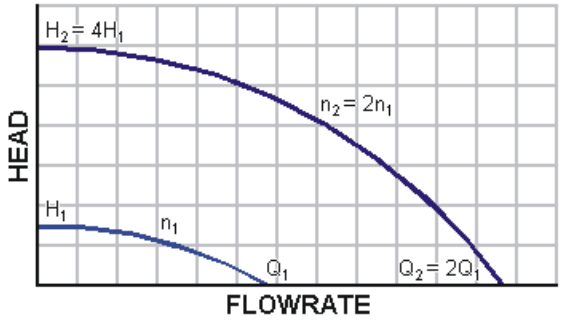
$$\frac{Q_1}{Q_2} = \frac{n_1}{n_2}$$

and

$$\frac{h_1}{h_2} = \left(\frac{n_1}{n_2}\right)^2$$

Where:

Q	=	Pump flow rate (m^3/s, cfs)
h	=	Pump head (m, ft.)
n	=	Pump speed (rpm)



Effect of Relative Speed on Pump Curve

Constant Horsepower Pumps

During preliminary studies, the exact characteristics of the constant horsepower pump may not be known. In these cases, the assumption is often made that the pump is adding energy to the water at a constant rate. Based on power-head-flow rate relationships for pumps, the operating point of the pump can then be determined. Although this assumption is useful for some applications, a constant horsepower pump should only be used for preliminary studies.

Note: It is not necessary to place a check valve on the pipe immediately downstream of a pump because pumps have built in check valves that prevent reverse flow.

This software currently models six different types of pumps:

Tip: Whenever possible, avoid using constant power or design point pumps. They are often enticing because they require less work on behalf of the engineer, but they are much less accurate than a pump curve based on several representative points.

- **Constant Power**—These pumps may be useful for preliminary designs and estimating pump size, but should not be used for any analysis for which more accurate results are desired.
- **Design Point (One-Point)**—A pump can be defined by a single design point (Hd @ Qd). From this point, the curve's interception with the head and discharge axes is computed as Ho = 1.33•Hd and Qo = 2.00•Qd. This type of pump is useful for preliminary designs but should not be used for final analysis.
- **Standard (Three-Point)**—This pump curve is defined by three points—the shutoff head (pump head at zero discharge), the design point (as with the single-point pump), and the maximum operating point (the highest discharge at which the pump performs predictably).
- **Standard Extended**—The same as the standard three-point pump but with an extended point at the zero pump head point. This is automatically calculated by the program.
- **Custom Extended**—The custom extended pump is similar to the standard extended pump, but allows you to enter the discharge at zero pump head.
- **Multiple Point**—This option allows you to define a custom rating curve for a pump. The pump curve is defined by entering points for discharge rates at various heads. Since the general pump equation, shown below, is used to simulate the pump during the network computations, the user-defined pump curve points are used to solve for coefficients in the general pump equation:

$$Y = A - (B \times Q^{C})$$

Where:	Y	=	Head (m, ft.)
	Q	=	Discharge (m^3/s, cfs)
	A,B,C	=	Pump curve coefficients

The Levenberg-Marquardt Method is used to solve for A, B and C based on the given multiple-point rating curve.

Valve Theory

There are several types of valves that may be present in a pressurized system. These valves have different behaviors and different responsibilities, but all valves are used for automatically controlling parts of the system. They can be opened, closed, or throttled to achieve the desired result.

Check Valves (CVs)

Check valves are used to maintain flow in only one direction by closing when the flow begins to reverse. When the flow is in the specified direction of the check valve, it is considered to be fully open.

WaterCAD V8*i* can model distribution system check valves in two ways.

1. A check valve can be specified as a property of a pipe. Flow is only permitted to go from the Start Node to the Stop Node.
2. A check valve node element can be placed in the network. In this case, flow is only permitted in the direction of the downstream pipe. If a check valve is to be used in a Hammer simulation, this type of check valve must be used.

Check valves are generally used on the suction side of pumps. WaterCAD V8*i* assumes that all pumps have a check valve on their downstream side. Therefore, a user should not specify a check valve there..

Flow Control Valves (FCVs)

FCVs are used to limit the maximum flow rate through the valve from upstream to downstream. FCVs do not limit the minimum flow rate or negative flow rate (flow from the To Pipe to the From Pipe). These valves are commonly found in areas where a water district has contracted with another district or a private developer to limit the maximum demand to a value that will not adversely affect the provider's system.

Pressure Reducing Valves (PRVs)

Pressure reducing valves are often used for separate pressure zones in water distribution networks. These valves prevent the pressure downstream from exceeding a specified level in order to avoid pressures that could have damaging effects on the system.

Pressure Sustaining Valves (PSVs)

A Pressure Sustaining Valve (PSV) is used to maintain a set pressure at a specific point in the pipe network. The valve can be in one of three states:

- Partially opened (i.e., active) to maintain its pressure setting on its upstream side when the downstream pressure is below this value.
- Fully open if the downstream pressure is above the setting.
- Closed if the pressure on the downstream side exceeds that on the upstream side (i.e., reverse flow is not allowed).

Pressure Breaker Valves (PBVs)

Pressure breaker valves create a specified headloss across the valve and are often used to model components that cannot be easily modeled using standard minor loss elements.

Throttle Control Valves (TCVs)

Throttle control valves simulate minor loss elements whose headloss characteristics change over time.

General Purpose Valves (GPVs)

GPVs are used to model situations and devices where you specify the flow-to-headloss relationship, rather than using standard hydraulic formulas. GPVs can be used to represent reduced pressure backflow prevention valves, well draw-down behavior, and turbines.

Friction and Minor Loss Methods

[Chezy's Equation]

[Colebrook-White Equation]

[Hazen-Williams Equation]

[Darcy-Weisbach Equation]

[Swamee and Jain Equation]

[Manning's Equation]

[Minor Losses]

Chezy's Equation

Chezy's equation is rarely used directly, but it is the basis for several other methods, including Manning's equation. Chezy's equation is:

$$Q = C \cdot A \cdot \sqrt{R \cdot S}$$

Where:	Q	=	Discharge in the section (m^3/s, cfs)
	C	=	Chezy's roughness coefficient ($m^{1/2}/s$, $ft.^{1/2}/sec.$)
	A	=	Flow area (m^2, $ft.^2$)
	R	=	Hydraulic radius (m, ft.)
	S	=	Friction slope (m/m, ft./ft.)

Colebrook-White Equation

The Colebrook-White equation is used to iteratively calculate for the Darcy-Weisbach friction factor:

Free Surface:

$$\frac{1}{\sqrt{f}} = -2\log\left[\frac{k}{12.0R} + \frac{2.51}{R_e\sqrt{f}}\right]$$

Full Flow (Closed Conduit):

$$\frac{1}{\sqrt{f}} = -2\log\left(\frac{k}{3.7D} + \frac{2.51}{R_e\sqrt{f}}\right)$$

Where:	f	=	Friction factor (unitless)
	k	=	Darcy-Weisbach roughness height (m, ft.)
	Re	=	Reynolds Number (unitless)
	R	=	Hydraulic radius (m, ft.)
	D	=	Pipe diameter (m, ft.)

Hazen-Williams Equation

The Hazen-Williams Formula is frequently used in the analysis of pressure pipe systems (such as water distribution networks and sewer force mains). The formula is as follows:

$$Q = k \cdot C \cdot A \cdot R^{0.63} \cdot S^{0.54}$$

Where:	Q	=	Discharge in the section (m^3/s, cfs)
	C	=	Hazen-Williams roughness coefficient (unitless)
	A	=	Flow area (m^2, $ft.^2$)
	R	=	Hydraulic radius (m, ft.)
	S	=	Friction slope (m/m, ft./ft.)
	k	=	Constant (0.85 for SI units, 1.32 for US units).

Darcy-Weisbach Equation

Because of non-empirical origins, the Darcy-Weisbach equation is viewed by many engineers as the most accurate method for modeling friction losses. It most commonly takes the following form:

$$h_L = f \frac{L}{D} \frac{V^2}{2g}$$

Where:	h_L	=	Headloss (m, ft.)
	f	=	Darcy-Weisbach friction factor (unitless)
	D	=	Pipe diameter (m, ft.)
	L	=	Pipe length (m, ft.)
	V	=	Flow velocity (m/s, ft./sec.)
	g	=	Gravitational acceleration constant (m/s^2, $ft./sec.^2$)

For section geometries that are not circular, this equation is adapted by relating a circular section's full-flow hydraulic radius to its diameter:

$D = 4R$

Where:	R	=	Hydraulic radius (m, ft.)
	D	=	Diameter (m, ft.)

This can then be rearranged to the form:

$$Q = A \cdot \sqrt{8g \cdot \frac{R \cdot S}{f}}$$

Where:	Q	=	Discharge (m^3/s, cfs)
	A	=	Flow area (m^2, $ft.^2$)
	R	=	Hydraulic radius (m, ft.)
	S	=	Friction slope (m/m, ft./ft.)
	f	=	Darcy-Weisbach friction factor (unitless)
	g	=	Gravitational acceleration constant (m/s^2, $ft./sec.^2$)

The Swamee and Jain equation can then be used to calculate the friction factor.

Swamee and Jain Equation

Note: The Kinematic Viscosity is used in determining the friction coefficient in the Darcy-Weisbach Friction Method. The default units are initially set by Bentley Systems.

$$f = \frac{1.325}{\left[\ln\left(\varepsilon/3.7D + 5.74/R_e^{0.9}\right)\right]^2}$$

Where:			
	f	=	Friction factor (unitless)
	ε	=	Roughness height (m, ft.)
	D	=	Pipe diameter (m, ft.)
	R_e	=	Reynolds Number (unitless)

The friction factor is dependent on the Reynolds number of the flow, which is dependent on the flow velocity, which is dependent on the discharge. As you can see, this process requires the iterative selection of a friction factor until the calculated discharge agrees with the chosen friction factor.

Manning's Equation

Note: Manning's roughness coefficients are the same as the roughness coefficients used in Kutter's equation.

Manning's equation, which is based on Chezy's equation, is one of the most popular methods in use today for free surface flow. For Manning's equation, the roughness coefficient in Chezy's equation is calculated as:

$$C = k \cdot \frac{R^{1/6}}{n}$$

Where:			
	C	=	Chezy's roughness coefficient ($m^{1/2}/s$, $ft.^{1/2}/sec.$)
	R	=	Hydraulic radius (m, ft.)
	n	=	Manning's roughness ($s/m^{1/3}$)
	k	=	Constant (1.00 $m^{1/3}/m^{1/3}$, 1.49 $ft.^{1/3}/ft.^{1/3}$)

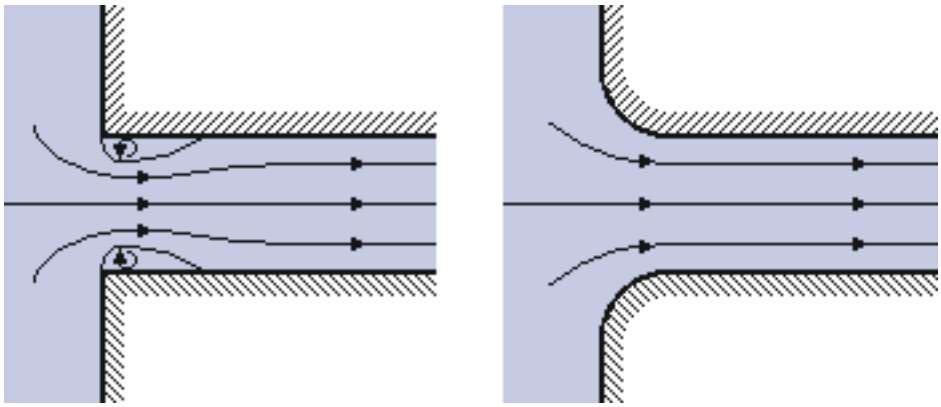
Substituting this roughness into Chezy's equation, you obtain the well-known Manning's equation:

$$Q = \frac{k}{n} \cdot A \cdot R^{2/3} \cdot S^{1/2}$$

Where:	Q	=	Discharge (m^3/s, cfs)
	k	=	Constant (1.00 $m^{1/3}$/s, 1.49 $ft.^{1/3}$/sec.)
	n	=	Manning's roughness (unitless)
	A	=	Flow area (m^2, $ft.^2$)
	R	=	Hydraulic radius (m, ft.)
	S	=	Friction slope (m/m, ft./ft.)

Minor Losses

Minor losses in pressure pipes are caused by localized areas of increased turbulence that create a drop in the energy and hydraulic grades at that point in the system. The magnitude of these losses is dependent primarily upon the shape of the fitting, which directly affects the flow lines in the pipe.



Flow Lines at Entrance

The equation most commonly used for determining the loss in a fitting, valve, meter, or other localized component is:

$$h_m = K\frac{V^2}{2g}$$

Where: h_m = Loss due to the minor loss element (m, ft.)

K = Loss coefficient for the specific fitting

V = Velocity (m/s, ft./sec.)

g = Gravitational acceleration constant (m/s^2, ft./sec. ^{2})

Typical values for fitting loss coefficients are included in the Fittings Table.

Generally speaking, more gradual transitions create smoother flow lines and smaller headlosses. For example, the figure below shows the effects of entrance configuration on typical pipe entrance flow lines.

Engineer's Reference

This section provides you with tables of commonly used roughness values and fitting loss coefficients.

Roughness Values—Manning's Equation

Commonly used roughness values for different materials are:

Manning's Coefficient (n) for Closed Metal Conduits Flowing Partly Full

Channel Type and Description	Minimum	Normal	Maximum
a. Brass, smooth	0.009	0.010	0.013
b. Steel			
1. Lockbar and welded	0.010	0.012	0.014
2. Riveted and spiral	0.013	0.016	0.017
c. Cast iron			
1. Coated	0.010	0.013	0.014

Manning's Coefficient (n) for Closed Metal Conduits Flowing Partly Full (Cont'd)

Channel Type and Description	Minimum	Normal	Maximum
2. Uncoated	0.011	0.014	0.016
d. Wrought iron			
1. Black	0.012	0.014	0.015
2. Galvanized	0.013	0.016	0.017
e. Corrugated metal			
1. Subdrain	0.017	0.019	0.021
2. Storm drain	0.021	0.024	0.030

Roughness Values—Darcy-Weisbach Equation (Colebrook-White)

Commonly used roughness values for different materials are:

Darcy-Weisbach Roughness Heights e for Closed Conduits

Pipe Material	ε (mm)	ε (ft.)
Glass, drawn brass, copper (new)	0.0015	0.000005
Seamless commercial steel (new)	0.004	0.000013
Commercial steel (enamel coated)	0.0048	0.000016
Commercial steel (new)	0.045	0.00015
Wrought iron (new)	0.045	0.00015
Asphalted cast iron (new)	0.12	0.0004
Galvanized iron	0.15	0.0005
Cast iron (new)	0.26	0.00085
Concrete (steel forms, smooth)	0.18	0.0006

Darcy-Weisbach Roughness Heights e for Closed Conduits

Pipe Material	ε (mm)	ε (ft.)
Concrete (good joints, average)	0.36	0.0012
Concrete (rough, visible, form marks)	0.60	0.002
Riveted steel (new)	0.9 ~ 9.0	0.003 - 0.03
Corrugated metal	45	0.15

Roughness Values—Hazen-Williams Equation

Commonly used roughness values for different materials are:

Hazen-Williams Roughness Coefficients (C)

Pipe Material	C
Asbestos Cement	140
Brass	130-140
Brick sewer	100
Cast-iron	
New, unlined	130
10 yr. Old	107-113
20 yr. Old	89-100
30 yr. Old	75-90
40 yr. Old	64-83
Concrete or concrete lined	
Steel forms	140
Wooden forms	120
Centrifugally spun	135
Copper	130-140
Galvanized iron	120
Glass	140

Hazen-Williams Roughness Coefficients (C)

Pipe Material	C
Lead	130-140
Plastic	140-150
Steel	
Coal-tar enamel, lined	145-150
New unlined	140-150
Riveted	110
Tin	130
Vitrified clay (good condition)	110-140
Wood stave (average condition)	120

Typical Roughness Values for Pressure Pipes

Typical pipe roughness values are shown below. These values may vary depending on the manufacturer, workmanship, age, and many other factors.

Comparative Pipe Roughness Values

Material	Manning's Coefficient n	Hazen-Williams C	Darcy-Weisbach Roughness Height	
			k (mm)	k (0.001 ft.)
Asbestos cement	0.011	140	0.0015	0.005
Brass	0.011	135	0.0015	0.005
Brick	0.015	100	0.6	2
Cast-iron, new	0.012	130	0.26	0.85
Concrete:				
Steel forms	0.011	140	0.18	0.6
Wooden forms	0.015	120	0.6	2
Centrifugally spun	0.013	135	0.36	1.2

Comparative Pipe Roughness Values (Cont'd)

Material	Manning's Coefficient n	Hazen-Williams C	Darcy-Weisbach Roughness Height	
Copper	0.011	135	0.0015	0.005
Corrugated metal	0.022	—	45	150
Galvanized iron	0.016	120	0.15	0.5
Glass	0.011	140	0.0015	0.005
Lead	0.011	135	0.0015	0.005
Plastic	0.009	150	0.0015	0.005
Steel				
Coal-tar enamel	0.010	148	0.0048	0.016
New unlined	0.011	145	0.045	0.15
Riveted	0.019	110	0.9	3
Wood stave	0.012	120	0.18	0.6

Fitting Loss Coefficients

For similar fittings, the K-value is highly dependent on things such as bend radius and contraction ratios.

Typical Fitting K Coefficients

Fitting	K Value	Fitting	K Value
Pipe Entrance		90° Smooth Bend	
Bellmouth	0.03-0.05	Bend Radius / D = 4	0.16-0.18
Rounded	0.12-0.25	Bend Radius / D = 2	0.19-0.25
Sharp-Edged	0.50	Bend Radius / D = 1	0.35-0.40
Projecting	0.80	Mitered Bend	
Contraction—Sudden		θ = 15°	0.05
D_2/D_1 = 0.80	0.18	θ = 30°	0.10

Typical Fitting K Coefficients (Cont'd)

Fitting	K Value	Fitting	K Value
$D_2/D_1 = 0.50$	0.37	$\theta = 45°$	0.20
$D_2/D_1 = 0.20$	0.49	$\theta = 60°$	0.35
Contraction—Conical		$\theta = 90°$	0.80
$D_2/D_1 = 0.80$	0.05	Tee	
$D_2/D_1 = 0.50$	0.07	Line Flow	0.30-0.40
$D_2/D_1 = 0.20$	0.08	Branch Flow	0.75-1.80
Expansion—Sudden		Cross	
$D_2/D_1 = 0.80$	0.16	Line Flow	0.50
$D_2/D_1 = 0.50$	0.57	Branch Flow	0.75
$D_2/D_1 = 0.20$	0.92	45° Wye	
Expansion—Conical		Line Flow	0.30
$D_2/D_1 = 0.80$	0.03	Branch Flow	0.50
$D_2/D_1 = 0.50$	0.08		
$D_2/D_1 = 0.20$	0.13		

Variable Speed Pump Theory

The variable speed pump (VSP) model within Bentley WaterCAD V8*i* lets you model the performance of pumps equipped with variable frequency drives. Variable frequency drives continually adjust the pump drive shaft rotational speed in order to maintain pressure and flow requirements in a network while improving energy efficiency and other operating characteristics as summarized by Lingireddy and Wood (1998);

- Minimization of excess pressures and energy usage,
- Leakage control through more precise pressure regulation,
- Flexible pump scheduling, improving off peak energy utilization,
- Control of tank drain and fill cycles,

- Improved system performance during emergency water usage events such as fires and main breaks,
- Reduction of transients produced when pumps start and stop,
- Simplification of flow control procedures.

Bentley WaterCAD V8*i* variable speed pumping feature will allow designers to make better decisions by empowering them to fully evaluate the advantages and disadvantages associated with VSPs for their unique application.

Within Bentley WaterCAD V8*i* there are two different ways to model VSPs depending on the data available to describe pump operations. The relative speed factor is a unitless number that quantifies the rotational speed of the pump drive shaft. 1) If the relative speed factor (or for EPS simulations a series of factors) is known, a pattern based VSP can be used. 2) If the relative speed factor is unknown, it can be estimated using the VSP with Bentley WaterCAD V8*i* new Automatic Parameter Estimation eXtension (APEX).

- **Pattern Based VSPs**—The variable speed pumping model lets you adjust pump performance using the relative speed factor. A single relative speed setting or a pattern of time varying relative speed factors can be applied to the pump. This is especially useful when modeling the operation of existing VSPs in your system.

  The Affinity Laws are used to adjust pump performance according to the relative speed factor setting.

  See Pump Theory for more information about pump curves.
- **VSPs with APEX**—APEX can be used in conjunction with the VSP model to estimate an unknown relative speed setting sufficient to maintain an operating objective. APEX uses an explicit algorithm to solve for unknown parameters directly (Boulos and Wood, 1990). This technique has proven to be powerful, robust, and computationally efficient for estimation of network parameters and has been improved to allow use for steady state and extended period simulations.

  To use APEX for estimating relative speed factors, the control node and control level setting for the pump must be selected and the pump curve and operating range for the pump must be defined. The following paragraphs provide guidelines for performing these tasks.
- **Control Node Location**—The location of the control node is an important consideration that affects pump operating efficiency, pressure maintenance performance, and, in rare instances, the stability of the parameter estimation calculation. The algorithm has been designed to allow multiple VSPs to operate within one pressure zone of a network; however, the pump and control node pairs should be decoupled from one another. In other words, a control node should be located such that only the pump it controls influences it. If the pressure zone of the model contains a tank or reservoir (hydraulic boundary conditions), consider making the

boundary condition the control node as opposed to selecting a pressure junction near the boundary. This will eliminate the possibility of specifying a set of hydraulic conditions that are impossible to maintain and thus reduce the possibility of computational failure.

- **Setting the Target Head**—The control node target head is the constant elevation of the hydraulic grade line (HGL) that the VSP will attempt to maintain. The target head at the control node must be within the physical limitations of the VSP as it has been defined (pump curve and maximum speed setting). If the target head is greater then the maximum head, the pump can generate at the demanded flow rate the pump will automatically revert to fixed speed operation at the maximum relative speed setting, and the target head will not be maintained.

  Tip: Navigating to the target head settings—The VSP target head for junction nodes can be set on the VSP tab of the Pump dialog box and for tanks on the Section tab of the Tank dialog box by adjusting the initial level.

- **Setting the Maximum Relative Speed Factor**—For flexible operation, a variable speed drive and pump should be configured such that it can efficiently operate over a range of speeds to satisfy the pressure and flow requirements it will be subject. The value selected for the maximum relative speed factor depends on the normal operating range of the drive motor. To set the proper maximum value, you must determine the drive motor's normal operating speed and maximum operating speed (the maximum speed at which the drive motor normally operates, not the speed at which the drive catastrophically fails). The relative speed factor is defined as the quotient of the current operating speed and the normal operating speed. Thus the maximum relative speed factor is the maximum operating speed of the drive divided by the normal operating speed. For example, a maximum relative speed factor of 2.0 means that the maximum speed is two times the normal operating speed, and a maximum relative speed factor of 1.0 means that the maximum operating speed is equal to the normal operating speed.
- **Defining the Pump Curve**—In order to determine the relative speed factor using APEX, the pump curve must be smooth and continuously differentiable; thus a one point or three point power function curve definition must be used. For best results, the curve should be defined for the normal operating speed of the pump (corresponding to a relative speed factor equal to 1.0, regardless of the maximum speed setting).

Variable speed pump theory includes:

VSP Interactions with Simple and Logical Controls

VSP Interactions with Simple and Logical Controls

The VSP model and APEX have been designed to fully integrate with the simple and rule based control framework within Bentley WaterCAD V8*i* . You must keep in mind that the definition of controls requires that the state (On, Off, Fixed Speed Override) and speed setting of a VSP be properly managed during the simulation. Therefore, the interactions between VSPs and controls can be rather complex. We have tried to the extent possible to simplify these interactions while maintaining the power and flexibility to model real world behaviors. The paragraphs that follow describe guidelines for defining simple and logical controls with VSPs.

- **Pattern based VSPs**—The pattern of relative speed factors specified for a VSP takes precedence over all simple and logical control commands. Therefore, the use of controls with pattern based VSPs is not recommended. Rather, the pattern of relative speed factors should be defined such that control objectives are implicitly met.
- **VSPs with APEX**—A VSP can be switched into any one of three different states. When the VSP is On, the APEX will estimate the relative speed sufficient to maintain a constant pressure head at the control node. When the VSP is Off, the relative speed factor and flow through the pump are set to zero, and the pressure head at the control node is a function of the prevailing network boundary and demand conditions. When the control state of a VSP is Fixed Speed Override, the pump will operate at the maximum speed setting and the target head will no longer be maintained. The Temporarily Closed state for a VSP indicates that the check valve (CV) within the pump has closed in response to prevailing hydraulic conditions, and that the target head cannot be maintained. The VSP control node can be specified at any junction node or tank in a network model. As described below, however, the behavior of simple and logical controls depends on the type of control node selected.
- **Junction Nodes**—When the VSP control node type selected is a junction node, the VSP will behave according to some automatic behaviors in addition to the controls defined for the pump. If the head at the control node is above the target head, the pump state will automatically switch to Off. If the head at the control node is less then the target head, the pump state will automatically switch to On. The VSP will automatically switch into and out of the Fixed Speed Override and Temporarily Closed states in order to maintain the fixed head at the control node and prevent reverse flow through the pump. Additional controls can be added to model more complex use cases.
- **Tanks**—When the VSP control node is a tank, you must manage the state of the pump through control definitions, allowing for flexible modeling of the complex control behaviors that may be desired for tanks. If a VSP has a state of On, the pump will maintain the current level of the tank. For example, at the beginning of a simulation, if a VSP has status of on it will maintain the initial level of the tank. As the simulation progresses and the pump happens to turn off, temporarily close, or go into fixed speed override, the level in the tank will be determined in

response to the hydraulic conditions prevailing in the network. When the VSP turns on again, it will maintain the current level of the tank, not the initial level. Thus control statements must be written that dictate what state the pump should switch to depending on the level in the tank. A pump station with a VSP and a fixed-speed pump operating in a coordinated fashion can be used to model tank drain and fill operations.

•

Performing Advanced Analyses

The VSP model is fully integrated with the Energy Cost Manager for easy estimation of pump operating costs. When comparing the energy efficiency of fixed speed and variable speed pumps, however, it is important to bear in mind that the pumps are not maintaining the same pressures in the network. The performance of the pumps should be compared in such a way that takes this difference into account; otherwise the comparison is of little value. For example, consider a comparison between a VSP and a fixed-speed pump is prepared, but the target head at the control node is greater than the head maintained there by the fixed speed pump. The VSP energy efficiency numbers will be disappointing because the VSP is maintaining higher pressures.

The concept of a minimum acceptable head (or pressure) can be useful when evaluating the performance of fixed speed and variable speed pumps. Both pumps should be sized and operated such that the pressure is equal to or greater than the minimum acceptable head. In this way, the heads maintained by the respective pumps can be used to define equivalency between the respective designs. When the comparison is thoughtfully designed and conducted, it is likely that the energy efficiency improvements possible with VSPs will come to light more clearly.

Hydraulic Equivalency Theory

This section outlines the rules that Skelebrator uses for creating equivalent pipes from parallel or series pipes.

These equations can be solved for equivalent diameter or roughness (C, n or k). With the Darcy-Weisbach equation, the equations are solved only for D because there are situations where the roughness can be negative. Both solutions are presented. In general, there will be one pipe that is the dominant pipe, and the properties of that pipe will be used when a decision must be made. There will be some default rule for picking the dominant pipe, but you will be able to override it.

You will not use equivalent lengths because you want to preserve the system geometry. For pipes in parallel, you will use the length of the dominant pipe while for pipes in series, you will add the lengths of the two pipes as follows:

$L_r = L_1 + L_2$

Principles

The equations derived below are based on the following principles. The equations below are for two pipes but can be extended to n pipes.

For pipes in series:

$Q_r = Q_1 = Q_2$

where Q = flow, r refers to the resulting pipe, and 1 and 2 refer to the pipes being removed.

$h_r = h_1 + h_2$

For pipes in parallel:

$Q_r = Q_1 + Q_2$

and

$h_r = h_1 = h_2$

As long as the units are consistent, then any appropriate units can be used. For example, if the diameters are in feet, then the resulting diameter will be in feet.

Hazen-Williams Equation

$$h = \frac{KL}{D^{4.87}}\left(\frac{Q}{C}\right)^{1.85}$$

K depends on the units but cancels out in equivalent pipe calculations.

Series Pipes

For series pipes, the length is based on the sum of the lengths.

Solved for C:

$$C_r = \frac{\frac{L_r^{0.54}}{D_r^{2.63}}}{\left(\sum \frac{L_i}{D_i^{4.87} C_i^{1.85}}\right)^{0.54}}$$

Solved for D:

$$D_r = \frac{\frac{L_r^{0.205}}{C_r^{0.38}}}{\left(\sum \frac{L_i}{D_i^{4.87} C_i^{1.85}}\right)^{0.205}}$$

Parallel Pipes

Solved for C:

$$C_r = \frac{L_r^{0.54}}{D_r^{2.63}} \sum \frac{C_i D_i^{2.63}}{L_i^{0.54}}$$

Solved for D:

$$D_r = \left(\frac{L_r^{0.54}}{C_r} \sum \frac{C_i D_i^{2.63}}{L_i^{0.54}}\right)^{0.38}$$

Manning's Equation

$$h = \frac{KL(nQ)^2}{D^{5.33}}$$

Series Pipes

Solved for n:

$$n_r = \frac{D_r^{2.66}}{L_r^{0.5}}\left(\sum \frac{L_i n_i^2}{D_i^{5.33}}\right)^{0.5}$$

Solved for D:

$$D_r = \left(\frac{L_r n_r^2}{\sum \frac{L_i n_r^2}{D_i^{5.33}}}\right)^{0.188}$$

Parallel Pipes

Solved for n:

$$n_r = \frac{\frac{D_r^{2.66}}{L_r^{0.5}}}{\sum \frac{D_i^{2.66}}{L_i^{0.5} n}}$$

Solved for D:

$$D_r = \left(L_r^{0.5} n \sum \frac{D_i^{2.66}}{L_i^{0.5} n}\right)^{0.376}$$

Darcy-Weisbach Equation

$$h = \frac{KLfQ^2}{D^5}$$

It is the roughness k—not f—that is a property of the pipe. While f behaves well, the roughness can take on negative values in the parallel pipe case. Therefore, only solutions for D will be developed.

The other problem with the Darcy-Weisbach equation is that D and f are not uniquely related and depend on the Reynolds number, which is a function of velocity. So the question that must be first answered is, Which value of f should be used in the equations? This is especially tricky when the individual pipes have different values of k. First, a velocity of 1 m/s will be used as a reference velocity to calculate Reynolds number for the individual pipes. Second, an iterative solution must be used to solve for D.

That is

1. Pick a D and k based on the dominant pipe.
2. Calculate f for the resultant pipe using Swamee-Jain formula.
3. Use that f for fr in the equations below.
4. Check if Dr is close enough to D used to calculate f.
5. Repeat until convergence.

The Swamee-Jain equation is

$$f = \frac{1.325}{\left[\ln\left(\frac{k}{3.7D} + \frac{5.74}{Re^{0.9}}\right)\right]^2}$$

where

$$Re = \frac{VD}{\nu}$$

ν must be selected so that the units cancel. Typical values are 1.00e-6 m^2/s or 1.088e-5 $ft.^2/sec$.

Series Pipes

$$D_r = \left(\frac{L_r f_f}{\sum \frac{L_i f_i}{D_i^5}} \right)^{0.2}$$

Parallel Pipes

$$D_r = \left(L_r f_r \left(\sum \frac{D_i^{2.5}}{(L_i f_i)^{0.5}} \right)^2 \right)^{0.2}$$

Check Valves

For series pipes, if any pipe has a check valve, then the resulting pipe will have a check valve. For parallel pipes, if both pipes have check valves, then the resulting pipe will have a check valve.

The degenerative case is when one of the parallel pipes has a check valve. This should not happen in terms of good engineering. If it does, the parallel pipes should not be combined and a warning message should be issued.

Minor Losses

For pipes in series, the minor loss coefficients should be added. The differences in diameter between the original pipe and the resulting pipe should be negligible. You should be given the option to ignore minor losses in series pipes.

For pipes in parallel, you should be given the option to ignore minor losses, not skeletonize pipes with significant minor losses (e.g., if total Km > 100) or account for them as a change in diameter.

One possible short heuristic for handling minor losses in parallel pipes is to realize that you are splitting the minor loss over two pipes. If the pipes are roughly the same length, roughness, and diameter, then the minor loss coefficient will be cut approximately in half. I worked through the math for coming up with an equivalent minor loss coefficient and it's a mess. Using half the minor loss coefficient isn't exactly correct, but it pretty much accounts for things.

Numerical Check

To check the equations, run through examples of each. Solve for head loss in each pipe individually and then combine to see how the head loss in the equivalent pipe compares for series pipes and for parallel, see how the flow compares. Stick with the SI units (i.e., flow in m^3/s, D, L and h in m).

Series

Use Q = 1 m^3/s and solve for head loss. Pipe 1 is the dominant pipe.

Comparison between the Sum of the Headlosses from the Two Pipes and the Headloss from the Equivalent Pipe

	Pipe 1	Pipe 2	Resulting, solve for D	Resulting, solve for C,n
Length	100	80	180	180
Diameter	1	0.75	0.88	0.75k, 0.855n
C	100	120	100	71
k	0.002	0.0015	0.002	X
n	0.013	0.012	0.013	0.0197
h (Hazen)	0.21	0.49	0.72	0.72
h (Manning)	0.17	0.55	0.72	0.72
h (Darcy)	0.20	0.58	0.77	X

Parallel

Use head loss = 1 m and solve for Q.

Comparison between the Sum of the Flows from the Two Pipes and the Flow from the Equivalent Pipe

	Pipe 1	Pipe 2	Resulting, solve for D	Resulting, solve for C,n
Length	100	80	100	100
Diameter	1	0.75	0.88	1.18n, 1.21k
C	100	120	100	163
k	0.002	0.0015	0.002	X
n	0.013	0.012	0.013	0.0083
Q (Hazen)	2.31	1.47	3.74	3.77
Q (Manning)	2.40	1.35	3.72	3.75
Q (Darcy)	2.26	1.31	3.55	X

Thiessen Polygon Generation Theory

Naïve Method

Plane Sweep Method

Naïve Method

A Thiessen polygon of a site, also called a Voronoi region, is the set of points that are closer to the site than to any of the other sites.

Let $P = \{p_1, p_2,\ldots p_n\}$ be the set of sites and $V = \{v(p_1), v(p_2),\ldots v(p_n)\}$ represent the Voronoi regions or Thiessen polygons for P_i, which is the intersection of all of the half planes defined by the perpendicular bisectors of p_i and the other sites. Thus, a naïve method for constructing Thiessen Polygons can be formulated as follows:

Step 1 For each i such that $i = 1, 2,\ldots, n$, generate n - 1 half planes $H(p_i,p_j)$, 1 </= j </ = n, i <> j, and construct their common intersection $v(p_i)$.

Step 2 Report $V = \{v(p_1), v(p_2),\ldots v(p_n)\}$ as the output and stop.

This naïve procedure is, however, very inefficient for generating Thiessen polygons. The computation time increases exponentially as the number of sites increases. There are many other more competent methods for constructing a Thiessen polygon.

Plane Sweep Method

The plane sweep technique is a fundamental method for solving two-dimensional geometric problems. It works with a special line called a sweepline, a vertical line sweeping the plane from left to right. It hits objects one by one as the sweepline moves. Whenever it crosses an object, a portion of the problem is solved. Therefore, it enables a two-dimensional problem to be solved in a sequence of one-dimension processing. Sweep plane technique provides a conceptually simple and efficient algorithm. Steven Fortune (1986; 1987) has developed a sweepline algorithm for constructing Thiessen polygons. This algorithm has been implemented in the WaterCAD V8*i* Thiessen Polygon Generator. The detailed working algorithm is given as follows:

1. Q <------- P.
2. Choose and delete the left-most point, say p_i from Q.
3. L <------- the list consisting of a single region $\varphi(V(p_i)$.
4. While Q is not empty, repeat Steps 1-3.
5. If w is a site, say $w = p_i$, do:
   a. Find region $\varphi(V(p_i)$ on L containing p_i.
   b. Replace $\varphi(V(p_i)$ on L by the sequence ($\varphi(V(p_j)$, $h^-(pi, pj)$, ($\varphi(V(p_i))$, $h^+(pi, pj)$, $\varphi(V(p_j)$.
   c. Add to Q the intersection of $h^-(pi, pj)$ with the intermediate lower half hyperbola on L and the intersection of $h^+(pi, pj)$ with the immediate upper half hyperbola on L.
6. If w is an intersection, say $w = \varphi(q_t)$, do:
   a. Replace sub-sequence ($h^{\pm}(p_i, p_j)$, $\varphi(V(p_i))$, $h^{\pm}(p_i, p_k)$) on L by $h = h^-(p_i, p_k)$ or $h = h^+(p_i, p_k)$ appropriately.
   b. Delete from Q any intersection of $h^{\pm}(p_i, p_j)$ or $h^{\pm}(p_i, p_k)$ with others.

c. Add to Q any intersection of h with its immediate upper half hyperbola and its immediate lower half parabola on L.

d. Mark $\varphi(q_t)$ as a Voronai vertex incident to $h^{\pm}(p_i, p_j)$, $h^{\pm}(p_i, p_k)$, and h.

7. Repeat all half hyperbolas ever listed on L, all the Voronai vertices marked in the preceding step, and the incidence relations among them.

The sweepline algorithm is an efficient technique for constructing a Thiessen polygon. The computation time required for the worst case is $O(n\log n)$. It produces a far more competent method than the naïve method and provides satisfactory performance for generating Thiessen polygons for a large number of points.

Method for Modeling Pressure Dependent Demand

A water distribution system does not always supply the required or normal demand to customers under all conditions. It is important for water companies to be informed to what degree or level that a water system is able to supply its customers when an emergency or calamity scenario occurs. A calamity event can be one or more than one element out of service. When such an event occurs, it is expected that the service can only be maintained to a certain level before the outage is fully recovered.

In order to deal with a recoverable calamity, the concept of water supply is introduced to quantify the supply capacity of a water distribution system. It is defined as a percentage of the supplied demand over the normal demand. Water companies are required to comply the minimum water supply level under a calamity of one element outage, which is expected to be fully repaired within 24 hours. The modeling approach for evaluating water supply level for the use cases as follows.

Use Cases

Supply Level Evaluation

Pressure Dependent Demand

Demand Deficit

Solution Methodology

Modified GGA Solution

Direct GGA Solution

Use Cases

In 1994, the Dutch water authority posted the guideline for water companies to evaluate the level of water supply while coping with calamity events. A tentative guideline requirement is that a water system must meet 75% of the original demand for the majority of customers and no large group of customers (2000 resident addresses) should receive less than 75% of their original demand.

The guideline is applicable to all the elements between the source and tap in a water system and is required to find the effect of every element. In order to calculate the water supply level under a calamity event, a hydraulic modeling approach is proposed:

1. Take one element at a time out of a model, copying the calamity event of element outage
2. Run the model for peak hours of all demand types and also the peak hours of tank filling. The actual demand needs to be modeled as a function of pressure; the supply is considered unaffected if the pressure is above the required pressure threshold
3. Evaluate the water supply level for each demand node. If there is less than 2000 resident customers receiving less than 75% of the normal demand, then the requirement is met. Repeat Step 1 to simulate another calamity event. If the requirement is not met, continue with step 4.
4. Perform 24 hours pressure dependent demand simulation for the maximum demand day under the calamity even
5. Sum up the actual demand for each node over 24 hours
6. Check if there is any node where the totalized demand over 24 hours is less than 75% of the maximum day demand; if not, the guideline is met. Otherwise an appropriate system improvement needs to be identified in order to meet the guideline.

UK water companies are required by law to provide water at a pressure that will, under normal circumstances, enable it to reach the top floor of a house. In order to assess if this requirement is satisfied, companies are required to report against a service level corresponding to a pressure head of 10 meters at a flow of 9 liters per minute. In addition, water companies are also required to report the supply reference for unplanned and planned service interruptions.

Both use cases provide some generality for water utilities world wide to evaluate the performance of water systems under emergency and low pressure conditions. An emergency event can be specified as one set of element outages. In order to quantify the water supply level under such an event, the demand must be modeled as a function of nodal pressure. Hydraulic model needs to be enhanced to perform pressure dependent demand simulation and to compute the level of certainty/supply level.

Supply Level Evaluation

Assume Q_i to be the normal demand at node *i*. $Q_i^s{}_{,j}$ represents the actual supplied demand at node *i* under calamity event *j*, the supply level at node *i* for event *j* is given as:

$$S_{i,j} = \frac{Q_{i,j}^s}{Q_i} \times 100\%$$

This gives the percentage of the demand that a system supplies to node *i* under calamity event *j*. The key is to calculate the actual supply demand Q_i^S under the outage that may cause lower than required junction pressure. The less the demand, the greater the impact the calamity is on the system supplied capacity and the more critical the element is to the system.

Pressure Dependent Demand

Whenever a calamity occurs, the systems pressures are affected. Some locations may not have the required pressure. Nodal demand, water available at a location, is dependent on the pressure at the node when the pressure is low. Unlike the conventional approach of demand driven analysis, demand is a function of pressure, Pressure Dependent Demand (PDD). However, it is believed that a junction demand is not affected by pressure if the pressure is above a threshold. The junction demand is reduced when the pressure is dropping below the pressure threshold and it is zero when the pressure is zero.

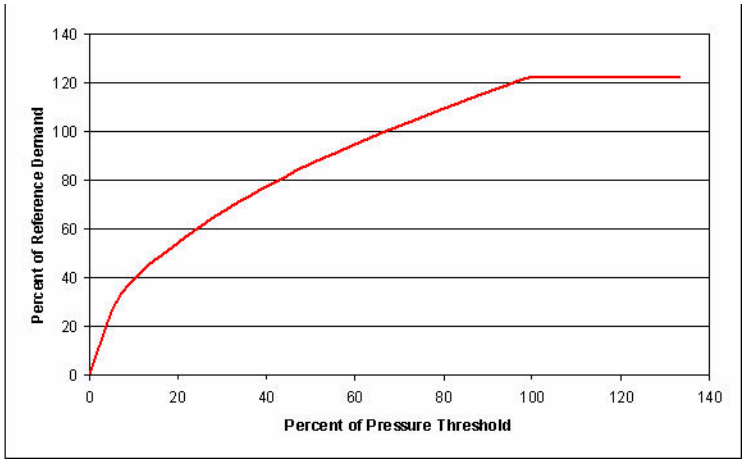
PDD can be defined as one of two pressure demand relationships including a power function and a pressure demand piecewise linear curve (table). The power function is given as:

$$\frac{Q_i^s}{Q_{ri}} = \begin{cases} 0 & H_i \le 0 \\ \left(\frac{H_i}{H_{ri}}\right)^{\alpha} & 0 < H_i < H_t \\ \left(\frac{H_t}{H_{ri}}\right)^{\alpha} & H_i \ge H_t \end{cases}$$

Where:

H_i = calculated pressure at node i
Q_{ri} = requested demand or reference demand at node i
Q^s_i = calculated demand at node i
H_{ri} = reference pressure that is deemed to supply full requested/reference demand
H_t = pressure threshold above which the demand is independent of nodal pressure
α = exponent of pressure demand relationship.

A typical PDD power function is illustrated below. The actual demand increases to the full requested demand (100%) as pressure increases but remains constant after the pressure is greater than the pressure threshold, namely the percent of pressure threshold is greater than 100%.



Pressure demand piecewise linear curve is specified as a table of pressure percentage vs. demand percentage. Pressure percentage is the ratio of actual pressure to a nodal threshold pressure while demand percentage is the ratio of the calculated demand to the reference demand.

Demand Deficit

When a calamity event is modeled, the total supplied demand may be less than the normal required demand. The difference between the calculated demand and the normal required demand is a demand deficit that is evaluated under a prescribed supply level threshold. The total system demand deficit under one possible calamity event *j:*

$$\Delta Q_j = \sum_{i=1}^{N} (Q_i - Q_{i,j}^s) \qquad when \quad S_{i,j} < S_t$$

Where ΔQ_j is the deficit demand at event j and S_t is the threshold of supply level. This formula provides the method for evaluating water supply level, element criticality, and modeling pressure dependent demand.

Solution Methodology

The key solution methodology is how to solve for the pressure dependent demand. Conventionally, nodal demand is a known value. Applying the mass conservation law to each node and energy conservation law to each loop, the network hydraulics solution can be obtained by iteratively solving a set of linear and non-linear equations. A unified formulation for solving network hydraulics is given as a global gradient algorithm (GGA).

$$\begin{bmatrix} A_{11} & \dots & A_{12} \\ \dots & \dots & \dots \\ A_{21} & \dots & 0 \end{bmatrix} \begin{bmatrix} Q \\ \dots \\ H \end{bmatrix} = \begin{bmatrix} -A_{10}H_0 \\ \dots \\ -q \end{bmatrix}$$

Where Q is the unknown pipe discharge and H is the unknown nodal head. q is the set of nodal demand that is not dependent on the nodal head H.

For pressure dependent demand, the demand is no longer a known value but a function of nodal pressure. The solution matrix becomes:

$$\begin{bmatrix} A_{11} & \dots & A_{12} \\ \dots & \dots & \dots \\ A_{21} & \dots & A_{22} \end{bmatrix} \begin{bmatrix} Q \\ \dots \\ H \end{bmatrix} = \begin{bmatrix} -A_{10}H_0 \\ \dots \\ -q \end{bmatrix}$$

A new diagonal matrix A_{22} is added to the solution matrix. The non-zero diagonal element is given as

$$A_{22}(i,i) = Q_i^s$$

Modified GGA Solution

By following the original derivation of GGA, pressure dependent demand formula can be solved as:

$$\begin{bmatrix} D_{11} & \dots & A_{12} \\ \dots & \dots & \dots \\ A_{21} & \dots & D_{22} \end{bmatrix} \begin{bmatrix} dQ \\ \dots \\ dH \end{bmatrix} = \begin{bmatrix} dE \\ \dots \\ dq \end{bmatrix}$$

The difference from the original GGA is the new diagonal matrix D22, which is the deviation of A22 of pressure head H.

$$D_{22}(i,i) = \begin{cases} 0 & P_i^s \le 0 \\ \alpha \left(\dfrac{H_i}{P_t} \right)^{\alpha - 1} \times Q_i & 0 < P_i^s < P_t \\ 0 & P_i^s \ge P_t \end{cases}$$

The modified GGA is to calculate D22 for each pressure dependent demand node and add at A(i, i) as follows:

$$A(i,i) = \sum_j p_{ij} - D_{22}(i,i)$$

where *j* denotes the pipe *j* that is connected with node *i*. This notation is the same as the EPANET2 engine code.

Direct GGA Solution

An alternative solution method is to directly apply GGA as derived but move the pressure dependent demand term to the right

$$\begin{bmatrix} A_{11} & \dots & A_{12} \\ \dots & \dots & \dots \\ A_{21} & \dots & 0 \end{bmatrix} \begin{bmatrix} Q \\ \dots \\ H \end{bmatrix} = \begin{bmatrix} -A_{10}H_0 \\ \dots \\ -A_{22}H - q \end{bmatrix}$$

This method will require no matrix modification of original GGA, but the program will update the nodal demand according to the pressure head of the left side of the matrix.

References

Babovic V., Wu Z. Y. & Larsen L. C., “Calibrating Hydrodynamic Models by Means of Simulated Evolution,” in Proceeding of Hydroinformatics, Delft, Netherlands, pp193-200, 1994.

Benedict, R. P., *Fundamentals of Pipe Flow*, John Wiley and Sons, Inc., New York, 1980.

Brater, Ernest F. and Horace W. King, *Handbook of Hydraulics*, McGraw-Hill Book Company, New York, 1976.

Boulos, P. F. and D. J. Wood, “Explicit Calculation of Pipe-Network Parameters,” *Journal of Hydraulic Engineering*, ASCE, 116(11) 1329-1344, 1987.

Cesario, A. Lee, *Modeling, Analysis, and Design of Water Distribution Systems*, AWWA, 1995.

Clark, R.M., “Chlorine demand and Trihalomethane formation kinetics: a second-order model,” *Journal of Environmental Engineering*, Vol. 124, No. 1, pp. 16-24, 1998.

Clark, R. M., W. M. Grayman, R. M. Males, and A. F. Hess, “Modeling Contaminant propagation in Drinking Water Distribution Systems,” *Journal of Environmental Engineering*, ASCE, New York, 1993.

Cohon, J.L., *Multi-objective Programming and Planning*. Academic Press, New York, 1978.

Computer Applications in Hydraulic Engineering, Fifth Edition, Waterbury, Connecticut, Haestad Press, 2002.

CulvertMaster User’s Guide, Waterbury, Connecticut, Haestad Methods, 2000.

Dunlop, E.J., *WADI Users Manual*, Local Government Computer Services Board, Dublin, Ireland, 1991.

Essential Hydraulics and Hydrology, Waterbury, Connecticut, Haestad Press, 1998.

FlowMaster PE Version 6.1 User's Guide, Waterbury, Connecticut, Haestad Methods, 2000.

George, A. & Liu, J. W-H., *Computer Solution of Large Sparse Positive Definite Systems*, Prentice-Hall, Englewood Cliffs, NJ, 1981.

Goldberg, D.E., *Genetic Algorithms in Search, Optimization and Machine Learning. Addison Wesley*, Reading, MA, 1989.

Goldberg, D. E., Korb, B., & Deb, K., "Messy genetic algorithms: Motivation, analysis, and first results," *Complex Systems*, 3, 493-530, 1989.

Goldberg, D. E., Deb, K., Kargupta, H., & Harik G., "Rapid, Accurate Optimization of Difficult Problems Using Fast Messy Genetic Algorithms," *IlliGAL Report* No. 93004, Illinois Genetic Algorithms Laboratory, University of Illinois at Urbana-Champaign, Urbana, IL 61801, 1993.

Hamam, Y.M., & Brameller, A., "Hybrid method for the solution of piping networks," *Proc. IEE*, Vol. 113, No. 11, pp. 1607-1612, 1971.

International Conference on Computer Applications for Water Supply and Distribution, Leicester Polytechnic, UK, September 8-10.

Koechling, M.T., *Assessment and Modeling of Chlorine Reactions with Natural Organic Matter: Impact of Source Water Quality and Reaction Conditions*, Ph.D. Thesis, Department of Civil and Environmental Engineering, University of Cincinnati, Cincinnati, Ohio, 1998.

Lingireddy, S. and D.J. Wood, "Improved Operation of Water Distribution Systems Using Variable Speed Pumps," *Journal of Energy Engineering*, ASCE, 124(3) 90-103, 1998.

Liou, C.P. and Kroon, J.R., "Modeling the propagation of waterborne substances in distribution networks," *J. AWWA*, 79(11), 54-58, 1987.

Males R. M., W. M. Grayman and R. M. Clark, "Modeling Water Quality in Distribution System," *Journal of Water Resources Planning and Management*, ASCE, New York, 1988.

Notter, R.H. and Sleicher, C.A., "The eddy diffusivity in the turbulent boundary layer near a wall," *Chem. Eng. Sci.*, Vol. 26, pp. 161-171, 1971.

Osiadacz, A.J., *Simulation and Analysis of Gas Networks*, E. & F.N. Spon, London, 1987.

Practical Guide to Hydraulics and Hydrology, Waterbury, Connecticut, Haestad Press, 1997.

Roberson, John A., John J. Cassidy, and Hanif M. Chaudhry, *Hydraulic Engineering*, Houghton Mifflin Company, Massachusetts, 1988.

Roberson, John A. and Clayton T. Crowe, *Engineering Fluid Mechanics 4th Edition*, Houghton Mifflin Company, Massachusetts, 1990.

Rossman, Lewis A., *EPANET User's Manual (AWWA Workshop Edition)*, Risk Reduction Engineering Laboratory, Office of Research and Development, USEPA, Ohio, 1993.

Rossman, Lewis A. et al., "Numerical Methods for Modeling Water Quality in Distribution Systems: A Comparison," *Journal of Water Resources Planning and Management*, ASCE, New York, 1996.

Rossman, Lewis A., R. M. Clark, and W. M. Grayman, "Modeling Chlorine Residuals in Drinking-water Distribution Systems," *Journal of Environmental Engineering*, ASCE, New York, 1994.

Rossman, L.A., Boulos, P.F., and Altman, T., "Discrete volume-element method for network water-quality models," *Journal of Water Resource Planning and Management*, Vol. 119, No. 5, 505-517, 1993.

Rossman, L.A., Clark, R.M., and Grayman, W.M., "Modeling chlorine residuals in drinking-water distribution systems," *Journal of Environmental Engineering*, Vol. 120, No. 4, 803-820, 1994.

Rossman, L.A. and Boulos, P.F., "Numerical methods for modeling water quality in distribution systems: A comparison," *Journal of Water Resource Planning and Management*, Vol. 122, No. 2, 137-146, 1996.

Rossman, L.A. and Grayman, W.M., "Scale-model studies of mixing in drinking water storage tanks," *Journal of Environmental Engineering*, Vol. 125, No. 8, pp. 755-761, 1999.

Salgado, R., Todini, E., & O'Connell, P.E., "Extending the gradient method to include pressure regulating valves in pipe networks," Proc. Inter. Symposium on Computer Modeling of Water Distribution Systems, University of Kentucky, May 12-13, 1988.

Sanks, Robert L., *Pumping Station Design*, Butterworth-Heinemann, Inc., Stoneham, Massachusetts, 1989.

Streeter, Victor L. and Wylie, E. Benjamin, *Fluid Mechanics*, McGraw-Hill Book Company, New York, 1985.

Todini, E. and S. Pilati, "A Gradient Algorithm for the Analysis of Pipe Networks," *Computer Applications in Water Supply, Volume 1 - Systems Analysis and Simulation*, ed. Bryan Coulbeck and Chun-Hou Orr, Research Studies Press Ltd., Letchworth, Hertfordshire, England.

Todini, E. & Pilati, S., "A gradient method for the analysis of pipe networks," 1987.

Walski, T.M., "Model Calibration Data: The Good, The Bad and The Useless," *J. AWWA*, 92(1), p. 94, 2000.

Walski, T. M., "Understanding the adjustments for water distribution system model calibration," *Journal of Indian Water Works Association*, April-June, 2001, pp151-157, 2001.

Walski, T.M., Chase, D.V. and Savic, D.A., *Water Distribution Modeling*, Haestad Press, Waterbury, CT, 2001.

Walski, Thomas M., *Water System Modeling Using CYBERNET*, Waterbury, Connecticut, Haestad Methods, 1993.

Wang Q.J., "The Genetic Algorithm and its Application to Conceptual Rainfall-Runoff Models," *Water Resources Research*, Vol.27, No.9, pp2467-2482, 1991.

Wu Z.Y., "Automatic Model Calibration by Simulating Evolution," M.Sc. Thesis, H.H. 191, International Institute for Infrastructure, Hydraulic and Environmental Engineering, Delft, Netherlands, 1994.

Wu, Z. Y., Boulos, P.F., Orr, C.H., and Ro, J.J., "An Efficient Genetic Algorithms Approach to an Intelligent Decision Support System for Water Distribution Networks," in *Proceedings of the Hydroinformatics 2000 Conference*, Iowa, IW, July 26-29, 2000.

Wu, Z. Y., Boulos P. F., Orr C.-H. and Ro J. J., "Rehabilitation of water distribution system using genetic algorithm," *Journal of AWWA*, Vol. 93, No. 11, pp74-85, 2001.

Wu Z.Y. & Larsen C.L., "Verification of hydrological and hydrodynamic models calibrated by genetic algorithms," Proc. of the 2nd International Conference on Water Resources & Environmental Research, Vol. 2, Kyoto, Japan, pp175-182, 1996.

Wu, Z. Y. and Simpson A. R., "An Efficient Genetic Algorithm Paradigm for Discrete Optimization of Pipeline Networks," International Congress on Modeling and Simulation, Hobart, Tasmania, Australia, 8-11 December, 1997b.

Wu, Z. Y. and Simpson A. R., “Competent Genetic Algorithm Optimization of Water Distribution Systems,” *Journal of Computing in Civil Engineering*, ASCE, Vol 15, No. 2, pp89-101, 2001.

Wu, Z. Y. and Simpson A. R., “Messy Genetic Algorithm for Optimal Design of Water Distribution Systems,” Research Report, No. 140, Department of Civil & Environmental Engineering, University of Adelaide, South Australia., 1996

Wu, Z. Y and Simpson A. R., “Optimal Rehabilitation of Water Distribution Systems Using a Messy Genetic Algorithm,” AWWA 17th Federal Convention Water in the Balance, Melbourne, Australia, 16-21 March 1997a.

Wu, Z. Y, Walski, T., Mankowski, R., Cook, J. Tryby, M. and Herrin G., “Optimal Capacity of Water Distribution Systems,” in *Proceeding of 1st Annual Environmental and Water Resources Systems Analysis (EWRSA) Symposium*, Roanoke, VA, May 19-22, 2002.

Zipparro, Vincent J. and Hasen Hans, Davis’ *Handbook of Applied Hydraulics*, McGraw-Hill Book Company, New York, 1993.

Technical Information Resources 19

docs.bentley.com

Bentley Services

Bentley Discussion Groups

Bentley on the Web

TechNotes/Frequently Asked Questions

BE Magazine

BE Newsletter

Client Server

BE Careers Network

Contact Bentley Systems

docs.bentley.com

Bentley ServicesBentley Discussion Groups

Bentley on the Web

TechNotes/Frequently Asked Questions

BE Magazine

BE NewsletterClient Server

BE Careers Network

docs.bentley.com

docs.bentley.com is your repository of product help files and books. You can browse through online help for specific information or download it to ensure you have the most recent help available on your computer. Also through this site, many product books are available as free, downloadable PDFs, or can be purchased pre-bound with a credit card.

Bentley Services

There are a variety of Bentley Services, including Bentley SELECT^{R} priority services, one-on-one consulting, training programs, MicroStation resellers, as well as your local technical support provider.

U.S./Canada/Latin America	support@bentley.com
Europe/Middle East/ Africa	support@bentley.nl
Asia/Pacific	support@bentley.com.au

Bentley SELECT^{R}

Bentley SELECT^{R} is the comprehensive delivery and support subscription program that features product updates and upgrades via Web downloads and MySELECT CD, around-the-clock technical support, exclusive licensing options, discounts on training and consulting services, as well as technical information and support channels. For more detailed information go online at http://www.bentley.com and click the Support link.

Bentley Professional Services

Bentley Professional Services is a team of project managers, technical managers, application specialists, and developers organized regionally and assigned by skill sets. By adding their extensive knowledge to your project, they provide customized services on a one-to-one basis to help you maximize your investment in Bentley technology. For more information visit http://www.bentley.com/Services/ and click the Bentley Professional Services link.

Bentley Institute

The Bentley Institute manages professional training programs to ensure consistent, high quality, user training for a variety of Bentley products and for varying levels of application experience.

Bentley Institute training is developed to maximize your productivity by using examples relevant to your day-to-day project efforts. Training is developed concurrently with software applications to provide knowledge of the latest tools and features. Additionally, all Bentley Institute faculty meet rigorous certification requirements.

To access the Bentley Institute home page directly from WaterCAD V8*i*, choose Help > Bentley Institute Training, or visit http://www.bentley.com/Training/.

Bentley Discussion Groups

Meet other users of Bentley products, exchange ideas, and discuss a wide range of technical subjects in Bentley's discussion groups. They can be accessed via most common discussion group newsreaders or Web browsers and are a good source of how-to tips, technical information, and programming techniques from Bentley employees and professionals who use our products.

A current list of discussion groups as well as helpful information regarding them can be found at http://discussion.bentley.com/help/.

Bentley on the Web

Visit Bentley on the web at http://www.bentley.com/. Here you will find links to products, services, industries, events and training, community information, and the latest corporate news announcements pertaining to Bentley Systems, Incorporated, your global provider of collaborative software solutions.

TechNotes/Frequently Asked Questions

TechNotes, FAQs and other technical support information are available online at Bentley's WaterCAD V8*i* Technical Support page, in the SELECTservices area: http://selectservices.bentley.com.

BE Magazine

The *BE Magazine* is a quarterly e-magazine focused on the Bentley community of users. It serves as a showcase for Bentley users and their work improving the world's infrastructure.

Each issue is an open forum for the world community of architecture, engineering, and construction professionals and owner-operators. Visit http://www.be.org and click the BE Magazine link to subscribe or to view the magazine online.

BE Newsletter

The *BE Newsletter* is an email newsletter covering industry news, Bentley updates and events, technical tips, and more. Visit http://www.be.org and click the BE Magazine link to subscribe or to view the newsletter online.

Client Server

Client Server is an online newsletter for Bentley SELECT subscribers. This online resource is filled with the latest technical news and information.

Archives of *Client Server* provide an abundant resource of technical information in the form of book excerpts, case studies, commentary and analysis, and productivity tips. For more detailed information go online to http://www.bentley.com and click the Support link.

BE Careers Network

The BE (Bentley Empowered) Careers Network is a program dedicated to supporting accredited academic institutions by providing the latest releases of Bentley products, as well as world-renowned support, online communities, and the latest engineering news and information. For details about the BE Careers Network go online at http://www.becareers.org/.

Contact Bentley Systems

Contact Bentley Systems if you want product information, to upgrade your software, or need technical support.

Sales

Bentley Systems' professional staff is ready to answer your questions. Please contact your sales representative for any questions regarding Bentley Systems' latest products and prices.

Toll-free U.S. Phone:	800-727-6555
Worldwide Phone:	+1-203-755-1666
Fax:	+1-203-597-1488
Email:	sales@bentley.com

Technical Support

We hope that everything runs smoothly and you never have a need for our technical support staff. However, if you do need support, our highly-skilled staff offers their services seven days a week and may be contacted by phone, fax, email, and the Internet. For information on the various levels of support that we offer, contact our sales team today and request information on our Bentley SELECT program, or visit our Web site.

When contacting us for support, in order to assist our technicians in troubleshooting your problem, please be in front of your computer and have the following information available:

- Your computer's operating system.
- Name and build number of the Bentley Systems software you are calling about. The build number can be determined by clicking **Help** > **About** Bentley WaterCAD V8*i* . The build number is the number in brackets located in the lower-left corner of the dialog box that opens.
- A note of exactly what you were doing when you encountered the problem.
- Any error messages or other information displayed on your screen.

When emailing us for support, please provide the following details, in addition to the above, to enable us to provide a more timely and accurate response:

- Company name, address, and phone number
- A detailed explanation of your concerns
- If you are emailing us, the Bentley WaterCAD V8*i* .log files located in the product directory (e.g., C:\Documents and Settings\<User Name>\Local Settings\Application Data\Bentley\<Product Name>\8 (under Windows 2003 Server/XP) or C:\Users\<User Directory>\AppData\Local\Bentley\<Product Name>\8 (under Windows Vista, Windows 7, and Server 2008).

:Available 24 hours a day, seven days a week. You can contact our technical support team at: http://selectservices.bentley.com

Addresses

Internet:	http://selectservices.bentley.com
Email:	sales@bentley.com
Mail:	Bentley Systems, Incorporated Haestad Methods Solutions Center Suite 200W 27 Siemon Company Drive Watertown, CT 06795

Element Properties Reference 20

Edit Element Properties

Pipe Attributes

Junction Attributes

Hydrant Attributes

Tank Attributes

Reservoir Attributes

Periodic Head-Flow Attributes

Pump Attributes

Pump Station Attributes

Variable Speed Pump Battery Attributes

Turbine Attributes

Valve Attributes

Valve With Linear Area Change Attributes

Check Valve Attributes

Orifice Between Pipes Attributes

Discharge To Atmosphere Attributes

Surge Tank Attributes

Hydropneumatic Tank Attributes

Air Valve Attributes

Surge Valve Attributes

Rupture Disk Attributes

Isolation Valve Attributes

Spot Elevation Attributes

Edit Element Properties

- Double-click the element in the drawing pane,
- Choose the element to edit, then choose **View** > **Properties,**
- Or press <**F4**>**.**

Properties displayed in the Property Editor are grouped into categories. An expanded category can be collapsed by clicking plus (+) next to the category heading. A collapsed category can be expanded by clicking minus (-) next to the category heading.

Pipe Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Material: The pipe's material type.
- Diameter: Value represents the internal diameter of a circular pipe or four times the hydraulic radius for non-circular cross-sections.
- Manning's n: Mannings N
- Hazen-Williams C: Hazen Williams C Factor
- Darcy-Weisbach e: Darcy-Weisbach roughness height for the pipe wall.

- Has User Defined Length?: Allows the calculated scaled pipe lengths to be over-ridden with a user defined value.
- Length (User Defined): A user defined pipe length that is not scaled from the underlying map dimensions.
- Has Check Valve?: Defines whether the pipe contains a check valve that limits flow to a single direction. Direction of flow is with the direction of the pipe.
- Specify Local Minor Loss?: If true then the minor coefficent for the element is manually set, otherwise the value is derived from the minor loss library.
- Minor Losses: List of all associated minor losses associated with the element, and can be used to generate the composite minor loss coefficient.
- Status (Initial): Choices: Open, Closed
- Specify Local Bulk Reaction Rate?: If true than a local Bulk Reaction Rate can be specified for the pipe, otherwise the bulk reaction rate associated with selected constituent will govern.
- Bulk Reaction Rate (Local): Coefficient defining how rapidly a constituent grows or decays over time.
- Specify Local Wall Rate?: If true then a local wall reaction rate can be specified for the selected pipe.
- Wall Reaction Rate (First Order): First order coefficient defining the rate at which a substance reacts with the wall of a pipe. Is available if global constituent is set to first order.
- Wall Reaction Rate (Zero Order): Zero order coefficient defining the rate at which a substance reacts with the wall of a pipe. Is available if the global constituent is set to zero order.
- Zone: Specify the zone for the element.
- Installation Year: Specify the install year of the element. It does not affect the calculations.
- Wave Speed: The speed with which a disturbance (i.e. pressure wave) moves through the fluid in the pipe.
- Flow (Initial): A value corresponding to flow in the pipe at the beginning of the transient simulation.
- Hydraulic Grade (Initial Start): The start node hydraulic grade elevation at the beginning of the transient simulation.
- Hydraulic Grade (Initial Stop): The stop node hydraulic grade elevation at the beginning of the transient simulation.
- Pressure (Start): Pressure at the start node of the pipe.
- Pressure (Stop): Pressure at the stop node of the pipe.
- Number of Breaks: The number of breaks that occurred on this pipe.

- Use Local Duration of Pipe Failure History?: Override the global duration of pipe failure history with a local value.
- Duration of Pipe Failure History: The duration of pipe failure history for this pipe. By default uses the global duration.
- Pipe Break Group: The pipe break group this pipe belongs to.
- Cost of Break: The cost of the break for this pipe.
- Flow: Total flow through the pipe. If the value is negative the flow is traveling from the stop node to the start node, and vice versa if positivie.
- Velocity: Velocity of fluid through the pipe.
- Headloss Gradient: The headloss per unit length in the pipe.
- Headloss: Total headloss occuring in the pipe, including both friction and minor headlosses and any minor losses from isolation valves.
- Pressure Loss Gradient: The pressure loss per unit length in the pipe.
- Pressure Loss: Total pressure loss occuring in the pipe, including both friction and minor pressure losses and any minor losses from isolation valves.
- Flow (Absolute): Absolute value of flow through the pipe.
- Hydraulic Grade (Start): Hydraulic grade at start node of pipe.
- Hydraulic Grade (Stop): Hydraulic grade at stop node of pipe.
- Length: Displays either the scaled length or the user defined length depending on which option is set for the pipe.
- Travel Time: The length of the pipe divided by the velocity of flow through pipe.
- Headloss (Minor): Headloss resulting from minor losses in the pipe only. (Excludes isolation valve minor losses).
- Headloss (Friction): Headloss through pipe resulting from friction. (Includes any isolation valve minor losses).
- Area Full: Cross-sectional area of pipe.
- Shear Stress: Shear stress at current time step.
- Length (3D): Length derived from x, y and z coordinates of bounding node.
- Status (Calculated): Choices: Open, Closed
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.
- Concentration (Start): Concentration at start end of the pipe.
- Concentration (Stop): Concentration at the stop end of the pipe.

- Trace (Start): Trace percentage at start end of the pipe.
- Trace (Stop): Trace percentage at stop end of the pipe.
- Age (Start): Water age at the start end of the pipe.
- Age (Stop): Water age at the stop end of the pipe.
- Flow (Minimum Absolute): Minimum flow magnitude through pipe over the course of the simulation.
- Flow (Maximum Absolute): Maximum flow magnitude through the pipe over the course of the simulation.
- Velocity (Maximum): Maximum velocity of flow that occurs in pipe over the course of the simulation.
- Velocity (Minimum): Minimum velocity of flow that occurs in pipe over the course of the simulation.
- Headloss Gradient (Minimum): Minimum headloss gradient that occurs at the selected pipe.
- Headloss Gradient (Maximum): Maximum headloss gradient that occurs at the selected pipe.
- Flow (Minimum): Minimum flow through pipe over the course of the simulation.
- Flow (Maximum): Maximum flow through the pipe over the course of the simulation.
- Age (Minimum): Minimum age in the pipe over the course of the simulation.
- Age (Maximum): Maximum age in the pipe over the course of the simulation.
- Trace (Minimum): Minimum trace in the pipe over the course of the simulation.
- Trace (Maximum): Maximum trace in the pipe over the course of the simulation.
- Concentration (Minimum): Minimum concentration in the pipe over the course of the simulation.
- Concentration (Maximum): Maximum concentration in the pipe over the course of the simulation.
- Age (Start) (Minimum): Minimum age at the start end of the pipe over the course of the simulation.
- Age (Start) (Maximum): Maximum age at the start end of the pipe over the course of the simulation.
- Age (Stop) (Minimum): Minimum age at the stop end of the pipe over the course of the simulation.
- Age (Stop) (Maximum): Maximum age at the stop end of the pipe over the course of the simulation.
- Trace (Start) (Minimum): Minimum trace at the start end of the pipe over the course of the simulation.

- Trace (Start) (Maximum): Maximum trace at the start end of the pipe over the course of the simulation.
- Trace (Stop) (Minimum): Minimum trace at the stop end of the pipe over the course of the simulation.
- Trace (Stop) (Maximum): Maximum trace at the stop end of the pipe over the course of the simulation.
- Concentration (Start) (Minimum): Minimum concentration at the start end of the pipe over the course of the simulation.
- Concentration (Start) (Maximum): Maximum concentration at the start end of the pipe over the course of the simulation.
- Concentration (Stop) (Minimum): Minimum concentration at the stop end of the pipe over the course of the simulation.
- Concentration (Stop) (Maximum): Maximum concentration at the stop end of the pipe over the course of the simulation.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Is Closed?: True if the current element is closed during the current time step.
- Is Open?: Set to true if open during the current time step.
- Is Initially Closed?: If true, the initial condition for the control element is "Closed" or "Off."
- Controlled?: Is true if a control action in the current control set references the selected element.
- Minor Loss Coefficient (Unified): Displays the current minor loss value for the element, depending on whether its derived or local.
- Head (Maximum, Transient): Maximum head at any point along the pipe over the course of the transient simulation.
- Head (Minimum, Transient): Minimum head at any point along the pipe over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at any point along the pipe over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at any point along the pipe over the course of the transient simulation.
- Flow (Maximum, Transient): Maximum flow at any point along the pipe over the course of the transient simulation.
- Flow (Minimum, Transient): Minimum flow at any point along the pipe over the course of the transient simulation.
- Velocity (Minimum, Transient): Minimum velocity at any point along the pipe over the course of the transient simulation.

- Velocity (Maximum, Transient): Maximum velocity at any point along the pipe over the course of the transient simulation.
- Wave Speed Adjustment Percent: The wave speed adjustment applied to this pipe (relative to the original wave speed) so that a sharp pressure-wave front can travel the length of one of the pipe's interior segments in one time step.
- Wave Speed Adjustment: The wave speed adjustment applied to this pipe so that a sharp pressure-wave front can travel the length of one of the pipe's interior segments in one time step.
- Length Adjustment Percent: The length adjustment applied to this pipe (relative to the original length) so that a sharp pressure-wave front can travel the length of one of the pipe's interior segments in one time step.
- Length Adjustment: The length adjustment applied to this pipe so that a sharp pressure-wave front can travel the length of one of the pipe's interior segments in one time step.
- Velocity (Initial, Transient): The flow velocity along the pipe at the beginning of the transient simulation.
- Pressure (Maximum at Stop Node, Transient): Maximum pressure at the pipe's stop node over the course of the transient simulation.
- Pressure (Maximum at Start Node, Transient): Maximum pressure at the pipe's start node over the course of the transient simulation.
- Pressure (Minimum at Stop Node, Transient): Minimum pressure at the pipe's stop node over the course of the transient simulation.
- Pressure (Minimum at Start Node, Transient): Minimum pressure at the pipe's start node over the course of the transient simulation.
- Head (Maximum at Stop Node, Transient): Maximum head at the pipe's stop node over the course of the transient simulation.
- Head (Maximum at Start Node, Transient): Maximum head at the pipe's start node over the course of the transient simulation.
- Head (Minimum at Start Node, Transient): Minimum head at the pipe's start node over the course of the transient simulation.
- Head (Minimum at Stop Node, Transient): Minimum head at the pipe's stop node over the course of the transient simulation.
- Upsurge Ratio at Start Node: Ratio of maximum pressure at the pipe's start node over the course of the transient simulation to the pressure at the beginning of the transient simulation.
- Upsurge Ratio at Stop Node: Ratio of maximum pressure at the pipe's stop node over the course of the transient simulation to the pressure at the beginning of the transient simulation.

- Head (Initial at Start Node, Transient): The head at the pipe's start node at the beginning of the transient simulation.
- Head (Initial at Stop Node, Transient): The head at the pipe's stop node at the beginning of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume along the pipe over the course of the transient simulation.
- Vapor Volume (Maximum, Transient): Maximum vapor volume along the pipe over the course of the transient simulation.
- Velocity (Maximum Flushing): The maximum achieved pipe velocity across all flushing events. If comparing against previous results (for other alternatives/ scenarios) this result is the maximum achieved velocity across all flushing events for which results exist.
- Shear Stress (Maxmum Flushing): The maximum achieved shear stress across all flushing events. If comparing against previous results (for other alternatives/ scenarios) this result is the maximum achieved shear stress across all flushing events for which results exist.
- Flushing Event: The flushing event that resulted in the pipe maximum achieved velocity. If comparing against previous results, this flushing event may be defined in another alternative/scenario.
- Satisfies Flushing Target Velocity?: True if the maximum achieved velocity for the pipe is greater than or equal to the target velocity.
- Satisfies Flushing Target Shear Stress?: True if the maximum achieved shear stress for the pipe is greater than or equal to the target shear stress.
- Break Rate: The break rate for the pipe over time.
- Break Rate (Pipe Group): The break rate for the group the pipe belongs to.
- Projected Breaks: The projected number of breaks for this pipe.
- Annual Expected Cost: The annual expected cost of the breaks for this pipe.
- Present Worth: How much the pipe is currently worth based on the projection cost.
- Break Rate (Scaled): A weighted combination of the individual pipe break rate and the pipe break rate of the group to which the pipe belongs.

Junction Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.

- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Specify Local Fire Flow Constraints?: If set to true then local fire flow constraints which override the global values can be set for the current junction.
- Fire Flow (Needed): The flow rate required at the junction to meet fire flow demands. This value will be added to or replace the junctions baseline demand, depending on the default setting for applying fire flows as specified in the Fire Flow Alternative dialog box.
- Fire Flow (Upper Limit): This input defines the maximum allowable fire flow that a junction can provide and the maximum allowable fire flow that can occur at any single withdrawal location.
- Pressure (Residual Lower Limit): Minimum residual pressure to occur at the junction node. The program determines the amount of fire flow available such that the residual pressure at the junction node does not fall below this target pressure.
- Pressure (Zone Lower Limit): Minimum pressure to occur at all junction nodes within the Zone you are testing. The model determines the available fire flow such that the minimum zone pressures do not fall below this target pressure.
- Pressure (System Lower Limit): Minimum pressure allowed at any junction in the entire system as a result of the fire flow withdrawal. If a node's pressure anywhere in the system falls below this constraint while withdrawing fire flow, fire flow will not be satisfied.
- Use Velocity Constraint: If set to true, then a velocity constraint can be specified for the node.
- Velocity (Upper Limit): Maximum velocity allowed in the associated pipe set.
- Use Minimum System Pressure Constraints?: If set to true then the fire flow analysis by pressure throughout the entire system.
- Emitter Coefficient: Discharge coefficient for an emitter (sprinkler or nozzle) placed at junction. Units are flow units at 1 unit of pressure drop (psi or m). Leave blank or set to 0 if no emitter is present.
- Percent of Demand that is Pressure Dependent: The percent of demand that is pressure dependent for the current junction. Overrides the global value that is set in the pressure dependent demand alternative
- Pressure (Reference): Overrides the reference pressure defined in the pressure dependent demand alternative for the current junction.
- Local Function: Defines the relationship between the pressure and the demand for the current junction. This function will be used instead of the global function defined in the pressure dependent demand alternative.
- Use Local Pressure Dependent Demand Data?: If set to true, then pressure dependent demand parameters that override the global default values can be set for the current junction.

- Vapor Volume (Initial): Volume of vapour at the node at the start of the transient simulation. If volume is nonzero, then liquid is at the vapour pressure. Only applicable at dead ends.
- Pressure Drop (Typical): Pressure drop across the orifice corresponding to the initial/typical flow.
- Flow (Typical): If the initial flow is zero, then this is a typical (positive) flow.
- Demand Collection: A collection of baseline demands and associated temporal patterns.
- Unit Demand Collection: A collection of unit demands, associated unit counts, and temporal patterns.
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.
- Is Constituent Source?: If true then the selected node can inject a set concentration of the global constituent into the system.
- Pattern (Constituent): Specify the pattern which dictates how the injected constituent concentration varies over time.
- Constituent Source Type: Choices: Concentration, Flow Paced Booster, Setpoint Booster, Mass Booster
- Concentration (Base): This data field allows you to specify the corresponding constituent concentration at this node over time.
- Mass Rate (Base): This data field allows you to specify the corresponding constituent mass rate at this node over time.
- Age (Initial): Specify the initial age of the fluid at the selected element.
- Pressure: Calculated pressure at node.
- Pressure Head: Calculated pressure head at node.
- Demand Shortage: Difference between the target demand and the demand the system can supply during the current time step.
- Demand (Cumulative): Total required demand volume at current node up to the current time step.
- Supply (Cumulative): Total volume of flow that the system can actually supply up to the current time step.
- Shortfall (Cumulative): The cumulative difference in volume between the target demand and the flow supplied up to the current time step.

- Supply Rate (Cumulative): The cumulative ratio of supply/demand up to the current time step.
- Demand (Target): The demand required at the node. Calculated from the nodes input data.
- Satisfies Fire Flow Constraints?: Set to true if hydraulic calculations met accuracy constraints within the allotted number of trials.
- Fire Flow (Available): Amount of flow available for fire protection while maintaining all fire flow pressure constraints.
- Pressure (Calculated Residual): Calculated pressure at the junction node during the fire flow withdrawal.
- Pressure (Calculated Zone Lower Limit): Minimum calculated pressure of all junctions in the same zone as this junction.
- Junction w/ Minimum Pressure (Zone): Label of the junction corresponding to the minimum zone pressure.
- Pressure (Calculated System Lower Limit): Minimum calculated pressure of all junctions in the system.
- Junction w/ Minimum Pressure (System): Junction corresponding to the minimum system pressure.
- Is Fire Flow Run Balanced?: If set to true then the fire flow analysis was able to solve.
- Fire Flow Iterations: Number of iterations required to hone in on the fire flow result.
- Flow (Total Needed): If fire flow is added to baseline demand this equals the sum of the calculated demand and the needed fire flow, otherwise is equivalent to the needed fire flow.
- Flow (Total Available): If fire flow is added to the baseline demand this equals the sum of the calculated demand and the available fire flow at the node, otherwise it is equivalent to the available fire flow.
- Fire Flow (Total Upper Limit): If fire flow is added to base line, this equals the sum of the demand at the junction plus the fire flow upper limit, otherwise it is equivalent to the fire flow upper limit.
- Junction w/ Minimum Pressure (Zone @ Total Flow Needed): If baseline flow is added to demand, this represents the junction with the minimum pressure in the zone as a result of the total needed demand and fire flow.
- Pressure (Calculated Residual @ Total Flow Needed): Lower limit for system pressure at node.
- Pressure (Calculated Zone Lower Limit @ Total Flow Needed): Lower limit for pressure in zone at node
- Pipe w/ Maximum Velocity: Label of pipe with max velocity

- Velocity of Maximum Pipe: Velocity in pipe with highest velocity.
- Demand (Minimum): Minimum demand at node over the course of the simulation.
- Demand (Maximum): Maximum demand at node over the course of the simulation.
- Hydraulic Grade (Maximum): Maximum calculated hydraulic grade at node over the course of the simulation.
- Hydraulic Grade (Minimum): Minimum calculated hydraulic grade at node over the course of simulation.
- Pressure (Minimum): Minimum pressure at node over the course of the simulation.
- Pressure (Maximum): Maximum pressure at node over the course of the simulation.
- Age (Minimum): Minimum age at node over the course of the simulation.
- Age (Maximum): Maximum age at node over the course of the simulation.
- Trace (Minimum): Minimum trace at node over the course of the simulation.
- Trace (Maximum): Maximum trace at node over the course of the simulation.
- Concentration (Minimum): Minimum concentration at node over the course of the simulation.
- Concentration (Maximum): Minimum concentration at node over the course of the simulation.
- Demand: Total calculated demand at selected element.
- Demand Adjusted Population: Population of area supplied by current node. This value is derived from the unit demand loads applied to the collection and their equivalent populations.
- Hydraulic Grade: Calculated hydraulic grade at node.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.
- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.

- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.

Hydrant Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Include Lateral Loss?: Specifies whether the lateral loss of the hydrant should be accounted for or not. If so, you can specify the properties of the hydrant lateral.
- Lateral Diameter: The diameter of the hydrant lateral.
- Lateral Minor Loss Coefficient: The minor loss coefficient of the hydrant lateral.
- Lateral Length: The length of the hydrant lateral.
- Hydrant Status: Choices: Open, Closed
- Specify Local Fire Flow Constraints?: If set to true then local fire flow constraints which override the global values can be set for the current junction.
- Fire Flow (Needed): The flow rate required at the junction to meet fire flow demands. This value will be added to or replace the junctions baseline demand, depending on the default setting for applying fire flows as specified in the Fire Flow Alternative dialog box.
- Fire Flow (Upper Limit): This input defines the maximum allowable fire flow that a junction can provide and the maximum allowable fire flow that can occur at any single withdrawal location.
- Pressure (Residual Lower Limit): Minimum residual pressure to occur at the junction node. The program determines the amount of fire flow available such that the residual pressure at the junction node does not fall below this target pressure.
- Pressure (Zone Lower Limit): Minimum pressure to occur at all junction nodes within the Zone you are testing. The model determines the available fire flow such that the minimum zone pressures do not fall below this target pressure.

- Pressure (System Lower Limit): Minimum pressure allowed at any junction in the entire system as a result of the fire flow withdrawal. If a node's pressure anywhere in the system falls below this constraint while withdrawing fire flow, fire flow will not be satisfied.
- Use Velocity Constraint: If set to true, then a velocity constraint can be specified for the node.
- Velocity (Upper Limit): Maximum velocity allowed in the associated pipe set.
- Use Minimum System Pressure Constraints?: If set to true then the fire flow analysis by pressure throughout the entire system.
- Emitter Coefficient: Discharge coefficient for an emitter (sprinkler or nozzle) placed at junction. Units are flow units at 1 unit of pressure drop (psi or m). Leave blank or set to 0 if no emitter is present.
- Percent of Demand that is Pressure Dependent: The percent of demand that is pressure dependent for the current junction. Overrides the global value that is set in the pressure dependent demand alternative
- Pressure (Reference): Overrides the reference pressure defined in the pressure dependent demand alternative for the current junction.
- Local Function: Defines the relationship between the pressure and the demand for the current junction. This function will be used instead of the global function defined in the pressure dependent demand alternative.
- Use Local Pressure Dependent Demand Data?: If set to true, then pressure dependent demand parameters that override the global default values can be set for the current junction.
- Vapor Volume (Initial): Volume of vapour at the node at the start of the transient simulation. If volume is nonzero, then liquid is at the vapour pressure. Only applicable at dead ends.
- Pressure Drop (Typical): Pressure drop across the orifice corresponding to the initial/typical flow.
- Flow (Typical): If the initial flow is zero, then this is a typical (positive) flow.
- Demand Collection: A collection of baseline demands and associated temporal patterns.
- Unit Demand Collection: A collection of unit demands, associated unit counts, and temporal patterns.
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.

- Is Constituent Source?: If true then the selected node can inject a set concentration of the global constituent into the system.
- Pattern (Constituent): Specify the pattern which dictates how the injected constituent concentration varies over time.
- Constituent Source Type: Choices: Concentration, Flow Paced Booster, Setpoint Booster, Mass Booster
- Concentration (Base): This data field allows you to specify the corresponding constituent concentration at this node over time.
- Mass Rate (Base): This data field allows you to specify the corresponding constituent mass rate at this node over time.
- Age (Initial): Specify the initial age of the fluid at the selected element.
- Pressure: Calculated pressure at node.
- Pressure Head: Calculated pressure head at node.
- Demand Shortage: Difference between the target demand and the demand the system can supply during the current time step.
- Demand (Cumulative): Total required demand volume at current node up to the current time step.
- Supply (Cumulative): Total volume of flow that the system can actually supply up to the current time step.
- Shortfall (Cumulative): The cumulative difference in volume between the target demand and the flow supplied up to the current time step.
- Supply Rate (Cumulative): The cumulative ratio of supply/demand up to the current time step.
- Demand (Target): The demand required at the node. Calculated from the nodes input data.
- Satisfies Fire Flow Constraints?: Set to true if hydraulic calculations met accuracy constraints within the allotted number of trials.
- Fire Flow (Available): Amount of flow available for fire protection while maintaining all fire flow pressure constraints.
- Pressure (Calculated Residual): Calculated pressure at the junction node during the fire flow withdrawal.
- Pressure (Calculated Zone Lower Limit): Minimum calculated pressure of all junctions in the same zone as this junction.
- Junction w/ Minimum Pressure (Zone): Label of the junction corresponding to the minimum zone pressure.
- Pressure (Calculated System Lower Limit): Minimum calculated pressure of all junctions in the system.

- Junction w/ Minimum Pressure (System): Junction corresponding to the minimum system pressure.
- Is Fire Flow Run Balanced?: If set to true then the fire flow analysis was able to solve.
- Fire Flow Iterations: Number of iterations required to hone in on the fire flow result.
- Flow (Total Needed): If fire flow is added to baseline demand this equals the sum of the calculated demand and the needed fire flow, otherwise is equivalent to the needed fire flow.
- Flow (Total Available): If fire flow is added to the baseline demand this equals the sum of the calculated demand and the available fire flow at the node, otherwise it is equivalent to the available fire flow.
- Fire Flow (Total Upper Limit): If fire flow is added to base line, this equals the sum of the demand at the junction plus the fire flow upper limit, otherwise it is equivalent to the fire flow upper limit.
- Junction w/ Minimum Pressure (Zone @ Total Flow Needed): If baseline flow is added to demand, this represents the junction with the minimum pressure in the zone as a result of the total needed demand and fire flow.
- Pressure (Calculated Residual @ Total Flow Needed): Lower limit for system pressure at node.
- Pressure (Calculated Zone Lower Limit @ Total Flow Needed): Lower limit for pressure in zone at node
- Pipe w/ Maximum Velocity: Label of pipe with max velocity
- Velocity of Maximum Pipe: Velocity in pipe with highest velocity.
- Demand (Minimum): Minimum demand at node over the course of the simulation.
- Demand (Maximum): Maximum demand at node over the course of the simulation.
- Hydraulic Grade (Maximum): Maximum calculated hydraulic grade at node over the course of the simulation.
- Hydraulic Grade (Minimum): Minimum calculated hydraulic grade at node over the course of simulation.
- Pressure (Minimum): Minimum pressure at node over the course of the simulation.
- Pressure (Maximum): Maximum pressure at node over the course of the simulation.
- Age (Minimum): Minimum age at node over the course of the simulation.
- Age (Maximum): Maximum age at node over the course of the simulation.

- Trace (Minimum): Minimum trace at node over the course of the simulation.
- Trace (Maximum): Maximum trace at node over the course of the simulation.
- Concentration (Minimum): Minimum concentration at node over the course of the simulation.
- Concentration (Maximum): Minimum concentration at node over the course of the simulation.
- Demand: Total calculated demand at selected element.
- Demand Adjusted Population: Population of area supplied by current node. This value is derived from the unit demand loads applied to the collection and their equivalent populations.
- Hydraulic Grade: Calculated hydraulic grade at node.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.
- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.

Tank Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.

- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Include in Energy Calculation?: If set to true, cost generated by the element will be included in the calculations, otherwise they will be excluded.
- Has Separate Inlet?: Specifies whether this tank has a separate inlet pipe (only significant when modeling top fill tanks or throttling inlets).
- Inlet Pipe: Specifies which pipe will be used to model a top fill inlet, throttling inlet, or both.
- Tank Fills From Top?: Set this to true if you wish to model a top fill inlet.
- Level (Inlet Invert): Specify the invert level for the inlet. If the upstream HGL is lower than this level the tank will not fill.
- Inlet Valve Throttles?: Set this to true if there is a throttling valve (such as a float valve) on the inlet.
- Discharge Coefficient (Fully Open): Specifies the discharge or flow coefficient of the inlet valve in its fully open position.
- Level Inlet Valve Fully Closes: Specifies the level at which the throttling valve becomes fully closed allowing no more flow to pass into the tank.
- Valve Characteristics: Specifies the valve characteristics definition to be used for this valve. If the Valve Characteristic Curve is not defined then a default curve will be used. The default curve will have (Relative Closure, Relative Area) points of (0,1) and (1,0).
- Valve Type: Choices: Butterfly, Needle, Circular Gate, Globe, Ball, User Defined
- Elevation (Base): Elevation of the storage tank base used as a reference when entering water surface elevations in the tank in terms of levels.
- Elevation (Maximum): Highest allowable water surface elevation or level. If the tank fills above this point, it will be automatically shut off from the system.
- Level (Maximum): A reference level to compare the hydraulic grade in the tank. Does not influence the calculations.
- Diameter: Diameter of tank with constant circular cross-section.
- Area (Average): Cross-Sectional area of tank for constant x-section tanks.
- Volume Full (Input): The full active volume of the variable area tank (i.e., the volume at 100% depth), exclusive of any inactive volume.
- Operating Range Type: Choices: Elevation, Level
- Section: Choices: Circular, Non-Circular, Variable Area
- Cross-Section Curve: Defines a curve which specifies the relationship between depth and volume.

- Specify Local Bulk Rate?: If true than a local Bulk Reaction Rate can be specified for the tank, otherwise the bulk reaction rate associated with selected constituent will govern.
- Bulk Reaction Rate (Local): Coefficient defining how rapidly a constituent grows or decays over time.
- Tank Mixing Model: Choices: 2-Compartment, Completely Mixed, FIFO, LIFO
- Compartment 1: Percent of available storage that makes up the first compartment. Inflow and outflow is assumed to take place in the first compartment.
- Compartment 2: Percent of available storage that makes up the second compartment. The second compartment receives overflow from the first, and this overflow is completely mixed.
- Elevation (Minimum): Lowest allowable water surface elevation or level. If the tank drains below this point, it will be automatically shut off from the system.
- Volume (Inactive): The inactive volume of the tank. This volume is the inaccessible volume of the tank that is below the tank active operating range and can become important in water quality simulations subject to the selected mixing model.
- Level (Minimum): Lowest allowable water surface elevation or level. If the tank drains below this point, it will be automatically shut off from the system.
- Elevation (High Alarm): The elevation above which the high level alarm is generated. Calculation notifications are produced to advise you of any alarm level violations.
- Level (High Alarm): The level above which the high level alarm is generated. Calculation notifications are produced to advise you of any alarm level violations.
- Elevation (Low Alarm): The elevation below which the low level alarm is generated. Calculation notifications are produced to advise you of any alarm level violations.
- Level (Low Alarm): The level below which the low level alarm is generated. Calculation notifications are produced to advise you of any alarm level violations.
- Use High Alarm?: Specifies whether or not to check high alarm levels during Steady State/EPS calculation and generate messages if the levels are violated.
- Use Low Alarm?: Specifies whether or not to check low alarm levels during Steady State/EPS calculation and generate messages if the levels are violated.
- Elevation (Initial): Starting water surface elevation/level in the tank.
- Level (Initial): Starting water surface elevation/level in the tank.
- Installation Year: Specify the install year of the element. It does not affect the calculations.

- Elevation (Initial, Transient): Enter a value only if a check valve is installed (i.e., case of a one-way surge tank), or there is an initial inflow/outflow head loss. By default, the intial water surface level is taken equal to the head in the adjacent pipe.
- Report Period (Transient): Number of time steps between successive printouts of operation. By default, this printout is suppressed.
- Demand Collection: A collection of baseline demands and associated temporal patterns.
- Unit Demand Collection: A collection of unit demands, associated unit counts, and temporal patterns.
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.
- Is Constituent Source?: If true then the selected node can inject a set concentration of the global constituent into the system.
- Pattern (Constituent): Specify the pattern which dictates how the injected constituent concentration varies over time.
- Constituent Source Type: Choices: Concentration, Flow Paced Booster, Setpoint Booster, Mass Booster
- Concentration (Base): This data field allows you to specify the corresponding constituent concentration at this node over time.
- Mass Rate (Base): This data field allows you to specify the corresponding constituent mass rate at this node over time.
- Age (Initial): Specify the initial age of the fluid at the selected element.
- Relative Closure (Calculated, Inlet Valve): The relative closure of the tank throttling inlet valve at the current time step. (Only applies if the inlet throttles).
- Discharge Coefficient Setting (Calculated, Inlet Valve): The discharge coefficient of the throttling inlet valve at the current time step. (Only applies if the inlet throttles).
- Headloss (Inlet Valve): The headloss across the separate inlet valve at the current time step.
- Hydraulic Grade (Inlet Valve, From): Calculated hydraulic grade at the entrance of the separate inlet valve.
- Hydraulic Grade (Inlet Valve, To): Calculated hydraulic grade at the exit of the separate inlet valve.

- Status (Calculated, Inlet Valve): Choices: Active, Inactive, Closed
- Volume Full (Calculated): The full active volume of the tank between the limits of the defined operating range, exclusive of any inactive volume.
- Level (Calculated): Difference between calcuted hydraulic grade and the base elevation of the tank.
- Volume (Calculated): Total volume of fluid in tank including the inactive volume.
- Percent Full: The ratio of tank active volume to the calculated tank full active volume. Active volume is the tank volume within the operating range and is exclusive of inactive volume.
- Status (Calculated): Choices: Empty, Emptying, Filling, Full, Stagnant
- Flow (Out net): Net flow out of the element.
- Flow (In net): Net flow into the element.
- Demand Adjusted Population: Population of area supplied by current node. This value is derived from the unit demand loads applied to the collection and their equivalent populations.
- Hydraulic Grade: Calculated hydraulic grade at node.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.
- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.

Reservoir Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.

- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Hydraulic Grade Pattern: Allows you to apply a pattern for changes to the reservoirs hydraulic grade line over time for extended period simulations.
- Elevation (Inlet/Outlet Invert): Elevation of the reservoir inlet/outlet invert.
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.
- Is Constituent Source?: If true then the selected node can inject a set concentration of the global constituent into the system.
- Pattern (Constituent): Specify the pattern which dictates how the injected constituent concentration varies over time.
- Constituent Source Type: Choices: Concentration, Flow Paced Booster, Setpoint Booster, Mass Booster
- Concentration (Base): This data field allows you to specify the corresponding constituent concentration at this node over time.
- Mass Rate (Base): This data field allows you to specify the corresponding constituent mass rate at this node over time.
- Age (Initial): Specify the initial age of the fluid at the selected element.
- Flow (Out net): Net flow out of the element.
- Flow (In net): Net flow into the element.
- Hydraulic Grade: Calculated hydraulic grade at node.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.

- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.

Periodic Head-Flow Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Sinusoidal: If sinusoidal, then mean value, amplitude and phase are entered; otherwise, a table of values is required. A sinusoidal quantity X has the form: X = X0 + A sin(2 * PI * t / T + Phase).
- Mean Value (Head): The mean head value. Required only if sinusoidal data specified.
- Amplitude (Head): The amplitude of the sinusoidal head curve. Required only if sinusoidal data specified.
- Phase: Phase of the sinusoidal flow or head curve. Default option is 0 such that periodic component of head or flow is zero at time zero.
- Period: Oscillation period of the sinusoidal flow or head curve (must be positive), or the period after which a tabular flow or head pattern repeats.
- Mean Value (Flow): The mean flow value. Required only if sinusoidal data specified.
- Amplitude (Flow): The amplitude of the sinusoidal flow curve. Required only if sinusoidal data specified.
- Transient Parameter: Choices: Head, Flow
- Flow Pattern: A collection of time/flow pairs representing the transient flow pattern.

- Head Pattern: A collection of time/head pairs representing the transient flow pattern.
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.
- Is Constituent Source?: If true then the selected node can inject a set concentration of the global constituent into the system.
- Pattern (Constituent): Specify the pattern which dictates how the injected constituent concentration varies over time.
- Constituent Source Type: Choices: Concentration, Flow Paced Booster, Setpoint Booster, Mass Booster
- Concentration (Base): This data field allows you to specify the corresponding constituent concentration at this node over time.
- Mass Rate (Base): This data field allows you to specify the corresponding constituent mass rate at this node over time.
- Age (Initial): Specify the initial age of the fluid at the selected element.
- Discharge (Calculated): Calculated discharge from the node.
- Pressure: Calculated pressure at node.
- Pressure Head: Calculated pressure head at node.
- Hydraulic Grade: Calculated hydraulic grade at node.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.
- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.

- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.

Pump Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Pump Definition: Select the pump definition to apply to the selected pump.
- Is Variable Speed Pump?: If set to true then the pump will act as a Variable Speed Pump.
- Relative Speed Factor (Maximum): The highest relative speed factor that the pump can be set at to meet the target head at the control node. If the target head cannot be met when the pump is set at the maximum relative speed factor, the maximum will be used.
- VSP Type: Choices: Pattern Based, Fixed Head, Fixed Flow
- Flow (Target): The relative speed of a VSP of type "Fixed Flow" will be adjusted to meet the Flow (Target).
- Pattern (Relative Speeds): Select the pattern by which the relative speed factor is adjusted over the course of the simulation. (Note that patterns override settings changes made by controls).
- Control Node: The node that the VSP checks to determine whether to increase, maintain, or decrease its relative speed factor.
- Hydraulic Grade (Target): The Head that the VSP will attempt to maintain for the Control Node.
- Control Node on Suction Side?: Specifies if the VSP has a suction side control node.
- Relative Speed Factor (Initial): Determines the initial speed of the pump impeller relative to the speed at which the pump curve is defined.
- Status (Initial): Choices: On, Off
- Include in Energy Calculation?: If set to true, cost generated by the element will be included in the calculations, otherwise they will be excluded.

- Energy Pricing: Specify which energy pricing definition is to be used when calculating costs of the corresponding pump.
- Diameter (Pump Valve): Diameter refers to the valve at full opening, typically equal to the internal diameter of the discharge flange.
- Flow (Nominal): Rated or duty flow for the pump, often at or near the best efficiency point.
- Head (Nominal): Rated or duty head for the pump, often at or near the best efficiency point.
- Relative Speed (Initial, Transient): The initial pump relative speed to be used in the transient analysis.
- Torque (Nominal): Specifies the nominal torque that, when multiplied by the Operating Rule's pattern multiplier values will result in the torque values used by the engine.
- Pump Type (Transient): Choices: Shut Down After Time Delay, Constant Speed - No Pump Curve, Constant Speed - Pump Curve, Variable Speed/Torque, Pump Start - Variable Speed/Torque
- Time (Delay until Shut Down): Time at which power to pump motor is shut off. By default, there is no time delay.
- Time (For Valve to Close): The time taken for the pump discharge control valve to close after the transient simulation begins.
- Time (For Valve to Operate): Time to close check valve (or to open it if initial flow is zero). If the check valve allows flow only in one direction, enter 0.
- Control Variable: Choices: Speed, Torque
- Status (Initial, Transient): Choices: On, Off
- Pump Valve Type: Choices: Check Valve, Control Valve
- Operating Rule: Specifies the operation of the valve during a transient simulation.
- Report Period (Transient): Number of time steps between successive printouts of operation. By default, this printout is suppressed.
- Pump Station: The Pump Station to which this Pump belongs.
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.
- Age (Initial): Specify the initial age of the fluid at the selected element.
- Installation Year: Specify the install year of the element. It does not affect the calculations.
- Trace (Initial): Specify the initial trace amount at the current location.

- Zone: Specify the zone for the element.
- Relative Speed Factor (Calculated): Current relative speed factor of pump at current time step.
- Hydraulic Grade (Suction): Calculated hydraulic grade at suction side of the pump.
- Hydraulic Grade (Discharge): Calculated hydraulic grade at discharge side of the pump.
- Flow (Total): Total flow pumped by standard pump or the pump battery.
- Pump Head: Head gain between suction and discharge side of the pump.
- Pressure (Suction): Calculated pressure at suction side of the pump.
- Pressure (Discharge): Calculated pressure at discharge side of the pump.
- Flow (Absolute): The magnitude of flow through the pump regardless of flow direction.
- Pump Exceeds Operating Range?: Is true if the system demands on the pump exceeds its capabilities.
- Status (Calculated): Choices: On, Off, Pump Cannot Deliver Head (Closed), Pump Result Cannot Deliver Flow (Open)
- Peak Power: Displays the peak energy usage, as calculated during the extended period simulation. This result is displayed even if Peak Demand Charges are not applied.
- Time of Peak Energy Cost: Time when energy cost is maximum.
- Demand Charge: The charge applied per kW.
- Demand Charge Period: Time over which demand charge is averaged in order to get $/day.
- Peak Power Cost: Displays the energy cost as calculated during the extended period simulation. If no Peak Demand Charge has been applied to the associated Energy Price Definition, this field will display as zero.
- Peak Power Cost (Daily): The cost associated with the Peak Demand Charge.
- Volume Pumped (Incremental): Total volume of flow pumped during current time step.
- Volume Pumped (Cumulative): Total volume of flow pumped up to the current time step.
- Water Power: The amount of energy transferred to the water by the pump.
- Pump Efficiency: The Pump Efficiency value is representative of the ability of the pump to transfer the mechanical energy generated by the motor to Water Power.
- Wire to Water Efficiency: The ratio of the Water Power to the Wire Power.
- Wire Power: The amount of energy delivered to the pump motor.

- Energy Used (Incremental): Total energy used during current time step.
- Energy Used (Cumulative): Total amount of energy used up to the current time step.
- Energy Price: Cost per unit of energy.
- Energy Cost (Incremental): The energy cost during the current time step.
- Energy Cost (Cumulative): The total energy cost up to the current time step.
- Cost per Unit Volume: Cost per unit of volume pumped for current time step.
- Relative Speed Factor (Energy Cost Engine): Relative speed of pump at current time step.
- Motor Efficiency: The Motor Efficiency value is representative of the ability of the motor to transform electrical energy to rotary mechanical energy.
- Time of Use: The amount of time the pump is turned on over the course of the simulation.
- Utilization: Percentage of total time during the EPS that the pump was On.
- Volume Pumped (Total): The total volume of fluid pumped during the simulation.
- Water Power (Average): The average amount of energy transferred to the water by the pump over the course of the simulation.
- Pump Efficiency (Average): The average pump efficiency during the simulation.
- Wire to Water Efficiency (Average): The average ratio of the Water Power to the Wire Power.
- Wire Power (Average): The average amount of energy delivered to the pump motor during the simulation.
- Energy Usage (Total): The total energy used during the simulation.
- Energy Use Cost (Total): Total cost of energy used during simulation.
- Energy Usage (Daily): Amount of energy used during a 24-hour period.
- Energy Use Cost (Daily): The cost of the energy used during a 24-hour period, determined by the calculated energy usage and the energy pricing pattern.
- Cost per Unit Volume (Summary): Cost per unit of volume pumped over course of simulation.
- Head (Shutoff): Displays the shutoff head of the referenced pump definition if applicable.
- Head (Design): Displays the design head of the referenced pump definition if applicable.
- Flow (Design): Displays the design flow of the referenced pump definition if applicable.

- Head (Maximum Operating): Displays the maximum operating head of the referenced pump definition if applicable.
- Flow (Maximum Operating): Displays the maximum operating flow of the referenced pump definition if applicable.
- Flow (Maximum Extended): Displays the maximum extended flow of the referenced pump definition if applicable.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Is Closed?: True if the current element is closed during the current time step.
- Is Open?: Set to true if open during the current time step.
- Is Initially Closed?: If true, the initial condition for the control element is "Closed" or "Off."
- Controlled?: Is true if a control action in the current control set references the selected element.
- Cannot Deliver Flow or Head?: If true then the cannot deliver head or cannot deliver flow warning was generated for the element for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.
- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Speed (Maximum, Transient): Maximum pump speed over the course of the transient simulation.
- Speed (Minimum, Transient): Minimum pump speed over the course of the transient simulation.

Pump Station Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Geometry: Specify the geometric coordinates for this entity.
- Scaled Area: A polygon area value obtained from the underlying map dimensions.
- Is Active?: Specifies whether this element is active in the current scenario.
- Controls: Opens a filtered controls editor that only displays the controls associated with the pumps in the selected pump station.
- Pumps: Pump elements that belong to this pump station.
- Time of Use: The total number of pump hours run during the simulation.
- Volume Pumped: The total volume of fluid pumped during the simulation.
- Water Power: The total amount of energy transferred to the water by the pumps in the pump station over the course of the simulation.
- Efficiency (Average) Pump Station: Water power out divided by motor brake power in as a percentage.
- Wire to Water Efficiency (Average): Water power out divided by wire power in as a percentage.
- Wire Power (Total): The total amount of energy delivered to all the pump motors in the pump station during the simulation.
- Energy Usage (Total): The total energy used during the simulation.
- Energy Use Cost (Total): Total cost of energy used during the simulation.
- Energy Usage (Daily): Amount of energy used during the 24-hour period.
- Energy Use Cost (Daily): The cost of energy used during a 24-hour period, determined by the calculated energy usage and the energy pricing pattern.
- Cost per Unit Volume (Summary): Cost of energy divided by the volume pumped over the course of the simulation.
- Flow (Total): Total flow pumped by pumps in the pump station at the current time step.
- Volume Pumped (Incremental): Total volume of flow pumped during the current time step.

- Volume Pumped (Cumulative): Total volume of flow pumped up to the current time step.
- Water Power: The amount of power transferred to the water by the pumps in the pump station at the current time step.
- Efficiency Pump Station: Water power out from the station divided by the motor brake power in at the current time step, expressed as a percentage.
- Wire to Water Efficiency: Water power from the station divided by the wire power in to the station at the current time step, expressed as a percentage.
- Wire Power: The amount of power delivered to the pump motors at the pump station at the current time step.
- Energy Used (Incremental): Total energy used during the current time step.
- Energy Used (Cumulative): Total amount of energy used up to the current time step.
- Energy Price: Cost per unit of energy at the current time step.
- Energy Cost (Incremental): The cost of energy used during the current time step.
- Energy Cost (Cumulative): The total cost of energy used up to the current time step.
- Cost per Unit Volume: Cost per unit of volume pumped for the current time step.

Variable Speed Pump Battery Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Battery Pump Definition: Select pump definition for the lead and lag pumps in the battery.
- Control Node: The node that the battery checks to determine whether to increase, maintain, or decrease its relative speed factor.
- Hydraulic Grade (Target): The Head that the battery will attempt to maintain for the Control Node.
- Relative Speed Factor (Maximum): The highest relative speed factor that the pump can be set at to meet the target head at the control node. If the target head cannot be met when the pump is set at the maximum relative speed factor, the maximum will be used.

- Lag Pump Count: Number of lag pumps (identical to the lead pump) whose relative speed factor is adjusted to maintain the target head for a fixed head VSPB. (Lag pumps are not used for constant flow VSPBs).
- Control Node on Suction Side?: Specifies if the VSPB has a suction side control node.
- Target Flow: The relative speed of the lead pump will be adjusted to meet the Flow (Target). (Lag pumps are not used for constant flow VSPBs).
- VSBP Type: Choices: Fixed Head, Fixed Flow
- Number of Running Lag Pumps (Initial): The initial number of running lag pumps for the transient simulation.
- Relative Speed Factor (Initial): Determines the initial speed of the pump impeller relative to the speed at which the pump curve is defined.
- Status (Initial): Choices: On, Off
- Include in Energy Calculation?: If set to true, cost generated by the element will be included in the calculations, otherwise they will be excluded.
- Energy Pricing: Specify which energy pricing definition is to be used when calculating costs of the corresponding pump.
- Diameter (Pump Valve): Diameter refers to the valve at full opening, typically equal to the internal diameter of the discharge flange.
- Flow (Nominal): Rated or duty flow for the pump, often at or near the best efficiency point.
- Head (Nominal): Rated or duty head for the pump, often at or near the best efficiency point.
- Relative Speed (Initial, Transient): The initial pump relative speed to be used in the transient analysis.
- Torque (Nominal): Specifies the nominal torque that, when multiplied by the Operating Rule's pattern multiplier values will result in the torque values used by the engine.
- Pump Type (Transient): Choices: Shut Down After Time Delay, Constant Speed - No Pump Curve, Constant Speed - Pump Curve, Variable Speed/Torque, Pump Start - Variable Speed/Torque
- Time (Delay until Shut Down): Time at which power to pump motor is shut off. By default, there is no time delay.
- Time (For Valve to Close): The time taken for the pump discharge control valve to close after the transient simulation begins.
- Time (For Valve to Operate): Time to close check valve (or to open it if initial flow is zero). If the check valve allows flow only in one direction, enter 0.
- Control Variable: Choices: Speed, Torque

- Status (Initial, Transient): Choices: On, Off
- Pump Valve Type: Choices: Check Valve, Control Valve
- Operating Rule: Specifies the operation of the valve during a transient simulation.
- Report Period (Transient): Number of time steps between successive printouts of operation. By default, this printout is suppressed.
- Pump Station: The Pump Station to which this Pump belongs.
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.
- Age (Initial): Specify the initial age of the fluid at the selected element.
- Installation Year: Specify the install year of the element. It does not affect the calculations.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.
- Flow (Lead Pump): Flow contributed by the lead pump in the pump battery.
- Number of Running Lag Pumps: Number of pump battery lag pumps running duing the current time step.
- Lag Pump Results: The collection of results for each lag pump in the battery.
- Relative Speed Factor (Calculated): Current relative speed factor of pump at current time step.
- Hydraulic Grade (Suction): Calculated hydraulic grade at suction side of the pump.
- Hydraulic Grade (Discharge): Calculated hydraulic grade at discharge side of the pump.
- Flow (Total): Total flow pumped by standard pump or the pump battery.
- Pump Head: Head gain between suction and discharge side of the pump.
- Pressure (Suction): Calculated pressure at suction side of the pump.
- Pressure (Discharge): Calculated pressure at discharge side of the pump.
- Flow (Absolute): The magnitude of flow through the pump regardless of flow direction.
- Pump Exceeds Operating Range?: Is true if the system demands on the pump exceeds its capabilities.
- Status (Calculated): Choices: On, Off, Pump Cannot Deliver Head (Closed), Pump Result Cannot Deliver Flow (Open)

- Peak Power: Displays the peak energy usage, as calculated during the extended period simulation. This result is displayed even if Peak Demand Charges are not applied.
- Time of Peak Energy Cost: Time when energy cost is maximum.
- Demand Charge: The charge applied per kW.
- Demand Charge Period: Time over which demand charge is averaged in order to get $/day.
- Peak Power Cost: Displays the energy cost as calculated during the extended period simulation. If no Peak Demand Charge has been applied to the associated Energy Price Definition, this field will display as zero.
- Peak Power Cost (Daily): The cost associated with the Peak Demand Charge.
- Volume Pumped (Incremental): Total volume of flow pumped during current time step.
- Volume Pumped (Cumulative): Total volume of flow pumped up to the current time step.
- Water Power: The amount of energy transferred to the water by the pump.
- Pump Efficiency: The Pump Efficiency value is representative of the ability of the pump to transfer the mechanical energy generated by the motor to Water Power.
- Wire to Water Efficiency: The ratio of the Water Power to the Wire Power.
- Wire Power: The amount of energy delivered to the pump motor.
- Energy Used (Incremental): Total energy used during current time step.
- Energy Used (Cumulative): Total amount of energy used up to the current time step.
- Energy Price: Cost per unit of energy.
- Energy Cost (Incremental): The energy cost during the current time step.
- Energy Cost (Cumulative): The total energy cost up to the current time step.
- Cost per Unit Volume: Cost per unit of volume pumped for current time step.
- Relative Speed Factor (Energy Cost Engine): Relative speed of pump at current time step.
- Motor Efficiency: The Motor Efficiency value is representative of the ability of the motor to transform electrical energy to rotary mechanical energy.
- Time of Use: The amount of time the pump is turned on over the course of the simulation.
- Utilization: Percentage of total time during the EPS that the pump was On.
- Volume Pumped (Total): The total volume of fluid pumped during the simulation.

- Water Power (Average): The average amount of energy transferred to the water by the pump over the course of the simulation.
- Pump Efficiency (Average): The average pump efficiency during the simulation.
- Wire to Water Efficiency (Average): The average ratio of the Water Power to the Wire Power.
- Wire Power (Average): The average amount of energy delivered to the pump motor during the simulation.
- Energy Usage (Total): The total energy used during the simulation.
- Energy Use Cost (Total): Total cost of energy used during simulation.
- Energy Usage (Daily): Amount of energy used during a 24-hour period.
- Energy Use Cost (Daily): The cost of the energy used during a 24-hour period, determined by the calculated energy usage and the energy pricing pattern.
- Cost per Unit Volume (Summary): Cost per unit of volume pumped over course of simulation.
- Head (Shutoff): Displays the shutoff head of the referenced pump definition if applicable.
- Head (Design): Displays the design head of the referenced pump definition if applicable.
- Flow (Design): Displays the design flow of the referenced pump definition if applicable.
- Head (Maximum Operating): Displays the maximum operating head of the referenced pump definition if applicable.
- Flow (Maximum Operating): Displays the maximum operating flow of the referenced pump definition if applicable.
- Flow (Maximum Extended): Displays the maximum extended flow of the referenced pump definition if applicable.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Is Closed?: True if the current element is closed during the current time step.
- Is Open?: Set to true if open during the current time step.
- Is Initially Closed?: If true, the initial condition for the control element is "Closed" or "Off."

- Controlled?: Is true if a control action in the current control set references the selected element.
- Cannot Deliver Flow or Head?: If true then the cannot deliver head or cannot deliver flow warning was generated for the element for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.
- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.

Turbine Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Time (Delay until Valve Operates): The time delay prior to operating the spherical valve.
- Time For Valve To Operate: Time required to operate the spherical valve. By default, it is set equal to one time step.
- Diameter (Spherical Valve): The diameter of the spherical valve.
- Efficiency: The efficiency of the turbine. A typical value is 90.
- Moment of Inertia: The (weight) moment of inertia accounts for the turbine, generator, and entrained water.
- Speed (Rotational): Also known as synchronous speed for a turbine. The power it generates depends on it.
- Pattern (Gate Opening): Operating Rule describes the percent wicket gate opening vs time.

- Specific Speed: This represents the type of turbine. HAMMER ships with 4-quadrant curves for: 30, 45, or 60 (US units), 115, 170, or 230 (metric units). You can add your own curves to this library.
- Flow (Rated): Nominal or rated flow of the turbine.
- Head (Rated): Nominal or rated head of the turbine.
- Operating Case: Choices: Instant Load Rejection, Load Rejection, Load Acceptance, Load Variation
- Turbine Curve: Turbine Curve is only required for a steady run. For a transient run, HAMMER uses a 4-quadrant curve based on Specific Speed, Rated Head and Rated Flow.
- Electrical Torque Curve: Defines the time vs torque response for the turbine. Only applies to the Load Rejection operating case.
- Report Period (Transient): Number of time steps between successive printouts of operation. By default, this printout is suppressed.
- Status (Initial): Choices: Open, Closed
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.
- Age (Initial): Specify the initial age of the fluid at the selected element.
- Installation Year: Specify the install year of the element. It does not affect the calculations.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.
- Flow: Total flow through the turbine.
- Headloss: Change in head across turbine.
- Hydraulic Grade (From): Calculated hydraulic grade at the entrance of the turbine.
- Hydraulic Grade (To): Calculated hydraulic grade at the exit of the turbine.
- Pressure (From): Calculated pressure at the entrance of the turbine.
- Pressure (To): Calculated pressure at the exit to the turbine.
- Flow (Absolute): Magnitude of flow through the selected turbine.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.

- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Is Closed?: True if the current element is closed during the current time step.
- Is Open?: Set to true if open during the current time step.
- Is Initially Closed?: If true, the initial condition for the control element is "Closed" or "Off."
- Controlled?: Is true if a control action in the current control set references the selected element.
- Cannot Deliver Flow or Head?: If true then the cannot deliver head or cannot deliver flow warning was generated for the element for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.
- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Speed (Maximum, Transient): Maximum turbine speed over the course of the transient simulation.
- Speed (Minimum, Transient): Minimum turbine speed over the course of the transient simulation.

Valve Attributes

Pressure Reducing Valve (PRV) Attributes

Pressure Breaker Valve (PBV) Attributes

Flow Control Vale (FCV) Attributes

Throttle Control Valve (TCV) Attributes

General Purpose Valve (GPV) Attributes

Pressure Reducing Valve (PRV) Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Discharge Coefficient (Transient): Discharge coefficient, Cv, is defined as: Flow / (Pressure Drop) ^ 0.5.
- Operating Rule: Specifies the operation of the valve during a transient simulation.
- Hydraulic Grade Setting (Initial, Transient): The initial HGL setting for the transient simulation.
- Valve Characteristics: Specifies the valve characteristics definition to be used for this valve. If the Valve Characteristic Curve is not defined then a default curve will be used. The default curve will have (Relative Closure, Relative Area) points of (0,1) and (1,0).
- Valve Type: Choices: Butterfly, Needle, Circular Gate, Globe, Ball, User Defined
- Setting Type: Choices: Pressure, Hydraulic Grade
- Pressure Setting (Initial): Specify the initial pressure setting for the valve.
- Hydraulic Grade Setting (Initial): Specify the initial hydraulic grade setting for the valve.
- Pattern (Valve Settings): Allows you to apply a pattern to make changes to the valve's setting over time for use in extended period simulations. For settings patterns to work the valve must have a Status (Initial) equal to Active. For pressure valves the setting applies to the valve's effective pressure setting irrespective of the Setting Type. Note that changes made to a valve's setting by patterns will override any settings changes made by controls.
- Status (Initial): Choices: Active, Inactive, Closed
- Diameter (Valve): Inside diameter of the valve. Used to calculate the velocity through the valve and a corresponding minor loss when a minor loss coefficient is entered.
- Valve Coefficient Type: Specifies which type of coefficient to enter for the control valve. If entering discharge coefficient, the value is internally converted into an equivalent headloss coefficient (minor loss).
- Discharge Coefficient (Fully Open): The discharge coefficient of the valve when fully open. Used in lieu of minor loss for valves of this coefficient type.

- Specify Local Minor Loss?: If True, the minor coefficient for the element is manually set in the Minor Loss Coefficient (Local) field; otherwise the value is derived from the minor loss library.
- Minor Loss Coefficient (Local): User-input minor loss coefficient. You can either type the value directly in this field or select the value from the minor loss library. The minor loss is applied to the valve when it is fully open (inactive). Note that minor losses do not apply to the following valve types: General Purpose Valve and Valve With Linear Area Change. These two valve types do not support a (fully) open status and always apply the head/flow relationship defined by their headloss curve and discharge coefficient, respectively.
- Valve Type: Specify the type of valve. Choices are Butterfly, Needle, Circular Gate, Globe, Ball, and User Defined.
- Modulate Valve During Transient?: If True, the valve closure will be automatically adjusted to maintain constant valve outlet pressure.
- Opening Rate Coefficient: A constant that relates PRV opening rate during a transient simulation to the difference between the PRV pressure setting and the computed PRV outlet pressure. Units are change in the valve relative closure per second per unit of pressure difference.
- Closing Rate Coefficient: A constant that relates PRV closure rate during a transient simulation to the difference between the PRV pressure setting and the computed PRV outlet pressure. Units are change in the valve relative closure per second per unit of pressure difference.
- Operating Rule: Specifies the operation of the valve during a transient simulation.
- Status (Initial, Transient): Choices: Active, Inactive, Closed
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.
- Age (Initial): Specify the initial age of the fluid at the selected element.
- Installation Year: Specify the install year of the element. It does not affect the calculations.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.
- Hydraulic Grade Setting (Calculated): Hydraulic Grade Setting during current time step.
- Pressure Setting (Calculated): Pressure setting for valve at current time step.
- Flow: Total flow through the valve.
- Velocity: Velocity of flow traveling through the valve.

- Headloss: Change in head across the valve.
- Pressure Loss: Change in pressure across the valve.
- Hydraulic Grade (From): Calculated hydraulic grade at the entrance of the valve.
- Hydraulic Grade (To): Calculated hydraulic grade at the exit of the valve.
- Pressure (From): Calculated pressure at the entrance of the valve.
- Pressure (To): Calculated pressure at the exit to the valve.
- Flow (Absolute): Magnitude of flow through the selected valve.
- Status (Calculated): Choices: Active, Inactive, Closed
- Minor Loss Coefficient (Unified): Displays the current minor loss value for the element, depending on whether its derived or local.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Is Closed?: True if the current element is closed during the current time step.
- Is Open?: Set to true if open during the current time step.
- Is Initially Closed?: If true, the initial condition for the control element is "Closed" or "Off."
- Controlled?: Is true if a control action in the current control set references the selected element.
- Cannot Deliver Flow or Head?: If true then the cannot deliver head or cannot deliver flow warning was generated for the element for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.
- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.

Pressure Sustaining Valve (PSV) Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Discharge Coefficient (Transient): Discharge coefficient, Cv, is defined as: Flow / (Pressure Drop) ^ 0.5.
- Operating Rule: Specifies the operation of the valve during a transient simulation.
- Hydraulic Grade Setting (Initial, Transient): The initial HGL setting for the transient simulation.
- Valve Characteristics: Specifies the valve characteristics definition to be used for this valve. If the Valve Characteristic Curve is not defined then a default curve will be used. The default curve will have (Relative Closure, Relative Area) points of (0,1) and (1,0).
- Valve Type: Choices: Butterfly, Needle, Circular Gate, Globe, Ball, User Defined
- Setting Type: Choices: Pressure, Hydraulic Grade
- Pressure Setting (Initial): Specify the initial pressure setting for the valve.
- Hydraulic Grade Setting (Initial): Specify the initial hydraulic grade setting for the valve.
- Pattern (Valve Settings): Allows you to apply a pattern for changes to the valve's setting over time for extended period simulations. (Note that patterns override settings changes made by controls).
- Status (Initial): Choices: Active, Inactive, Closed
- Diameter (Valve): Inside diameter of the valve. Used to calculate the velocity through the valve and a corresponding minor loss when a minor loss coefficient is entered.
- Valve Coefficient Type: Specifies which type of coefficient to enter for the control valve. If entering discharge coefficient, the value is internally converted into an equivalent headloss coefficient (minor loss).
- Discharge Coefficient (Fully Open): The discharge coefficient of the valve when fully open. Used in lieu of minor loss for valves of this coefficient type.
- Specify Local Minor Loss?: If True, the minor coefficient for the element is manually set in the Minor Loss Coefficient (Local) field; otherwise the value is derived from the minor loss library.

- Minor Loss Coefficient (Local): User-input minor loss coefficient. You can either type the value directly in this field or select the value from the minor loss library. The minor loss is applied to the valve when it is fully open (inactive). Note that minor losses do not apply to the following valve types: General Purpose Valve and Valve With Linear Area Change. These two valve types do not support a (fully) open status and always apply the head/flow relationship defined by their headloss curve and discharge coefficient, respectively.
- Status (Initial, Transient): Choices: Active, Inactive, Closed
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.
- Age (Initial): Specify the initial age of the fluid at the selected element.
- Installation Year: Specify the install year of the element. It does not affect the calculations.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.
- Hydraulic Grade Setting (Calculated): Hydraulic Grade Setting during current time step.
- Pressure Setting (Calculated): Pressure setting for valve at current time step.
- Flow: Total flow through the valve.
- Velocity: Velocity of flow traveling through the valve.
- Headloss: Change in head across the valve.
- Pressure Loss: Change in pressure across the valve.
- Hydraulic Grade (From): Calculated hydraulic grade at the entrance of the valve.
- Hydraulic Grade (To): Calculated hydraulic grade at the exit of the valve.
- Pressure (From): Calculated pressure at the entrance of the valve.
- Pressure (To): Calculated pressure at the exit to the valve.
- Flow (Absolute): Magnitude of flow through the selected valve.
- Status (Calculated): Choices: Active, Inactive, Closed
- Minor Loss Coefficient (Unified): Displays the current minor loss value for the element, depending on whether its derived or local.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.

- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Is Closed?: True if the current element is closed during the current time step.
- Is Open?: Set to true if open during the current time step.
- Is Initially Closed?: If true, the initial condition for the control element is "Closed" or "Off."
- Controlled?: Is true if a control action in the current control set references the selected element.
- Cannot Deliver Flow or Head?: If true then the cannot deliver head or cannot deliver flow warning was generated for the element for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.
- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.

Pressure Breaker Valve (PBV) Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Flow (Initial): This is a typical (positive) flow through the valve.
- Pressure Drop (Initial): Pressure drop corresponding to the typical flow.
- Setting Type: Choices: Pressure, Hydraulic Grade
- Pressure Setting (Initial): Specify the initial pressure setting for the valve.

- Hydraulic Grade Setting (Initial): Specify the initial hydraulic grade setting for the valve.
- Pattern (Valve Settings): Allows you to apply a pattern for changes to the valve's setting over time for extended period simulations. (Note that patterns override settings changes made by controls).
- Status (Initial): Choices: Active, Inactive, Closed
- Diameter (Valve): Inside diameter of the valve. Used to calculate the velocity through the valve and a corresponding minor loss when a minor loss coefficient is entered.
- Valve Coefficient Type: Specifies which type of coefficient to enter for the control valve. If entering discharge coefficient, the value is internally converted into an equivalent headloss coefficient (minor loss).
- Discharge Coefficient (Fully Open): The discharge coefficient of the valve when fully open. Used in lieu of minor loss for valves of this coefficient type.
- Specify Local Minor Loss?: If True, the minor coefficient for the element is manually set in the Minor Loss Coefficient (Local) field; otherwise the value is derived from the minor loss library.
- Minor Loss Coefficient (Local): User-input minor loss coefficient. You can either type the value directly in this field or select the value from the minor loss library. The minor loss is applied to the valve when it is fully open (inactive). Note that minor losses do not apply to the following valve types: General Purpose Valve and Valve With Linear Area Change. These two valve types do not support a (fully) open status and always apply the head/flow relationship defined by their headloss curve and discharge coefficient, respectively.
- Status (Initial, Transient): Choices: Active, Inactive, Closed
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.
- Age (Initial): Specify the initial age of the fluid at the selected element.
- Installation Year: Specify the install year of the element. It does not affect the calculations.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.
- Hydraulic Grade Setting (Calculated): Hydraulic Grade Setting during current time step.
- Pressure Setting (Calculated): Pressure setting for valve at current time step.
- Flow: Total flow through the valve.

- Velocity: Velocity of flow traveling through the valve.
- Headloss: Change in head across the valve.
- Pressure Loss: Change in pressure across the valve.
- Hydraulic Grade (From): Calculated hydraulic grade at the entrance of the valve.
- Hydraulic Grade (To): Calculated hydraulic grade at the exit of the valve.
- Pressure (From): Calculated pressure at the entrance of the valve.
- Pressure (To): Calculated pressure at the exit to the valve.
- Flow (Absolute): Magnitude of flow through the selected valve.
- Status (Calculated): Choices: Active, Inactive, Closed
- Minor Loss Coefficient (Unified): Displays the current minor loss value for the element, depending on whether its derived or local.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Is Closed?: True if the current element is closed during the current time step.
- Is Open?: Set to true if open during the current time step.
- Is Initially Closed?: If true, the initial condition for the control element is "Closed" or "Off."
- Controlled?: Is true if a control action in the current control set references the selected element.
- Cannot Deliver Flow or Head?: If true then the cannot deliver head or cannot deliver flow warning was generated for the element for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.
- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.

- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.

Flow Control Vale (FCV) Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Flow Setting (Initial): Initial flow setting for the flow control valve.
- Discharge Coefficient (Transient): Discharge coefficient, Cv, is defined as: Flow / (Pressure Drop) ^ 0.5.
- Operating Rule: Specifies the operation of the valve during a transient simulation.
- Flow (Initial, Transient): The initial flow to be used in the transient analysis.
- Pattern (Valve Settings): Allows you to apply a pattern for changes to the valve's setting over time for extended period simulations. (Note that patterns override settings changes made by controls).
- Valve Characteristics: Specifies the valve characteristics definition to be used for this valve. If the Valve Characteristic Curve is not defined then a default curve will be used. The default curve will have (Relative Closure, Relative Area) points of (0,1) and (1,0).
- Valve Type: Choices: Butterfly, Needle, Circular Gate, Globe, Ball, User Defined
- Status (Initial): Choices: Active, Inactive, Closed
- Diameter (Valve): Inside diameter of the valve. Used to calculate the velocity through the valve and a corresponding minor loss when a minor loss coefficient is entered.
- Valve Coefficient Type: Specifies which type of coefficient to enter for the control valve. If entering discharge coefficient, the value is internally converted into an equivalent headloss coefficient (minor loss).
- Discharge Coefficient (Fully Open): The discharge coefficient of the valve when fully open. Used in lieu of minor loss for valves of this coefficient type.
- Specify Local Minor Loss?: If True, the minor coefficient for the element is manually set in the Minor Loss Coefficient (Local) field; otherwise the value is derived from the minor loss library.

- Minor Loss Coefficient (Local): User-input minor loss coefficient. You can either type the value directly in this field or select the value from the minor loss library. The minor loss is applied to the valve when it is fully open (inactive). Note that minor losses do not apply to the following valve types: General Purpose Valve and Valve With Linear Area Change. These two valve types do not support a (fully) open status and always apply the head/flow relationship defined by their headloss curve and discharge coefficient, respectively.
- Status (Initial, Transient): Choices: Active, Inactive, Closed
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.
- Age (Initial): Specify the initial age of the fluid at the selected element.
- Installation Year: Specify the install year of the element. It does not affect the calculations.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.
- Flow Setting (Calculated): Flow setting at selected valve for current time step.
- Flow: Total flow through the valve.
- Velocity: Velocity of flow traveling through the valve.
- Headloss: Change in head across the valve.
- Pressure Loss: Change in pressure across the valve.
- Hydraulic Grade (From): Calculated hydraulic grade at the entrance of the valve.
- Hydraulic Grade (To): Calculated hydraulic grade at the exit of the valve.
- Pressure (From): Calculated pressure at the entrance of the valve.
- Pressure (To): Calculated pressure at the exit to the valve.
- Flow (Absolute): Magnitude of flow through the selected valve.
- Status (Calculated): Choices: Active, Inactive, Closed
- Minor Loss Coefficient (Unified): Displays the current minor loss value for the element, depending on whether its derived or local.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.

- Is Closed?: True if the current element is closed during the current time step.
- Is Open?: Set to true if open during the current time step.
- Is Initially Closed?: If true, the initial condition for the control element is "Closed" or "Off."
- Controlled?: Is true if a control action in the current control set references the selected element.
- Cannot Deliver Flow or Head?: If true then the cannot deliver head or cannot deliver flow warning was generated for the element for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.
- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.

Throttle Control Valve (TCV) Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Headloss Coefficient Setting (Initial): Set the initial headloss coefficient for the valve.
- Discharge Coefficient (Initial): The discharge coefficient used at the start of a steady state or EPS run.
- Relative Closure (Initial): The initial relative closure used at the start of a steady state or EPS run. (A relative closure of 0%% means the valve is 0%% closed, or 100%% open. Conversely, a relative closure of 100%% means the valve is 100%% closed, or 0%% open).

- Coefficient Type: Choices: Headloss Coefficient, Discharge Coefficient, Valve Characteristics Curve
- Discharge Coefficient (Transient): Discharge coefficient, Cv, is defined as: Flow / (Pressure Drop) ^ 0.5.
- Operating Rule: Specifies the operation of the valve during a transient simulation.
- Pattern (Valve Settings): Allows you to apply a pattern for changes to the valve's setting over time for extended period simulations. (Note that patterns override settings changes made by controls).
- Pattern (Relative Closures): Allows you to apply a pattern for changes to the valve's relative closure over time for extended period simulations. (Note that patterns override settings changes made by controls).
- Discharge Coefficient (Fully Open): The discharge coefficient of the valve when fully open. Used in lieu of minor loss for valves of this coefficient type.
- Relative Closure (Initial Transient): The initial relative closure of the valve at the start of the transient calculation.
- Valve Characteristics: Specifies the valve characteristics definition to be used for this valve. If the Valve Characteristic Curve is not defined then a default curve will be used. The default curve will have (Relative Closure, Relative Area) points of (0,1) and (1,0).
- Valve Type: Choices: Butterfly, Needle, Circular Gate, Globe, Ball, User Defined
- Status (Initial): Choices: Active, Inactive, Closed
- Specify Local Minor Loss?: If true then the minor coefficent for the element is manually set, otherwise the value is derived from the minor loss library.
- Minor Losses: List of all associated minor losses associated with the element, and can be used to generate the composite minor loss coefficient.
- Diameter (Valve): Inside diameter of the valve. Used to calculate the velocity through the valve and a corresponding minor loss when a minor loss coefficient is entered.
- Status (Initial, Transient): Choices: Active, Inactive, Closed
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.
- Age (Initial): Specify the initial age of the fluid at the selected element.
- Installation Year: Specify the install year of the element. It does not affect the calculations.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.

- Headloss Coefficient Setting (Calculated): TCV headloss coefficient setting at the current time step.
- Discharge Coefficient Setting (Calculated): TCV discharge coefficient setting at the current time step.
- Relative Closure (Calculated): TCV relative closure at the current time step. (A relative closure of 0%% means the valve is 0%% closed, or 100%% open. Conversely, a relative closure of 100%% means the valve is 100%% closed, or 0%% open).
- Flow: Total flow through the valve.
- Velocity: Velocity of flow traveling through the valve.
- Headloss: Change in head across the valve.
- Pressure Loss: Change in pressure across the valve.
- Hydraulic Grade (From): Calculated hydraulic grade at the entrance of the valve.
- Hydraulic Grade (To): Calculated hydraulic grade at the exit of the valve.
- Pressure (From): Calculated pressure at the entrance of the valve.
- Pressure (To): Calculated pressure at the exit to the valve.
- Flow (Absolute): Magnitude of flow through the selected valve.
- Status (Calculated): Choices: Active, Inactive, Closed
- Minor Loss Coefficient (Unified): Displays the current minor loss value for the element, depending on whether its derived or local.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Is Closed?: True if the current element is closed during the current time step.
- Is Open?: Set to true if open during the current time step.
- Is Initially Closed?: If true, the initial condition for the control element is "Closed" or "Off."
- Controlled?: Is true if a control action in the current control set references the selected element.
- Cannot Deliver Flow or Head?: If true then the cannot deliver head or cannot deliver flow warning was generated for the element for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.

- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.

General Purpose Valve (GPV) Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Flow (Initial): This is a typical (positive) flow through the valve.
- Pressure Drop (Initial): Pressure drop corresponding to the typical flow.
- Discharge Coefficient (Transient): Discharge coefficient, Cv, is defined as: Flow / (Pressure Drop) ^ 0.5.
- Operating Rule: Specifies the operation of the valve during a transient simulation.
- Transient Analysis Behavior: Choices: Orifice, Valve
- Valve Characteristics: Specifies the valve characteristics definition to be used for this valve. If the Valve Characteristic Curve is not defined then a default curve will be used. The default curve will have (Relative Closure, Relative Area) points of (0,1) and (1,0).
- Valve Type: Choices: Butterfly, Needle, Circular Gate, Globe, Ball, User Defined
- Status (Initial): Choices: Active, Closed
- Specify Local Minor Loss?: If true then the minor coefficent for the element is manually set, otherwise the value is derived from the minor loss library.
- Minor Losses: List of all associated minor losses associated with the element, and can be used to generate the composite minor loss coefficient.

- Diameter (Valve): Inside diameter of the valve. Used to calculate the velocity through the valve and a corresponding minor loss when a minor loss coefficient is entered.
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.
- Age (Initial): Specify the initial age of the fluid at the selected element.
- Installation Year: Specify the install year of the element. It does not affect the calculations.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.
- Flow: Total flow through the valve.
- Velocity: Velocity of flow traveling through the valve.
- Headloss: Change in head across the valve.
- Pressure Loss: Change in pressure across the valve.
- Hydraulic Grade (From): Calculated hydraulic grade at the entrance of the valve.
- Hydraulic Grade (To): Calculated hydraulic grade at the exit of the valve.
- Pressure (From): Calculated pressure at the entrance of the valve.
- Pressure (To): Calculated pressure at the exit to the valve.
- Flow (Absolute): Magnitude of flow through the selected valve.
- Status (Calculated): Choices: Active, Inactive, Closed
- Minor Loss Coefficient (Unified): Displays the current minor loss value for the element, depending on whether its derived or local.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Is Closed?: True if the current element is closed during the current time step.
- Is Open?: Set to true if open during the current time step.
- Is Initially Closed?: If true, the initial condition for the control element is "Closed" or "Off."

- Controlled?: Is true if a control action in the current control set references the selected element.
- Cannot Deliver Flow or Head?: If true then the cannot deliver head or cannot deliver flow warning was generated for the element for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.
- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.

Valve With Linear Area Change Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Status (Initial): Choices: Active, Closed
- Specify Local Minor Loss?: If true then the minor coefficent for the element is manually set, otherwise the value is derived from the minor loss library.
- Minor Losses: List of all associated minor losses associated with the element, and can be used to generate the composite minor loss coefficient.
- Diameter (Valve): Inside diameter of the valve. Used to calculate the velocity through the valve and a corresponding minor loss when a minor loss coefficient is entered.
- Status (Initial, Transient): Choices: Active, Closed
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.

- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.
- Age (Initial): Specify the initial age of the fluid at the selected element.
- Installation Year: Specify the install year of the element. It does not affect the calculations.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.
- Flow: Total flow through the valve.
- Velocity: Velocity of flow traveling through the valve.
- Headloss: Change in head across the valve.
- Pressure Loss: Change in pressure across the valve.
- Hydraulic Grade (From): Calculated hydraulic grade at the entrance of the valve.
- Hydraulic Grade (To): Calculated hydraulic grade at the exit of the valve.
- Pressure (From): Calculated pressure at the entrance of the valve.
- Pressure (To): Calculated pressure at the exit to the valve.
- Flow (Absolute): Magnitude of flow through the selected valve.
- Status (Calculated): Choices: Active, Inactive, Closed
- Minor Loss Coefficient (Unified): Displays the current minor loss value for the element, depending on whether its derived or local.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Is Closed?: True if the current element is closed during the current time step.
- Is Open?: Set to true if open during the current time step.
- Is Initially Closed?: If true, the initial condition for the control element is "Closed" or "Off."
- Controlled?: Is true if a control action in the current control set references the selected element.
- Cannot Deliver Flow or Head?: If true then the cannot deliver head or cannot deliver flow warning was generated for the element for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.

- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.

Check Valve Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Flow (Typical): This value is 0 should the valve be initially closed.
- Pressure (Threshold): The pressure difference between upstream and downstream side to (re)open the (closed) valve. If 0 entered, the valve (re)opens when the upstream pressure exceeds the downstream pressure.
- Closure Time: Time to close the valve, from the fully open position, after reverse flow is sensed. This establishes the rate of closure in case the valve's opening is partial.
- Open Time: Time to open the valve, from the fully closed position, after specified pressure difference is exceeded. This establishes the rate of opening in case the valve's closure is partial.
- Allow Disruption of Operation?: Determines whether an operation (opening or closing) can be terminated prematurely due to a signal to reverse.
- Located At Wye?: Specifies whether the check valve is simulated as a simple check valve in a run of pipe, or if it is simulated as a wye connection.
- Flow Direction: Choices: Towards Wye, Away from Wye
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.

- Age (Initial): Specify the initial age of the fluid at the selected element.
- Installation Year: Specify the install year of the element. It does not affect the calculations.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.
- Flow: Total flow through the check valve.
- Flow (Absolute): Magnitude of flow through the selected check valve.
- Pressure: Calculated pressure at the check valve.
- Hydraulic Grade: Calculated hydraulic grade at the check valve.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Is Closed?: True if the current element is closed during the current time step.
- Is Open?: Set to true if open during the current time step.
- Is Initially Closed?: If true, the initial condition for the control element is "Closed" or "Off."
- Controlled?: Is true if a control action in the current control set references the selected element.
- Cannot Deliver Flow or Head?: If true then the cannot deliver head or cannot deliver flow warning was generated for the element for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.
- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.

Orifice Between Pipes Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Pressure Drop (Typical): Pressure drop corresponding to the typical flow.
- Flow (Typical): This is a typical (positive) flow through the orifice or valve.
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.
- Age (Initial): Specify the initial age of the fluid at the selected element.
- Installation Year: Specify the install year of the element. It does not affect the calculations.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.
- Flow: Total flow through the orifice.
- Headloss: Change in head across orifice.
- Hydraulic Grade (From): Calculated hydraulic grade at the entrance of the orifice.
- Hydraulic Grade (To): Calculated hydraulic grade at the exit of the orifice.
- Pressure (From): Calculated pressure at the entrance of the orifice.
- Pressure (To): Calculated pressure at the exit to the orifice.
- Flow (Absolute): Magnitude of flow through the selected orifice.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Is Closed?: True if the current element is closed during the current time step.
- Is Open?: Set to true if open during the current time step.

- Is Initially Closed?: If true, the initial condition for the control element is "Closed" or "Off."
- Controlled?: Is true if a control action in the current control set references the selected element.
- Cannot Deliver Flow or Head?: If true then the cannot deliver head or cannot deliver flow warning was generated for the element for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.
- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.

Discharge To Atmosphere Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Discharge Element Type: Choices: Orifice, Valve, Rating Curve
- Gas Volume (Initial): The accumulated air at the orifice at the beginning of the simulation.
- Time to Start Operating: Valve starts to operate after this time.
- Time to Fully Open or Close: Time to close (or open, if zero initial flow) the valve.
- Valve Initial Status: Choices: Open, Closed
- Pressure Head vs. Flow: Specify a collection of Pressure Head vs. Flow points.
- Report Period (Transient): Number of time steps between successive printouts of operation. By default, this printout is suppressed.

- Pressure Drop (Typical): Pressure drop corresponding to the typical flow.
- Flow (Typical): This is a typical (positive) flow through the orifice or valve.
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.
- Is Constituent Source?: If true then the selected node can inject a set concentration of the global constituent into the system.
- Pattern (Constituent): Specify the pattern which dictates how the injected constituent concentration varies over time.
- Constituent Source Type: Choices: Concentration, Flow Paced Booster, Setpoint Booster, Mass Booster
- Concentration (Base): This data field allows you to specify the corresponding constituent concentration at this node over time.
- Mass Rate (Base): This data field allows you to specify the corresponding constituent mass rate at this node over time.
- Age (Initial): Specify the initial age of the fluid at the selected element.
- Discharge (Calculated): Calculated discharge from the node.
- Pressure: Calculated pressure at node.
- Pressure Head: Calculated pressure head at node.
- Hydraulic Grade: Calculated hydraulic grade at node.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.
- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.

- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.

Surge Tank Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Diameter (Orifice): Specifies the diameter of the tank inlet orifice. Only used by the transient engine.
- Ratio of Losses: Ratio of the head losses for equal inflows to / outflows from the tank via the orifice. Default value is 2.5.
- Headloss Coefficient: Applies to flow from the tank to the pipe/riser. This must be a positive number.
- Surge Tank Type: Choices: Simple, Differential
- Has Check Valve?: Specify whether there is a check valve installed on the tank inlet/outlet. For the case of steady state and EPS simulations, a surge tank with a check valve is simulated as a pressure junction.
- Diameter (Internal Riser): This is the upper riser.
- Elevation (Top of Internal Riser): The top of the upper riser.
- Elevation (Junction of Risers): Elevation at which the external and internal risers meet.
- Diameter (External Riser): This is the lower riser.
- Elevation (Orifice from Internal Riser to Tank): Elevation of the internal riser orifice.
- Elevation (Top of Tank Base): The elevation of the top of the hemisherical base of the tank. For a cylindrical tank, this is equal to the pipe elevation.
- Weir Length: The width of the weir.
- Treat as Junction?: Specifies whether or not to treat the surge tank as a junction in steady state and EPS simulations.

- Elevation (Base): Elevation of the storage tank base used as a reference when entering water surface elevations in the tank in terms of levels.
- Elevation (Maximum): Highest allowable water surface elevation or level. If the tank fills above this point, it will be automatically shut off from the system.
- Level (Maximum): A reference level to compare the hydraulic grade in the tank. Does not influence the calculations.
- Diameter: Diameter of tank with constant circular cross-section.
- Area (Average): Cross-Sectional area of tank for constant x-section tanks.
- Volume Full (Input): full active volume of the variable area tank (i.e., the volume at 100% depth), exclusive of any inactive volume.
- Operating Range Type: Choices: Elevation, Level
- Section: Choices: Circular, Non-Circular, Variable Area
- Cross-Section Curve: Defines a curve which specifies the relationship between depth and volume.
- Specify Local Bulk Rate?: If true than a local Bulk Reaction Rate can be specified for the tank, otherwise the bulk reaction rate associated with selected constituent will govern.
- Bulk Reaction Rate (Local): Coefficient defining how rapidly a constituent grows or decays over time.
- Tank Mixing Model: Choices: 2-Compartment, Completely Mixed, FIFO, LIFO
- Compartment 1: Percent of available storage that makes up the first compartment. Inflow and outflow is assumed to take place in the first compartment.
- Compartment 2: Percent of available storage that makes up the second compartment. The second compartment receives overflow from the first, and this overflow is completely mixed.
- Elevation (Minimum): Lowest allowable water surface elevation or level. If the tank drains below this point, it will be automatically shut off from the system.
- Volume (Inactive): The inactive volume of the tank. This volume is the inaccessible volume of the tank that is below the tank active operating range and can become important in water quality simulations subject to the selected mixing model.
- Level (Minimum): Lowest allowable water surface elevation or level. If the tank drains below this point, it will be automatically shut off from the system.
- Elevation (High Alarm): The elevation above which the high level alarm is generated. Calculation notifications are produced to advise you of any alarm level violations.
- Level (High Alarm): The level above which the high level alarm is generated. Calculation notifications are produced to advise you of any alarm level violations.

- Elevation (Low Alarm): The elevation below which the low level alarm is generated. Calculation notifications are produced to advise you of any alarm level violations.
- Level (Low Alarm): The level below which the low level alarm is generated. Calculation notifications are produced to advise you of any alarm level violations.
- Use High Alarm?: Specifies whether or not to check high alarm levels during Steady State/EPS calculation and generate messages if the levels are violated.
- Use Low Alarm?: Specifies whether or not to check low alarm levels during Steady State/EPS calculation and generate messages if the levels are violated.
- Elevation (Initial): Starting water surface elevation/level in the tank.
- Level (Initial): Starting water surface elevation/level in the tank.
- Installation Year: Specify the install year of the element. It does not affect the calculations.
- Elevation (Initial, Transient): Enter a value only if a check valve is installed (i.e., case of a one-way surge tank), or there is an initial inflow/outflow head loss. By default, the intial water surface level is taken equal to the head in the adjacent pipe.
- Report Period (Transient): Number of time steps between successive printouts of operation. By default, this printout is suppressed.
- Demand Collection: A collection of baseline demands and associated temporal patterns.
- Unit Demand Collection: A collection of unit demands, associated unit counts, and temporal patterns.
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.
- Is Constituent Source?: If true then the selected node can inject a set concentration of the global constituent into the system.
- Pattern (Constituent): Specify the pattern which dictates how the injected constituent concentration varies over time.
- Constituent Source Type: Choices: Concentration, Flow Paced Booster, Setpoint Booster, Mass Booster
- Concentration (Base): This data field allows you to specify the corresponding constituent concentration at this node over time.

- Mass Rate (Base): This data field allows you to specify the corresponding constituent mass rate at this node over time.
- Age (Initial): Specify the initial age of the fluid at the selected element.
- Volume Full (Calculated): The full active volume of the tank between the limits of the defined operating range, exclusive of any inactive volume.
- Level (Calculated): Difference between calcuted hydraulic grade and the base elevation of the tank.
- Volume (Calculated): Total volume of fluid in tank including the inactive volume.
- Percent Full: The ratio of tank active volume to the calculated tank full active volume. Active volume is the tank volume within the operating range and is exclusive of inactive volume.
- Status (Calculated): Choices: Empty, Emptying, Filling, Full, Stagnant
- Flow (Out net): Net flow out of the element.
- Flow (In net): Net flow into the element.
- Demand Adjusted Population: Population of area supplied by current node. This value is derived from the unit demand loads applied to the collection and their equivalent populations.
- Hydraulic Grade: Calculated hydraulic grade at node.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.
- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.

Hydropneumatic Tank Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Volume of Gas (Initial): The initial volume of gas in the pressure vessel at the start of the simulation. During the transient event, this gas volume expands or compresses, depending on the transient pressures in the system. Not used in steady state or EPS analyses.
- Diameter (Tank Inlet Orifice): The size of the opening between the gas vessel and the main pipe line. It is typically smaller than the main pipe size.
- Ratio of Losses: For same flow magnitude, ratio of inflow head loss to outflow loss. Default value is 2.5.
- Gas Law Exponent: Refers to the exponent to be used in the gas law equation. The usual range of this exponent is 1.0 to 1.4.
- Pressure (Gas-Preset): If there is a bladder, this is the pressure of the gas prior to exposing the tank to pipeline pressure; otherwise, this should be omitted as it is ignored.
- Liquid Elevation (Mean): The mean elevation of the liquid at the gas-liquid interface. (Liquid level referenced from a datum of 0).
- Elevation Type: Choices: Fixed, Mean Elevation, Variable Elevation
- Variable Elevation Curve: Defines the gas vessel chamber geometry as a function of liquid elevation versus equivalent diameter.
- Minor Loss Coefficient (Outflow): Dimensionless quantity, typical value = 2.5. This property is used only for transient analysis, to restrict the flow out of the hydropneumatic tank.
- Elevation (Base): Elevation of the storage tank base used as a reference when entering water surface elevations in the tank in terms of levels.
- Treat as Junction? - Selects whether or not the hydropneumatic tank is treated as a junction in steady state and EPS simulations. Note that if you wish to use the steady state/EPS results as input for a HAMMER transient analysis and you set this field to **True**, you will need to manually enter the **Volume of Gas (Initial)** for the tank for HAMMER

- Volume of Gas (Initial) - The initial volume of gas in the pressure vessel at the start of the simulation. During the transient event, the gas volume expands or compresses, depending on the transient pressures in the system. This value is not used in steady state or EPS analyses.
- Operating Range Type: Choices: Elevation, Level
- Tank Calculation Model: Choices: Constant Area Approximation, Gas Law Model
- Volume (Effective): The effective volume of the constrant area approximation hydropneumatic tank.
- HGL On: The lowest operational hydraulic grade desired for the hydropneumatic tank. You should define a simple or logical control that uses this hydraulic grade as the minimum operational value. For example, define a control to turn on a pump.
- HGL Off: The highest operational hydraulic grade desired for the hydropneumatic tank. You should define a simple or logical control that uses this hydraulic grade as the maximum operational value. For example, define a control to turn off a pump.
- Atmospheric Pressure Head: This field represents atmospheric pressure and is used in the gas law model computation of the hydropneumatic tank.
- HGL (Initial): Starting water surface elevation/level in the tank. Used in steady state and EPS analyses.
- Level (Initial): Starting water surface elevation/level in the tank. Used in steady state and EPS analyses.
- Liquid Volume (Initial): Starting liquid volume in the tank. For constant area approximation tanks this volume includes the inactive volume of the tank that lies below the effective volume. Only used in steady state and EPS analyses.
- Air Flow Curve (Air Inflow Orifice): Curve that defines orifice behavior for the injection of air into the pipeline.
- Air Flow Curve (Air Outflow Orifice): Curve that defines discharge of air when the volume is greater than or equal to the transition volume (TV).
- Air Flow Calculation Method: Choices: Orifice Diameter, Air Flow Curve
- Installation Year: Specify the install year of the element. It does not affect the calculations.
- Elevation (Initial, Transient): Enter a value only if a check valve is installed (i.e., case of a one-way surge tank), or there is an initial inflow/outflow head loss. By default, the intial water surface level is taken equal to the head in the adjacent pipe.
- Report Period (Transient): Number of time steps between successive printouts of operation. By default, this printout is suppressed.

- Demand Collection: A collection of baseline demands and associated temporal patterns.
- Unit Demand Collection: A collection of unit demands, associated unit counts, and temporal patterns.
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.
- Is Constituent Source?: If true then the selected node can inject a set concentration of the global constituent into the system.
- Pattern (Constituent): Specify the pattern which dictates how the injected constituent concentration varies over time.
- Constituent Source Type: Choices: Concentration, Flow Paced Booster, Setpoint Booster, Mass Booster
- Concentration (Base): This data field allows you to specify the corresponding constituent concentration at this node over time.
- Mass Rate (Base): This data field allows you to specify the corresponding constituent mass rate at this node over time.
- Age (Initial): Specify the initial age of the fluid at the selected element.
- Gas Volume (Calculated): The calculated volume of gas in the hydropneumatic tank.
- Pressure (Calculated): The calculated pressure in the hydropenumatic tank.
- Liquid Volume (Calculated): The calculated liquid volume in the hydropneumatic tank.
- Percent Full: The ratio of the fluid volume in the tank to the calculated full volume of the tank.
- Flow (Out net): Net flow out of the element.
- Flow (In net): Net flow into the element.
- Demand Adjusted Population: Population of area supplied by current node. This value is derived from the unit demand loads applied to the collection and their equivalent populations.
- Hydraulic Grade: Calculated hydraulic grade at node.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.

- Concentration (Calculated): Concentration at selected element for current time step.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.
- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Gas Pressure (Maximum, Transient): Maximum gas pressure at node over the course of the transient simulation.
- Gas Pressure (Minimum, Transient): Minimum gas pressure at node over the course of the transient simulation.
- Gas Volume (Maximum, Transient): Maximum gas volume at node over the course of the transient simulation.
- Gas Volume (Minimum, Transient): Minimum gas volume at node over the course of the transient simulation.
- Water Level (Maximum, Transient): Maximum water level in tank over the course of the transient simulation.
- Water Level (Minimum, Transient): Minimum water level in tank over the course of the transient simulation.

Air Valve Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.

- Air Volume (Initial): Volume of air near the valve at the start of the simulation. The default is zero. If volume is nonzero, the pressure must be zero.
- Time to Close: For an air valve, adiabatic compression (i.e., gas law exponent = 1.4) is assumed.The valve starts to close linearly with respect to area only when air begins to exit from the pipe. If air subsequently re-enters, then the valve opens fully again. It is possible for liquid to be discharged through this valve for a period after the air has been expelled.
- Diameter (Small Air Outflow Orifice): Diameter of the air outflow orifice (the orifice through which air is expelled from the pipeline) when the local air volume is less than the transition volume (TV), or the air pressure is greater than the transition pressure (TP) (depending on which trigger is used to switch the outflow orifice size). This diameter is typically small enough for the injected air to be compressed, which can help prevent severe transient pressures. Generally air flows out the large air outflow orifice for some time before switching to the small air outflow orifice for the final stages of air release.
- Diameter (Large Air Outflow Orifice): Refers to the discharge of air when the local air volume is greater than or equal to the transition volume (TV), or the air pressure is less than or equal to the transition pressure (TP) (depending on which trigger is used to switch the outflow orifice size). This diameter is typically large enough that there is little or no restriction to air outflow. Generally air flows out the large air outflow orifice for some time before switching to the small air outflow orifice for the final stages or air release.
- Diameter (Air Inflow Orifice): Diameter of the air inflow orifice (the orifice through which air enters the pipeline when the pipe internal pressure is less than atmospheric pressure). This diameter should be large enough to allow the free entry of air into the pipeline. If set to zero, this diameter is considered infinite (i.e. there is no restriction to air inflow).
- Diameter (Air Outflow Orifice): Diameter of the air outflow orifice (the orifice through which air is expelled from the pipeline).
- Transition Pressure: The local internal system air pressure at the air valve above which the transient solver switches from using the large air orifice to the small air orifice (in order to minimize transients).
- Transition Volume: The local volume of air at the air valve below which the transient solver switches from using the large air orifice to the small air orifice (in order to minimize transients). This volume often corresponds to the volume of the body of the air valve.
- Air Flow Curve (Small Air Outflow Orifice): Curve that defines discharge of air when the air volume is less than the transition volume (TV), or the air pressure is greater than the transition pressure (TP).
- Air Flow Curve (Large Air Outflow Orifice): Curve that defines discharge of air when the air volume is greater than or equal to the transition volume (TV), or the air pressure is less than or equal to the transition pressure (TP).

- Air Valve Type: Choices: Slow Closing, Double Acting, Triple Acting, Vacuum Breaker
- Trigger to Switch Outflow Orifice Size: Select whether the transient solver switches from the large air outflow orifice to the small air outflow orifice based on Transition Volume or Transition Pressure.
- Report Period (Transient): Number of time steps between successive printouts of operation. By default, this printout is suppressed.
- Treat Air Valve as Junction?: Specifies whether or not to treat the air-valve as a junction element in steady state and EPS simulations. If false, the valve may allow part full flow subject to the prevailing hydraulic conditions.
- Air Flow Curve (Air Inflow Orifice): Curve that defines orifice behavior for the injection of air into the pipeline.
- Air Flow Curve (Air Outflow Orifice): Curve that defines discharge of air when the volume is greater than or equal to the transition volume (TV).
- Air Flow Calculation Method: Choices: Orifice Diameter, Air Flow Curve
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.
- Is Constituent Source?: If true then the selected node can inject a set concentration of the global constituent into the system.
- Pattern (Constituent): Specify the pattern which dictates how the injected constituent concentration varies over time.
- Constituent Source Type: Choices: Concentration, Flow Paced Booster, Setpoint Booster, Mass Booster
- Concentration (Base): This data field allows you to specify the corresponding constituent concentration at this node over time.
- Mass Rate (Base): This data field allows you to specify the corresponding constituent mass rate at this node over time.
- Age (Initial): Specify the initial age of the fluid at the selected element.
- Pressure: Calculated pressure at node.
- Pressure Head: Calculated pressure head at node.
- Hydraulic Grade: Calculated hydraulic grade at node.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.

- Concentration (Calculated): Concentration at selected element for current time step.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.
- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.

Surge Valve Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Diameter (SAV): The valve's characteristics are determined by its Cv and type, so that the diameter is only used for descriptive purposes.
- Threshold Pressure (SAV): Pressure below which the SAV opens.
- Time for SAV to Open: Time for the SAV to open fully after being triggered.
- Time SAV Stays Fully Open: Time that SAV remains fully open (i.e., time between the end of the opening phase and the start of the closing phase).
- Time for SAV to Close: Time for the SAV to close fully, measured from the time that it was completely open.
- Discharge Coefficient (when SAV Fully Open): Discharge coefficient, Cv, is defined as: Flow / (Pressure Drop) ^ 0.5.
- Threshold Pressure (SRV): Pressure above which the SRV opens.
- Diameter (SRV): The diameter of the SRV.

- Spring Constant (SRV): Change in restoring force of the return spring per unit lift off seat. A possible value is 150 lb/in. (26.27 N/mm).
- SAV / SRV Type: Choices: Surge Anticipator Valve, Surge Relief Valve, Surge Anticipator & Relief Valve
- Valve Type: Choices: Needle, Circular Gate, Globe, Ball, Butterfly
- SAV Closure Trigger: Choices: Time, Threshold Pressure
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.
- Is Constituent Source?: If true then the selected node can inject a set concentration of the global constituent into the system.
- Pattern (Constituent): Specify the pattern which dictates how the injected constituent concentration varies over time.
- Constituent Source Type: Choices: Concentration, Flow Paced Booster, Setpoint Booster, Mass Booster
- Concentration (Base): This data field allows you to specify the corresponding constituent concentration at this node over time.
- Mass Rate (Base): This data field allows you to specify the corresponding constituent mass rate at this node over time.
- Age (Initial): Specify the initial age of the fluid at the selected element.
- Pressure: Calculated pressure at node.
- Pressure Head: Calculated pressure head at node.
- Hydraulic Grade: Calculated hydraulic grade at node.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.
- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.

- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.

Rupture Disk Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Pressure Threshold: The pressure above which the rupture disk breaks to vent the liquid to atmosphere.
- Pressure Drop (Typical): Pressure drop corresponding to the typical flow.
- Flow (Typical): This is a typical (positive) flow through the orifice or valve.
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Trace (Initial): Specify the initial trace amount at the current location.
- Zone: Specify the zone for the element.
- Concentration (Initial): Specify the initial concentration for the global concentration at the selected element.
- Is Constituent Source?: If true then the selected node can inject a set concentration of the global constituent into the system.
- Pattern (Constituent): Specify the pattern which dictates how the injected constituent concentration varies over time.
- Constituent Source Type: Choices: Concentration, Flow Paced Booster, Setpoint Booster, Mass Booster
- Concentration (Base): This data field allows you to specify the corresponding constituent concentration at this node over time.
- Mass Rate (Base): This data field allows you to specify the corresponding constituent mass rate at this node over time.

- Age (Initial): Specify the initial age of the fluid at the selected element.
- Pressure: Calculated pressure at node.
- Pressure Head: Calculated pressure head at node.
- Hydraulic Grade: Calculated hydraulic grade at node.
- Age (Calculated): Age at selected element for current time step.
- Trace (Calculated): Trace at selected element for current time step.
- Concentration (Calculated): Concentration at selected element for current time step.
- Has Calculation Messages Now?: If true then the current element has associated calculation warning messages for the current time step.
- Head (Maximum, Transient): Maximum head at node over the course of the transient simulation.
- Head (Minimum, Transient): Minimum head at node over the course of the transient simulation.
- Pressure (Maximum, Transient): Maximum pressure at node over the course of the transient simulation.
- Pressure (Minimum, Transient): Minimum pressure at node over the course of the transient simulation.
- Air Volume (Maximum, Transient): Maximum air volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.
- Vapor Volume (Maximum, Transient): Maximum vapor volume at node over the course of the transient simulation. Not applicable to Reservoirs and Rating curves.

Isolation Valve Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Diameter (Valve): Inside diameter of the valve. Used to calculate the velocity through the valve and a corresponding minor loss when a minor loss coefficient is entered.
- Minor Loss Coefficient: K value in the minor headloss equation.
- Is Operable?: If true, valve can be used in identifying segments.

- Status (Initial): Choices: Open, Closed
- Elevation: Elevation at centroid of junctions, valves, and pumps; the ground elevation at tanks; the hydraulic grade at reservoirs.
- Installation Year: Specify the install year of the element. It does not affect the calculations.
- Zone: Specify the zone for the element.
- Hydraulic Grade: Hydraulic Grade at valve location on pipe.
- Pressure: Pressure at valve location on pipe.
- Flow: Magnitude of flow through isolation valve.
- Velocity: Velocity through the isolation valve.
- Distance from End Point (Unified): Presents the active Distance From End Point for the current isolation valve.
- Is Closed?: True if current isolation valve is closed during the current time step.

Spot Elevation Attributes

- ID: Unique identifier assigned to this element.
- Label: Descriptive label for this element.
- Notes: Additional information about this element.
- GIS-IDs: List of associated IDs on the GIS/data-source side.
- Hyperlinks: Associate one or more web link, photo, word processing document, or other file with this element.
- Hydraulic Grade (Enhanced)Interpolated hydraulic grade at this location.
- Pressure (Enhanced)Pressure based on the interpolated hydraulic grade.

Glossary 21

Glossary

A B C D E F G H I L M N O P R S T V W X

A

Age:	An analysis for the age of water determines how long the water has been in the system, and is a general water quality indicator.
Available Fire Flow:	Amount of flow available at a node for fire protection while maintaining all fire flow pressure constraints.

B

.bak:	Extension for backup files.
Base Elevation & Level:	Elevation from which all tank levels are measured. For example, a tank level of two meters represents a water surface elevation two meters above the base elevation.
Boundary Node:	Node with a known hydraulic grade. It may be static (unchanging with time), such as a reservoir, or dynamic (changes with time), such as a tank. Every pipe network must contain at least one boundary node. In order to compute the hydraulic grade at the other nodes in the network, they must be reachable from a boundary.
Bulk Reaction Coefficient:	Coefficient used to define how rapidly a constituent grows or decays over time. It is expressed in units of 1/time, for first-order reactions.

C

Calc. Min. System Pressure: Minimum calculated pressure of all junctions in the system during fire flow withdrawal at a node.

Calc. Min. Zone Pressure: Minimum calculated pressure of all junctions in the same zone as the node where fire flow withdrawal occurs.

Calc. Residual Pressure: Calculated pressure at the junction node where the fire flow withdrawal occurs.

Calculation Unready: An element that does not have all the required information for performing an analysis is considered to be calculation unready.

C-Coefficient: Roughness coefficient used in the Hazen-Williams Equation.

Check Valve: Prevents water from flowing backwards through the pipe. In other words, water can only flow from the From Node to the To Node.

Closed/Inactive Status: You can control the status of a valve to be either inactive or closed. Inactive means that the valve will act like an open pipe where flow can occur in either direction, and the headloss across the valve will be calculated using the valve's minor loss factor. Closed means that no flow will occur through the valve.

Constituent: Any substance, such as chlorine or fluoride, for which the growth or decay can be adequately described through the use of a bulk reaction coefficient and a wall reaction coefficient.

Context Menu: A shortcut menu opened by right-clicking a project element or data entry field. Commands on the context menu are specific to the current state of the selected item.

Control Status: A pressure pipe can be either Open or Closed. Open means that flow occurs in the pipe, and Closed means that no flow occurs in the pipe.

Conveyance Element: A pipe or channel used to transport water.

Coordinates: Distances perpendicular to a set of reference axes. Some areas may have predefined coordinate systems, while other coordinate systems may be arbitrary. Coordinates may be presented as X and Y values or may be defined as Northing and Easting values, depending on individual preferences.

Cross Section Type:	Tanks can have either a constant area cross section or a variable area cross section. The cross section of a tank with a constant area is the same throughout the depth. The cross section of a tank with a variable area varies throughout the depth.
Crosshair:	The cursor that looks like a plus sign (+).
Current Storage Volume:	The volume of water currently stored in a tank. It includes both the hydraulically active volume and the hydraulically inactive volume.
CV:	Check valve.

D

.dgn:	Drawing information in MicroStation.
.dwg:	Drawing information in AutoCAD.
.dwh:	Drawing information in Stand-Alone.
Database Connections:	A connection represented by a group of database links. There may be a single linked external file within a connection, or there may be several external file links within a single connection.
Dataset:	A Bentley WaterCAD V8*i* project.
DBMS:	An acronym that stands for Database Management System. These systems can be relational (RDBMS) or non-relational.
DEM:	Digital elevation model.
Demand:	Represents the total demand from an individual junction for the current time period. It is based on the information from the Demand tab of the Junction Editor.
Design Point:	Point at which a pump was originally intended to operate, and is typically the best efficiency point (BEP) of the pump. At discharges above or below this point, the pump is not operating under optimum conditions.
Diameter:	Refers to a pipe or valve's inside diameter. It is the distance between two internal points directly opposite each other.
Discharge:	Volumetric rate of flow given in units of length^3/time.
DLG:	Digital line graph.
Double-Click:	To click the left mouse button twice in rapid succession.

Drag:	To hold down one of the mouse buttons while you move the mouse.

E

Element:	An object such as a tank, junction node, or pipe in a drawing.
Elevation:	The distance from a datum plane to the center of the element. Elevations are often referenced with mean sea level as the datum elevation.
Energy Grade Line (EGL):	Sum of datum (base elevation), elevation, velocity head, and pressure head at a section.
EPS:	Extended Period Simulation.
Extended Edit:	A small button with an ellipsis (…) as the label. Extended edit buttons are located next to drop-down choice lists, and provide further editing for the associated choice list items.
External Files:	Any file outside of this program that can be linked. These include database files (such as Dbase) and spreadsheets (such as Excel). Throughout the documentation, all of these file types will be referred to as databases or external files interchangeably.
Extrapolate:	To infer a value based on other known values, with the desired value lying outside the known range. Often based upon extending the slope of the line connecting the previous known values to the desired point. See also: interpolate.

F

Feature Class:	1. A classification describing the format of geographic features and supporting data in a coverage. Coverage feature classes for representing geographic features include point, arc, node, route-system, route, section, polygon and region. One or more coverage features are used to model geographic features; for example, arcs and nodes can be used to model linear features such as street centerlines. The tic, annotation, link, and boundary feature classes provide supporting data for coverage data management and viewing.

	2. The conceptual representation of a geographic feature. When referring to geographic features, feature classes include point, line, area, and surface.
Feature Dataset:	A feature dataset is a collection of feature classes that share the same spatial reference.
Field Links:	Define the actual mapping between model element attributes and columns within each database table.
File Extension:	The period and three characters, typically, at the end of a filename. A file extension usually identifies the kind of information the file contains. For example, files you create in AutoCAD have the extension *.DWG.
Fire Flow Upper Limit:	The maximum allowable fire flow that can occur at a withdrawal location. This is a user-specified practical limit that will prevent this program from computing unrealistically high fire flows at locations such as primary system mains, which have large diameters and high service pressures. Remember that a system's ability to deliver fire flows is ultimately limited by the size of the hydrant opening and service line, as well as the number of hydrants available to combat a fire at a specific location.
Flow:	Represents the calculated value of the pipe, valve, or pump discharge at the given time.
From Node:	Represents the pipe's starting node. Positive flow rates are in the direction of from towards to. Negative flow rates are in the opposite direction.
From Pipe:	The pipe that connects to the upstream side of a valve or pump.

G

GA:	Genetic algorithm.
GEMS Datastore:	The relational database that Bentley WaterCAD V8*i* uses to store model data. Each Bentley WaterCAD V8*i* project uses two main files for data storage, the datastore (.sqlite) and the Bentley WaterCAD V8*i* Modeler-specific data (.wtg). Although the Bentley WaterCAD V8*i* datastore is an .sqlite file, **cannot** be a geodatabase.

Generations:	The maximum value for genetic algorithm generations is determined by the Maximum Era Number and Era Generation Number you set in the GA Parameters. The actual number of generations that get calculated depend on the Stopping Criteria you set.
Geodatabase:	Short for geographic database, a geodatabase stores spatial and descriptive data in an efficient manner. Geodatabases are the standard file format for ArcGIS v8 and later.

H

Headloss:	Represents the energy lost due to friction and minor losses. The headloss field displays the pipe, valve, or pump's total headloss at the given time.
Headloss Gradient:	Presents the headloss in the pipe as a slope, or gradient. This allows you to more accurately compare headlosses for pipes of different lengths.
Hydraulic Grade:	Elevation to which water would rise under zero pressure. For open surfaces, such as reservoirs and tanks, this is equal to the water surface elevation. The hydraulic grade field presents the hydraulic grade for the element at the current time period as calculated based on the system flow rates and head changes.
Hydraulic Grade Setting:	The constraint to which a valve regulates, expressed in units of head (Length). Depending on the type of valve, it may refer to either the upstream or downstream hydraulic grade or the headloss across the valve.

I

:Inactive Volume:	The volume of water below the minimum elevation of the tank. This volume of water is always present, even when the tank reaches its minimum elevation and closes itself off from the system. Therefore, it is hydraulically inactive. It is primarily used for water quality calculations.
Inflow & Outflow:	An inflow is a flow into a node from the system, while an outflow is a flow from the node into the system. A negative outflow is the same as a positive inflow, and a negative inflow is the same as a positive outflow.

Inheritance:	Refers to the parent-child relationships used by scenarios and alternatives. Just as in the natural world, inheritance is used to refer to the situation where an entity receives something from its parent. For example, we speak of a child inheriting blue eyes from a parent. Unlike in the natural world, inheritance in scenarios and alternatives is dynamic. If the parent's attribute changes, the child's attribute automatically changes at the same time, unless the value is explicitly changed in a child.
Initial Settings:	Sets the status of an element for a steady-state analysis or the first time step in an extended period simulation. The initial settings for a pipe, pump, or valve can be set using the elemental dialog boxes or a table.
Initial Water Quality:	Represents the starting conditions at a node for age, trace, or constituent concentration. The initial value will be slightly different depending on the analysis type.
Interpolate:	Estimating a value between two known values assuming a linear relationship. See also: extrapolate.
Invert:	Lowest point of a pipe opening. Sometimes referred to as the flow line.

L

Label:	The unique name by which an element will be referenced in reports, error messages, and tables.
Length:	Represents the distance on a pipe from the From Node to the To Node, according to the scaled length of the pipe. To enter an overriding length, click the **User Defined Length** field and type in your desired length value.
LIDAR:	Light Detection and Ranging.

M

.sqlite:	The open database file.
Manning's Coefficient:	Roughness coefficient used in Manning's Formula.
Material:	The selection of a pipe's construction material. This material will be used to determine a default value for the pipe's roughness.

Maximum Elevation: The highest allowable water surface elevation in a tank. If the tank fills above this point, it will automatically shut off from the system.

Max. Extended Operating Point: The absolute maximum discharge at which a pump can operate, with zero head being added to the system. This value may be computed by the program or entered manually.

Maximum Operating Point: The highest discharge for which a pump is actually intended to run. At discharges above this point, the pump may behave unpredictably, or its performance may decline rapidly.

Menu: A menu of available commands or actions you can perform. Access menus from the menu bar at the top of the main program window.

Messages: The section that contains information generated during the calculation of the model, such as warnings, errors, and status updates.

Messages Light: A light that appears on the Tab of the Messages sheet. The light will be red if errors occurred during the analysis, yellow if there are warnings or cautions, and green if there are no warnings or errors.

Metadata: Additional information (aside from tabular and spatial data) that makes the data useful. Includes characteristics and information that are required to use the data but are not contained within the data itself.

Minimum Elevation: The lowest allowable water surface elevation in a tank. If the tank drains below this point, it will automatically shut off from the system.

Minimum System Junction: The junction where the calculated minimum system pressure occurs.

Minimum System Pressure: The minimum pressure allowed at any junction in the entire system as result of fire flow withdrawal. If the pressure at a node anywhere in the system falls below this constraint while withdrawing fire flow, fire flow will not be satisfied. A fire flow analysis may be configured to ignore this constraint.

Minimum Zone Junction: The junction where the calculated minimum zone pressure occurs.

Minimum Zone Pressure: The minimum pressure to maintain at all junction nodes within a Zone. The model determines the available fire flow such that the minimum zone pressures do not fall below this target pressure. Each junction has a zone

	associated with it, which can be specified in the junction's input data. If you do not want a junction node to be analyzed as part of another junction node's fire flow analysis, move it to another Zone.
Minor Loss:	The field that presents the total minor loss K value for a pipe or valve. If an element has more than one minor loss, each can be entered individually by clicking the **Ellipsis (…)** button.
Modeler/Stand-Alone:	The Bentley software environment, and not the AutoCAD one.
Mouse Buttons:	The left mouse button is the primary button for selecting or activating commands. The right mouse button is used to activate shortcut context menus and help. Note that the mouse button functions can be redefined using the Windows Control Panel. If your mouse is equipped with a mouse wheel, you can use it for various panning and zooming functions.

N

.nrg:	File containing energy cost results.
Needed Fire Flow:	The flow rate required at a junction to satisfy fire flow demands.
Network Element:	An element that forms part of the network model. Annotation elements, such as polylines, borders, and text, are not network elements.
Number:	The number of parallel conveyance elements in a model.
Notes:	The field that allows you to enter text relevant to the model. It may include a description of an element, a summary of your data sources, or any other information of interest.

O

.out:	File with complete scenario results.
ODBC:	Open Database Connectivity (ODBC) is a standard programming interface developed by Microsoft for accessing data in relational and non-relational database management systems (DBMS).

On/Off Status:	The status of a pump can be either on or off. On means that flow will occur in the downstream direction, and the pump will add head to the system according to it's characteristic curve. Off means that no flow will occur, and no head will be added.
Open/Closed Status:	The status of a pipe can be either open or closed. Open means that flow can occur in either direction. Closed means that no flow will occur through the pipe.

P

.pv8:	The previous version for files upgraded to new.
PBV:	Pressure breaker valve.
Percent Full:	The ratio of the current storage volume to the total storage volume, multiplied by 100.
Pipe Status:	Indicates whether the pipe is open or closed. As input, this determines how the pipe begins the simulation. As output, it shows the calculated status of the pipe at the given time.
Polyline:	A composite element that consists of a series of line segments. Each line segment begins and ends at a vertex. A vertex may be another element such as a junction, tank, or pump.
Power:	Represents the water horsepower of a pump. This is the horsepower that is actually transferred from the pump into the water. Depending on the pump's efficiency, the actual power consumed (brake horsepower) may vary.
Pressure:	The field that displays the pressure for the current time period.
Pressure Setting:	The constraint to which a valve regulates, expressed in units of pressure (Force per Length^2). Depending on the type of valve, it may refer to either the upstream or downstream pressure or the pressure drop.
PRV:	Pressure reducing valve.
PSV:	Pressure sustaining valves.
Pump Status:	A pump can have two different status conditions: On, which is normal operation, or Off, which is no flow under any condition.

R

.rpc:	The file with scenario messages.
RDBMS:	An acronym that stands for Relational Database Management System.
Relate:	A temporary connection between table records using a common item shared by both tables. Each record in one table is connected to those records in the other table that share the same value for the common item.
Relational Database:	A database in which the data is structured in such a way as to associate tables according to attributes that are shared by the tables.
Relational Join:	The process of merging two attribute tables using a common item.
Relative Speed Factor:	Defines the characteristics of a pump relative to the speed for which the pump curve was entered, in accordance with the affinity laws. A speed factor of 1.00 would indicate pump characteristics identical to those of the original pump curve.
Residual Pressure:	The minimum residual pressure to occur at a junction node. The program determines the amount of fire flow available such that the residual pressure at a junction node does not fall below this target pressure.
Reynolds Number:	Ratio of viscous forces relative to inertial forces. A high Reynold's number indicates turbulent flow, while a low number indicates laminar flow.
Roughness:	A measure of a pipe's resistance to flow. Pipes of different ages, construction material, and workmanship may have different roughness values.
Roughness Coefficient:	A value used to represent the resistance of a conveyance element to flow. In the Manning's equation, this value is inversely proportional to flow. The smaller the roughness coefficient, the greater the flow.

S

Satisfies Fire Flow:	A true or false statement indicating whether this junction node meets the fire flow constraints. A check mark in the box means the Fire Flow Constraints were satisfied for that node. If there is no check mark, the Fire Flow Constraints were NOT satisfied.

Schema:	A diagrammatic representation; an outline or model. Essentially, a schema represents the number of tables, the columns they contain, the data types of the columns, and any relationships between the tables.
Select:	The process of adding one or more elements to an active selection set.
Selection Set:	The active group of selected elements. A selection set allows editing or an action, such as move or delete, to be performed on a group of elements.
Shape:	The cross-sectional geometric form of a conveyance element (i.e., circular, box, arch, etc.).
Shapefile:	A file format that stores spatial and attribute data for the spatial features within the dataset. A shapefile consists of a main file, an index file, and a dBASE table. Shapefiles were the standard file storage format for ArcView 3.x and earlier.
Shutoff Point:	The point at which a pump will have zero discharge. Typically the maximum head point on a pump curve.
Size:	Inside diameter of a pipe section for a circular pipe.
Spatial Reference:	The spatial reference for a feature class describes its coordinate system (for example, geographic, UTM, and State Plane), its spatial domain, and its precision. The spatial domain is best described as the allowable coordinate range for x, y coordinates, m- (measure) values, and z-values. The precision describes the number of system units per one unit of measure. A spatial reference with a precision of 1 will store integer values, while a precision of 1000 will store three decimal places.
Stand-Alone/Modeler:	The Bentley Systems software environment, and not the AutoCAD one.
Starting Elevation:	The value that is used as the beginning condition for an extended period simulation.
Status Pane:	The area at the bottom of the window used for displaying status information.
Storage Node:	Special type of node where a free water surface exists, and the hydraulic head is the elevation of the water surface above sea level.

T

Table Links:	A table link must be created for every database table or spreadsheet worksheet that is to be linked to the current model. Any number of Table Links may reference the same database file.
TCV:	Throttle control valve.
To Node:	Represents a pipe's ending node. Positive flow rates are in the direction of from towards to. Negative flow rates are in the opposite direction.
To Pipe:	The pipe that connects to the downstream side of a valve or pump.
Total Active Volume:	The volume of water between minimum elevation and maximum elevation of a tank. This is an input value for variable area tanks.
Total Storage Volume:	The holding capacity of a tank. It is the sum of the maximum hydraulically active storage volume and the hydraulically inactive storage volume.
Total Needed Fire Flow:	If you choose to add the fire flow to the baseline demand, the Total Needed Fire Flow is equal to the Needed Fire Flow plus the baseline demand. If you choose not to add the fire flow to the baseline demand, the Total Needed Fire Flow is equal to the Needed Fire Flow.
Trace (Source Ident.):	Determines what percentage of water at any given point originated at a chosen tank, reservoir, or junction.
Trials:	The maximum value for genetic algorithm trials is determined by what you set for Stopping Criteria. Note that you can set a number larger than (Maximum Era Number)*(Era Generation Number)*(Population Size), but calculations beyond that number (for this example, the value is 45,000) are less likely to produce significant improvements in optimization.

V

Valve Status:	A valve can have several different status conditions: Closed (no flow under any condition), Active (throttling, opening, or closing dependent on system pressures and flows), and Inactive (wide open, with no regulation).

Velocity:	The field that displays the calculated value for a pipe, valve, or pump velocity at a given time. It is found by dividing the element's flow rate by its cross-sectional area.
Vertex:	An element in a topological network.

W

.wtg:	File that displays WaterCAD V8*i* information.
wtg.sqlite:	To distinguish between the WaterCAD V8*i* modeling data file and another programs data file. The most important file because it contains all of the modeling data.
Wall Reaction Coefficient:	Defines the rate at which a substance reacts with the wall of a pipe, and is expressed in units of length/time.
Bentley WaterCAD V8*i* Datastore:	The relational database that Bentley WaterCAD V8*i* uses to store model data. Each Bentley WaterCAD V8*i* project uses two main files for data storage, the datastore (.sqlite) and the Bentley WaterCAD V8*i* specific data (.wtg).
WaterCAD V8*i* File Types:	The following lists different types of files that can be used with WaterCAD V8*i*. .bak – backup of most files GEMS Data Store – modeling data Geodatabase – topology (in ArcGIS version) .dwh, .dgn, .dwg – drawing information in stand-alone, Microstation, AutoCAD .out – complete results by scenario .rpc – scenario messages .nrg – energy cost results .pv8 – previous version for files upgraded to new .xml – used for libraries
WaterObjects:	The object model used by Bentley WaterCAD V8*i* , which allows for the extension and customization of the core software functions.
Water Quality:	The field that displays the water quality for the current time period.

Water Quality Analysis:	An analysis that can be one of three types: Age, Trace, or Constituent.

X

.xml:	File used for libraries.

Symbols

.BAK 1135
.MDB 1135

A

about dialog box 9
accelerated redraw 265
accuracy 563
action
    rehabilitation 1111
actions tab 871
Active Topology 880
active topology 696, 880
Active Topology Alternative 696
active topology child alternative 699
add a background layer 219
add a background layer folder 218
add a FlexTable folder 1262
add a help topic 7
add or remove a button 31
Add To Selection Set dialog box 426
Adding and Removing Toolbar Buttons 30
Adding Annotations 1226
adding annotations 1226
adding color coding 1233
Adding Color-Coding 1233
adding design option groups 1108
adding elements 402
Adding Folders 1226
address
    See contacting Bentley Systems. 1448
Addresses 1448
Adjustment groups 1056
Advantages of Automated Scenario Management 669
affinity laws 1405
After One Branch Collapsing 630
After Two Branch Collapsing 631
Age 1527
age
    alternative 712
    analysis 779
Age Alternatives 712
Air valve 367

alarm 311
Allocation strategies 576
alternative 673
Alternative Editor Dialog Box 692
Alternative Editor dialog box 692
Alternative Manager 690, 696
Alternatives 689
alternatives 68, 669, 689
  base 693
  child 693
  creating 694
  editing 694
  hydrology 709
  initial conditions 705
  merge 689
  overview 669, 689
analysis
  constituent 780
  fire flow 774, 775
  hydraulic 750, 751, 752, 1396
  trace 781
  water age 779
  water quality 779, 780, 781
Analysis Menu 1384
Analysis menu 1384
Analysis Toolbar 13
Analysis toolbar 13
analyzing improvement suggestions 681
Animating Profiles 1256
animating profiles 1256
Animation Control Manager 752
Animation Controls 1251
animation options 754
Animation Options Dialog Box 754
Animation Options dialog box 754
Annotating Your Model 1222
annotation 90, 91
annotation properties 1228
Annotation Properties dialog box 1228
annotations 1222, 1228
  adding 1226
  deleting 1227
  editing 1227
  renaming 1227
Application Window Layout 9
Apply Demand and Pattern to Selection Dialog Box 609
apply minor losses 656

applying a zone to a junction 304
applying a zone to a pump 319
applying a zone to a reservoir 313
applying a zone to a tank 309
applying a zone to a valve 346
applying an HGL pattern to a reservoir 313
Applying Elevation Data 561
applying minor losses to a valve 346
applying zone to hydrant 305
ArcSDE 560
assigning demands to a junction 303
Attribute 673
Attribute Inheritance 676
attributes
    editing 413
    scenario 673
AutoCAD 226, 227, 239, 240
    commands 236, 245
    drawing synchronization 243
    entities 236, 245
    integrating with SewerGEMS 240
    undo/redo 247
AutoCAD Mode 226
AutoCAD mode 226, 227, 238, 239, 240
    graphical layout 229
    menus 241
    project files 242
Autodesk 226, 239
automated scenario management 669
automated skeletonization 624
Automated Skeletonization Techniques 627
Available Fire Flow 1527
Average Day Conditions 678

B

backflow preventer 772
background layer 219, 220
background layer files
    using with ProjectWise 284
background layer folder 218, 219
Background Layer manager 216
Background Layers 215
background layers 216
    deleting 220
    dxf files 225

editing 220
image compression 223
shapefiles 224
supported image types 216
backing up your model 664
base alternative 689
Base alternatives 693
base alternatives 693
Base and Child Scenarios 685
base elevation 1528
Base Elevation & Level 1527
Base Scenarios 685
Batch Assign Isolation Valves dialog box 407
batch pipe split 410
batch run 642, 687
Batch Run Editor Dialog Box 688
Batch Run Editor dialog box 688
Batch Runs 687
batch runs 687
Batch Split Pipe dialog box 409
BE Careers Network 1447
BE Magazine 1446
BE Newsletter 1447
Before Branch Collapsing 630
Bend command 406
benefit 1113, 1129
cost versus benefit 1129
design objectives 1114
maximize 1119
Pareto 1127, 1129
total 1122
versus cost 1127
benefit type 1114
Bentley discussion groups 1446
Bentley Institute 1445
Bentley Professional Services 1445
Bentley SELECT 8, 1445
Bentley services 1445
Bentley Systems 1443
addresses 1447
contacting 1447
email addresses 1448
program update 8
Web site 1448
Bentley Water 1375
Bernoulli equation 1397
Billing Meter aggregation 578

Border Editor dialog box 1335
border properties for graphs 1335
Border tool 399
border tool 398
Boundary Node 1527
boundary node 1528
Boundary Overrides 1053
Boundary Overrides tab 1053, 1093
boundary polygon feature classes 605
Branch Collapsing 630
branch collapsing
    See Skelebrator. 627
Branch Trimming 627
branch trimming 627, 630, 650
browse topics 6
buffering point area percentage 604, 605
build number 9
building cost function 1138
bulk reaction
    coefficient 1528
Bulk Reaction Coefficient 1527

C

C coefficient 1410, 1528
CAD 213
Calc. Min. System Pressure 1528
Calc. Min. Zone Pressure 1528
Calc. Residual Pressure 1528
calculating cost 1139
calculation
    unready 1528
Calculation Options 833
Calculation Summary 1364
calculation summary 1364
Calculation Summary Graph Series Options dialog box 1365
Calculation Unready 1528
calculator 331
calibration 758, 1046
Calibration Criteria tab 1058
Calibration export to scenario dialog 1073
calibration manager 1046
Calibration Nodes 566
calibration nodes 566
calibration options 1059
calibration options formulae 1059

Calibration Solutions 1070
Calibration Studies 1048
Calibration Study 1049
C-Coefficient 1528
Change Series Title dialog box 1342
change the position of a background layer 220
changing the drawing view 207
Changing Units, Format, and Precision in FlexTables 1268
characteristic curve
    pump 1405
    pumps 1404, 1405
Chart Options 1297
Chart Options Dialog Box 1297
Chart Options dialog box 1297
    Chart Tab 1298
    Export tab 1332
    Print tab 1334
    Series Tab 1323
    Tools tab 1331
Chart Tools Gallery dialog box 1342
check data 760
check run 756
Check Valve 1528
check valve 389, 1407
check valves 1407
chemical analysis 780
Chezy's Equation 1409
Chezy's equation 1409, 1413
child alternative
    creating active topology 699
Child Scenarios 685
child scenarios 685
Cholesky 1403
clearing element selection 405
Client Server 1447
Closed/Inactive Status 1528
coefficient 1537
    roughness 1537
coefficients
    engineer's reference 1414
Colebrook-White
    equation 1409
    typical values 1415
collapse a subtopic 6
collapsing branch
    See Skelebrator. 627
collections

minor loss 296
color coding 79, 92, 1231
adding 1233
deleting 1234
editing 1234
renaming 1235
color coding legend 1235
Color Coding Your Model 1231
Color dialog box 1337
Color Editor dialog box 1337
Color-Coding Properties dialog box 1236
column headings
editing for FlexTables 1268
Combination Pump Curves 337
commands (AutoCAD mode) 236, 245
comparing cost results 1171
Components Menu 1386
Components menu 1386
Composite Action 874
Composite Condition 870
Composite Logical Action 872
Compress Database command 1392
compressing large database files 1392
Compute Toolbar 16
concentration 780
Concentration (Base) 715
Concentration (Initial) 715
Conditions List 872
Conditions tab 864
conditions tab 864
conjugate gradient method 1403
connection
synchronization 243, 244
Connections manager 511
connectivity
explicit 534
implicit 534
conservation
of mass & energy 1398
consider pressure benefit 1100
Constant Area Approximation 375
constant horsepower pump 1406
constant power pump 1406
Constituent 1528
constituent 1528
alternative 715
analysis 780

Constituent Source Type 716
Constituents manager 719
constructing a query 472, 1272
contacting Bentley Systems
  email 1448
  fax 1448
  hours 1448
  mail 1448
  technical support 1448
  telephone 1448
Context Menu 1528
context menu 1528
contour 1238, 1239, 1240
  smoothing 1239, 1240
Contour Browser 1237, 1241
contour labels 247
Contour Manager 1236
contour maps 563
Contour Plot 1240
Contours 1236
control
  status 1528
  valve 1407
Control Manager 859
Control Sets tab 875
Control Status 1528
Controlling Toolbars 30
controls tab 860
Conveyanc Element 1528
Coordinates 1528
copy FlexTable data 1279
copy graph data 1287
copying
  FlexTables 1279
Copying, Exporting, and Printing FlexTable Data 1278
Correct Data Format 536
correcting an error 680
Correlation Graph Dialog Box 1072
Correlation Graph dialog box 1048
cost
  design 1122
  rehabilitation 1122
  total 1122
Costs/Properties tab 1107
create a FlexTable report 1279
create a new Alternative 694
create a new FlexTable 1265

create a new profile 1251
create a new scenario 686
create a new System Head Curve 769
create a new Totalizing Flow Meter 765
create an active topology alternative 699
create Observed Data 1294
Create Selection Set dialog box 424
creating
    graph 1285
Creating a New FlexTable 1265
Creating a Project Inventory Report 1282
creating a query 471
Creating a Scenario Summary Report 1282
Creating Alternatives 694
creating alternatives 694
Creating an Active Topology Child Alternative 699
creating dynamic 424
creating queries 472, 1272
creating reports 1281
Creating Scenarios 685
creating selection sets 424
criticality analysis 819
cross section of a variable area tank 310
Cross Section Type 1529
Crosshair 1529
Current Storage Volume 1529
curve
    pump 1404, 1405, 1406
curved pipes 406
custom AutoCAD entities 236, 245
custom extended
    pump 1406
custom results path 4
custom sort 1273
Customization Editor 490
customize
    drawing 242
customize a graph 1355
customizing
    FlexTables 1274
Customizing a Graph 1355
customizing graphs 1355
Customizing Managers 34
Customizing the Toolbars 30
customizing toolbars and buttons 30
Customizing WaterGEMS Toolbars and Buttons 30
Customizing Your FlexTable 1274

cut probability 1066
CV 1529

D

Darcy Weisbach
    Colebrook-White equation 1409
    equation 1411
    roughness values 1415
Darcy-Weisbach equation 1411, 1426
Darwin 1046
Darwin calibration 1064
Darwin Calibrator dialog box 1046
Darwin Calibrator troubleshooting tips 1079
Darwin Designer 1084
Darwin manager 1046
Darwin Scheduler 1173, 1178
dashed line 408
data
    check 760
    entry 43
    organization 689
    validation 760
data check 756
Data Format Needs Editing 536
Data Scrubbing 627
data scrubbing 627, 629
data source tables 536
data types for user data extensions 483
Database Connections 1529
Database Utilities 1392
Dataset 1529
DBMS 1529
DDF 569
DE Geodatabase 534
dead-end pipes 627
decimal point 417
default units 273
default workspace 34
defining pump settings 320
defining user data extensions 478
delete a background layer 220
delete a background layer folder 219
delete a FlexTable folder 1262
deleting
    FlexTables 1266

Deleting Annotations 1227
deleting annotations 1227
Deleting Background Layers 220
deleting background layers 220
deleting color coding 1234
deleting elements 405
Deleting FlexTables 1266
Deleting Folders 1226
deleting groups of elements in a selection set 426
Deleting Profiles 1255
deleting profiles 1255
DEM 565, 569, 1529
Demand 1529
demand
    multipliers 857
Demand Adjustments 1054
Demand Adjustments tab 1055, 1096
demand allocation 575
Demand Alternatives 704
Demand Collection dialog box 304
Demand Control Center 606
demand deficit 1435
Demand Groups 1056
Demand inversing 968
demand multiplier 1093
demand projection 581
Demand tab 1056
design costs 1107
design event editor 1089
design events 1117
Design Events tab 1089
design group
    adding 1104
    editing 1106
design groups 1124
Design Groups tab 1102, 1117
Design Point 1529
design point 1406
design run 1116
    computing 1121
design study 1085
design type tab 1113
designer data verification summary 1136
Diameter 1529
Digital Elevation Models 566
digital elevation models (DEMs) 563
    level one 565

level three 565
level two 565
type A 565
type B 565
type C 565
digital ortho-rectified photogrammetry 563
dimensionless benefit 1114
direct GGA solution 1437
Discharge 1529
discharge 772
discharge coefficient 345
discharge to atmosphere 390
display a topic 7
display format 417
Display Precision 417
display precision 416
display topics 6
displaying multiple projects 261
Distributed Scenarios 670, 671
DLG 1529
docked dynamic manager 35
docked static manager 35
dominant pipe criteria 653, 655
Double Acting 369
Double Click 1529
Drag 1530
drag 1530
drawing
setup (AutoCAD mode) 242
synchronization (AutoCAD mode) 243
drawing scale 271
drawing style 213
duplicate labels 433
DWG 243
DXF 569
DXF Properties 225
DXF Properties dialog box 225, 424, 426
Dynamic Inheritance 675
dynamic inheritance 675

E

edit a FlexTable 1267
edit a profile 1254
edit a scenario 687
Edit Hyperlink dialog box 462

Edit Menu 1384
Edit menu 1384
edit the properties of a background layer 220
Edit Toolbar 12
Edit toolbar 12
editable 727
editing
    FlexTables 1267
    numerous elements at once 1269
Editing Alternatives 694
editing alternatives 694
editing annotations 1227
editing attributes 1452
editing color coding 1234
editing column headings
    FlexTables 1268
Editing Column-Heading Text 1268
editing design options groups 1108
editing element attributes 413
Editing FlexTables 1267
Editing Scenarios 686
editing scenarios 686
editing units
    FlexTables 1268
EGL 1398
Element 1530
element
    deleting 235
    modify 235
    moving 236, 246
element attributes 1452
Element Attributes Reference 1451
element label project files 276
element labeling settings 276
element relabeling 1275
Element Symbology Manager 1222
    using folders in 1225
Element Symbology manager 1222
element symbols 213
elements 294
    adding in the middle of a pipe 405
    adding to your model 402
    clearing selection of 405
    deleting 403
    editing attributes 413
    globally editing data in numerous elements 1269
    moving 403

overview 294
reporting on 1285
selecting 403
selecting all 404
selecting all of the same type 404
selecting by polygon 404
validation 756
viewing in selection sets 423
elevated tanks 867
Elevation 1530
elevation 1528, 1534
base 1528
calibration nodes 566
determining pressure 561
maximum 1534
obtaining data 563
value 562
Elevation Data 561
elevation data 561
elevation data source 569
email 1448
email address 1448
energy
conservation 1398, 1399
equation 1397
grade line 1398, 1530
principle 1396
Energy Cost Alternative 729
energy cost alternative 732
Energy Cost Analysis Calculations 1166
Energy Cost Results 1166
Energy Costs 1151
energy costs 1151
energy equation 1397
Energy Grade Line (EGL) 1530
Engine Compatibility 850
engineering libraries 453, 455
overview 452
sharing on a network 455
working with 453
engineering libraries dialog box 455
Enhanced Pressure Contours 1242
enhanced pressure contours 1242
entering data 413
entities
in AutoCAD 236, 245
enumerated user data extensions 486

Enumeration Editor dialog box 486
EPS 751, 1530
  analysis 751, 752
equally distributed 631, 655
equivalent pipe method 653, 655
era generations number 1065
error messages 530, 760
errors 761
ESRI ArcGIS Geodatabase functionality 532
estimate 1531, 1534
exclamation point in circle 118
existing loads 631
existing projects 260
exit WaterGEMS 4
expand a subtopic 5
explicit connectivity 534
explode elements (AutoCAD mode) 245
export 1375
export FlexTable data 1279
export to scenario 1132
Export to Scenario dialog box 1048
exporting
  FlexTables 1279
exporting a DXF file 1377
exporting FlexTables 1278
Extended Edit Button 1530
extended edit button 1531
Extended Period Analysis 858
extended period analysis 751
  lesson 2 60
External Files 1530
external files 1531
External Tools 883
Extrapolate 1530
extrapolate 1531

F

fax 1448
FCV 352
Feature Class 1530
Feature Dataset 1531
field
  links 1531
Field Data Snapshots tab 1050
Field Links 1531

File Extension 1531
file format update 1135
File Menu 1381
File menu 1381
File Upgrade Wizard 1378
filter
    resetting 1272
filter a FlexTable 1271
Filter dialog box 728
filtering a FlexTable 1271
finalizing the project 681
Find 414
Find Logical Action dialog box 872
finding elements 414
fire flow
    alternative 723, 724, 727
    analysis 774, 775
    results 775
    theory 774
fire flow checks 777
Fire Flow Results Browser 776
Fire Flow System Data 727
Fire Flow Upper Limit 1531
fire flow upper limit 1534
fire hydrants 1009
fire hydrants as flow emitters 1012
fitness 1122
fitness tolerance 1065
fitness type 1059
fitting loss coefficients 1413, 1418
Fixed Point 417
FlexTable Dialog Box 1263
FlexTable dialog box 1263
FlexTable Setup Dialog Box 1276
FlexTable Setup dialog box 1276
FlexTables 1257
    copying 1278
    copying data 1279
    creating 1265
    customizing 1274
    deleting 1266
    editing 1267
    editing column headings 1268
    editing globally 1269
    editing units 1268
    exporting 1278
    exporting data 1279

filtering 1271
global editing 1269
navigating in 1268
opening 1265
ordering columns 1270
printing 1278, 1279
renaming 1266
reports 1279
saving as text 1279
shortcut keys 1268
sorting column order 1270
FlexTables Manager 1258
folders in 1262
FlexTables manager 1258
floating manager 34
Flow 1531
flow 1534
flow arrows 226
flow constraints 1100, 1125
flow control valve 1407
flow control valves 1408
flow distribution 579
flow emitters 772, 1012
flow per fitness point 1059
Flow Tolerance 844
flushing 989
folders
in Element Symbology Manager 1225
in FlexTables Manager 1262
format
unit 416
formulas 1414
Francis 358
Free Form 1229
friction and minor loss methods 1409
From Node 1531
from node 1534
From Pipe 1531
from pipe 1534

G

GA 1046, 1531
Gas Law Model 375
Gaussian elimination method 1404
GEMS Datastore 1531

General 417
general purpose valves 1408
general settings 264
Generations 1532
genetic algorithms 1046, 1439, 1441
  calibration tips 1077
  optimized calibration 1061
  optimized calibration advanced options 1065
Geodatabase 1532
Geodatabase feature 532
geodatabase support 532
Geometric data source 508
Geometric Networks 533
Getting Started in Bentley WaterGEMS 1
GIS
  demand allocation 575
GIS style 213
GIS-ID 537, 538
global edit 1270
global edit FlexTable column 1269
global editing
  FlexTables 1269
global settings 263
Global tab 264
globally editing data 1269
GO button 757
Google Earth 248
GPV 353
grade line
  energy 1398
  hydraulic 1398
gradient algorithm 1400
  derivation 1400
Gradient Editor dialog box 1336
graph
  copying and pasting data 799, 1291
  data 799, 1291
  new 1285
Graph Dialog Box 1287
Graph dialog box 1127, 1288
graph dialog box
  Darwin Designer 1127
Graph Manager 1285
Graph Series Options dialog box 1293
graphical layout
  AutoCAD 229
graphing 1285

changing total time period 1286
Graphs 1285
graphs 1285
customizing 1355
printing 1287
grid 569
groundwater well 1006

H

Haestad Methods
program update 8
Haestad.log 1448
HAMMER elements 401
Hatch Brush Editor dialog box 1338
Hazen-Williams
typical values 1416
Hazen-Williams equation 1410, 1424
coefficients 1417
roughness values 1416
Hazen-Williams Formula 1410
head 772
head loss 353
head per fitness point 1059
Headloss 1532
headloss 1534
headloss curves for GPVs 347
Headloss Gradient 1532
headloss gradient 1534
Help 19
help files and books 1444
Help Menu 1393
Help menu 1393
Help Toolbar 19
HGL 1398, 1534
HGL setting 1534
high alarm 310
history of what-if analyses 670
Hydrant Flow Curve editor 306
Hydrant Flow Curve manager 305
hydrant flow curves 305
hydrants 305, 1009
hydrants as flow emitters 1012
hydraulic analysis 751
hydraulic equivalency 632
Hydraulic Equivalency Theory 1423

Hydraulic Grade 1532
hydraulic grade 1534
hydraulic grade line 1398
Hydraulic Grade Setting 1532
hydraulic grade setting 1534
hydraulically close tanks 1009
hydrology alternatives 709
hydropneumatic tank 375, 1006
Hydropower Plants 360
hyperlinks 459

I

identifying elements for costing 1139
image compression 223
Image Filter 222
Image Properties Dialog Box 222
Image Properties dialog box 222
i-models 492
impeller 1405
implicit connectivity 534
import 540, 545, 549, 1374
import Bentley Water Model 1375
import database 1373
Import dialog box 487
import observed target 1075
import snapshots 1074
importing and exporting Epanet files 1374
importing field data 1074
importing/exporting skelebrator settings 666
impulse turbine 357
In 1397
Inactive elements 880
inactive pipes 1136
Inactive Volume 1532
inactive volume 1534
individual elements
    adding to your model 402
inflow 1534
Inflow & Outflow 1532
Inheritance 674, 1533
inheritance 674, 676, 1534
    dynamic 675
    overriding 675
initial conditions alternative 705
initial conditions of networks 1286

initial flow equals zero 1286
Initial Settings 1533
initial settings 1534
    alternative 705
Initial Water Quality 1533
initial water quality 1534
Initialize From Selection set dialog 1053
Initialize Table from Selection Set dialog box 1115
inline isolation valve replacement 634
installation 2
instant load rejection 362
integrating AutoCAD with SewerGEMS 240
intermediate node removal 628
Interpolate 1533
interpolate 1534
Invert 1533
invert 1534
Is Constituent Source? 716
Is offset to the left of referenced link? 346
isolation valve 408

J

junction conditions and tolerances 663
junctions 303

K

K coefficients 1418
Kaplan 358
KnowledgeBase 8

L

Label 1533
label 1534
labeling elements 416
LandXML 569
lateral loss 308
laws
    affinity 1405
    conservation of mass and energy 1398
layout 46, 47, 48
    AutoCAD 229

layout settings 266
layout tool 402
Layout Toolbar 20
Layout toolbar 20
legend 1235
Length 1533
length 1534
level 1528
Levenberg-Marquardt method 1407
library types 453
license 2
LIDAR 564, 1533
light 1534
    messages 1534
Line tool 400
line tool 398
linear system equation solver 1403
linear theory method 1400
load acceptance 362
load distribution strategy 650, 655
Load from Model dialog box 1115
Load rejection 360
LoadBuilder 582
    manager 582
    run summary 598
    wizard 583
Local and Inherited Values 676
local and inherited values 676
logical control 863
    dialog box 861
    manager 859
    set editor 878
logical control:
    See operational controls alternative.
Logical controls 862
logical controls
    overview 859
loop retaining sensitivity 659
loop-based algorithms 1400
losses
    friction 1402, 1411
    minor 1404, 1409, 1413, 1414
low alarm 311

M

mail 1448
Management controls 855
Manning's Coefficient 1533
Manning's coefficient 1534
Manning's equation 1412, 1425
  roughness values 1414
  typical values 1417
manual cost estimating 1137
Manual Design Run 1120
Manual Scenarios 672
manual selection 1120
manual skeletonization 636, 647
mass conservation 1398
Mass Rate (Base) 715
material 1534
maximize benefit 1114
maximum
  era number 1065
  extended operating point 1534
  increment 1061
  number of removal levels 653
  number of trimming levels 650
  operating point 1534
  trials 1065
Maximum Day Conditions 679
maximum trials 1119
menu
  context 1528
Menus 1381
merge
  merge
    alternatives 689
merging pipes by 656
merging pipes of the same diameter 656
messages 1534
  light 1534
meter aggregation 578
meter assignment 576
Microstation Mode 226
minimize cost 1114
minimum
  increment 1061
  system junction 1534
  system pressure 1528

zone pressure 1528
minor loss 352
Minor Loss Coefficients dialog box 299
minor loss collection 296
Minor Loss Collection dialog box 297
minor loss strategy 653
minor losses 1404, 1409, 1413, 1428
fitting 1418
mixed flow turbine 358
model 750
model and optimize distribution system 750
Model Spot Elevation 569
ModelBuilder 540, 545, 549
errors and warnings 530
supported formats 507
using 507
ModelBuilder Connections manager 511
ModelBuilder wizard 514
modeler definition 1535
modeling fire hydrants as flow emitters 1012
modeling pressure dependent demand 1432
modeling tips 1005, 1014
modeling variable speed pumps 1014
modified GGA solution 1437
moment of inertia 364
motor
pump 1421
motor and pump inertia 331
move
elements 236, 246
labels 237, 246
move a toolbar 31
moving elements 405
moving toolbars 31
multi-objective trade-off 1114
multiple 773, 1017
pump curve 1406, 1407
multiple elements
selecting 404
multiple point pump 1406
multiple projects
maximum number of 260
Multipliers 857
Municipal License Administrator 2
mutation probability 1065

N

naive method 1430
named views 418
Naming and Renaming FlexTables 1266
navigating in a FlexTables 1268
Navigating in Tables 1268
network connectivity 534
network hydraulics theory 1395
Network Navigator 427
network navigator 410
network review 410
network walking algorithm 636
New Logical Action dialog box 872
nodal demand vector 1401
node 1528, 1534
    boundary 1528
    from 1534
non-convergence 751
non-improvement generations 1065, 1119
Notes tab 1115, 1120
Number 417
number
    Reynolds 1537
numerical check 1429
Numerical Value of Elevation 562

O

Observed Data 1294
Observed Target 1052
Observed Target tab 1052
Obtaining Elevation Data 563
Obtaining elevation data 563
open a manager 34
open Darwin Designer 1084
open FlexTables 1265
open Help 5
open the registration dialog box 9
Opening FlexTables 1265
Opening Managers 34
opening managers 34
operation 1270
Operational Alternative 859
operational alternative 709

operational controls alternative 709
optimized calibration 1064, 1065
options 263
  calculation 833
  design run 1118
Options Dialog Box
  ProjectWise settings 277
Options dialog box 264, 269
options groups tab 1107
Oracle 559, 560
ordering
  FlexTable columns 1270
organize data 689
orphaning of pipes 629
Outage Segments 822
outflow 1534
output
  tables 1257
output data 842
override scenario demand alternative 1093
Overriding Inheritance 675
overriding inheritance 675
overview 1046

P

Pan tool 207
panning 207
  using a mousewheel to 208
parallel 773, 1017
Parallel Pipe Merging 633
parallel pipes 1007
  modeling 1007
  removal 633, 652
parallel pumps 1008
parent scenario 685
Pareto optimal defined 1129
pareto plot 1127
pattern 853, 854
  demand multipliers 854
  extended period analysis 752, 858
  pattern editor 854
  time steps 854
Pattern (Constituent) 716
Pattern Manager 855
patterns 549

PBV 353
peak demands 1171
Peak Hour Conditions 680
Pelton 357
Periodic Head-Flow 366
physical alternative 700, 701
physical properties 700
Pipe 1452
pipe 1534
  diameter 656
  from 1534
  length 1534
  material 1534
  merging 628
  merging same diameters 656
  parallel 1007
Pipe Attributes 1452
Pipe Break Analysis 1030
pipe conditions and tolerances 662
pipe inventory 1282
pipe material 295
pipe option group
  adding 1112
Pipe Renewal Planner 1020
pipe size usage plot 1127
Pipe-by-pipe 827
pipes 295
  modeling with curves 406
  splitting 405
plane sweep 1431
point demand assignment 581
Pointer dialog box 1341
polygons
  used to select elements 404
Polyline Vertices dialog box 407
PondPack
  build number 9
  installation 2
  upgrade 8
  upgrades and updates 2
  version number 9
population size 1066
Post Calculation Processor 770
predefined queries 467
Presenting Your Results 1221
preserve network integrity 659
pressure

head 1397, 1398
pressure breaker valve 1407
pressure breaker valves 1408
pressure constraints 1098, 1124
pressure dependent demand 1434
Pressure Dependent Demands 616
pressure engine 401
pressure pipes
adding a minor loss collection to 296
typical values 1417
pressure reducing valves 1408
pressure sustaining valve 1407
pressure sustaining valves 1408
Pressure Threshold 621
pressure vessel 375
pressure zone export 444
Pressure Zone Manager 434
pressurized tank 1006
principles 1424
Print Preparation 1368
print preview
FlexTables 1279
Print Preview Window 1366
printing
FlexTables 1279
Printing a Graph 1287
printing FlexTables 1278
printing graphs 1287
proejct queries 467
profile
editing 1254
profile setup 1244
Profile Viewer 1249
Profile Viewer dialog box 1255
profiles 1242
animating 1256
creating 1251
deleting 1255
renaming 1255
viewing 1255
Profiles manager 1242
Profiles Series Options dialog box 1248
Program Maintenance Dialog Box 8
project
files 231, 242, 243
project inventory 1282
Project Properties dialog box 262

Project tab 269
projection 581
projects 260
ProjectWise 278
    closing projects 279
    general guidelines for using 279
    using background layer files with 284
    viewing status 281
ProjectWise options 277
properties
    editing 413
Property Editor 413, 1452
    using Find Element 414
Property Grid Customizations 489
proportional to coalesced pipe attributes 631
proportional to dominant criteria 655
proportional to existing load 656
protected elements manager 644
prototypes 446
pump 1008
    affinity laws 1404
    constant horsepower 1405, 1406
    curve 1404, 1405, 1406, 1407
    custom extended 1406
    groundwater well 1006
    impeller 1405
    motor 1421
    multiple point 1406
    operating point 1404, 1405, 1406
    parallel 1008
    series 1008
    static head 1405
    static lift 1404
    theory 1404
    three point 1406, 1421
    type 1406
    variable speed 1405
Pump Curve Definitions dialog box 320
Pump Curve dialog box 329, 330
pump curves 545
pump definitions 540
pump results 1169
pump settings 320
pump station 341
pump types 329, 330
Pump Usage summary 1167
pumps 319, 773, 1017

1404
defining settings for 320

Q

queries 467, 472, 1272
creating 471
in FlexTables 1271
predefined 467
project 467
shared 467
Queries Manager 467
Query Builder dialog box 473
Query Parameters 470

R

random seed 1065
ranking
FlexTable columns 1270
Rasters 569
reaction turbine 358
read-only 727
reconnect 406
Record Types 565
red circle 118
redo 246, 247
reference
engineer's 1414
Reference Pressure 621
References 1438
rehab groups 1124
Rehab Groups tab 1102, 1118
rehabilitation action 1111
rehabilitation cost
adding 1112
editing 1112
rehabilitation function manager
Darwin Designer 1113
rehabilitation group
adding 1104
editing 1106
rehabilitation option group
defining 1112
relabeling elements 416

relative speed factor 1537
remove orphaned nodes 659
removing elements from selection sets 426
rename a background layer 220
rename a background layer folder 219
rename a FlexTable folder 1262
rename FlexTables 1266
renaming
    FlexTables 1266
renaming annotations 1227
Renaming Folders 1226
Report Menu 1393
Report menu 1393
report options 1282
Report Viewer 1048
report viewer 1125
Reporting 1281
reporting
    on a group of elements in a selection set 426
Reporting Time Step 842
reports 79, 80, 83, 1281
    creating for elements 1285
    FlexTables 1279
    scenario 1282
    standard 1282
Representative Scenario 1051
reserviors 313
reset
    FlexTable filter 1272
reset a filter 1272
Reset Workspace 34
residual pressure 1537
results
    Darwin Designer 1121
    getting results from Darwin Designer 1121
Results Table 1366
Reynolds number 1537
roughness
    Chezy’s equation 1409
    coefficient 1414
    Colebrook-White equation 1409
    Darcy-Weisbach equation 1411
    Hazen-Williams equation 1410
    Manning’s equation 1412
Roughness Groups 1056
roughness height 1410, 1412, 1415
Roughness tab 1056

roughness values 1414
Colebrook-White 1415
Darcy-Weisbach 1415
Hazen-Williams 1416
Manning's 1414
typical 1417
rounding of numbers 417
rule based 859
Running Criticality Analysis 823
Running Multiple Scenarios at Once 687
running the model 757
rupture disk 390

S

SAV 388
SAV Closure Trigger 388
save
as drawing *.DWG 244
saving FlexTables as text 1279
Scenario 673
scenario
alternatives 68, 70, 73, 74, 75, 76, 77
child 68, 70, 72, 74, 75, 76
lesson 3 68
Scenario Attributes and Alternatives 673
scenario example 678
Scenario Inheritance 677
Scenario Management
Example 678
scenario management 68
Scenario Manager 683, 688
scenario summary 1282
Scenarios 682
scenarios 669, 1091
advantages of using 669
attribute inheritance 676
attributes 673
base 685
batch run 687
creating new 686
editing 686
inheritance 674
local and inherited values in 676
overview 669, 672, 682
Scenarios Toolbar 15

Scenarios toolbar 15
schema
    Darwin Designer 1135
    format 1135
Schema Augmentation 1135
schema definition 1538
Scientific 417
scrubbing
    See Skelebrator. 627
SDTS 564, 569
search for text 7
segmentation 827
select boundary polygon feature class 604
Select dialog box 1053
select the point 604
selecting all elements 404
selecting an element 403
selecting elements
    all of the same type 404
    by polygon 404
selecting multiple elements 404
Selection Set Element Removal dialog box 426
selection sets 420, 421, 424, 426
    adding a group of elements to 426
    adding elements to 425
    creating 424
    creating from queries 424
    group-level operations 426
    in FlexTables 1264, 1265
    removing elements from 426
    viewing elements in 423
Selection Sets Manager 421
Selection tool 21
Self-Contained Scenarios 671
Self-Contained scenarios 671
Series Pipe Merging 631
series pipe merging
    See Skelebrator. 629
Series Pipe Removal 628
series pipe removal 628, 631, 654
series pumps 1008
set field options 1135
Set Field Options dialog box 416
setting options 263
setup 242
Shapefile Properties 224
Shapefile Properties dialog box 224

Shared Field Specification dialog box 485
shared queries 467
sharing engineering libraries on a network 455
shortcut keys
    FlexTables 1268
Show Flow Arrows 226
SHP 569
SI 417
Simple Logical Action 872
simultaneous path adjustment method 1400
Skelebrator 629
    batch run 642
    branch trimming 630, 650
    conditions and tolerances 661
    data scrubbing 629
    parallel pipes removal 633, 652
    protected elements manager 644
    series pipe removal 631, 654
    skeletonization manager 638
    skeletonization preview 635
    troubleshooting 665
    using 637
    what it does 636
Skelebrator features 635
Skelebrator Progress Summary dialog box 664
Skelebrator-specific selection sets 644
skeletonization 624
    branch trimming 627
    data scrubbing 627
    example 625
    manager 638
    network walking algorithm 636
    series pipe removal 628
    Skelebrator 629
    techniques 627
    See also Skelebrator.
skeletonization and active topology 667
skeletonization and scenarios 665
Skeletonization Using Skelebrator, Skelebrator, Using Skelebrator 629
Slow Closing 368
Smart Pipe Removal 629, 659
smoothing contours 1239
snap menu (AutoCAD mode) 237, 246
Snapshot Data 1051
Software 1444
software
    upgrades 8

Software Updates via the Web and Bentley SELECT 8
solution methodology 1436
solutions 1071
solutions to keep 1119
solutions to modeling problems 1005
sort columns in FlexTable 1270
sort contents of FlexTable 1270
sorting
    FlexTable columns 1270
Sorting and Filtering FlexTable Data 1270
source
    tracing 781
sparse matrix 1400, 1403, 1404
spatial data 534
spatial reference 569
Spatial Reference System 286
specific speed 365
speed 773, 1017
splice probability 1066
split 405
splitting pipes 405
spot elevations 354
SRS 286
stand-alone definition 1538
Stand-Alone Editor 207
standard extended pump 1406
standard reports 1282
Standard toolbar 10
start WaterGEMS 2
Starting Bentley WaterGEMS 2
starting Bentley WaterGEMS 2
starting projects 260
static head
    pump 1405
static lift
    pump 1404
station 773, 1017
statistics 1280
Status Elements 1057
Status Elements tab 1057
statuses
    initial settings 1534
steady state analysis 751
steady-state analyses 751
Stieltjes 1403
stopping criteria 1119
storage 1170

storage chambers 501
storage volume 1534
  active 1539
  inactive 1534
Stored Prompt Responses dialog box 268
submodel 1374, 1375
supply level evaluation 1434
support 1448
  addresses 1448
  hours 1448
surge tank 393
surge-anticipator valve 388
Swamee and Jain equation 1412
SWG file 242
symbol
  visibility (AutoCAD mode) 242
synchronize (AutoCAD mode) 243
System Head Curve editor 769
System Head Curves 767, 768, 769
System Head Curves manager 768
system operating point 1404

T

Table
  Properties 1276
  Type 1276
table
  setup 1276
tables
  column headings 1268
  editing FlexTables 1267
  units 1268
tabular report 1257
tank
  hydraulically close 1009
  hydropneumatic 1006
  pressurized 1006
tanks 308
TCV 353
technical support 1446, 1448
TeeChart Gallery dialog box 1354
text 237, 246
Text tool 399
text tool 398
the energy principle 1396

The Importance of Accurate Elevation Data 561
The Scenario Cycle 672
theme folders
    renaming 1226
theme groups
    deleting 1226
theory 1419
    network hydraulics 1396
    valve 1408
Thiessen polygon generation 600
Thiessen Polygon Generation Theory 1430
three point pump 1406, 1421
Threshold Pressure (SAV) 388
throttle control valve 1407
throttle control valves 1408
Time Details summary 1167
Time for SAV to Close 388
Time for SAV to Open 388
time of simulation 1286
Time SAV Stays Fully Open 388
Time Series Field Data 1360
time step 843
TIN 569
toolbars 9
Tools Menu 1390
Tools menu 1390
Tools Toolbar 24
Tools toolbar 24
Tooltip customization 491
top feed/bottom gravity discharge tank 1011
top solutions 1119
topology 760, 1400
total active volume 1539
total benefit 1123
total cost 1122
Totalizing Flow Meter Editor 766
Totalizing Flow Meter editor 766
Totalizing Flow Meter manager 765
Totalizing Flow Meter Manager Dialog 765
trace
    alternative 720
trace alternative 720
trace analysis 781
transition pressure 369
TRex Terrain Extractor 566
TRex terrain extractor 566
TRex Wizard 568

TRex wizard 568
trimming
    See Skelebrator. 627
Triple Acting 369
Troubleshooting 8
troubleshooting 761
    Darwin Designer 1136
    knowledge database 8
turbine 355, 364
    inertia 364
turbine element reference 364
turn toolbars off 31
turn toolbars on 31
turning toolbars off 31
turning toolbars on 30

U

U.S. customary 417
unbalanced network 1020
Understanding Scenarios and Alternatives 669
Understanding shortfalls 823
Unit 417
unit carbon emissions 1165
Unit Demand Collection dialog box 304
Unit Demand Control Center 614
Unit Line Flow Method 598
unit of measurement 417
unitized benefit 1114
units 273
    editing for FlexTables 1268
units and formatting 416
update file format 1135
updates 2
updating PondPack via the Web 8
upgrade
    PondPack 8
upgrades 2
upstream node demand proportion 656
use 50/50 split 653
use cases 1433
use equivalent pipes 653, 655
use ignore minor losses 653
use skip pipe if minor loss > max 653
use the Graph Manager 1285
use the index 6

User Data Extensions 741
user data extensions 478
    data types 483
    enumerated 486
User Data Extensions dialog box 480
User Notification Details dialog box 764
User Notifications 761
user notifications 761, 763
User Notifications Manager 761, 763
user-defined ratio 631, 656
USGS 569
USGS DEM 565
USGS topological maps 563
Using Folders in the Element Symbology Manager 1225
Using Profiles 1242
using Skelebrator 637
Using Standard Reports 1282
Using the Totalizing Flow Meter 764
using with SewerGEMS 278

V

Vacuum Breaker 370
validation 756, 760, 761
valve 352, 1528
    check 1528
    theory 1407
valve characteristic 351
valve characteristics 349
valve types 344
valve with linear area change 393
valves 827
Variable 773, 1017
variable elevation curve 387
variable frequency drive 1014, 1419
variable speed pump 1419
    curve equations 1405
    theory 1419
Variable Speed Pump Battery 341
variable speed pump theory 1419
variable speed pumps 1405
vector 569
velocity
    head 1398
verification report 1136
verification summary 1136

version number 9
VFD 1014, 1419
view
  tabular 1257
View Menu 1387
View menu 1387
View Toolbar 18
Viewing and Editing Data in FlexTables 1257
viewing elements in a selection set 423
Viewing Profiles 1255
viewing profiles 1255
visibility of symbols 242
VLA 353
volume 1534
  inactive 1534
  total active 1539
VSP 773, 1015, 1016, 1017, 1420, 1421, 1422, 1423
VSPs 773, 1017

W

warning
  Darwin Designer 118
warning messages 530
warnings 761
water main 1009
water quality
  analysis options 778
Water Quality Analysis 778
Water Quality Batch Run 790
WaterCAD
  custom AutoCAD entities 236, 245
WaterCAD in AutoCAD 226, 239
WaterCAD Managers 34
WaterObjects 36
wave speed 301
WCD file 231
Web updates 8
Website 1448
Welcome dialog 259
Welcome dialog box 259
well 1006
  groundwater 1006
well groundwater 1007
What-If 670
white 727

table columns 1267
window color settings 265
Working with FlexTable Folders 1262
Working with Graph Data
Viewing and Copying 1287
Working with WTG Files 2
World Wide Web
See Web. 8

Y

yellow 727
table cells 1267

Z

zero flow at time 0 1286
zones 295
Zones manager 450
Zoom 210
Zoom Center dialog box 209
Zoom Dependent Visibility 211
Zoom Extents 208
Zoom Factor 210
Zoom In 209
Zoom Out 209
Zoom Previous
Zoom Next 210
Zoom Realtime 209
Zoom Toolbar 27
Zoom Window 209
zooming 207