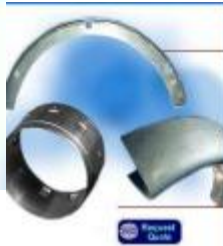


Sheet-metal forming

- Sheet metal forming is a process that materials undergo permanent deformation by cold forming to produce a variety of complex **three dimensional shapes**.
- The process is carried out in the **plane of sheet by tensile forces** with high ratio of surface area to thickness.
- **High rate of production** and formability is determined **by its mechanical properties**



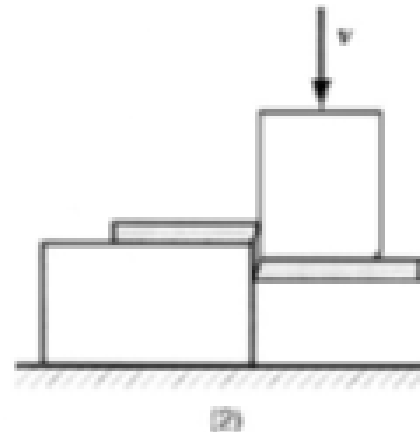
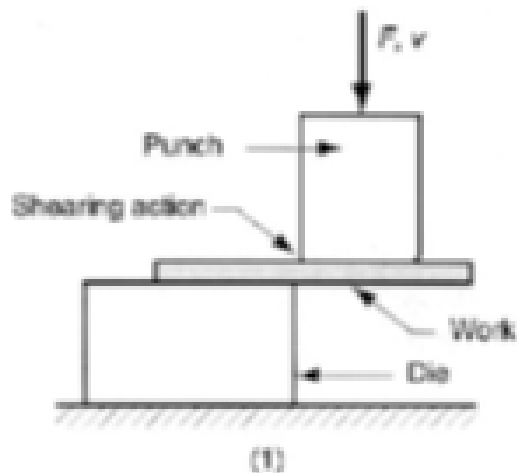
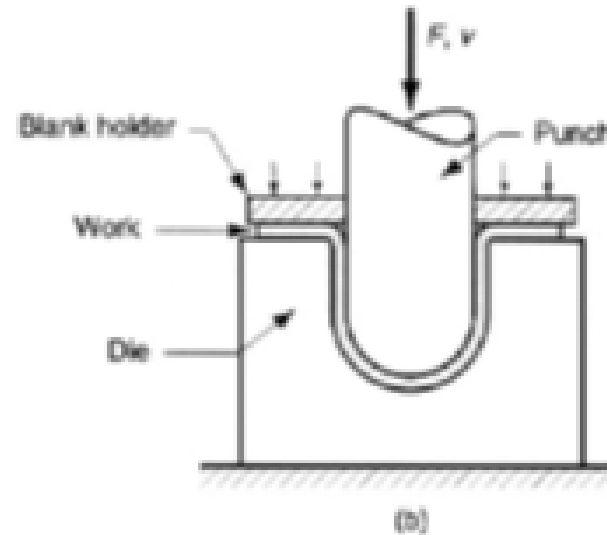
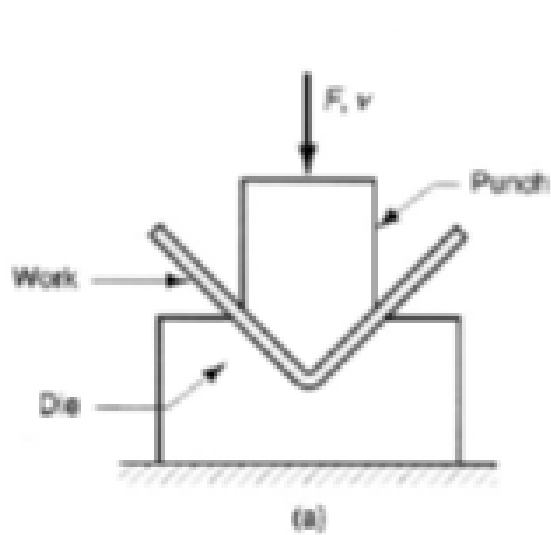
- **Cutting and forming** operations performed on relatively **thin sheets** of metal
- **Thickness** of sheet metal = **0.4 mm** (1/64 in) **to 6 mm** (1/4 in).



Three Major Categories of Sheet Metal Processes.

1. **Cutting** :**Shearing** to separate large sheets; or cut part perimeters or make holes in sheets
2. **Bending** :Straining sheet around a straight axis.
3. **Drawing** :Forming of sheet into **convex or concave** shapes.

Classification of sheet metal deformation



a) Bending

b) drawing

Shearing

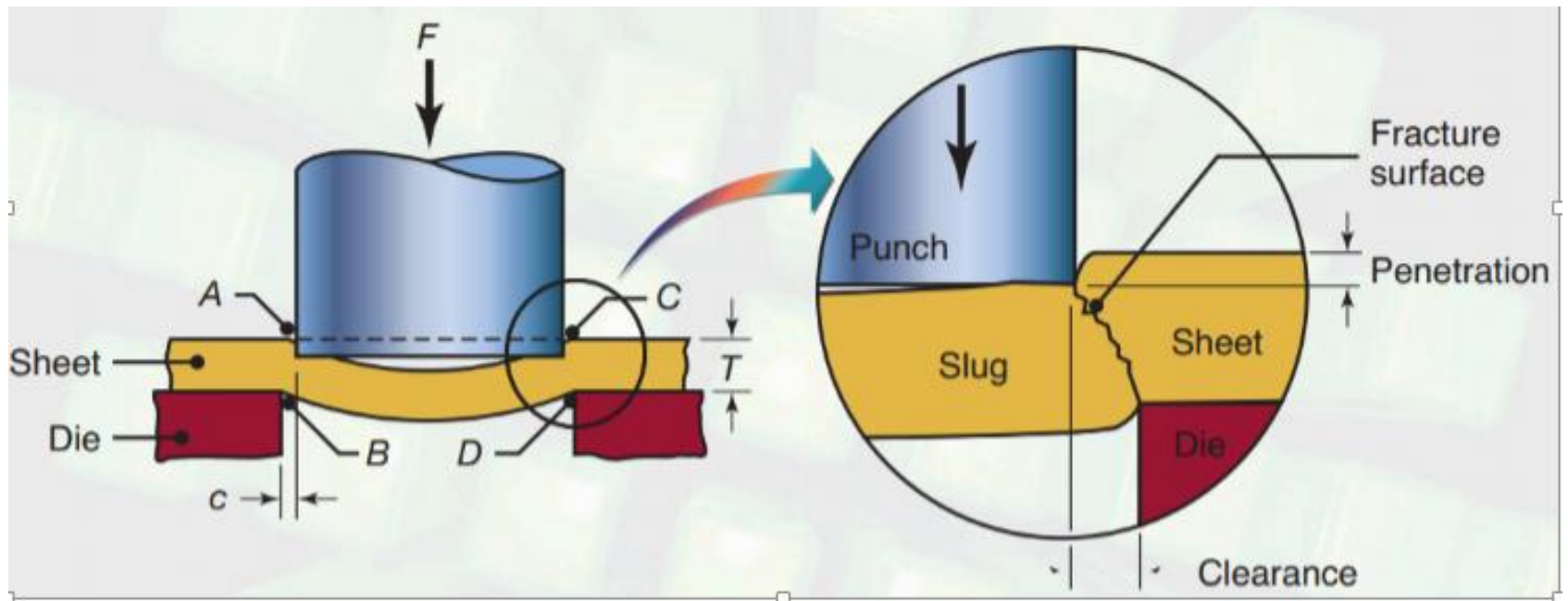
1) as punch first contacts sheet and

2) after cutting and

relative motion

Shearing processes

- **Shearing** is a process for cutting sheet metal to size out of a larger stock such as roll stock.



Three principal operations in press working that cut sheet metal:

- Shearing
- Blanking
- Punching

Shearing: The separation of metal by the movement of two blades operated based on shearing force.

- It is a metal cutting operation along a straight line between two cutting edges.
- Typically used to cut large sheets into smaller sections for subsequent operations

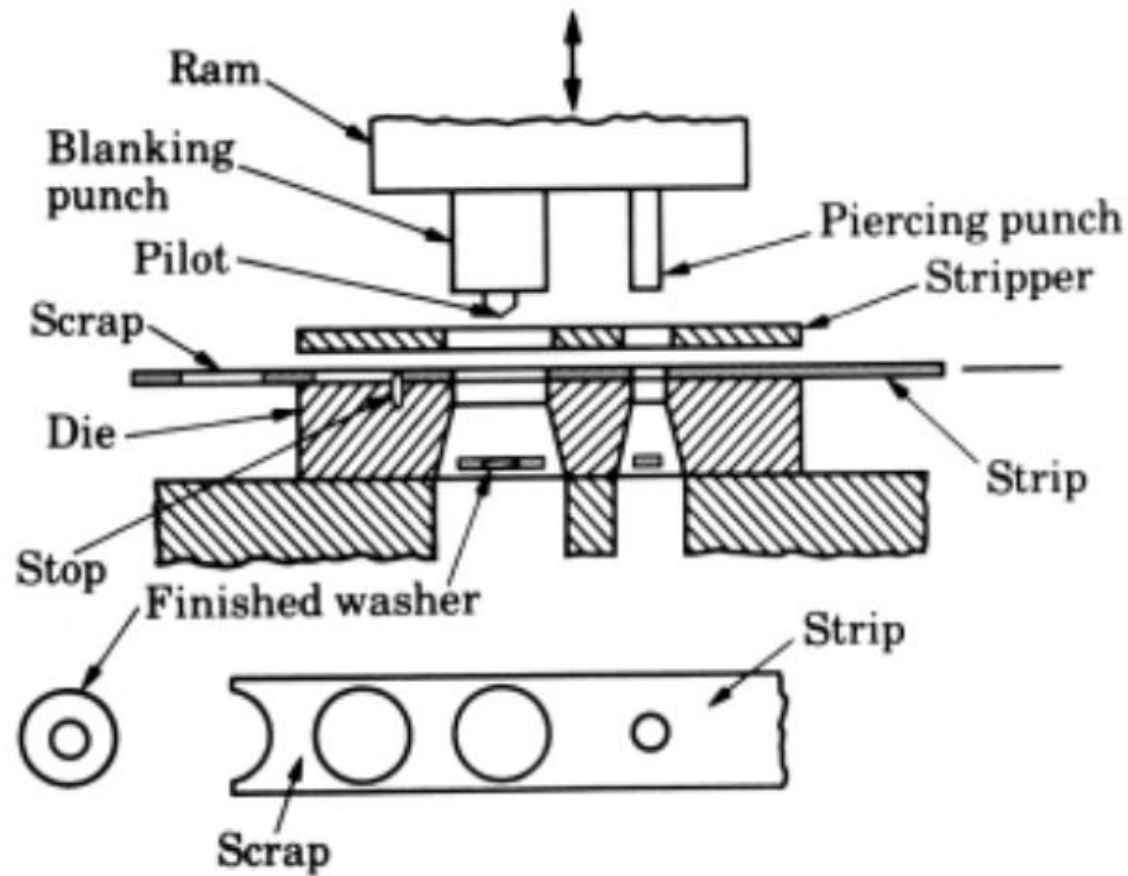
Blanking and Punching

❖ **Blanking** - sheet metal cutting to separate piece from surrounding stock

➤ Cut piece is the desired part, called a *blank*.

❖ **Punching or piercing** - sheet metal cutting similar to blanking except cut piece is scrap, called a *slug*

➤ Remaining stock is the desired part.



Schematic illustration of making a washer in a **progressive die**. NB. Operations such as piercing, blanking, forming drawing and cut off can be carried out on **progressive die**

Design Considerations

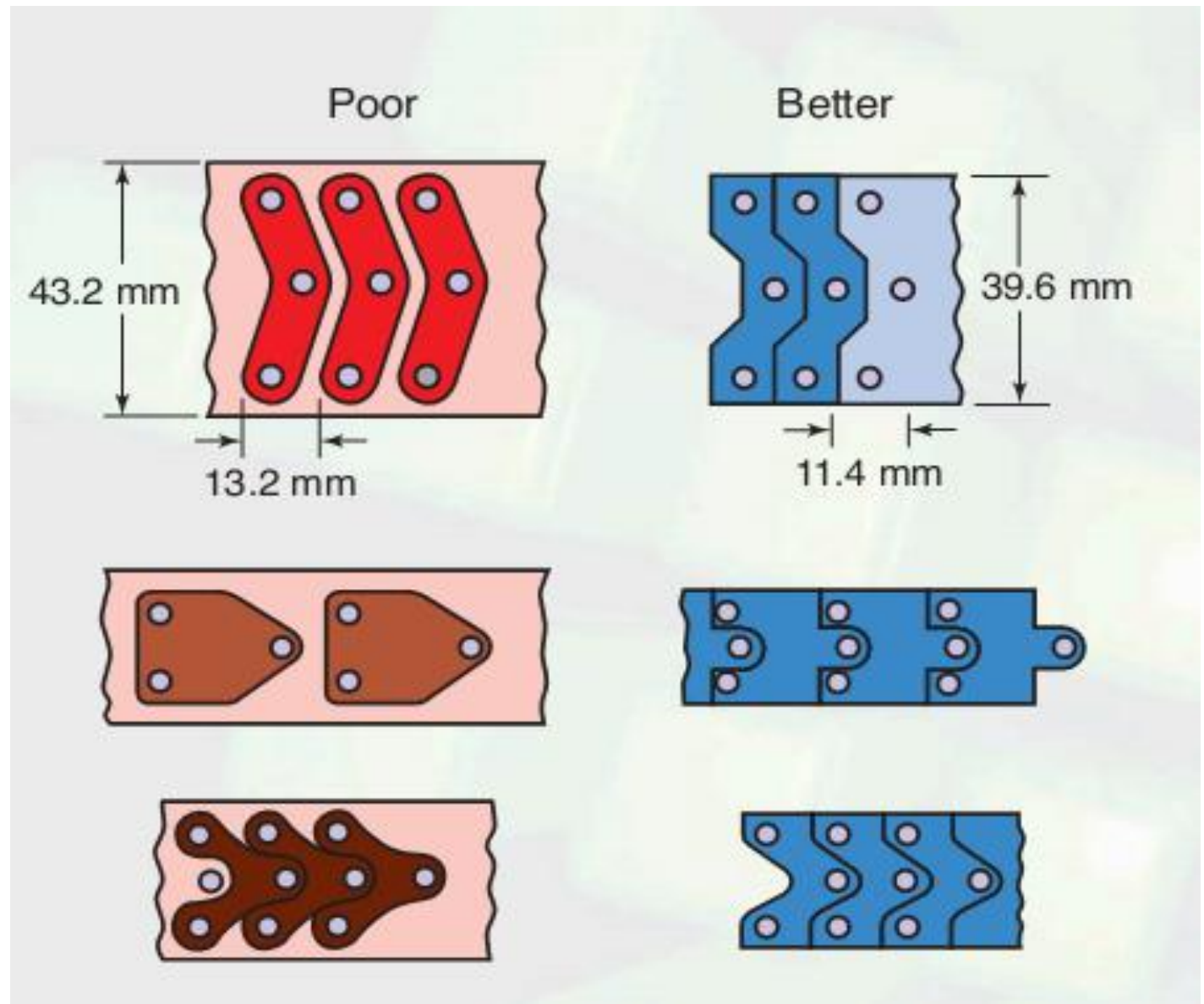
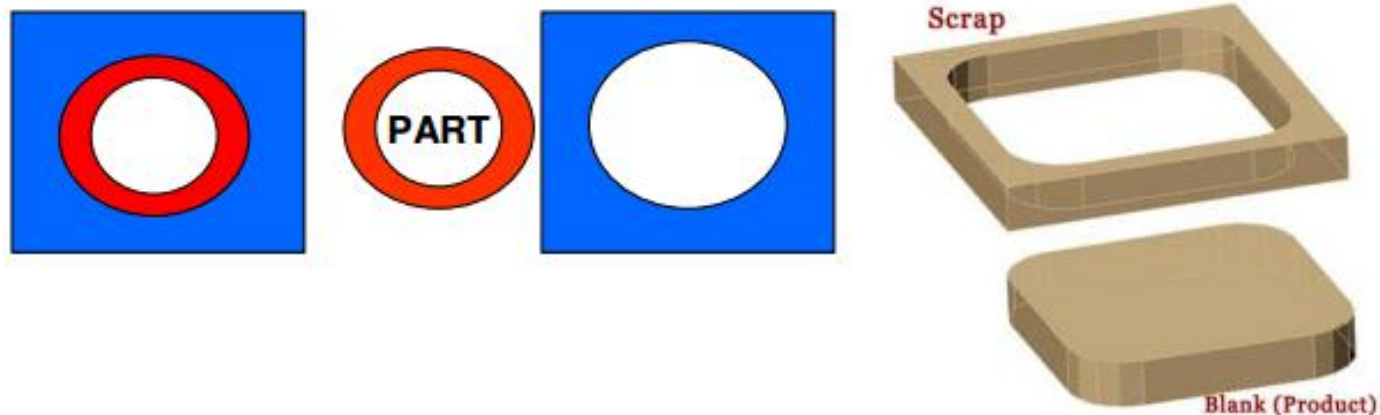
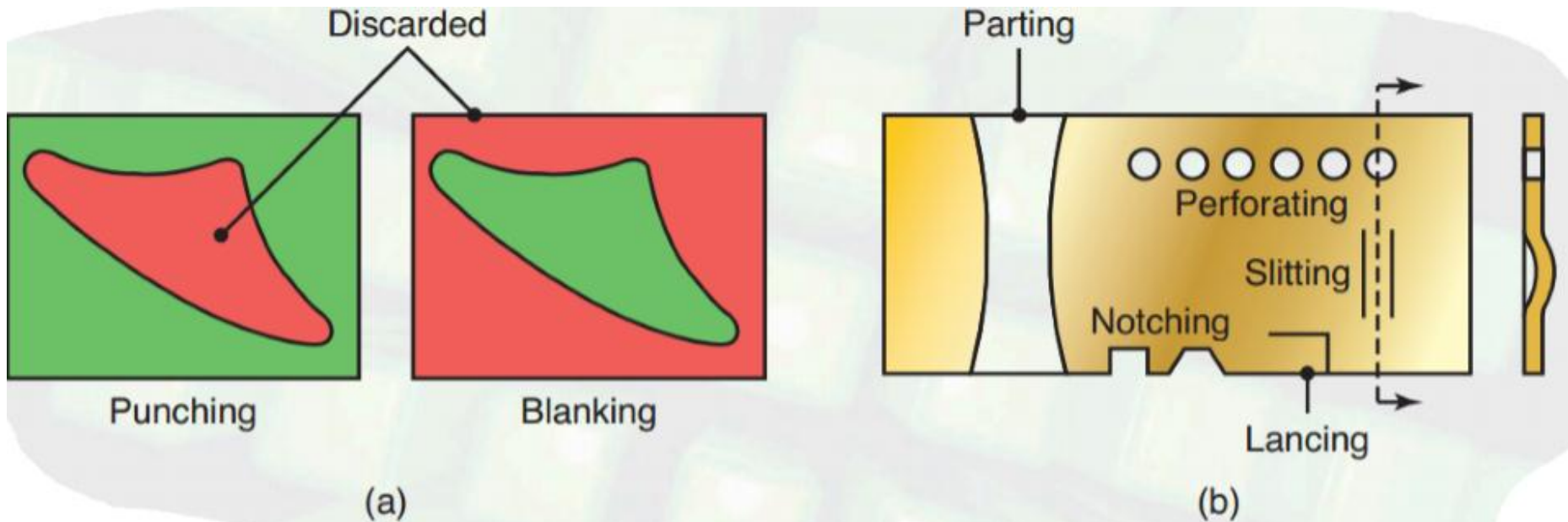


FIGURE 7.67 Efficient nesting of parts for optimum material utilization in blanking. Source: Society of Manufacturing

BLANKING:

- It is also a shearing operation.
- It enlarges earlier pierced hole.
- Removed metal is desired one.





(a) Punching and blanking. (b) Examples of shearing operations on sheet metal.

Clearance in Sheet Metal Cutting

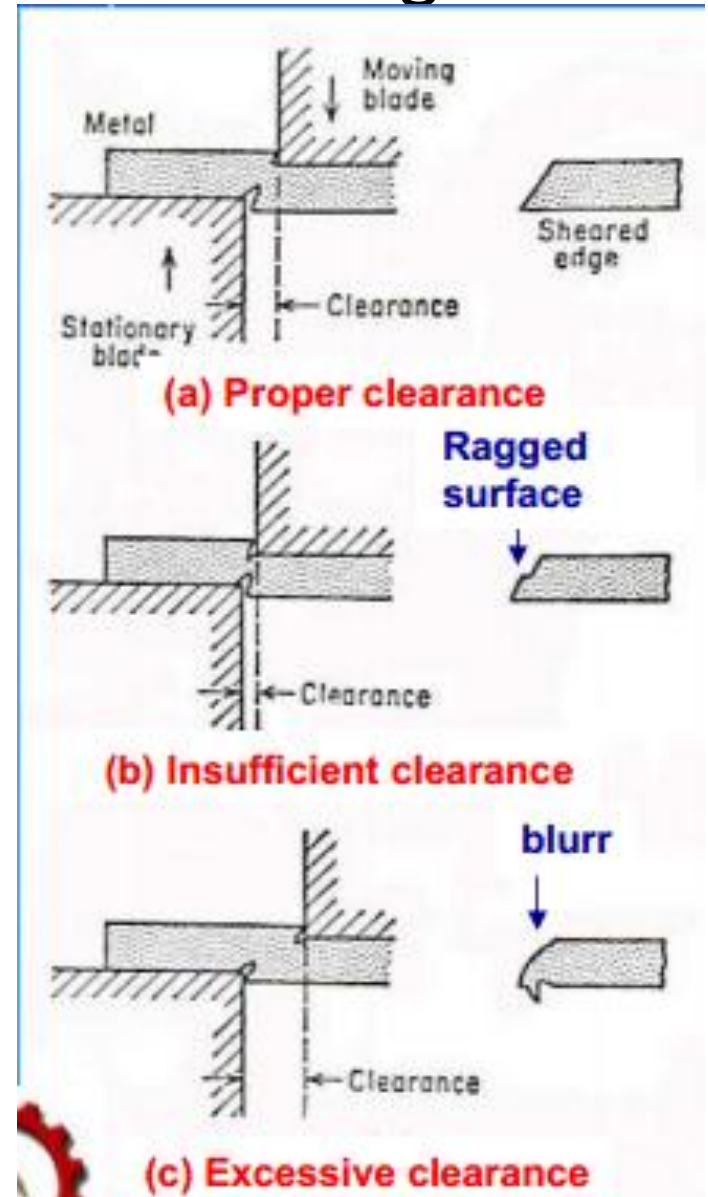
➤ Distance between the punch and die

Clearance 2-10% thickness

➤ Proper = clean fracture surface.

➤ Insufficient = ragged fracture surface.

➤ Excessive = greater distortion, greater energy



- Recommended clearance can be calculated by:

$$c = at$$

where c = clearance; a = allowance; and t = stock thickness

- Allowance a is determined according to type of metal

Allowance a for three Sheet Metal Groups

<u>Metal group</u>	<u>a</u>
1100S and 5052S aluminum alloys, all tempers	0.045
2024ST and 6061ST aluminum alloys; brass, soft cold rolled steel, soft stainless steel	0.060
Cold rolled steel, half hard; stainless steel, half hard and full hard	0.075

Punch and Die Sizes for Blanking and Punching

For a round *blank* of diameter D_b :

Blanking punch diameter = $D_b - 2c$

Blanking die diameter = D_b

where c = clearance

For a round *hole* of diameter D_h :

Hole punch diameter = D_h

Hole die diameter = $D_h + 2c$

where c = clearance

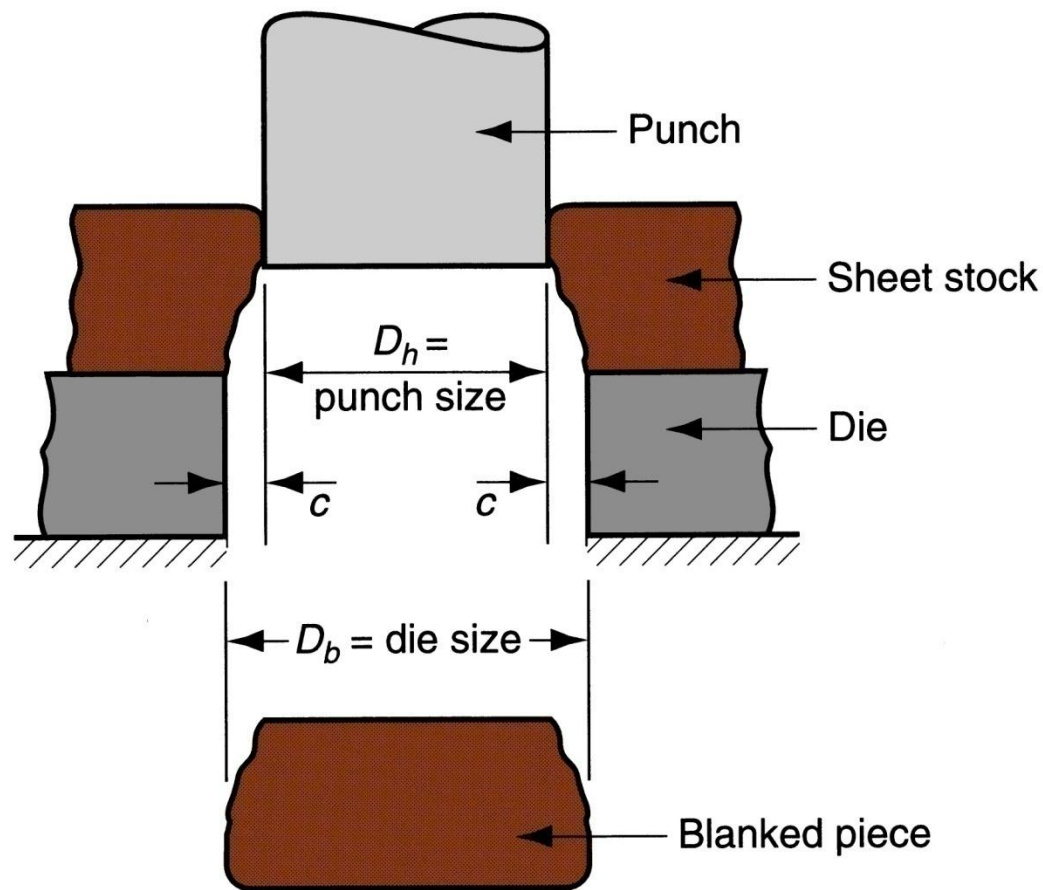


Figure - Die size determines blank size D_b ; punch size determines hole size D_h ; c = clearance

Angular Clearance

Purpose: allows slug or blank to drop through die

- Typical values: 0.25° to 1.5° on each side

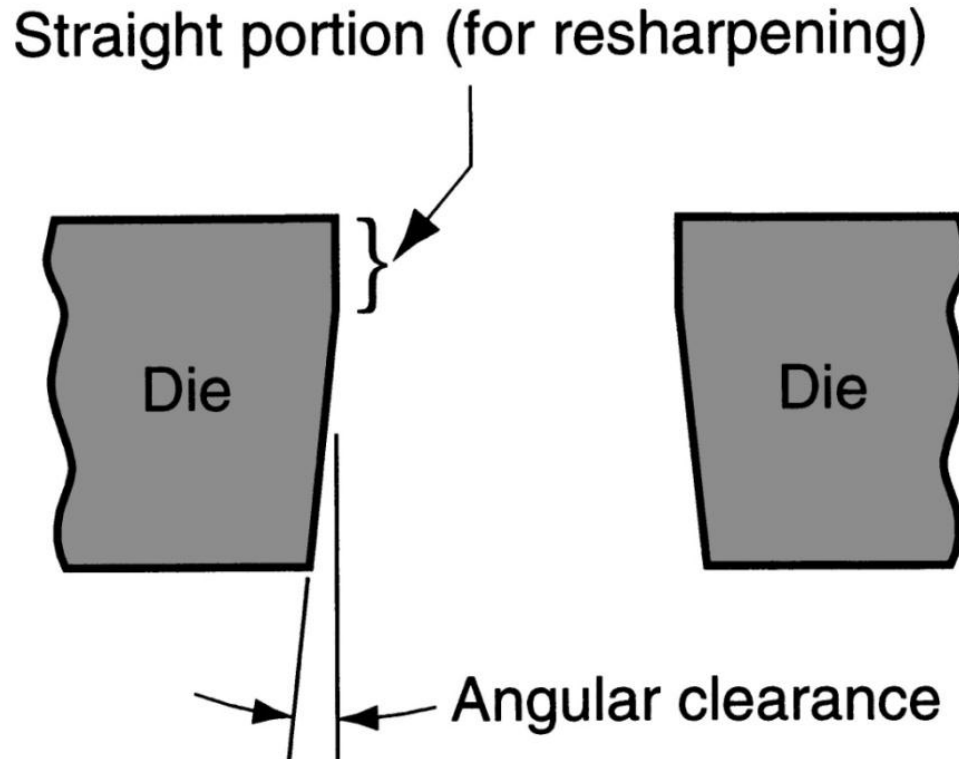


Fig Angular clearance

Cutting Forces

Important for determining press size (tonnage)

$$F = S t L$$

where S = shear strength of metal; t = stock thickness, and L = length of cut edge.

The shear strength S can be estimated by:

$$S = 0.7 * UTS; \text{ where}$$

UTS: the Ultimate Tensile Strength

The above formula does not consider other factors such as friction

Forces in a Shearing Operation

L = Total length Sheared

t = thickness

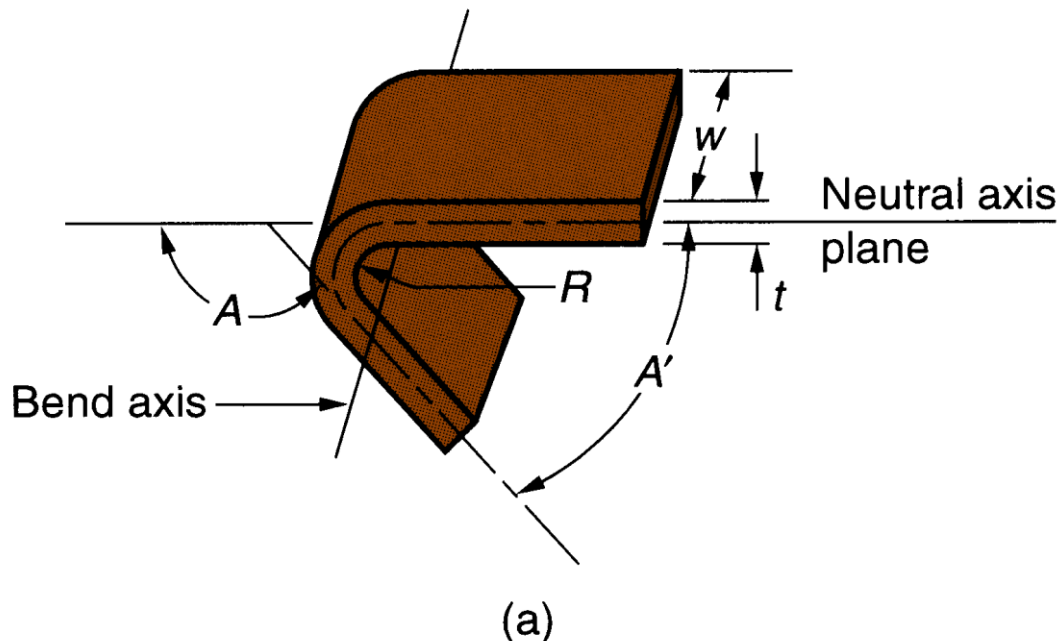
$$Force = 0.7 \times UTS \times t \times L$$

Example Estimate the force required for punching a 25mm diameter hole through a 3.2mm thick annealed titanium alloy Ti-6Al-4V sheet at room temperature. Data: UTS = 1000MPa

$$F = 0.7 \times UTS \times t \times L = 0.7 \times 1000MPa \times 0.0032m \times \pi \times 0.025 = 176kN$$

Bending

- A process by which a straight length is transformed into a curved length to take a permanent bend, produce channels, drums, tanks



A = bend angle = α

w = width of sheet

R = bend radius

t = sheet thickness

$A' = 180^\circ - \alpha$, "included" angle

Figure (a) Bending of sheet metal

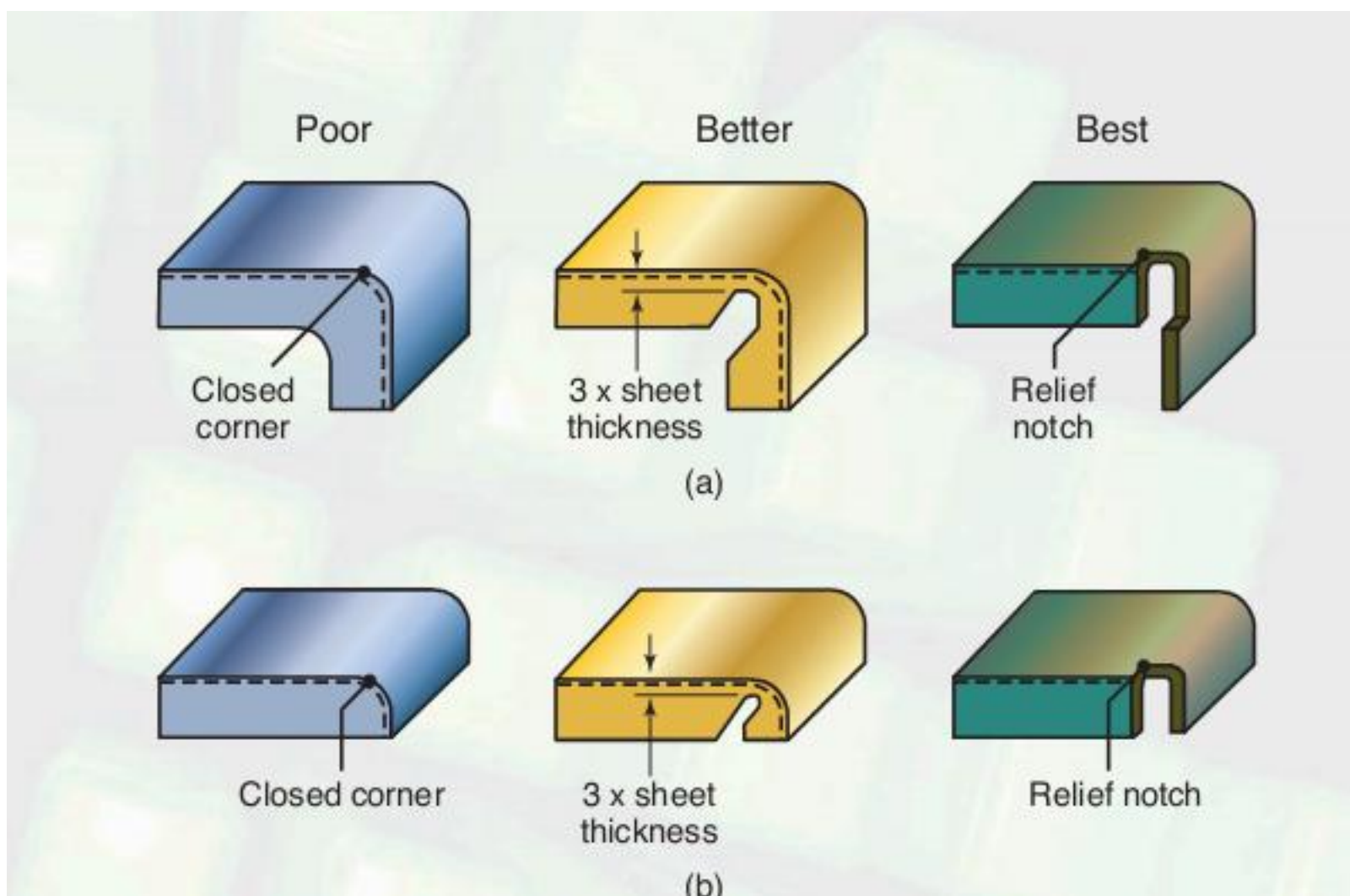


FIGURE 7.68 **Control of tearing and buckling** of a flange in a right-angle bend. Source: Society of Manufacturing Engineers.

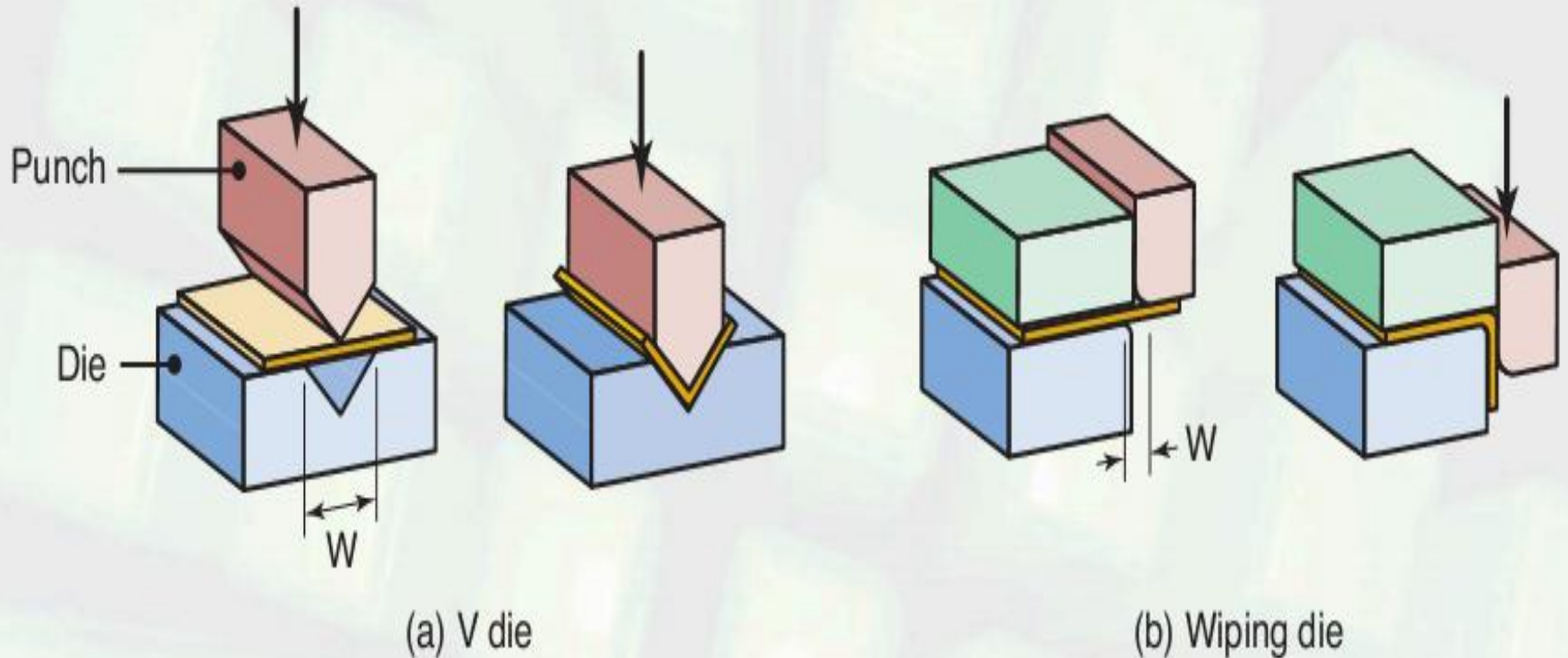
Types of Sheet metal Bending

- *V-bending* - performed with a V-shaped die
- *Edge bending* - performed with a wiping die

V-Bending

- In V-bending the sheet metal blank is bent
 - ✓ between a V-shaped punch and die. For low production
 - ✓ Performed on a *press brake*
 - ✓ V-dies are simple and inexpensive

Die-Bending Operations



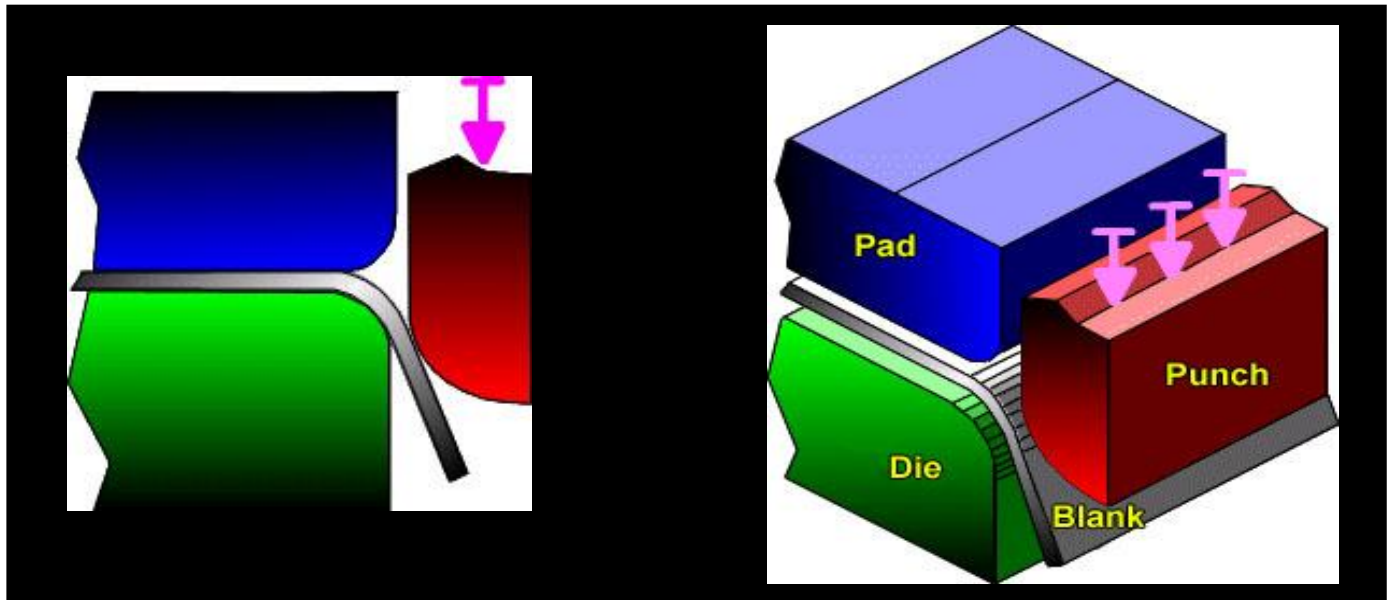


Fig1.30 edge (wipe)-bending setup

❖ Used for

- For high production
- Pressure pad required
- Dies are more complicated and costly

➤ Edge or wipe bending (conducted in lab) involves cantilever loading of the material.

➤ A pressure pad is used to apply a Force to hold the blank against the die, while the punch forces the work piece to yield and bend over the edge of the die.

Design variables

- Punch corner radius : A sharp punch corner leads local thinning and tearing.
- Die corner radius : Die curvature radius is required for smooth metal flow but a very high radius increases frictional forces during bending over the die.
- Clearance between the punch and the die should be enough to accommodate the increase in sheet thickness in the flange and insufficient clearance leads to ironing.

Bending Forces

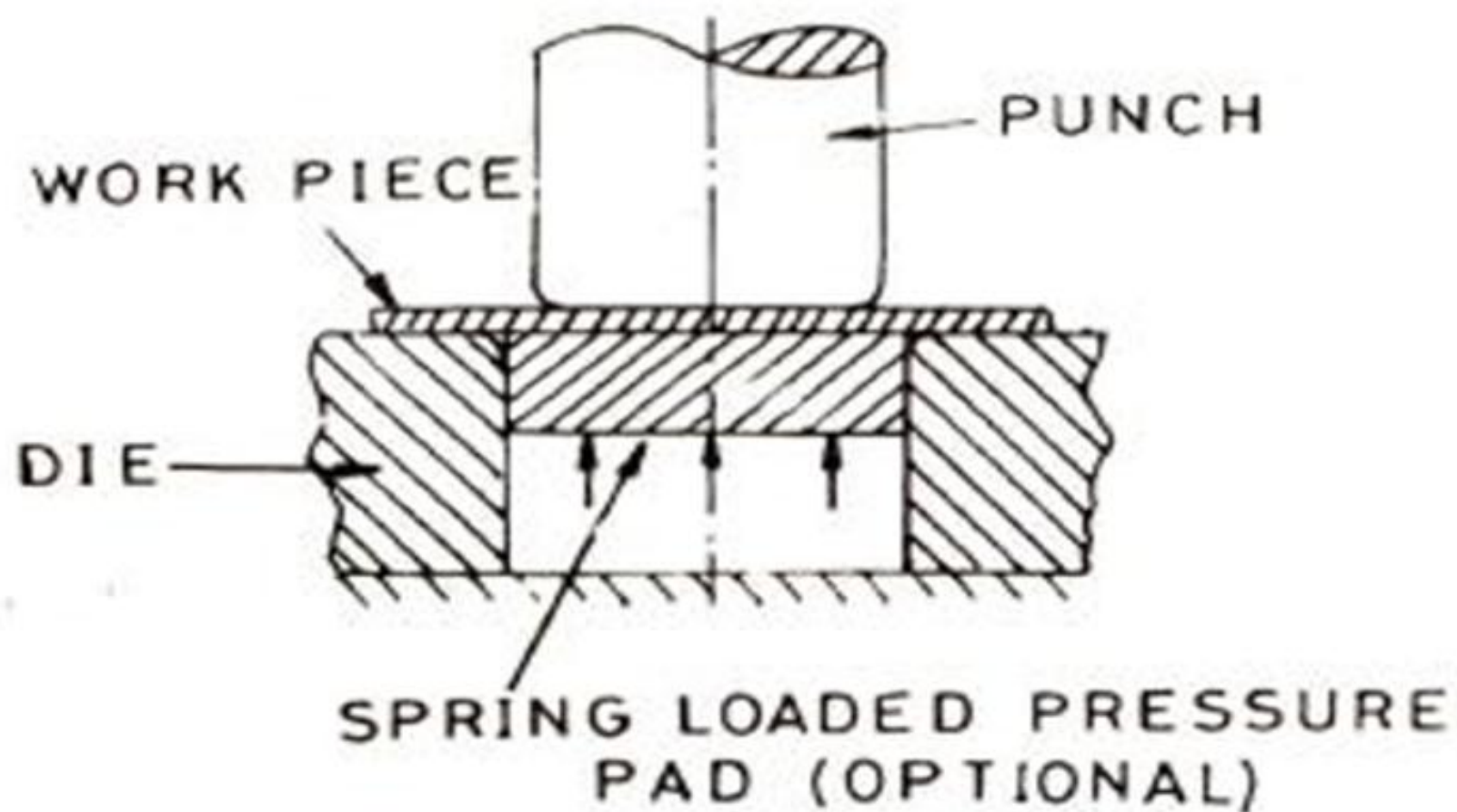


Fig. 7.7

The force required to bend a material is calculated by using the following formula :

$$F_b = \frac{K \cdot L \cdot f \cdot t^2}{W}$$

where

L = Contact length of work in cm.

t = Thickness of work material in cm.

f = Ultimate tensile strength in tonnes/cm².

W = Width of die (span of beam) in cm.

F_b = Bending force in tonnes.

K = Die opening factor.

For V-bends (air bending)

$$K = 1.33 \text{ for } W = 8t$$

$$= 1.2 \text{ for } W = 16t$$

For U-bending (channel bending)

$$K = 2.66 \text{ for } W = 8t$$

$$= 2.4 \text{ for } W = 16t$$

For edge bending

$$K = 0.67 \text{ for } W = 8t$$

$$= 0.6 \text{ for } W = 16t$$

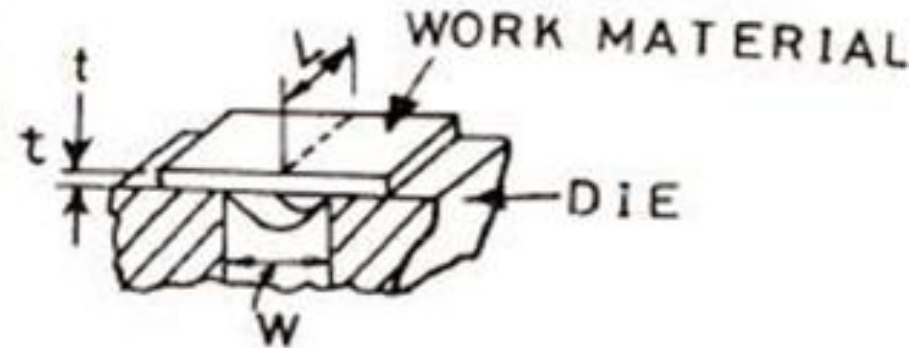
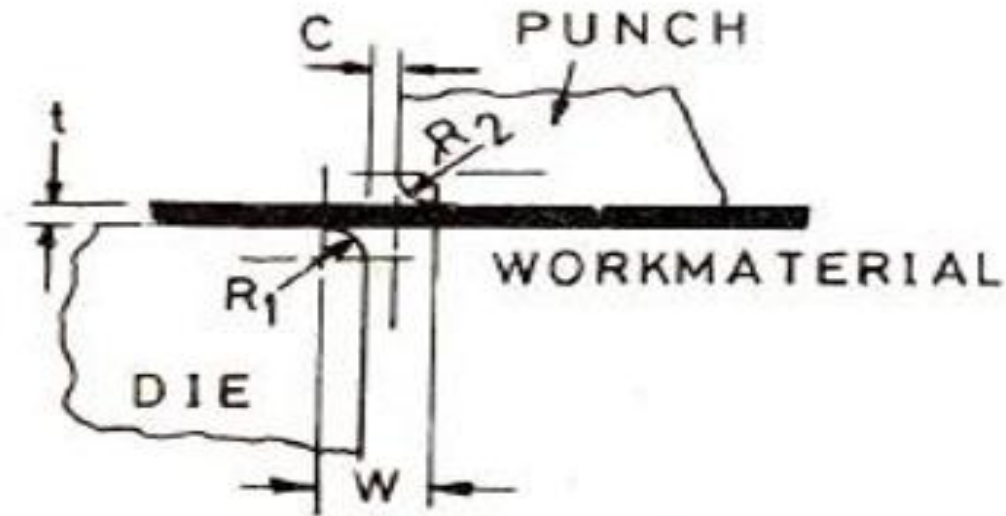
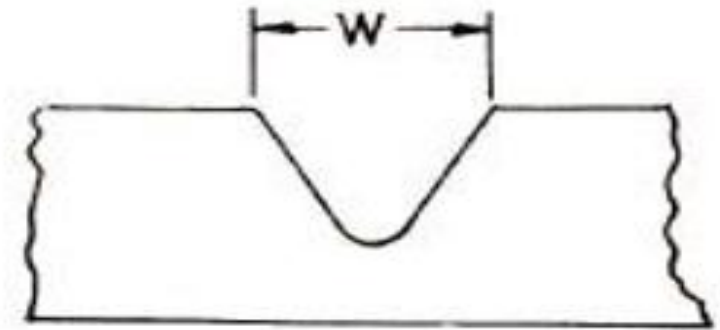


Fig. 7.8

For edge bending the width (W) between contact points on die is determined as follows (See Fig. 7.9)



(a)



(b)

Fig. 7.9

$$W = R_1 + R_2 + C$$

where

R_1 = Die radius

R_2 = Punch radius

t = Thickness of metallic sheet

Clearance

$$C = t$$

$$W = R_1 + R_2 + t.$$

Bend Allowance

In order to estimate the required flat work piece length to make a bend, calculation of bend allowance is essential.

To make a bend as shown in Fig. 7.10, the length of flat blank is determined as follows :

$$L = L_1 + L_2 + A$$

where L = Length of flat blank required to make the bend

L_1 = Length of bend leg 1

L_2 = Length of bend leg 2

A = Bend allowance

θ = Bend angle (Angle through which metal is bent)

R = Internal radius of bend

t = Metallic sheet thickness

$$A = \frac{2\pi\theta}{360}(R + Kt)$$

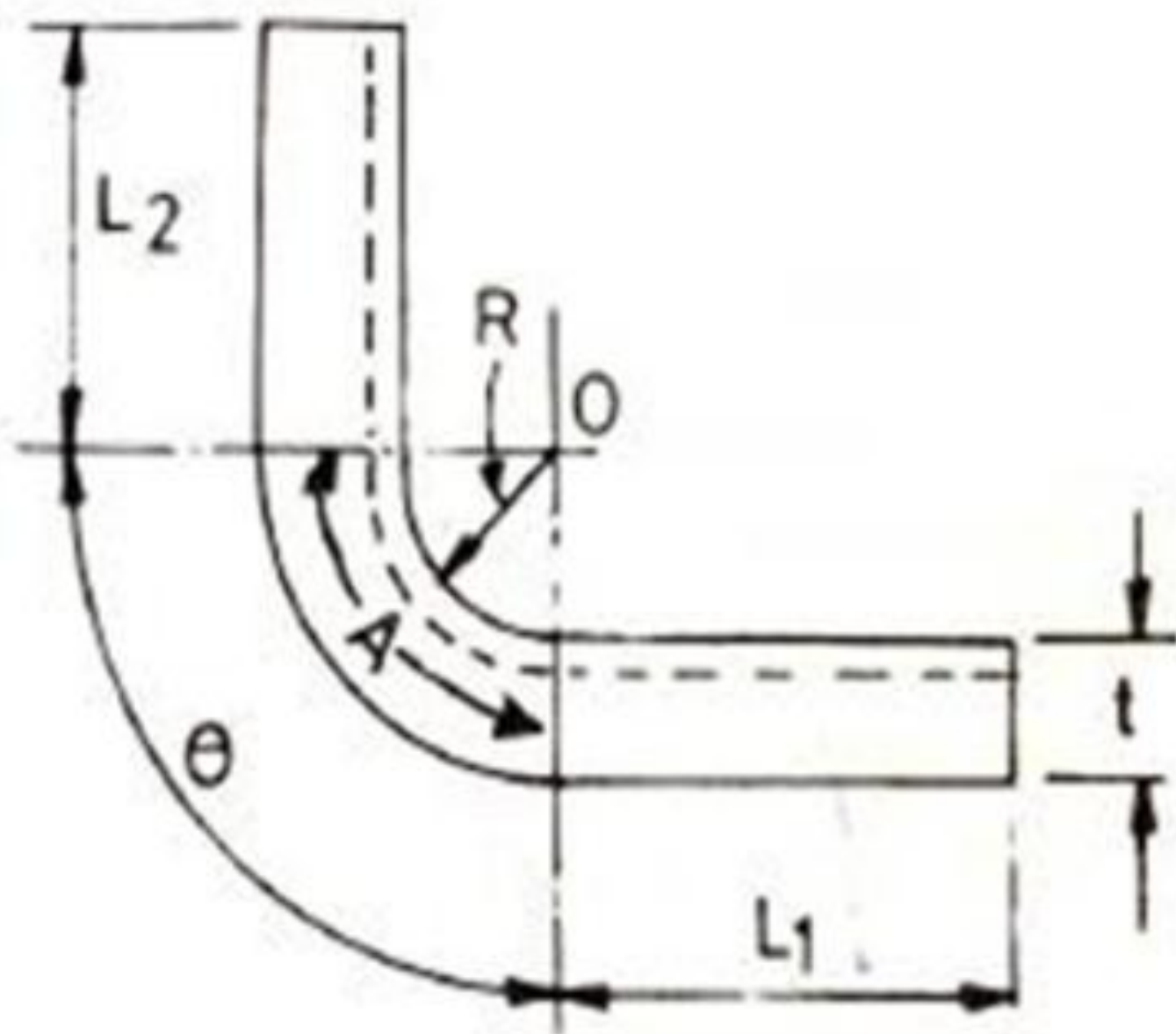
where

K = Constant

= $1/4$ when $R \leq t$

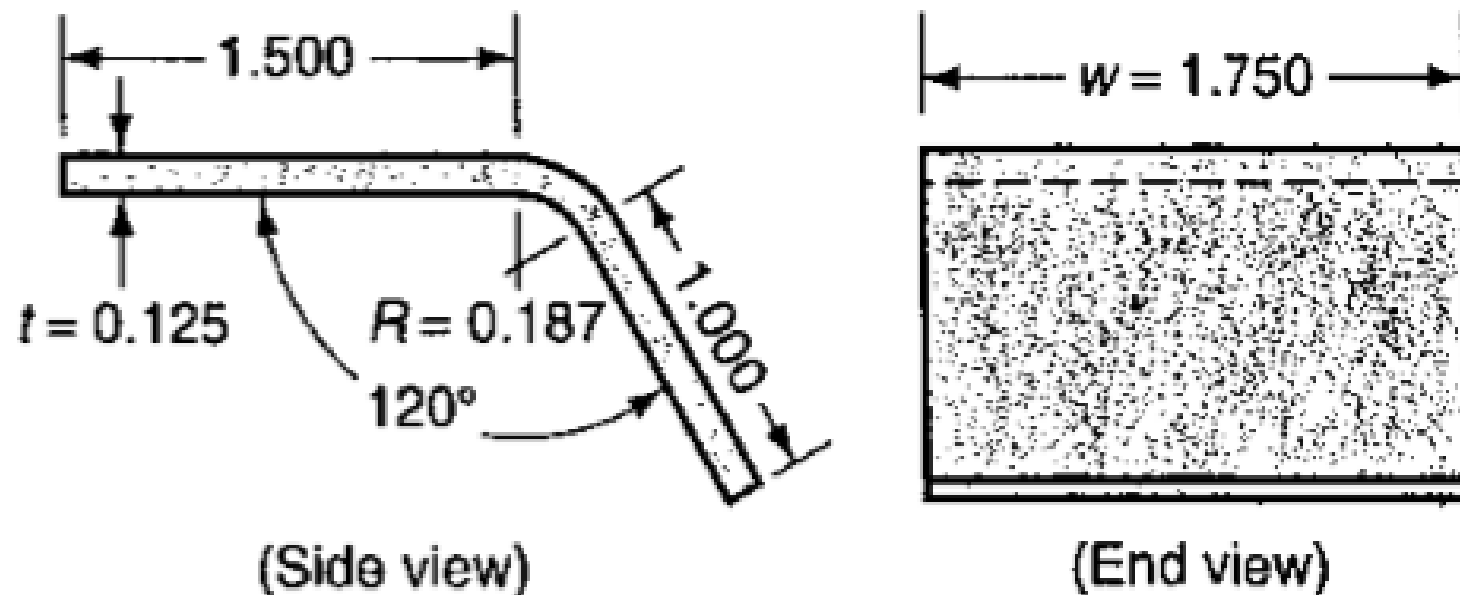
= $1/3$ when $R \leq 2t$

= $1/2$ when $R > 2t$.



Example:

A sheet material is to be bended according to the dimensions given in the figure. Determine the following: (a) starting blank size and (b) the bending force necessary if a V-die will be used with a die opening dimension of $W=1.0''$.
Data: $E=30000\text{ksi}$; Yield Strength= 40000psi and a tensile strength= 65000psi .



Length $L = 1.75'$, and the length of the part is: $1.5 + 1.00 + BA$.

$R/t = 0.187/0.125 = 1.5 < 2.0$, so $K_{ba} = 0.33$

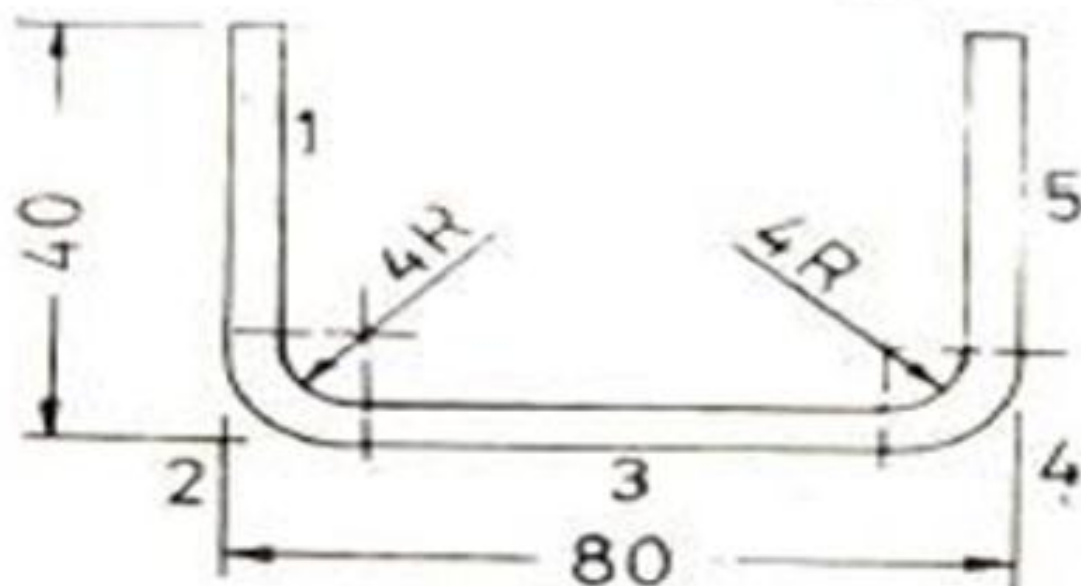
For an included angle $A' = 120^\circ$, then $A = 60^\circ$

$$L_b = \alpha(R + kt) = \left(\pi \frac{60}{180} \right) (0.187 + 0.33 \times 0.125) = 0.239"$$

$$\text{Length} = 1.5 + 1.0 + 0.239 = 2.739"$$

$$F_b = \frac{K \cdot L \cdot UTS \cdot t^2}{W} = \frac{1.33 \cdot 1.75 \cdot 65000 \cdot (0.125)^2}{1} = 2364 \text{ lb}$$

Example Calculate the blank length to make the part shown in Fig. Also determine the bending force required if the ultimate tensile strength of material is 3500 kg/cm^2 and die radius is 8 mm. The bend length being 120 cm.



Solution. (a) R = Inside radius
 $= 4 \text{ mm.}$

t = Thickness of metal $= 2 \text{ mm.}$

L_1 = Length of segment 1
 $= 40 - (t + R)$
 $= 40 - (2 + 4)$
 $= 34 \text{ mm}$

θ = Bend angle $= 90^\circ$

Bend Allowance, $L_2 = \frac{2\pi\theta}{360}[R + Kt]$

where

$K = 1/3 \text{ as } R = 2t$

$$L_2 = \frac{2\pi \times 90}{360} [4 + 1/3 \times 2]$$

$$= 7.33 \text{ mm}$$

$$L_3 = 80 - [t + t + R + R]$$

$$= 80 - [2 + 2 + 4 + 4]$$

$$= 68 \text{ mm}$$

$$L_4 = \text{Bend allowance}$$

$$L_4 = L_2 = 7.33$$

$$L_5 = \text{Length of segment 5}$$

$$= L_1$$

$$= 34 \text{ mm}$$

$$L = \text{Total blank length}$$

$$= \text{Length of all segments}$$

$$= L_1 + L_2 + L_3 + L_4 + L_5$$

$$= 34 + 7.33 + 68 + 7.33 + 34$$

$$= 150.66 \text{ mm}$$

(b)

t = Sheet thickness

$$= 2 \text{ mm} = 0.2 \text{ cm}$$

f = ultimate tensile strength

$$= 3500 \text{ kg/cm}^2$$

$$R_1 = R_2 = 8 \text{ mm}$$

W = Width of die

$$= R_1 + R_2 + C$$

where

$$C = t$$

$$= 8 + 8 + 2 = 18 \text{ mm}$$

$$= 1.8 \text{ cm.}$$

L = Length of part

$$= 120 \text{ mm.}$$

$K = 2.66$ for U bending

F_b = Bending force

$$= \frac{K \cdot L \cdot f t^2}{W} \quad ? \checkmark$$

$$= \frac{2.66 \times 120 \times 3500 \times 0.2^2}{1.8}$$

$$= 24800 \text{ kg}$$

$$= 24.8 \text{ tonnes.}$$

Example:

Estimate the force required for a 90 degrees bending of a St 50 steel of thickness of 2mm in a V die. The die opening can be taken as eight times the thickness. The length of the part is 1m.

Die Opening $W=8 \cdot 2=16\text{mm}$

UTS=500MPa

$$F_b = \frac{K \cdot L \cdot UTS \cdot t^2}{W} = \frac{1.33 \cdot 1 \cdot 500 \cdot (0.002)^2}{0.016} = 166.25\text{kN}$$

Spring back in Bending

- When the bending stress is removed at the end of the deformation process, elastic energy remains in the bent part causing it to partially recover to its original shape.
- In bending, this elastic recovery is called **spring back**.

Spring back = increase in included angle of bent part relative to included angle of forming tool after tool is removed

Reason for spring back:

- When bending pressure is removed, elastic energy remains in bent part, causing it to recover partially **toward its original shape** .

➤ It increases with

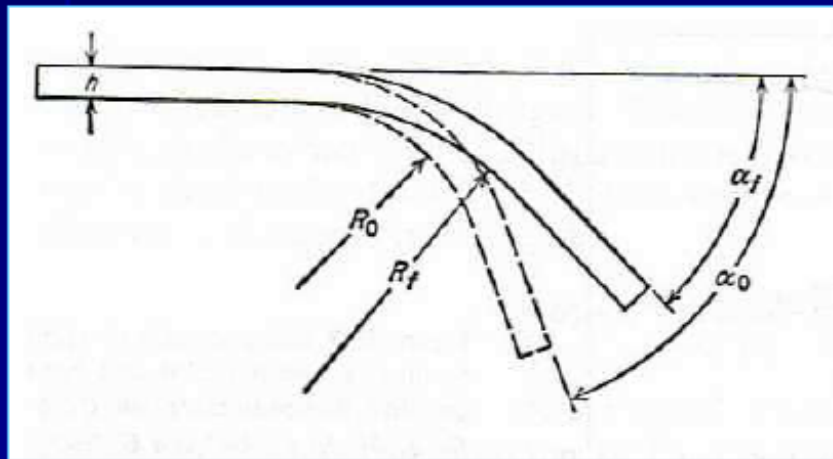
decreasing the modulus of elasticity, E , and
increasing the yield strength, Y , of a material.

After spring back:

- The bend angle will **decrease** (the included angle will **increase**) & the bend radius will increase

Springback

Dimensional change of the formed part after releasing the pressure of the forming tool due to the changes in strain produced by **elastic recovery**.



Yield stress



Elastic modulus



Plastic strain



Spring back



Springback is encountered in all forming operations, but most easily occurs in bending.



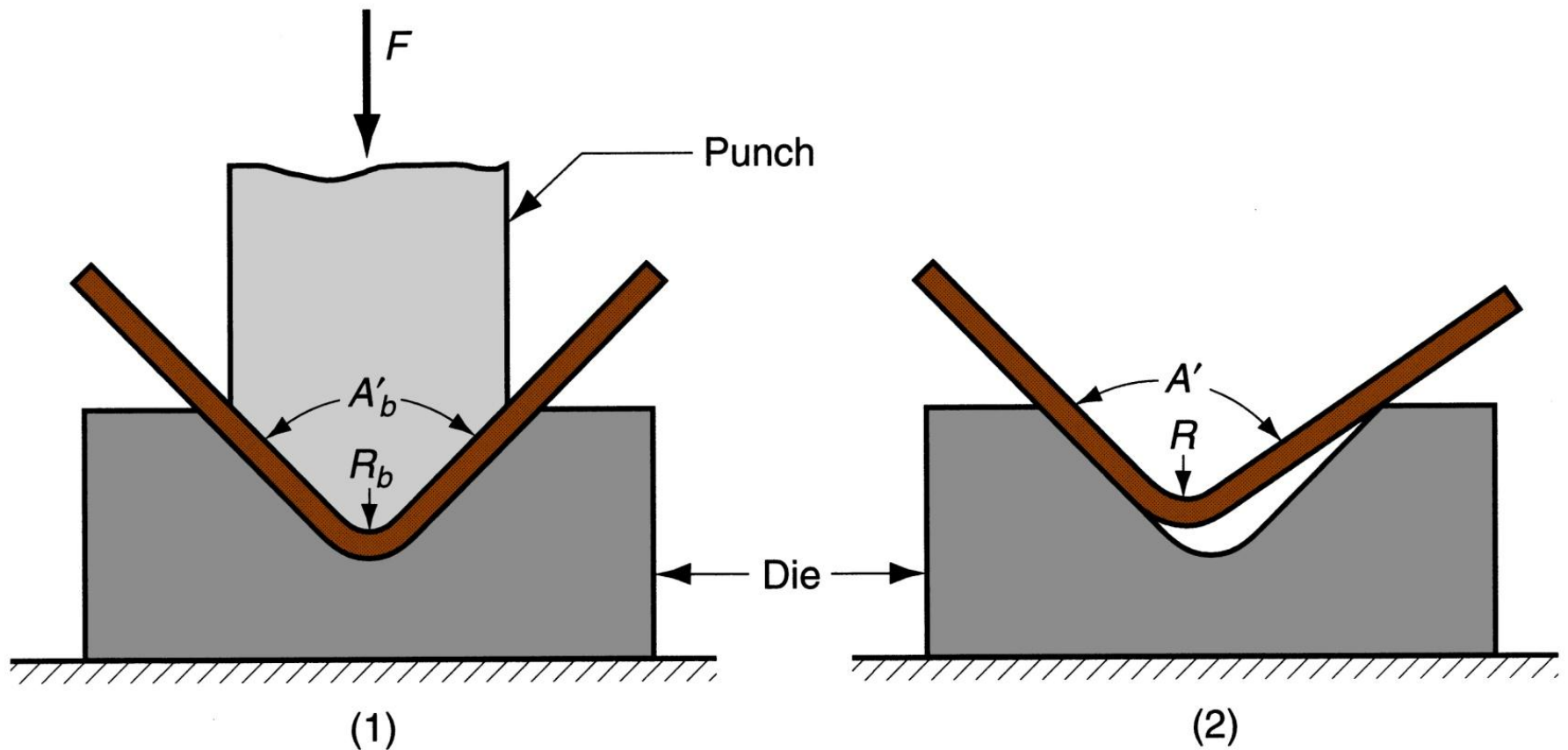
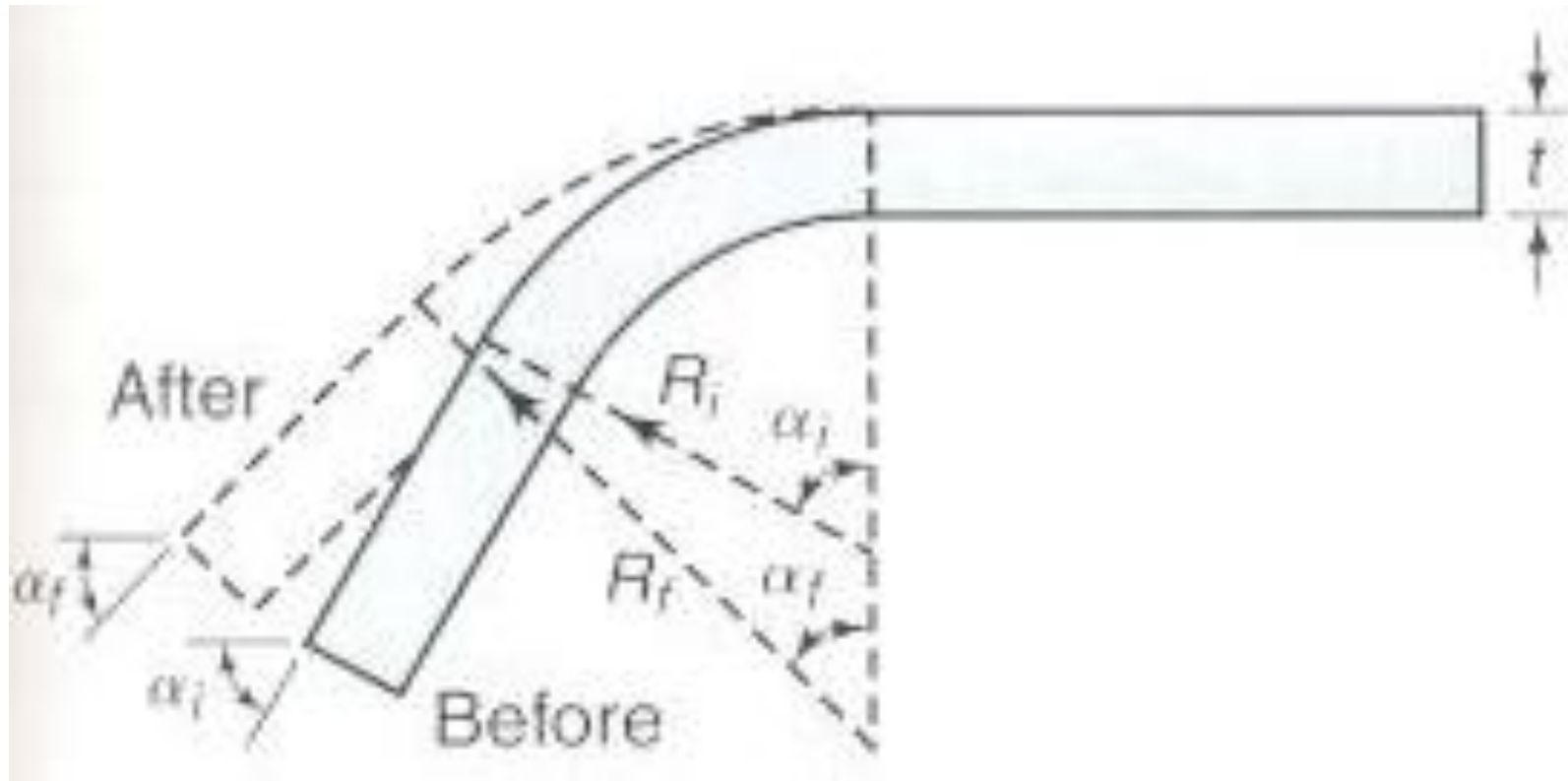


Figure **Spring back** in bending shows itself as a **decrease in bend angle and an increase in bend radius:**

Following is a schematic illustration of spring back in bending:



α_i : bend angle before spring back
 α_f : bend angle after spring back
 R_i : bend radius before spring back
 R_f : bend radius after spring back
Note: R_i and R_f are internal radii

➤ In order to estimate spring back, the following formula can be used:

$$\frac{R_i}{R_f} = 4 \left(\frac{R_i Y}{E \cdot t} \right)^3 - 3 \left(\frac{R_i Y}{E \cdot t} \right) + 1$$

where:

R_i , R_f : initial and final bend radii respectively

Y : Yield strength

E : Young's modulus

t : Sheet thickness

Example:

A 0.0359in thickness sheet (20-gage) is bent to a radius of 0.5in. Calculate the radius of the part after it is bent and the required bend angle to achieve a 90o bend after springback has occurred. Data: Yield Strength=40000psi; $E=29 \times 10^6$ psi

$$\frac{R_o}{R_f} = 4 \left(\frac{R_o \sigma_{yield}}{Et} \right)^3 - 3 \frac{R_o \sigma_{yield}}{Et} + 1$$

$$\frac{0.5}{R_f} = 4 \left(\frac{0.5 \cdot 40000}{29 \cdot 10^6 \cdot 0.0359} \right)^3 - 3 \frac{0.5 \cdot 40000}{29 \cdot 10^6 \cdot 0.0359} + 1 = 0.942$$

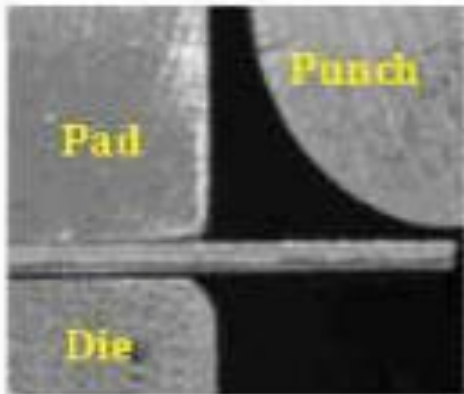
$$R_f = 0.531 \text{ in}$$

$$L_b = \alpha_o (R_o + kt) = \alpha_f (R_f + kt)$$

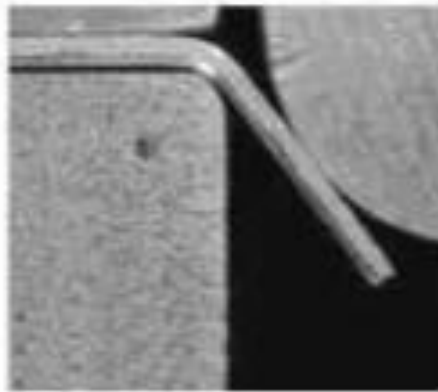
$$\alpha_o \left(0.5 + \frac{0.0359}{2} \right) = 90 \cdot \left(0.531 + \frac{0.0359}{2} \right)$$

$$\Rightarrow \alpha_o = 95.4^\circ$$

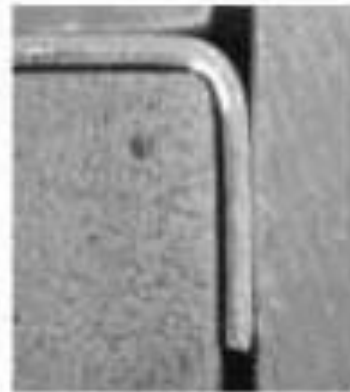
Sheet metal forming



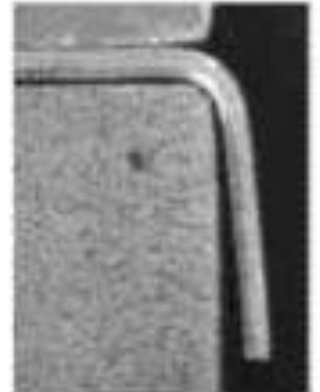
Start



During



Final

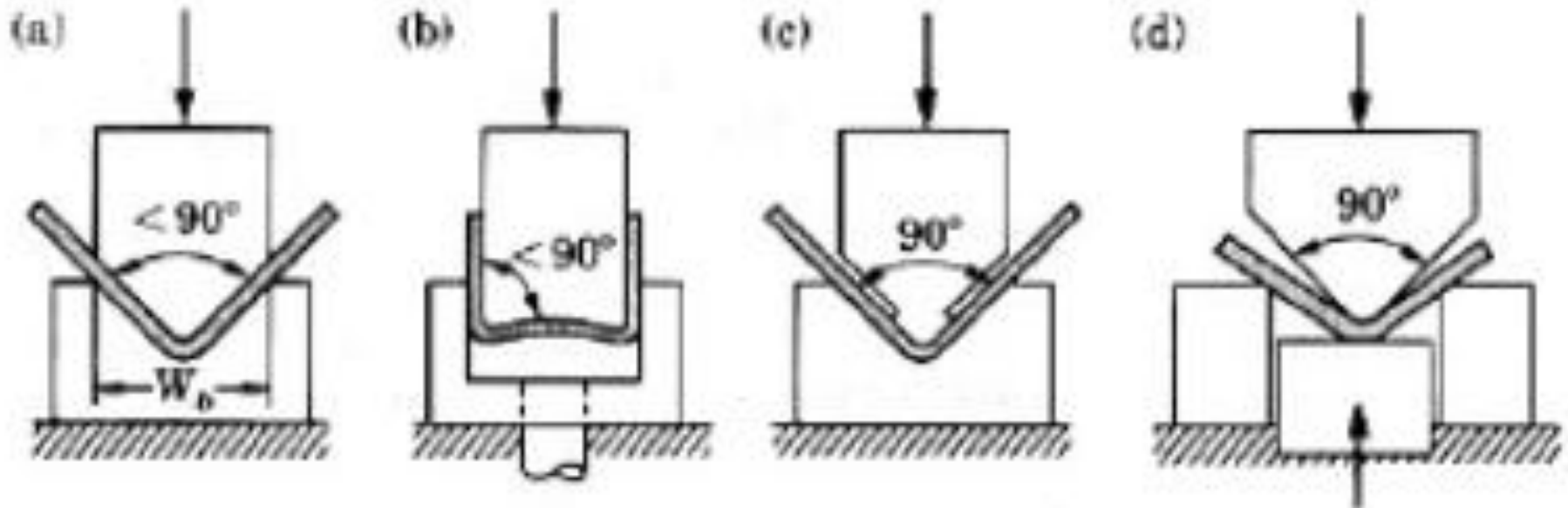


Springback

Compensation for Spring back

- Many ways can be used to compensate for spring back.
- Two common ways are:
 - Over bending
 - Bottoming (coining)
- When over bending is used in V-bending (for example), **the punch angle and radius are fabricated slightly smaller than the specified angle and radius of the final part.** This way the material can “spring back” to the desired value.
- Bottoming involves squeezing the part at the **end of the stroke**, thus plastically deforming it in the bend region.

Compensation for Spring back



Methods of reducing or eliminating spring back in bending operations.

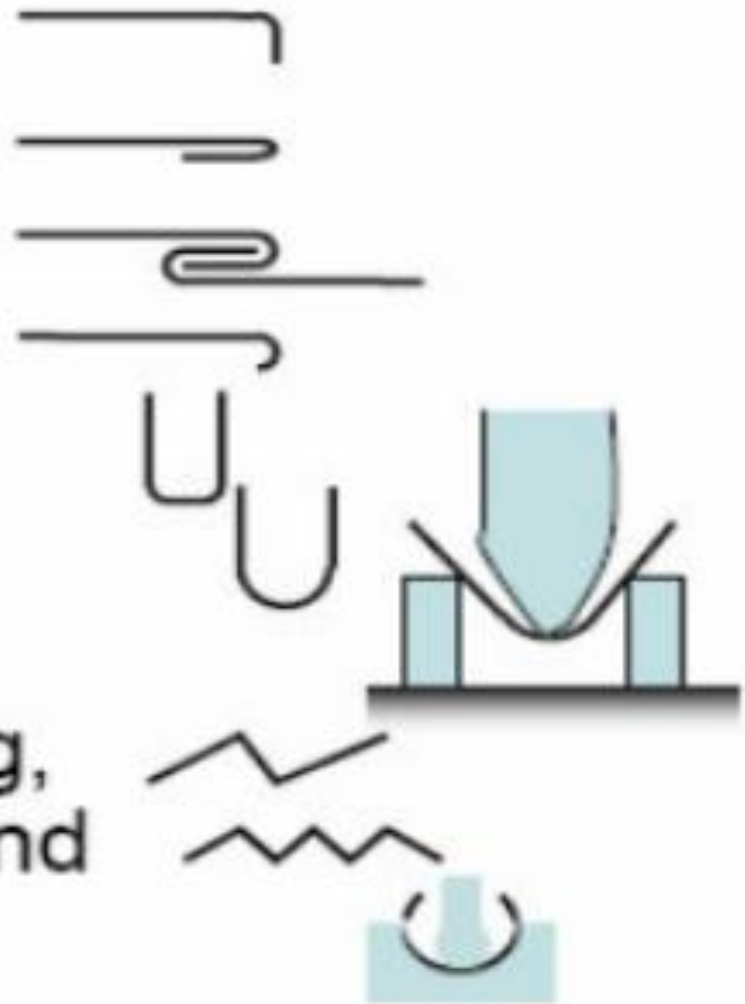
Compensation of spring back by

(a) and (b) *by over bending*

(c) and (d) *by bottoming* (squeezing the part at the end of the stroke)

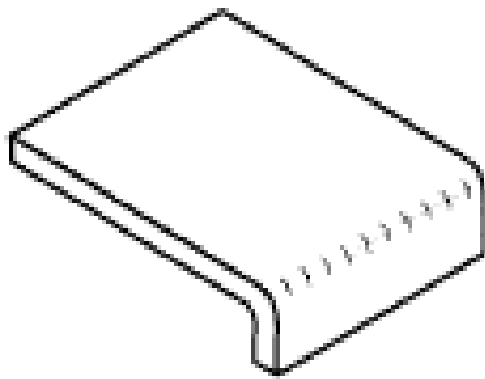
Other bending operations

- Flanging
- Hemming
- Seaming
- Curling
- Channel,
- U-bending
- Air bending,
- Offset bending,
- Corrugating and
- Tube forming

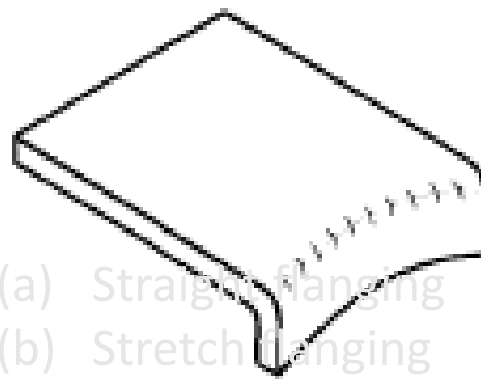


Other bending operations include:

- **Flanging** is a bending operation in which the **edge of a sheet metal is bent at a 90° angle** to form a rim or flange. It is often used **to strengthen or stiffen sheet metal**. The flange can be straight, or it can involve stretching or shrinking as shown in the figure below:

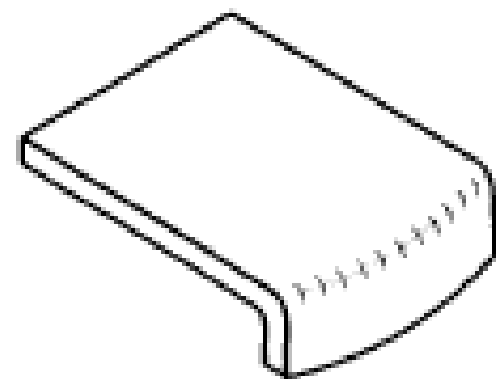


(a)



(a) Straight flanging

(b) Stretch flanging



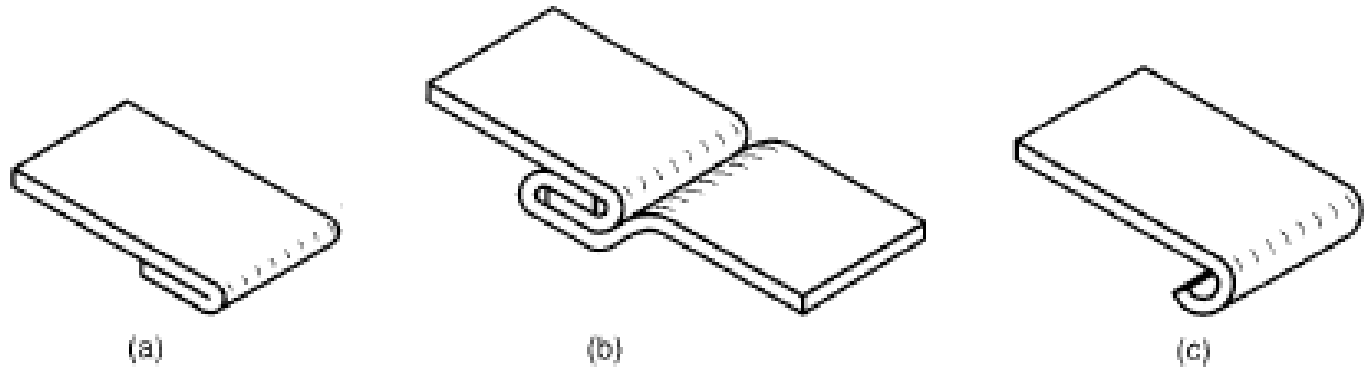
(c) Shrink flanging

(c)

• **Hemming** involves bending the edge of the sheet over onto itself in more than one bending step. This process is used to eliminate sharp edges, increase stiffness, and improve appearance, such as the edges in car doors. (see fig.a below)

• **Seaming** is a bending operation in which two sheet metal edges are joined together. (see fig.b below)

• **Curling** (or beading) forms the edges of the part into a roll. Curling is also used for safety, strength, and aesthetics. (see fig.c below)

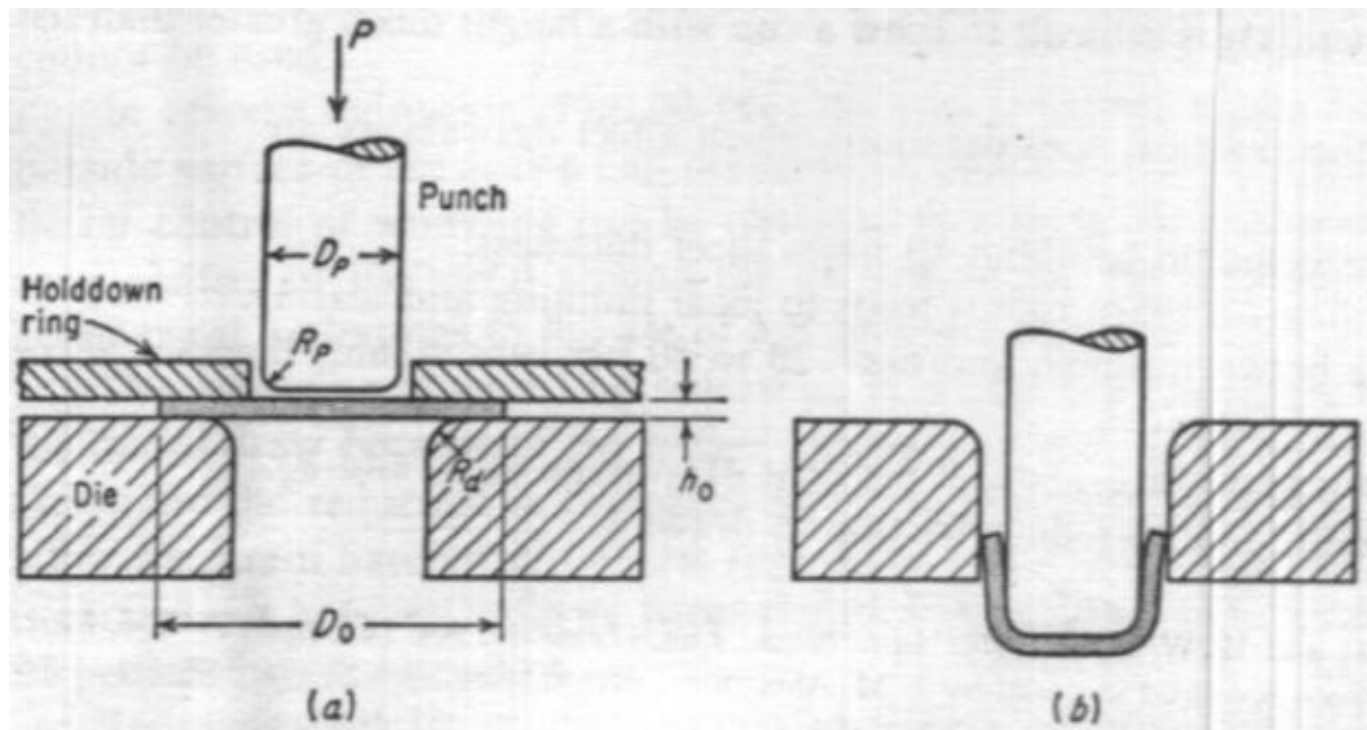


End of Sheet metal forming

N.B: The next topic is
deep drawing

Deep drawing

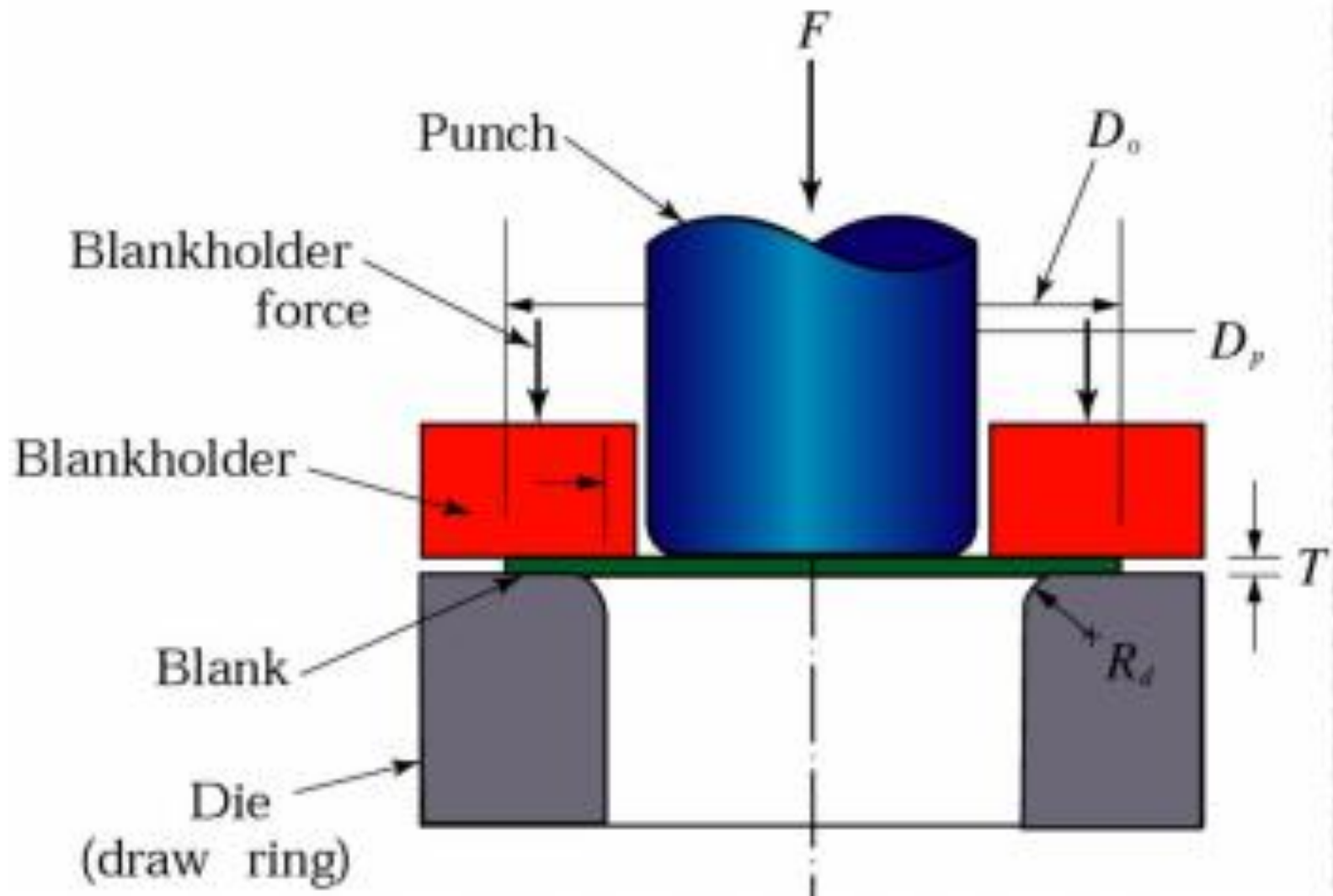
Deep drawing manufacturing technology is the stretching of sheet metal stock, commonly referred to as a blank, around a plug. The edges of the metal blank are restrained by rings and the plug is **deep drawn** into a top die cavity to achieve the end shape that is desired. There are many shapes that can be made through **deep drawing** and stamping such as **cups, pans, cylinders**, and hemispheres products.



Deep drawing of a cylindrical cup (a) before drawing and (b) after drawing

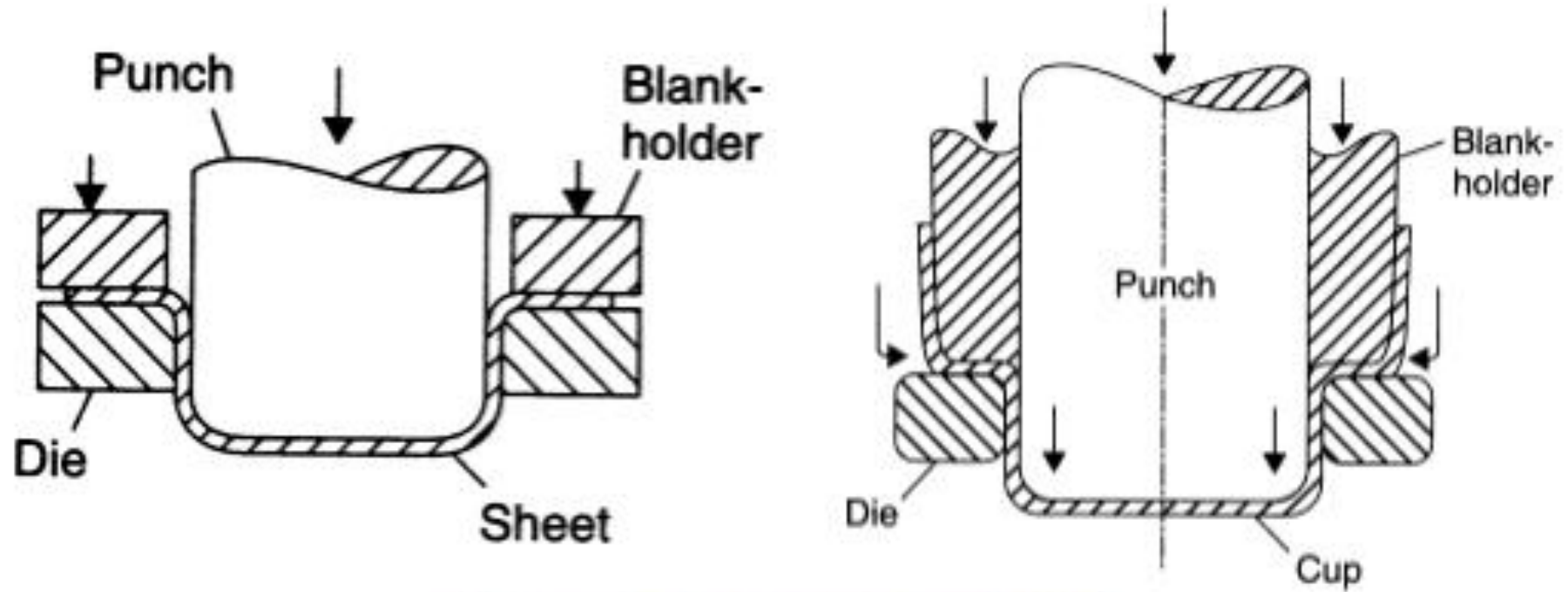
Process variables

- Blank holding pressure: Insufficient blank holding pressure causes wrinkling during drawing and a high BHP causes tearing at the die throat.
- Lubrication: Lubrication between blank and the die helps reduce the drawing force and hence drawability improves.



Process variables in deep drawing. Except for the punch force, F , all the parameters indicated in the figure are independent variables.

Deep drawing/redrawing



ERC NSM

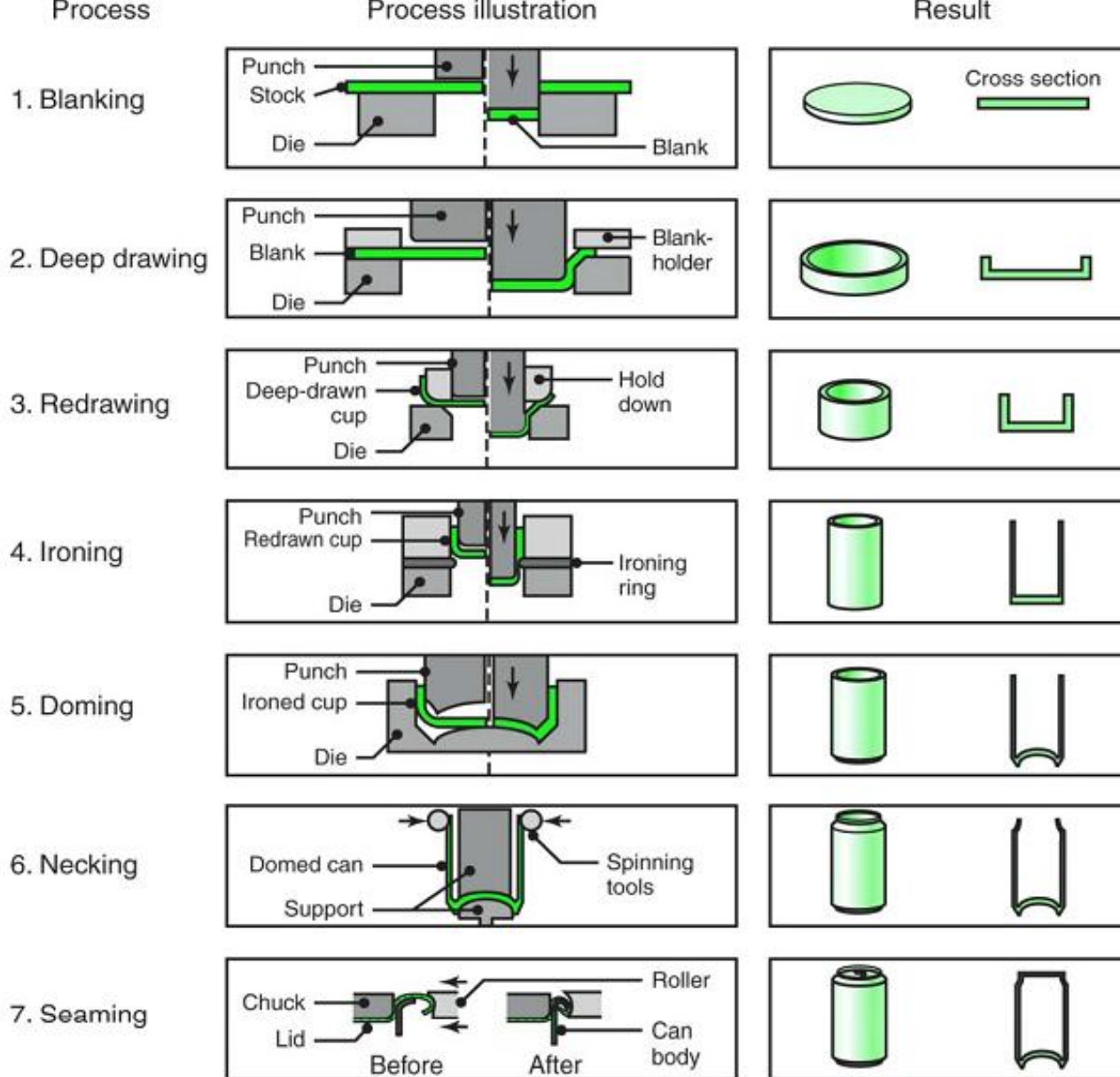
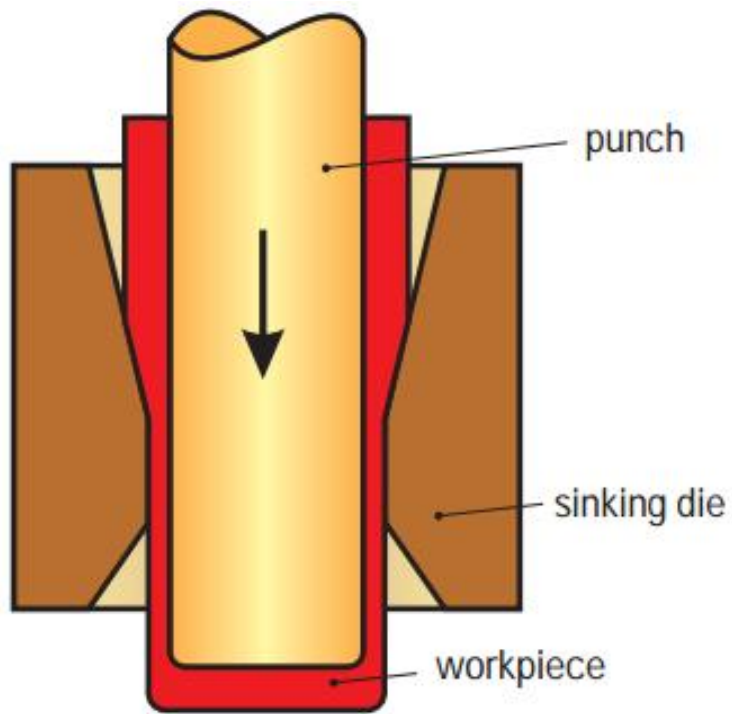


FIGURE 16.30 The metal-forming processes involved in manufacturing a two piece aluminum beverage can manufacturing a two - piece aluminum beverage can .

process involves reducing the wall thickness of deep-drawn or extruded hollow cups and lengthening of drawn cup called ironing



◀ Fig. 2.1.9 Can ironing

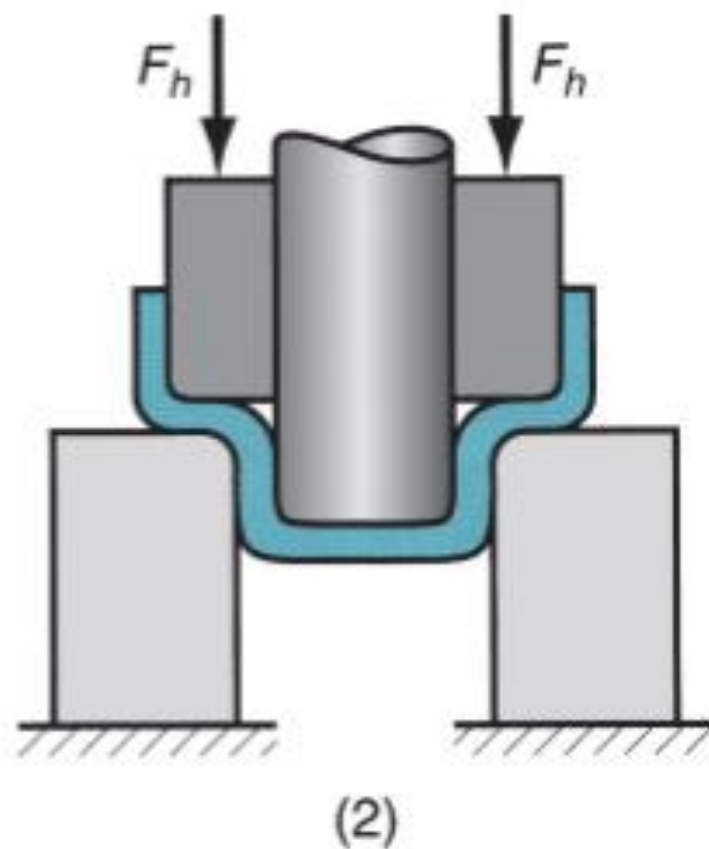
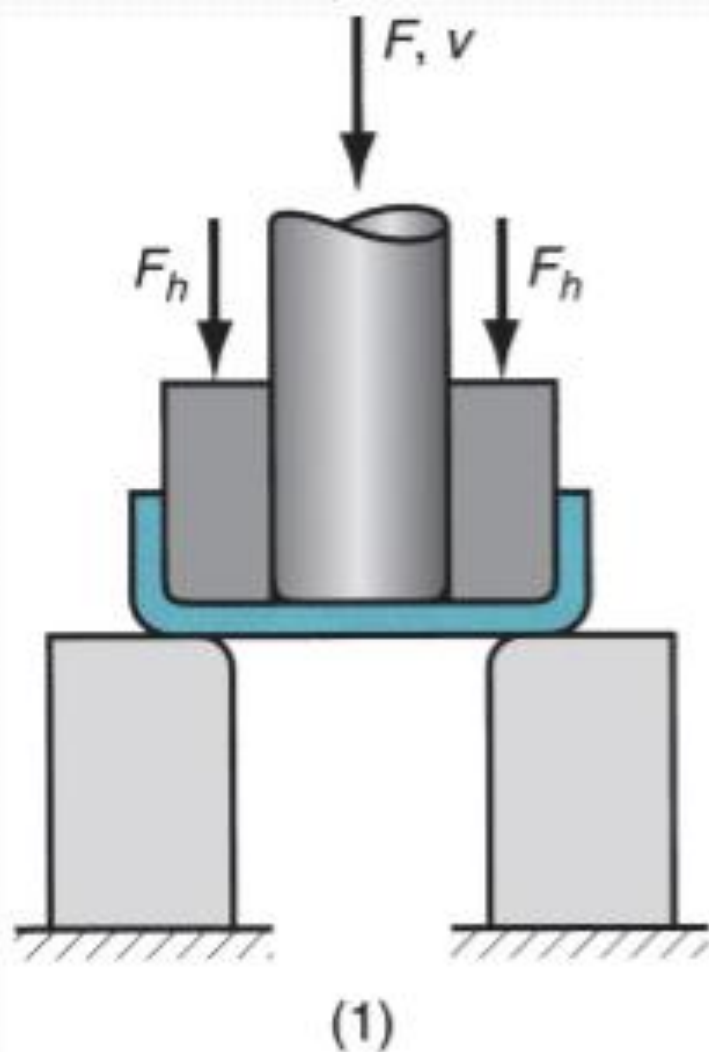
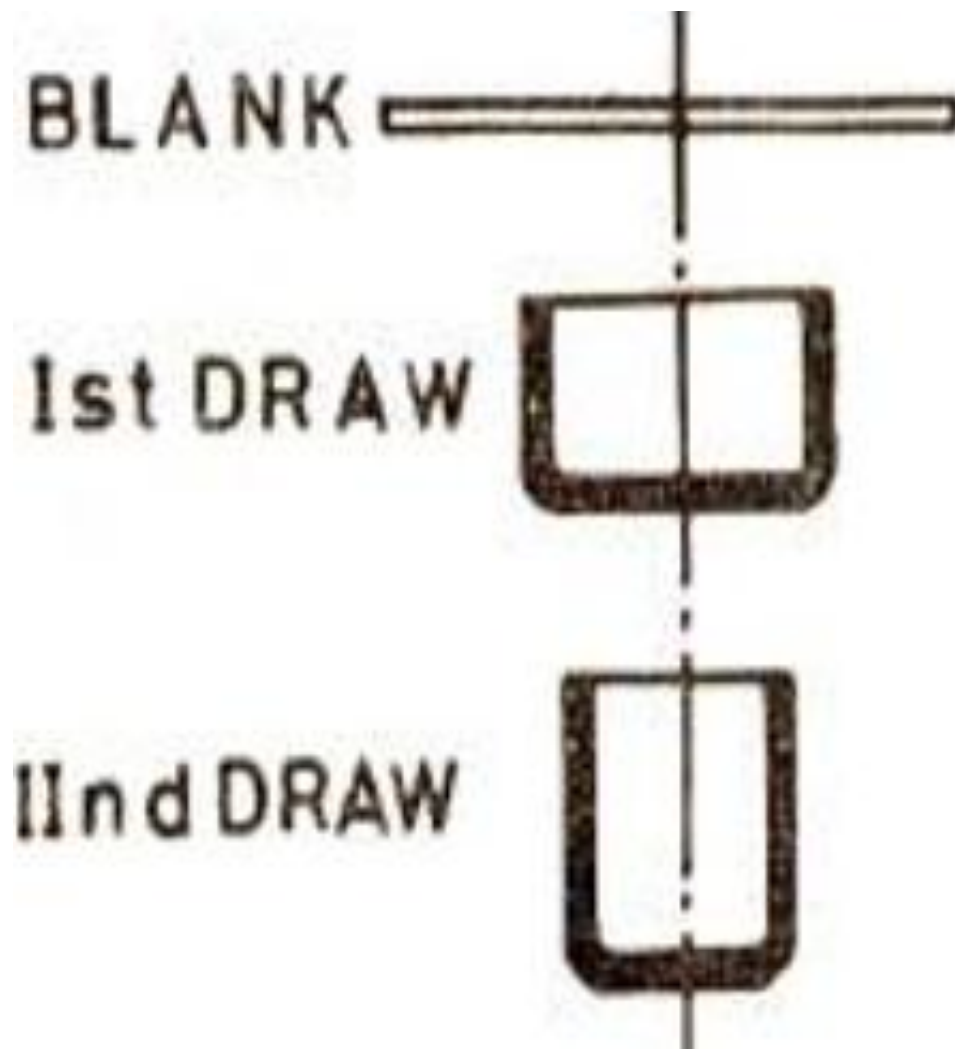


Figure 14.18

Redrawing of a cup



stages of deep drawing of a cup :

Drawing Force

To force required to draw a symmetrical cup is calculated as follows :

P = Drawing force

$$= \pi dt \sigma_y \left[\frac{D}{d} - K \right]$$

where

d = outside diameter of shell

D = Blank diameter

t = Thickness of metal

σ_y = Yield strength of metal

K = Constant to cover bending and friction.

= 0.6 – 0.7 for ductile materials

F = Total force to be exerted by the press

= Drawing force + Blank holding force + Ironing force.

$$F_{\max} = \pi D_p t (UTS) \left[\left(\frac{D_b}{D_p} \right) - 0.7 \right]$$

Example A cup 80 mm diameter and 70 mm deep is to be drawn from steel sheet of deep drawing quality 0.8 mm thick. The yield stress of the steel is 3200 kg/cm². Determine :

- (a) Blank diameter
- (b) Punch diameter for first draw,
- (c) Drawing force.

Solution. D = Blank diameter

d = Cup diameter
= 80 mm.

h = height of cup = 70 mm

$$\begin{aligned} D &= \sqrt{d^2 + 4dh} \\ &= \sqrt{80^2 + 4 \times 80 \times 70} \\ &= 169.7 \text{ mm.} \end{aligned}$$

P = Drawing force

$$\begin{aligned} &= \pi dt \sigma_y \left[\frac{D}{d} - k \right] \\ &= \pi \times 8 \times 0.08 \times 3200 \left[\frac{169.7}{80} - 0.65 \right] \\ &= 9453 \text{ kg} = 9.453 \text{ tons.} \end{aligned}$$

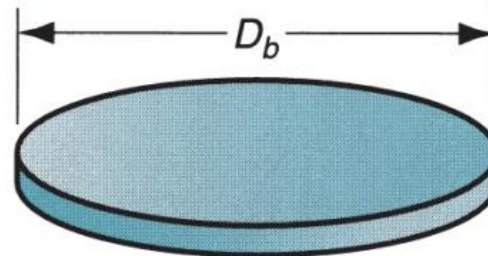
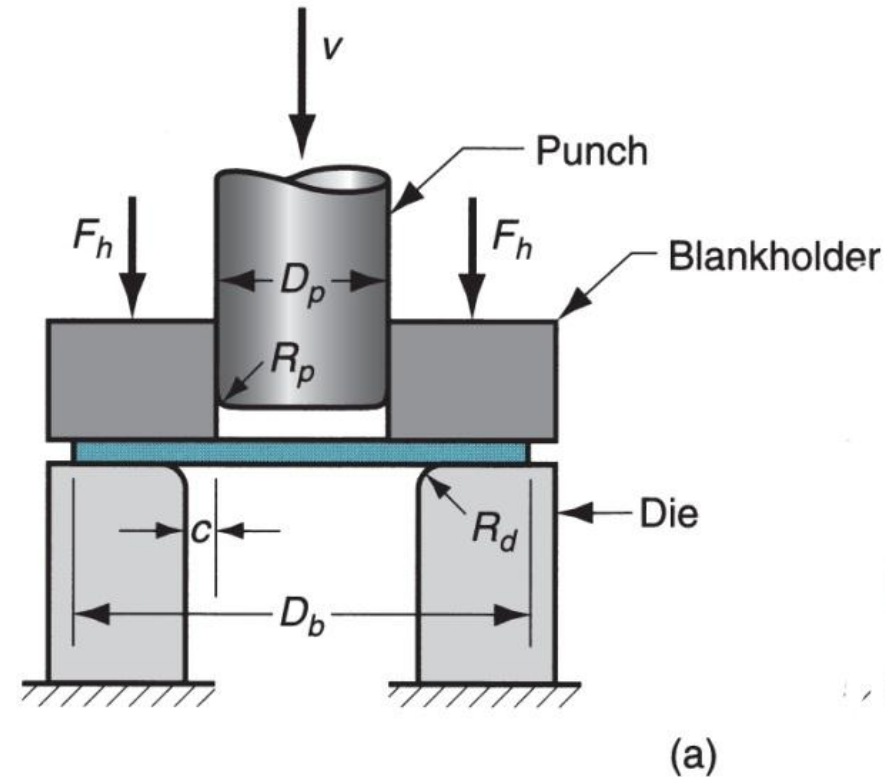
Clearance in Drawing

- Sides of punch and die separated by a
- clearance c given by:

$$c = 1.1 t$$

where t = stock thickness

- In other words, **clearance is about 10% greater than stock thickness**

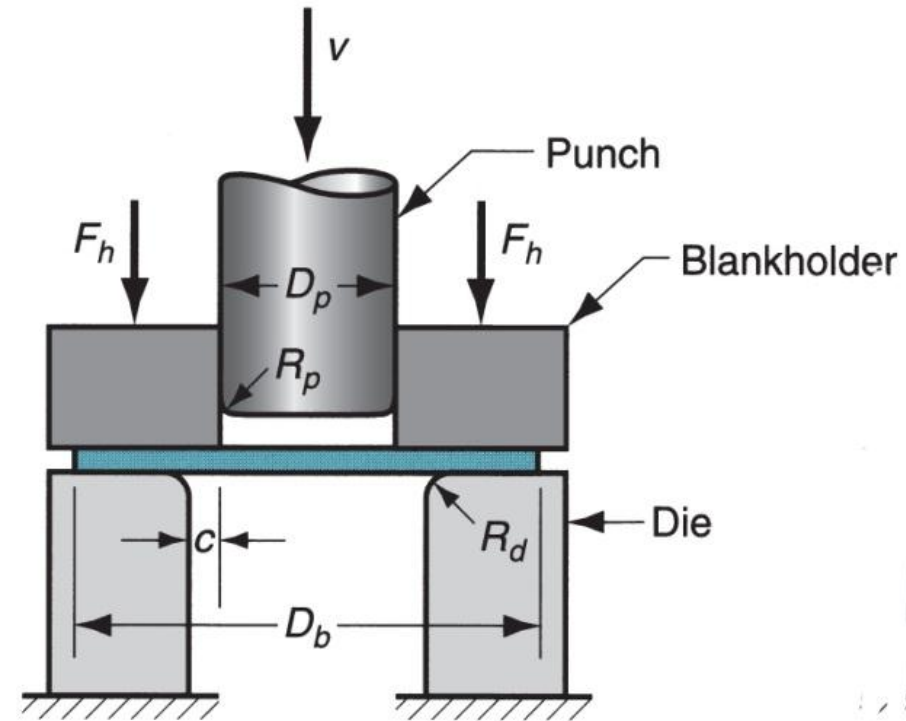


Reduction ratio

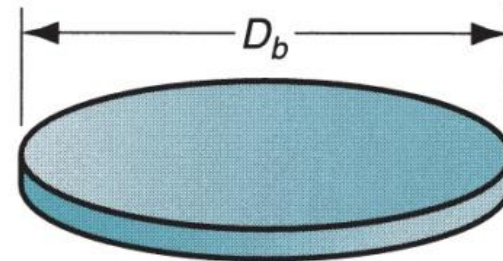
- Defined for
- cylindrical shape:

$$r = \frac{D_b - D_p}{D_b}$$

- Value of r should be less than 0.50



(a)



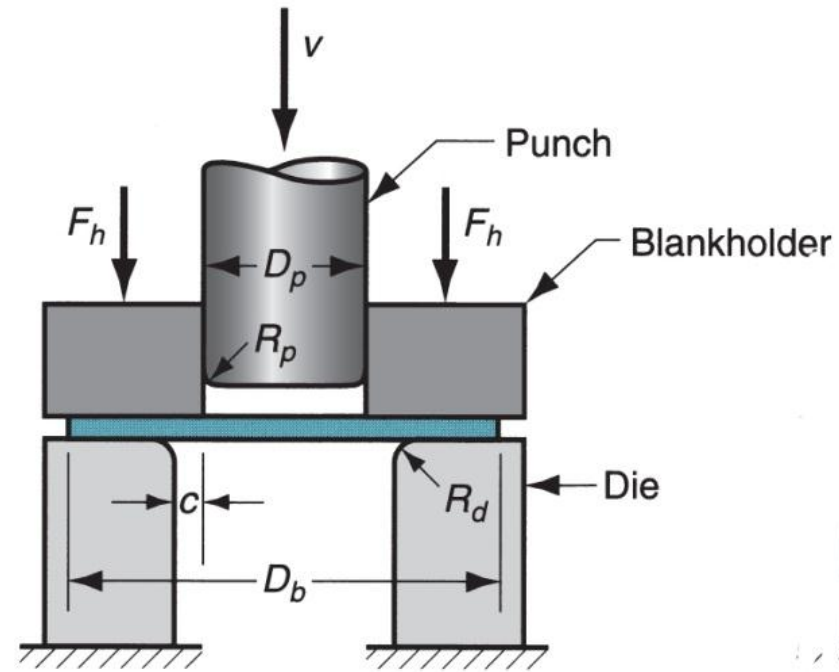
(1)

Thickness-to-Diameter

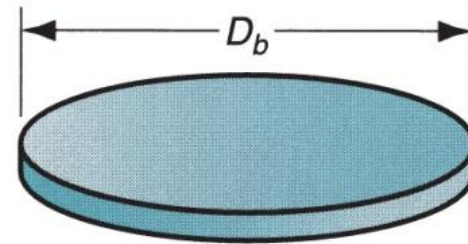
Ratio t/D_b

Thickness of starting blank divided by blank diameter

- Desirable for t/D_b ratio to be greater than 1%
- As t/D_b decreases, tendency for wrinkling increases



(a)



(1)

Common defects

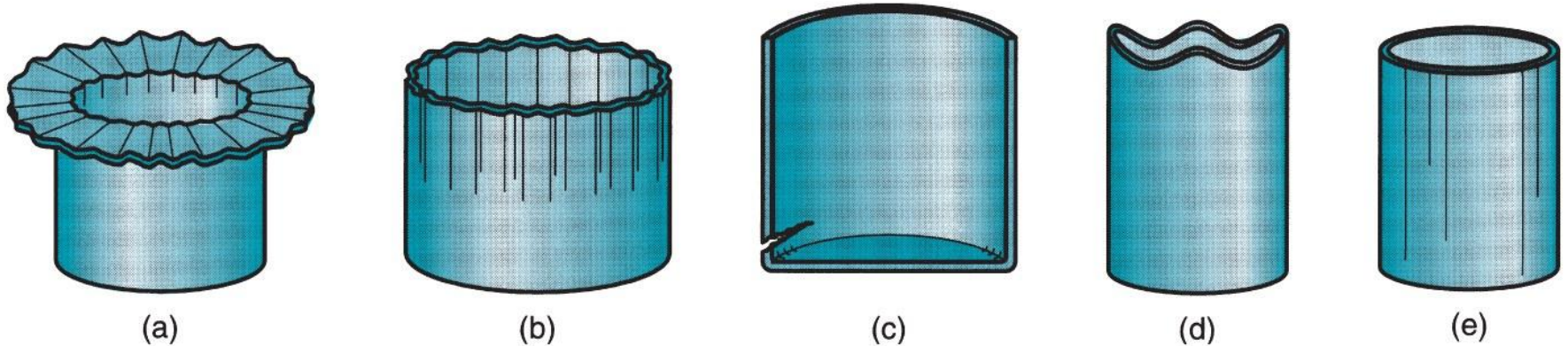


Figure 14.20
© John Wiley & Sons, Inc. All rights reserved.

- (a) Wrinkling in the flange (b) Wrinkling in the wall
(c) Tearing (d) Earing
(e) Scratches

Sheet and Plate Metal Products

- Sheet and plate metal parts for consumer and **industrial products such as**
 - Automobiles and trucks
 - Airplanes
 - Railway cars and locomotives
 - Farm and construction equipment
 - Small and large appliances
 - Office furniture
 - 8/20/2020 Computers and office equipment



Explosive forming



Fine blanking



Spinning



Tube bending.⁶⁹



Stretch forming



Deep drawing

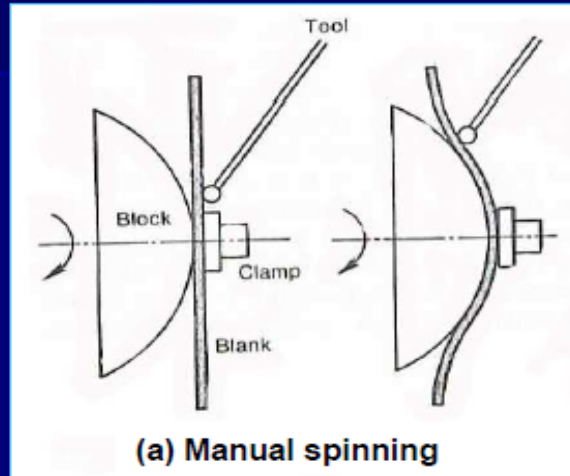


Spinning

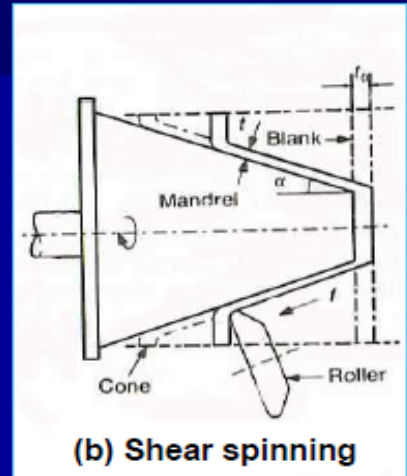
Spinning processes

- Deep parts of circular symmetry such as tank heads, television cones.

Materials: aluminium and alloys, high strength - low alloy steels, copper, brass and alloys, stainless steel,



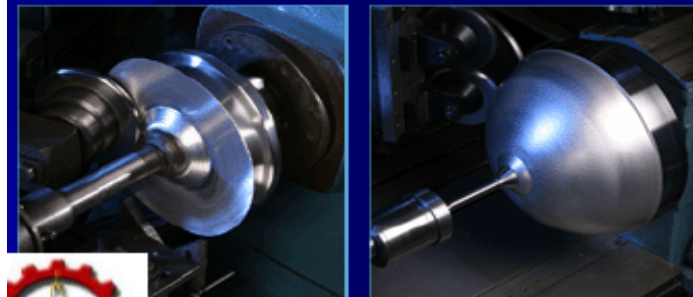
(a) Manual spinning



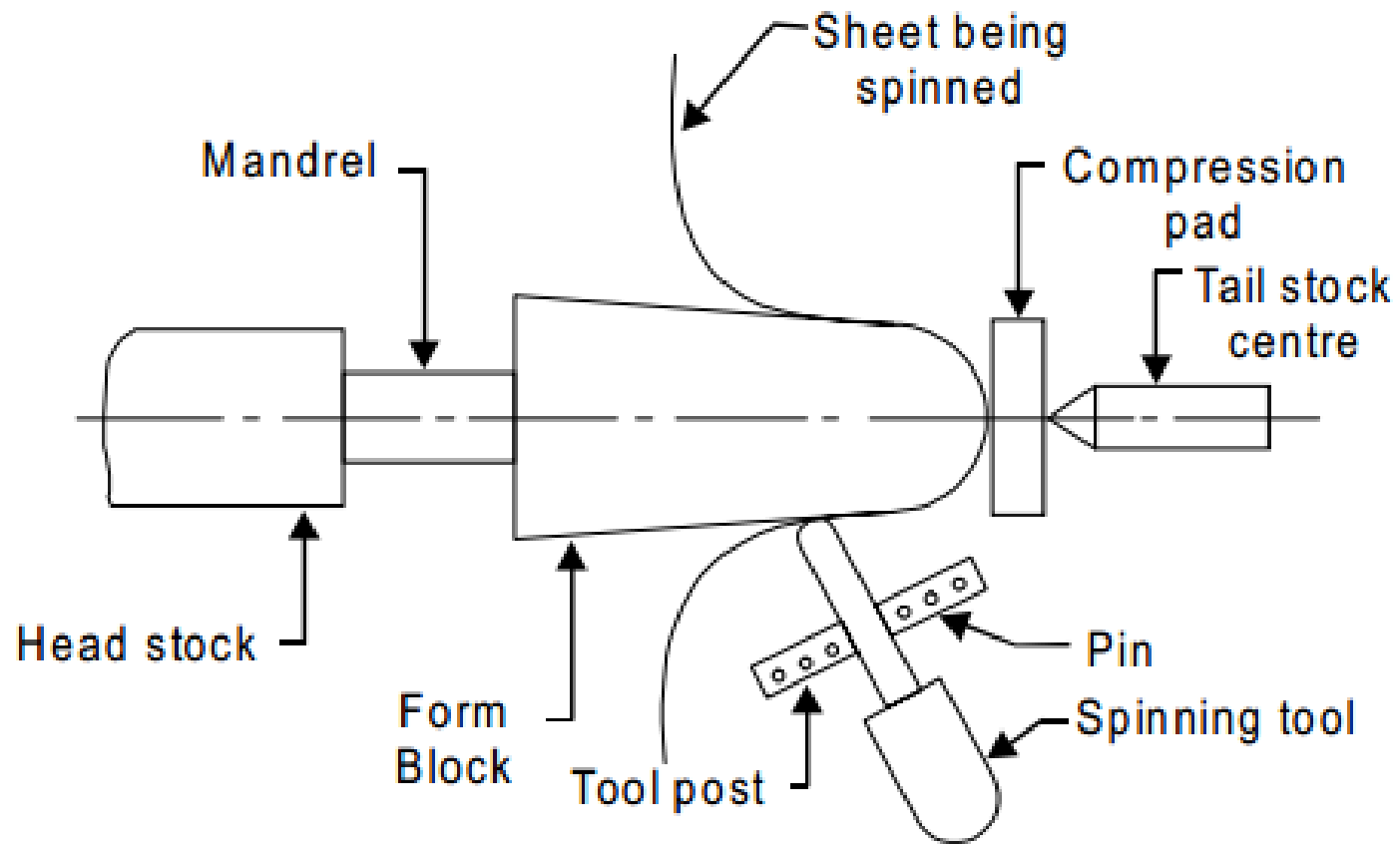
(b) Shear spinning

- The metal blank is clamped against a form block, which is rotated at high speed.
- The blank is progressively formed against the block, by a manual tool or by means of small-diameter work rolls.

Note: (a) no change in **thickness** but **diameter**,
(b) **diameter** equals to blank diameter but **thickness** stays the same.



Conventional Spinning



Hot spinning

Spinning

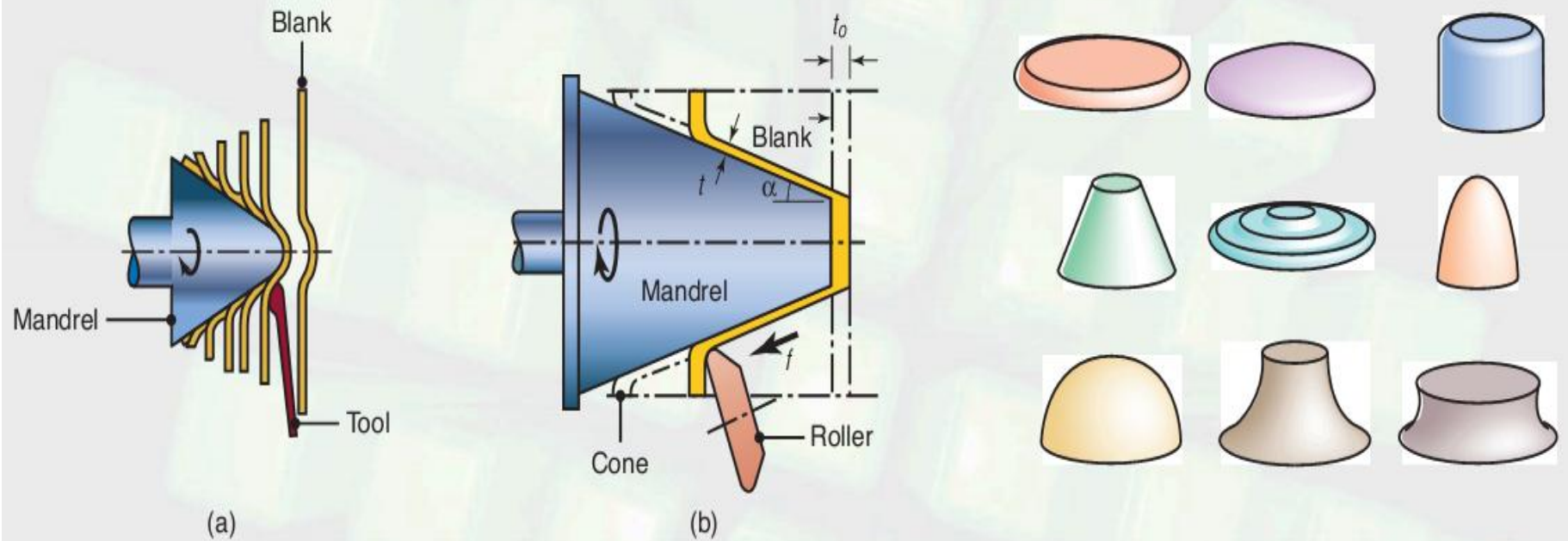


FIGURE 7.36 Schematic illustration of spinning processes:



fig10. Spinning machines

Stretch forming

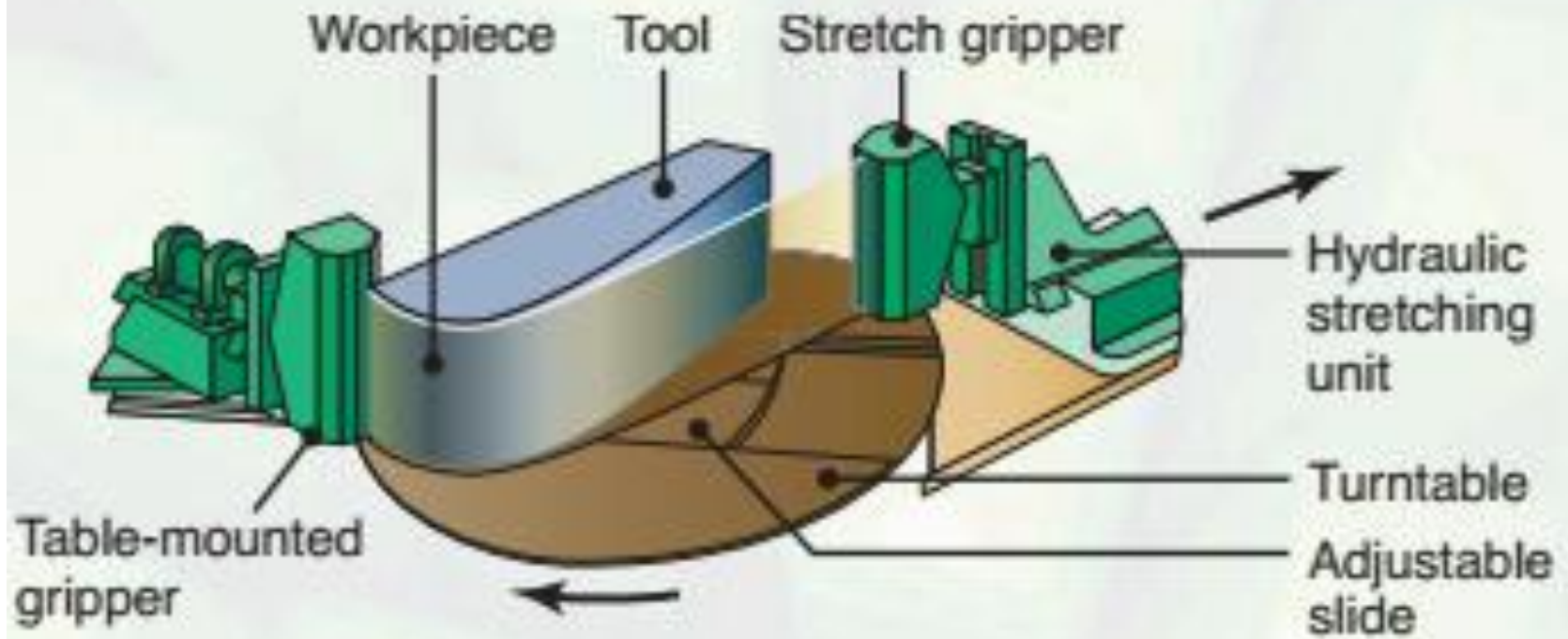
- Forming by using *tensile forces* to stretch the material over a tool or form block.
- used most extensively in the *aircraft industry* to produce parts of large radius of curvature. (normally for uniform cross section).
- required materials with *appreciable ductility*.
- *Springback* is largely eliminated because the stress gradient is relatively uniform.



Stretch forming feasible for aluminium, stainless steel, titanium.



Stretch-Forming

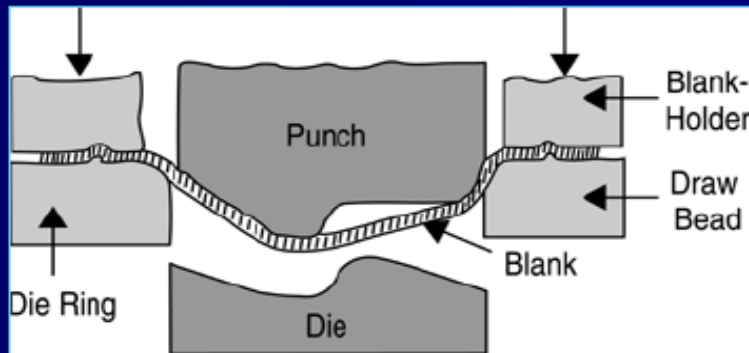


Schematic illustration of a stretch-forming operation. Aluminum skins for aircraft can be made by this process.

Tooling

Basic tools used with a metalworking press are the **punch** and the **die**.

- **Punch** → A **convex tool** for making holes by shearing, or making surface or displacing metal with a hammer.
- **Die** → A **concave die**, which is the female part as opposed to punch which is the male part.



Punch and die in stamping



Punches and dies

Die materials:

- High alloy steels heat treated for the punches and dies.



**Ceramic
extrusion dies**



www.capaalex.co.uk

steel extrusion dies



- **Die design** is at the heart of efficient extrusion production.
- Dies must withstand considerable amount of stresses, thermal shock, and oxidation.



www.capalex.co.uk

Die design

Die design consideration



- **Wall thickness:** different wall thicknesses in one section should be avoided.
- **Simple shapes:** the more simple shape the more cost effective.
- **Symetrical:** more accurate.
- **Sharp or rounded corners:** sharp corners should be avoided.
- **Size to weight ratio:**
- **Tolerlances:** tolerances are added to allow some distortions (industrial standards).

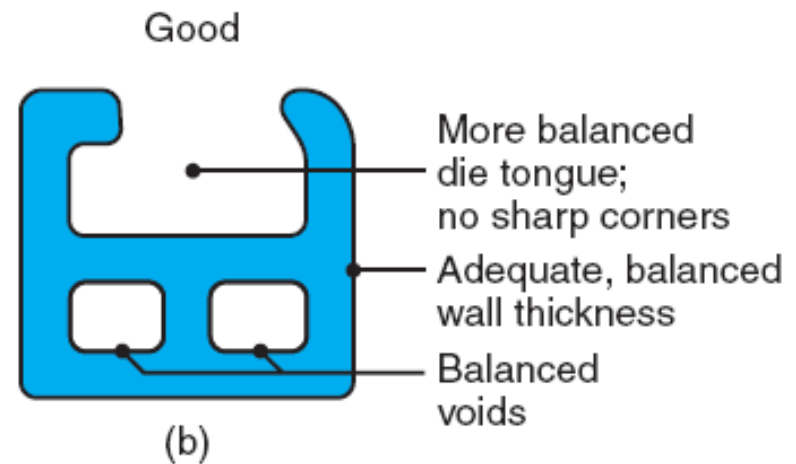
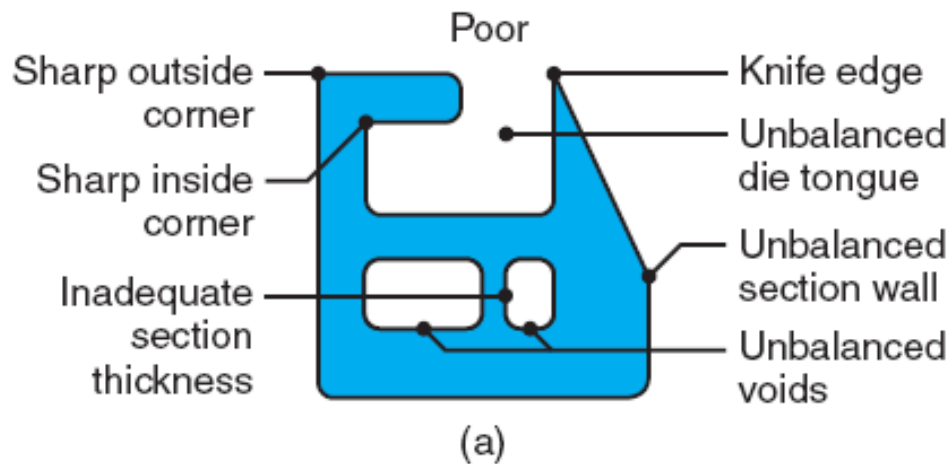
Die materials

- For hot extrusion - tool and alloy steels.
- Important properties of die materials are high wear resistance, high thermal conductivity to remove heat from the process.
- For cold extrusion - tool steels and cemented carbides.
- Carbides are used when high production rates, long die life, and good dimensional control are expected.

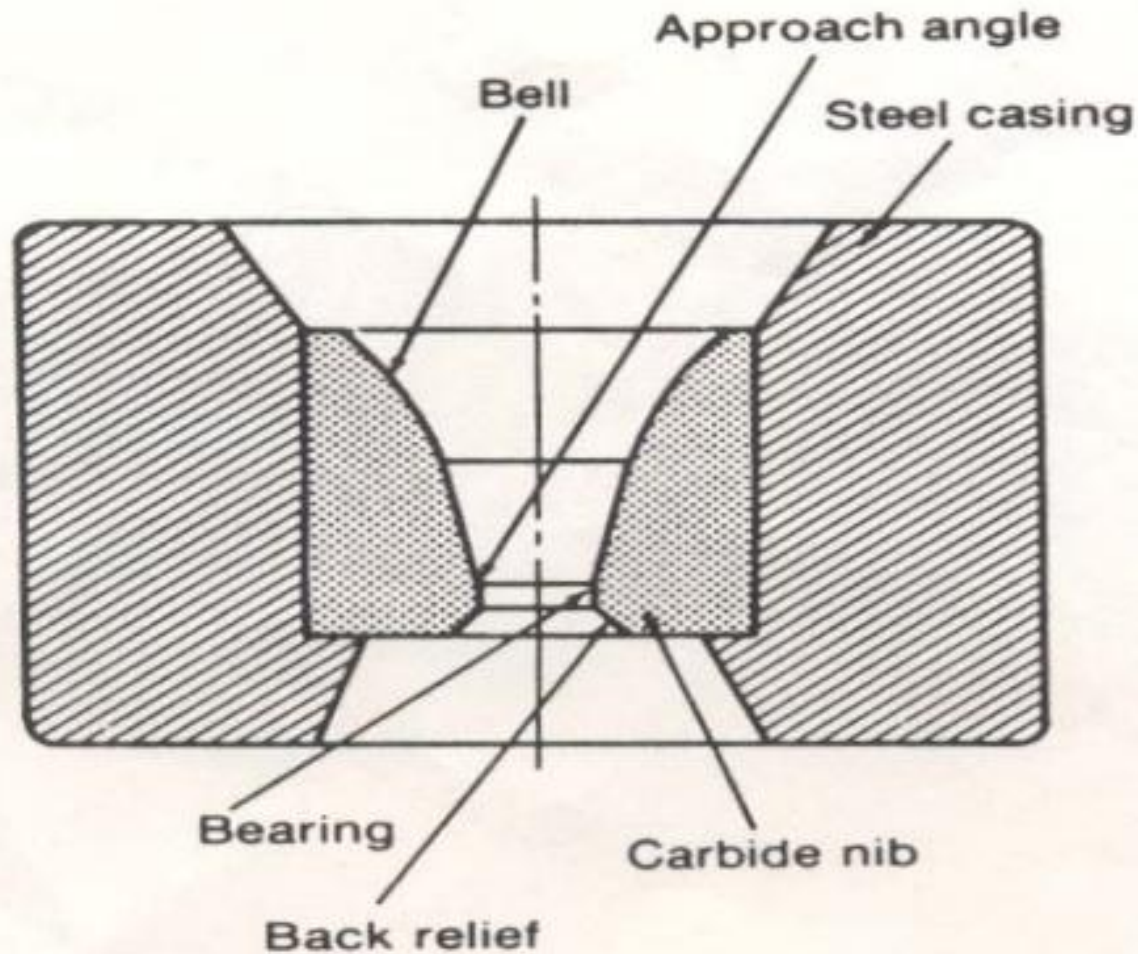
Hot Extrusion

Die Design

- Guidelines for proper die design in extrusion:
 - Symmetry of cross section
 - Avoidance of sharp corners
 - Avoidance of changes in die dimensions



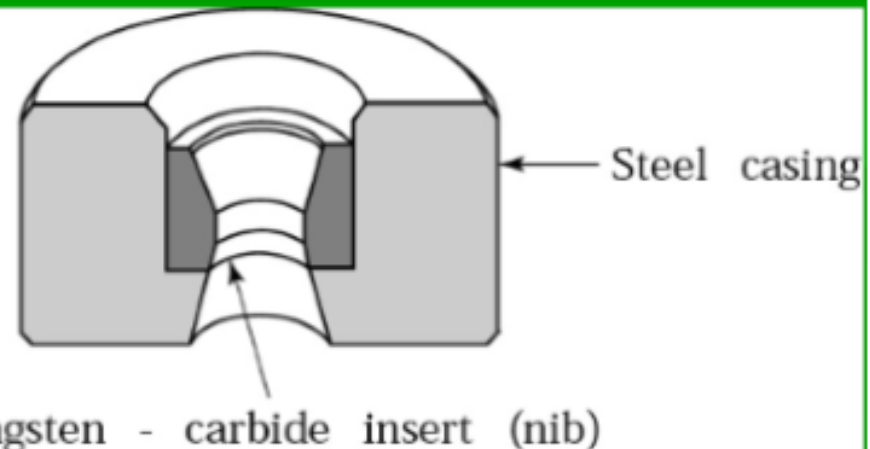
Dies for wire drawing



Die for Round Drawing



Figure 15.21 Tungsten- carbide die insert in a steel casing. Diamond dies, used in drawing thin wire, are encased in a similar manner.



- Purpose of the bearing surface (*land*) is to set the final diameter of the product (*sizing*)

End of chapter one!!

Thank you for your attention

