Chapter 20
SHEET METALWORKING

- Cutting Operations
- Bending Operations
- Drawing
- Other Sheet Metal Forming Operations
- Dies and Presses for Sheet Metal Processes
- Sheet Metal Operations Not Performed on Presses
- Bending of Tube Stock

Sheet Metalworking Defined

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)
- Thickness of plate stock > 6 mm
- Operations usually performed as cold working

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
  - Computers and office equipment

Advantages of Sheet Metal Parts

- High strength
- Good dimensional accuracy
- Good surface finish
- Relatively low cost
- For large quantities, economical mass production operations are available

Sheet Metalworking Terminology

1. "Punch-and-die"
   – Tooling to perform cutting, bending, and drawing
2. "Stamping press"
   – Machine tool that performs most sheet metal operations
3. "Stampings"
   – Sheet metal products

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

Cutting

Shearing between two sharp cutting edges

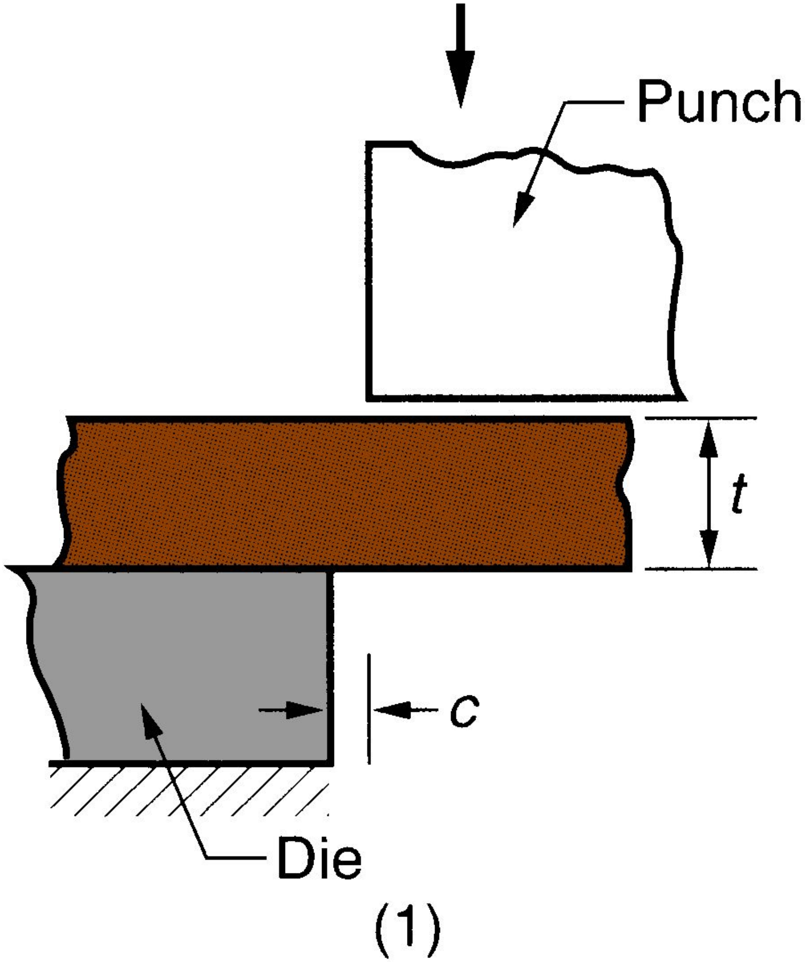


Figure 20.1 ‑ Shearing of sheet metal between two cutting edges:

(1) just before the punch contacts work

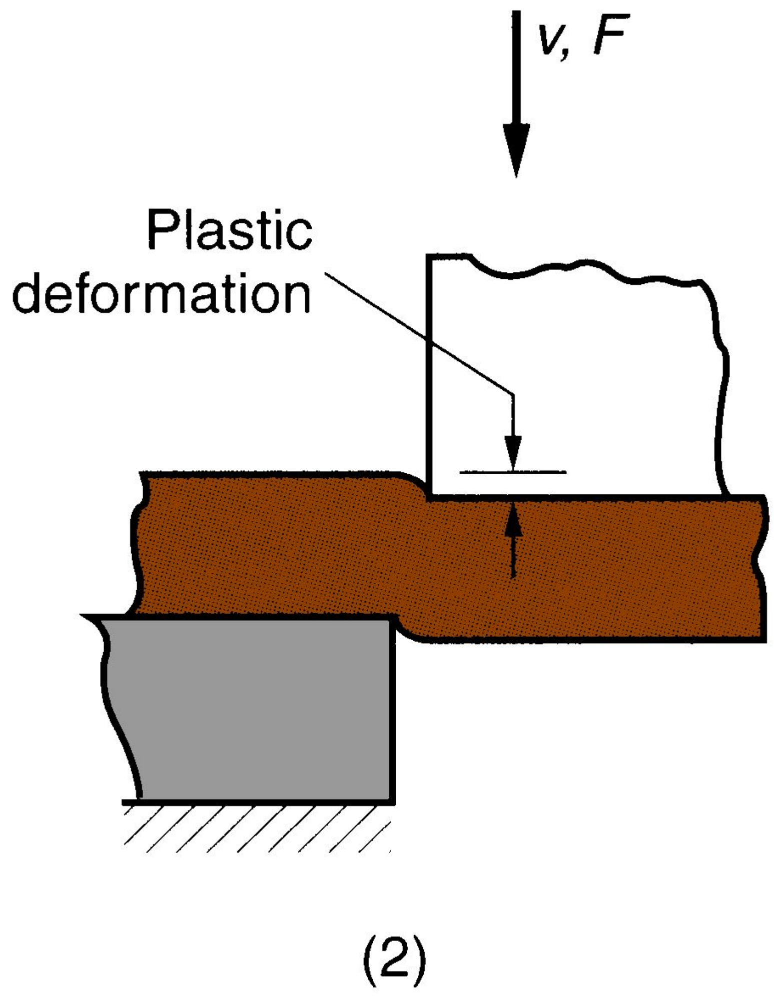


Figure 20.1 ‑ Shearing of sheet metal between two cutting edges:

(2) punch begins to push into work, causing plastic deformation

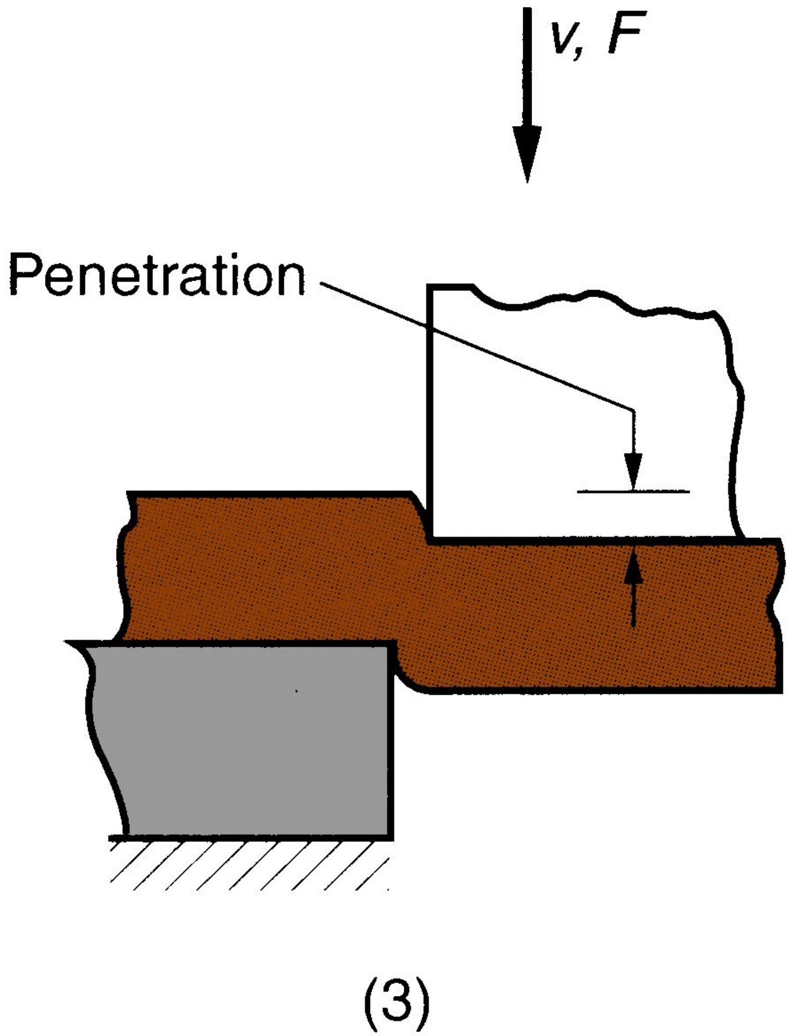


Figure 20.1 ‑ Shearing of sheet metal between two cutting edges:

(3) punch compresses and penetrates into work causing a smooth cut surface

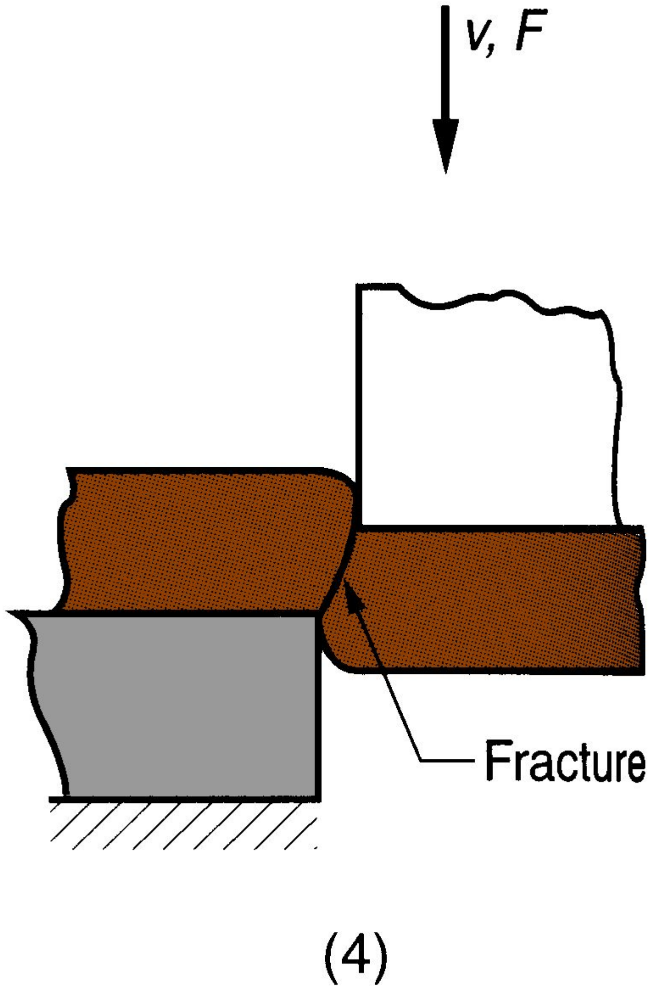


Figure 20.1 ‑ Shearing of sheet metal between two cutting edges:

(4) fracture is initiated at the opposing cutting edges which separates the sheet

Shearing, Blanking, and Punching

Three principal operations in pressworking that cut sheet metal:

- Shearing
- Blanking
- Punching

Shearing

Sheet metal cutting operation along a straight line between two cutting edges

- Typically used to cut large sheets into smaller sections for subsequent operations

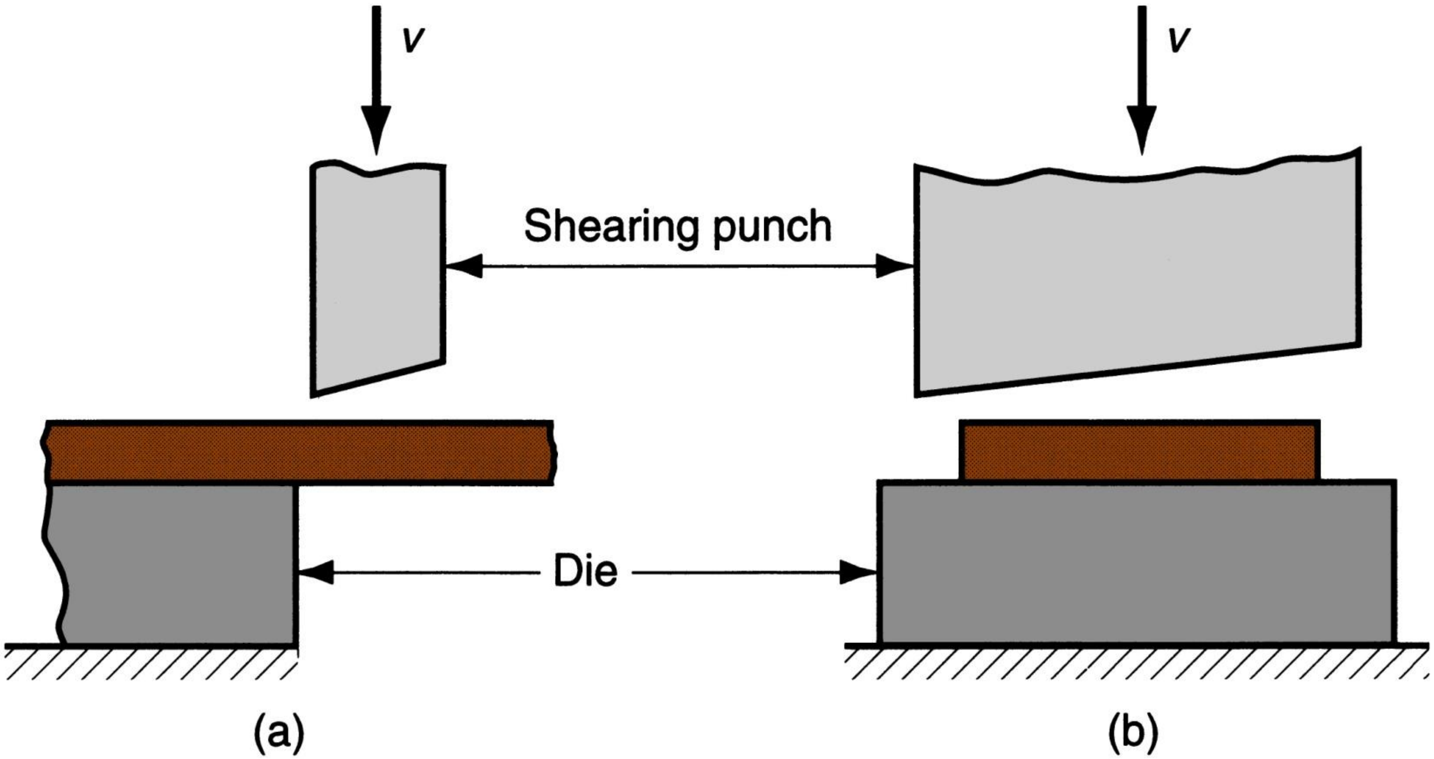


Figure 20.3 ‑ Shearing operation:
(a) side view of the shearing operation
(b) front view of power shears equipped with inclined upper cutting blade Symbol *v* indicates motion

Blanking and Punching

Blanking - sheet metal cutting to separate piece from surrounding stock

- Cut piece is the desired part, called a *blank*

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

- Remaining stock is the desired part

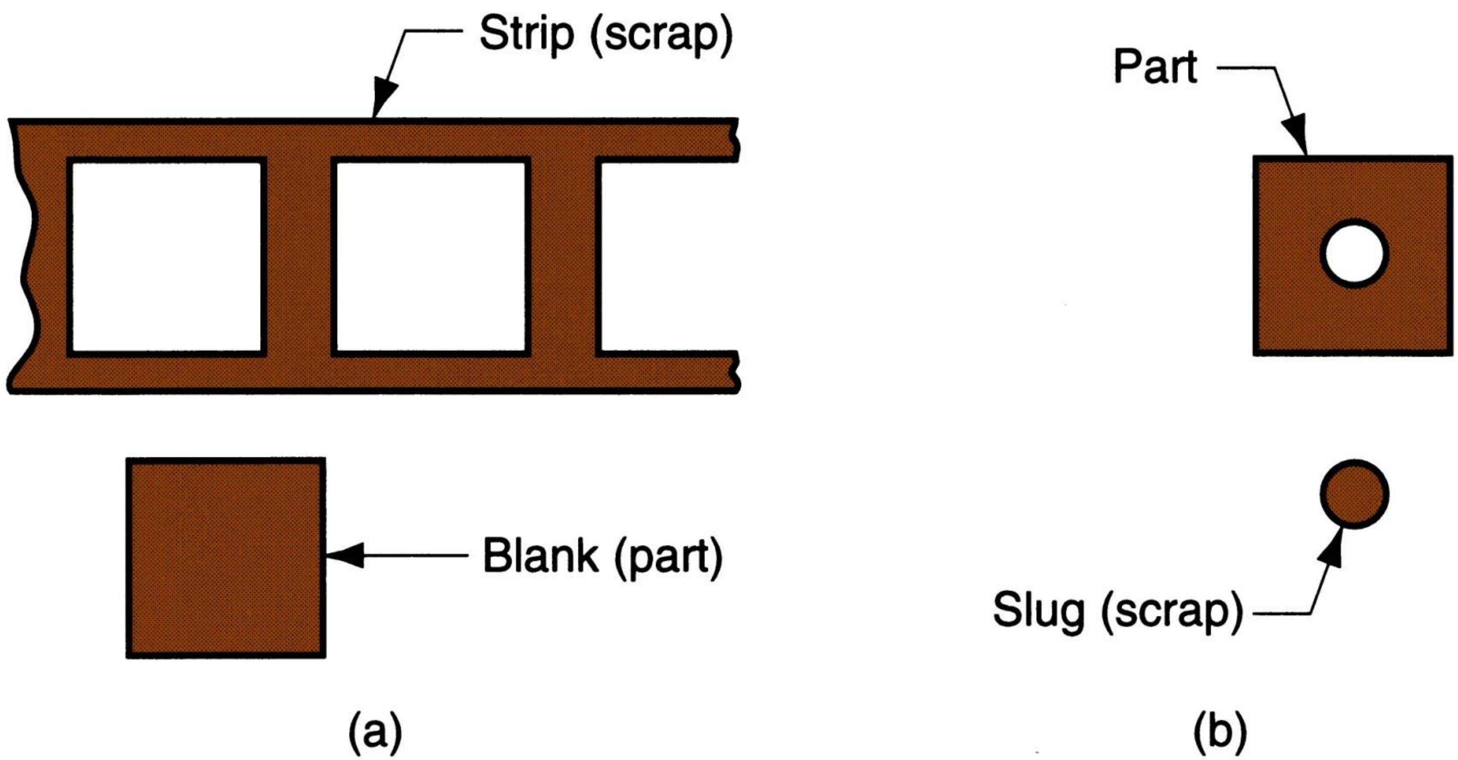


Figure 20.4 ‑ (a) Blanking and (b) punching

Clearance in Sheet Metal Cutting

Distance between the punch and die

- Typical values range between 4% and 8% of stock thickness
  - If too small, fracture lines pass each other, causing double burnishing and larger force
  - If too large, metal is pinched between cutting edges and excessive burr results

Clearance in Sheet Metal Cutting

- 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

Metal group	*a*
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

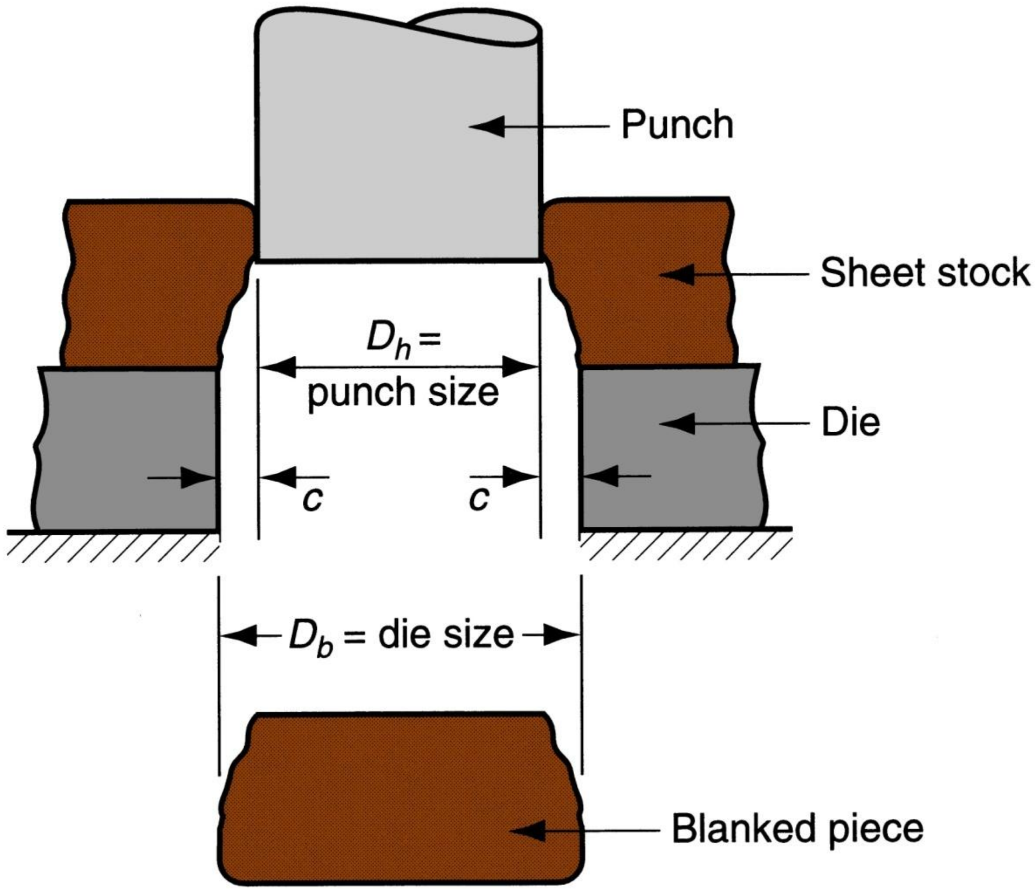


Figure 20.6 ‑ Die size determines blank size D_b; punch size determines hole size D_h.; c = clearance

Angular Clearance

Purpose: allows slug or blank to drop through die

- Typical values: 0

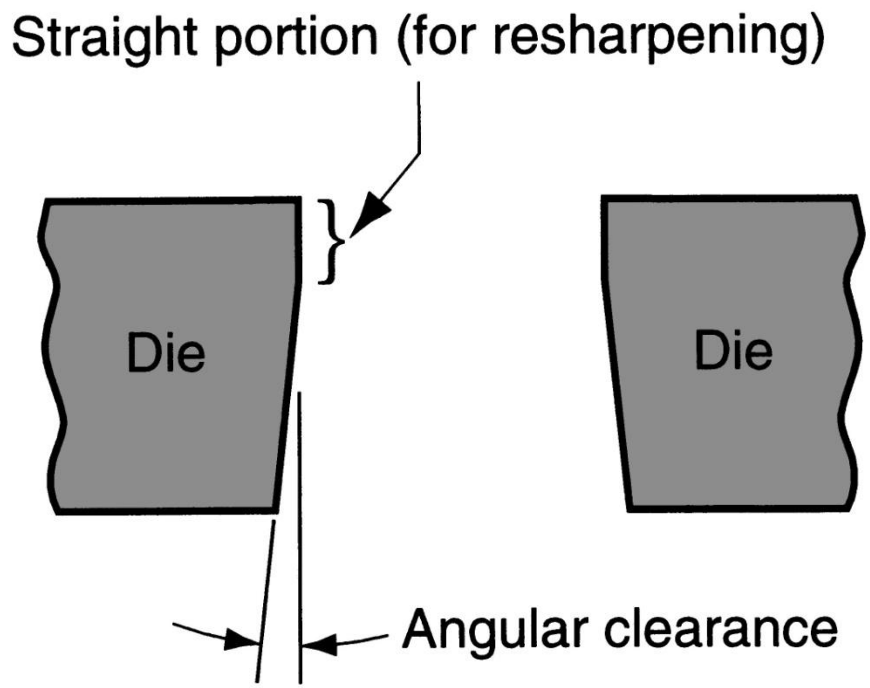


Figure 20.7 ‑ Angular clearance

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

Bending

Straining sheetmetal around a straight axis to take a permanent bend

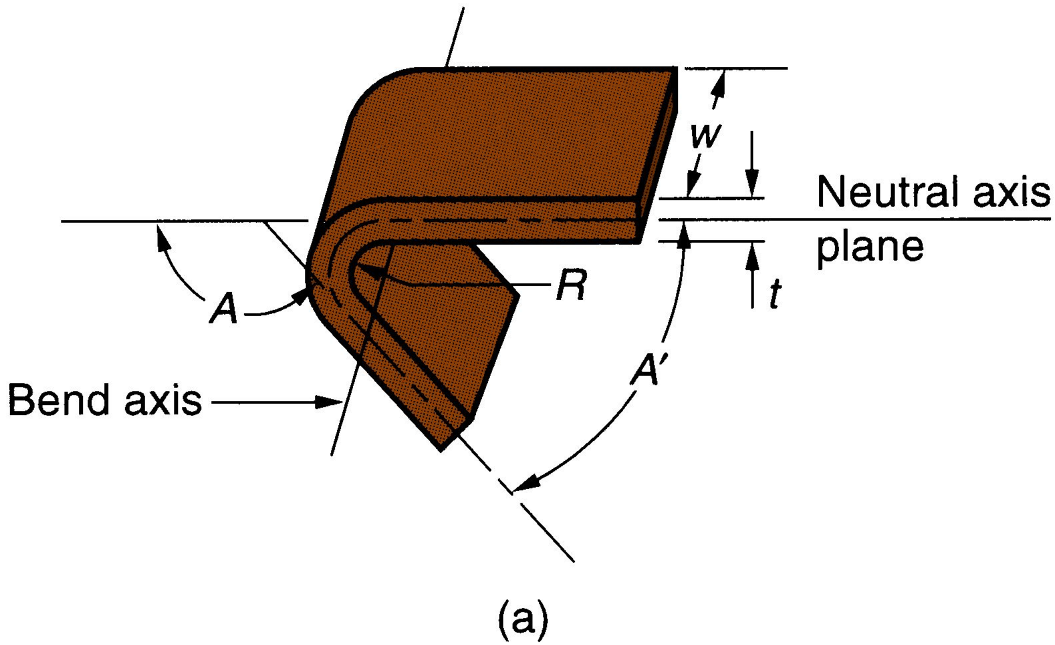
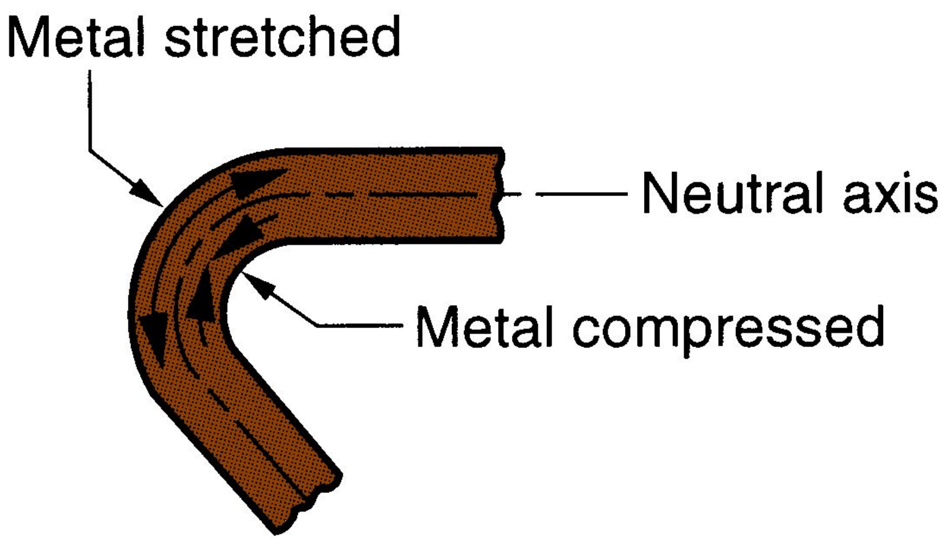


Figure 20.11 ‑ (a) Bending of sheet metal

Metal on inside of neutral plane is compressed, while metal on outside of neutral plane is stretched



(b)

Figure 20.11 ‑ (b) both compression and tensile elongation of the metal occur in bending

Types of Sheetmetal Bending

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

V-Bending

- For low production
- Performed on a *press brake*
- V-dies are simple and inexpensive

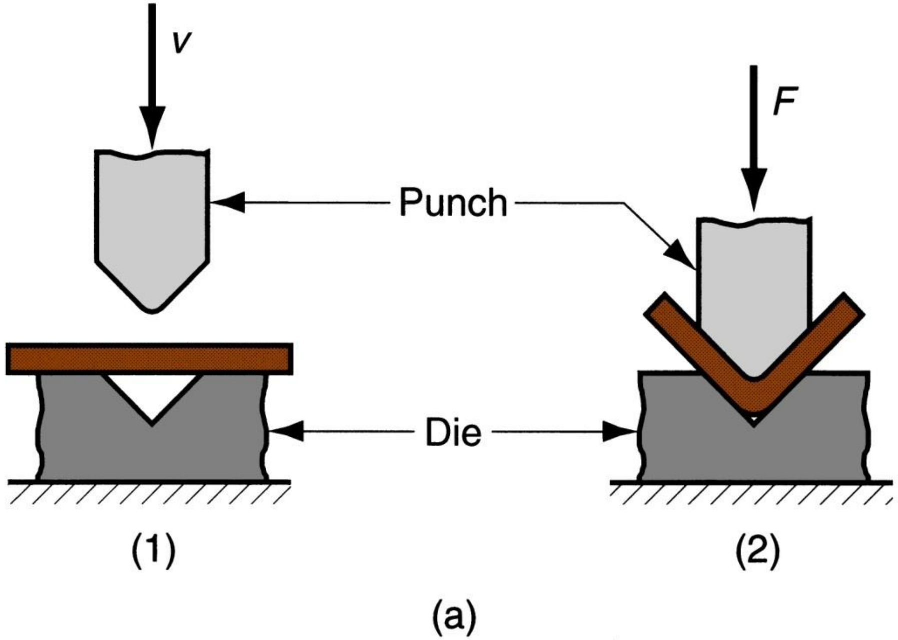


Figure 20.12 - (a) V-bending

Edge Bending

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

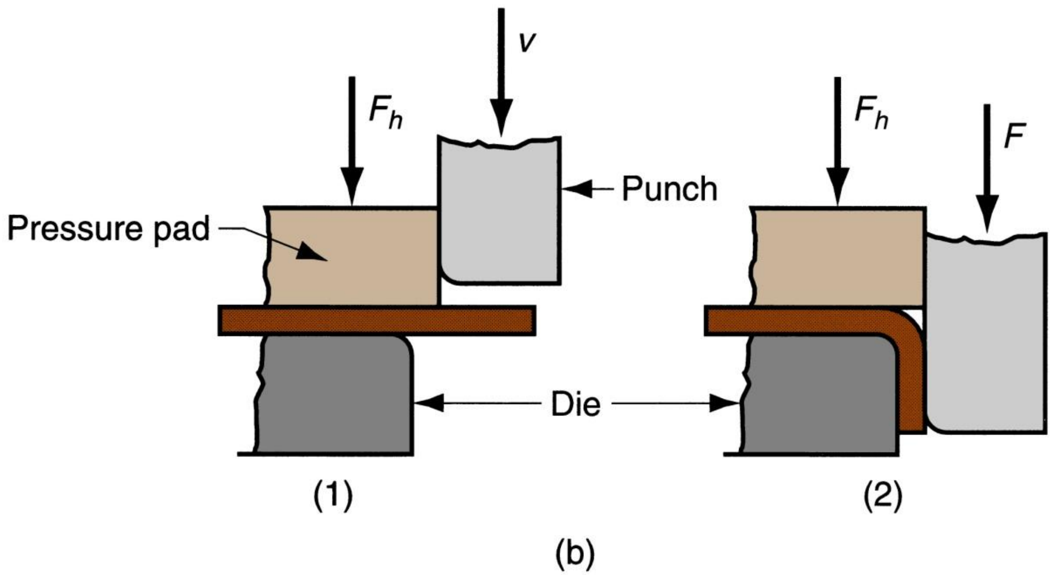


Figure 20.12 - (b) edge bending

Stretching during Bending

- If bend radius is small relative to stock thickness, metal tends to stretch during bending
- Important to estimate amount of stretching, so that final part length = specified dimension
- Problem: to determine the length of neutral axis of the part before bending

Bend Allowance Formula

$$BA = 2\pi \frac{A}{360}(R + K_{ba}t)$$

where BA = bend allowance; A = bend angle; R= bend radius; t = stock thickness; and K_{ba} is factor to estimate stretching

- If R < 2t, K_{ba} = 0.33
- If R ≥ 2t, K_{ba} = 0.50

Springback in Bending

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

- Reason for springback:
  - When bending pressure is removed, elastic energy remains in bent part, causing it to recover partially toward its original shape

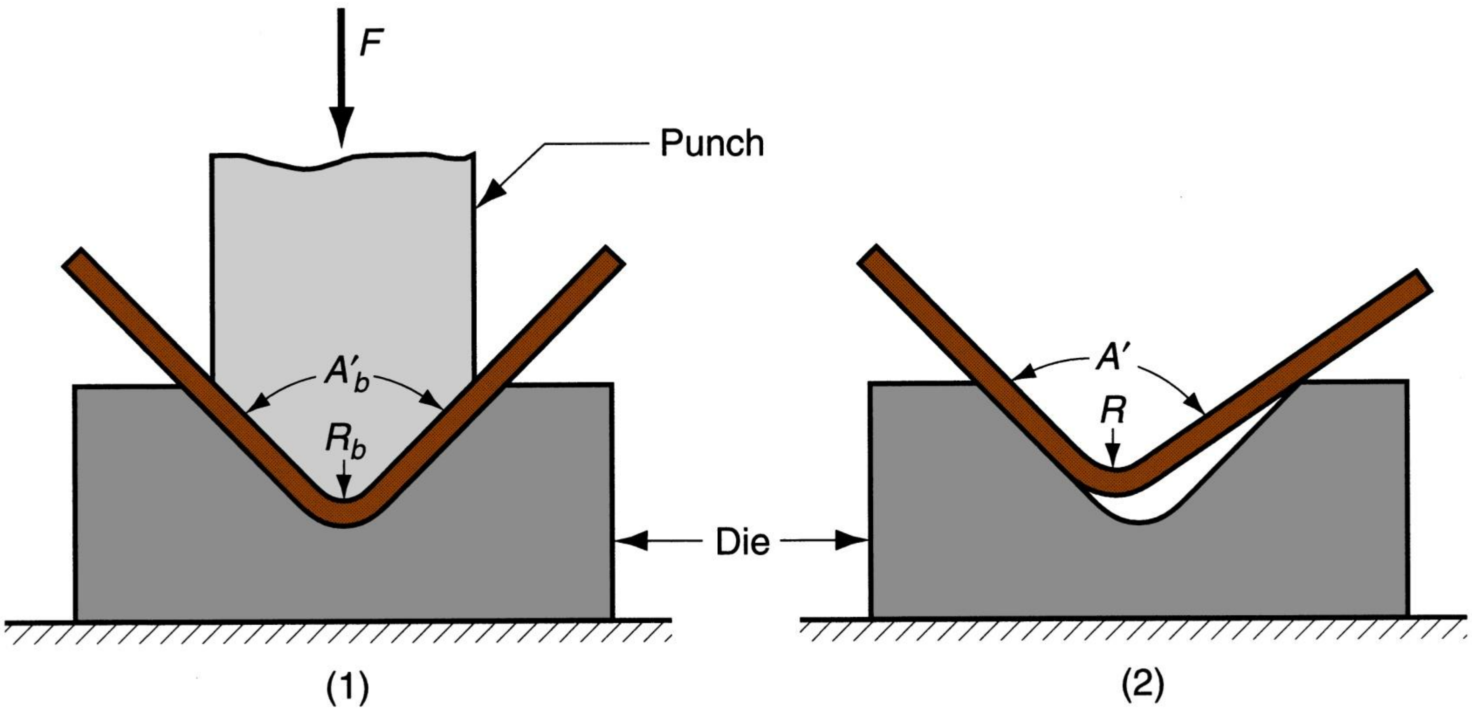


Figure 20.13 ‑ Springback in bending shows itself as a decrease in bend angle and an increase in bend radius: (1) during bending, the work is forced to take the radius R_b and included angle A_b' of the bending tool (punch in V‑bending), (2) after punch is removed, the work springs back to radius R and angle A'

Bending Force

Maximum bending force estimated as follows:

$$F = \frac{K_{bf} TS w t^2}{D}$$

where F = bending force; TS = tensile strength of sheet metal; w = part width in direction of bend axis; and t = stock thickness. For V- bending, K_{bf} = 1.33; for edge bending, K_{bf} = 0.33

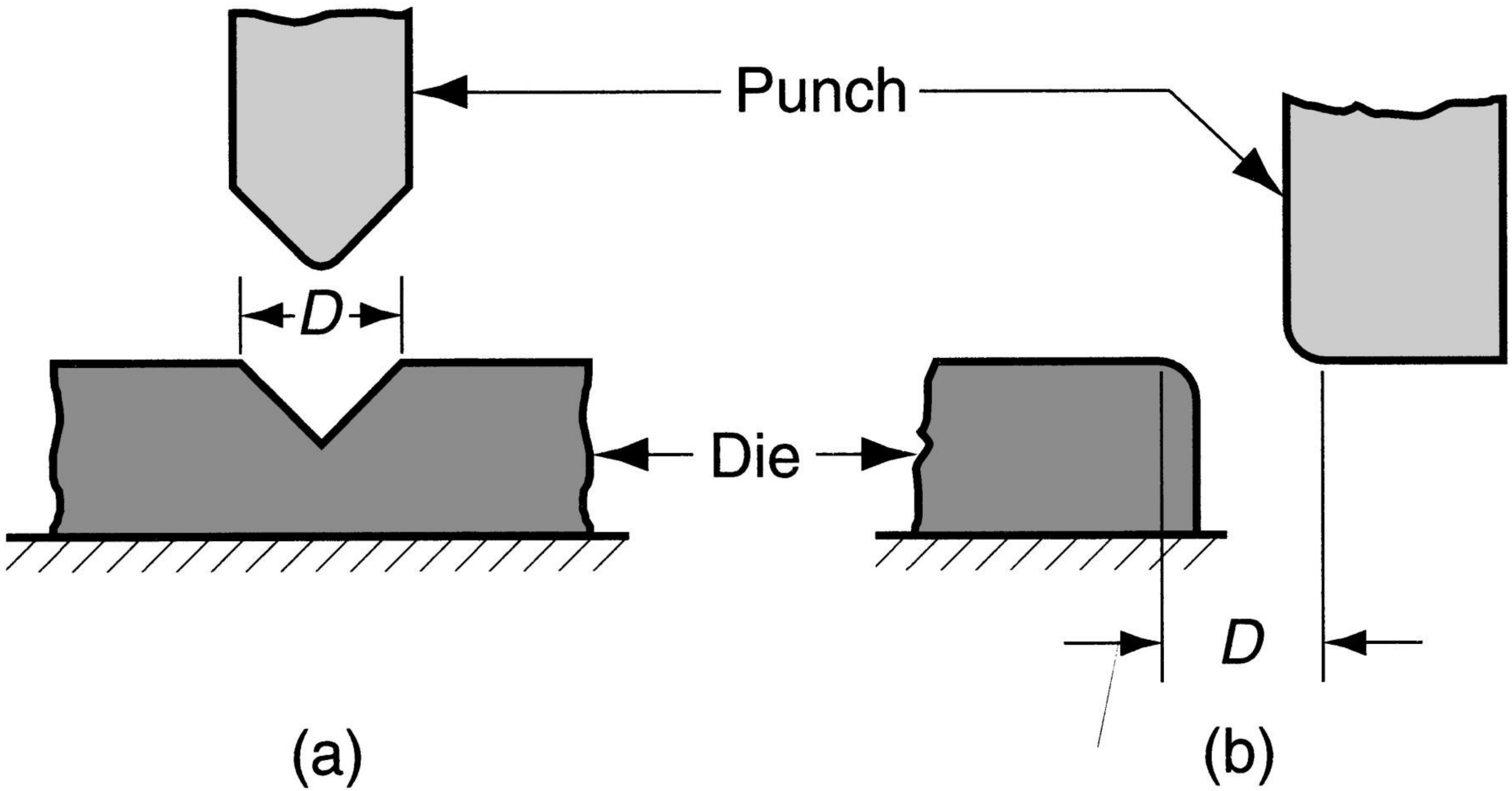


Figure 20.14 ‑ Die opening dimension D: (a) V‑die, (b) wiping die

Drawing

Sheet metal forming to make cup-shaped, box-shaped, or other complex-curved, hollow-shaped parts

- Sheet metal blank is positioned over die cavity and then punch pushes metal into opening
- Products: beverage cans, ammunition shells, automobile body panels

Figure 20.19 -

(a) Drawing of a cup-shaped part:

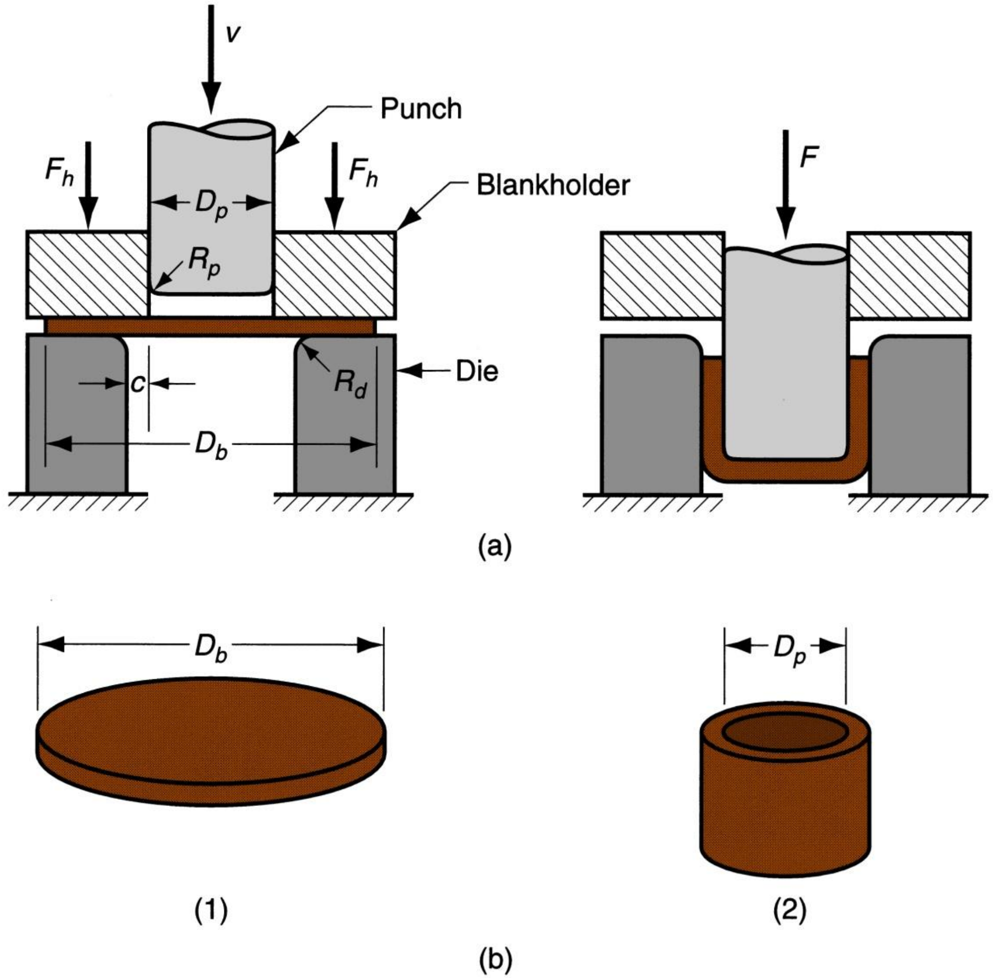
(1) start of operation before punch contacts work

(2) near end of stroke

(b) Corresponding workpart:

(1) starting blank

(2) drawn part



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 = about 10% greater than stock thickness

Drawing Ratio *DR*

Most easily defined for cylindrical shape:

$$DR = \frac{D_b}{D_p}$$

where D_b = blank diameter; and D_p = punch diameter

- Indicates severity of a given drawing operation
  - Upper limit = 2.0

Reduction *r*

- Again, defined for cylindrical shape:

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

- Value of *r* should be less than 0.50

Thickness-to-Diameter Ratio

Thickness of starting blank divided by blank diameter

Thickness-to-diameter ratio = t/D_b

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

Blank Size Determination

- For final dimensions of drawn shape to be correct, starting blank diameter D_b must be right
- Solve for D_b by setting starting sheet metal blank volume = final product volume
- To facilitate calculation, assume negligible thinning of part wall

Shapes other than Cylindrical Cups

- Square or rectangular boxes (as in sinks),
- Stepped cups,
- Cones,
- Cups with spherical rather than flat bases,
- Irregular curved forms (as in automobile body panels)

- Each of these shapes presents its own unique technical problems in drawing

Other Sheet Metal Forming on Presses

Other sheet metal forming operations performed on conventional presses

- Operations performed with metal tooling
- Operations performed with flexible rubber tooling

Ironing

- Makes wall thickness of cylindrical cup more uniform
- Examp tillery shells

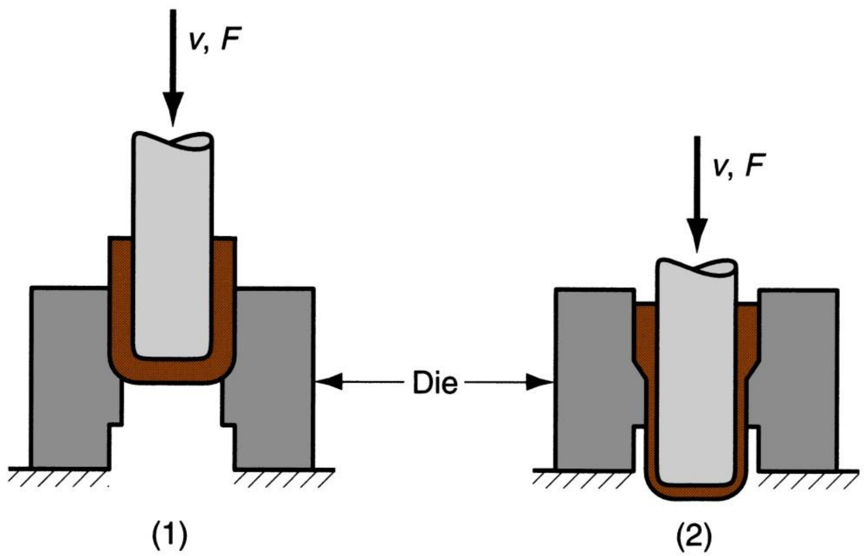


Figure 20.25 ‑ Ironing to achieve a more uniform wall thickness in a drawn cup: (1) start of process; (2) during process

Note thinning and elongation of walls

Embossing

- Used to create indentations in sheet, such as raised (or indented) lettering or stren

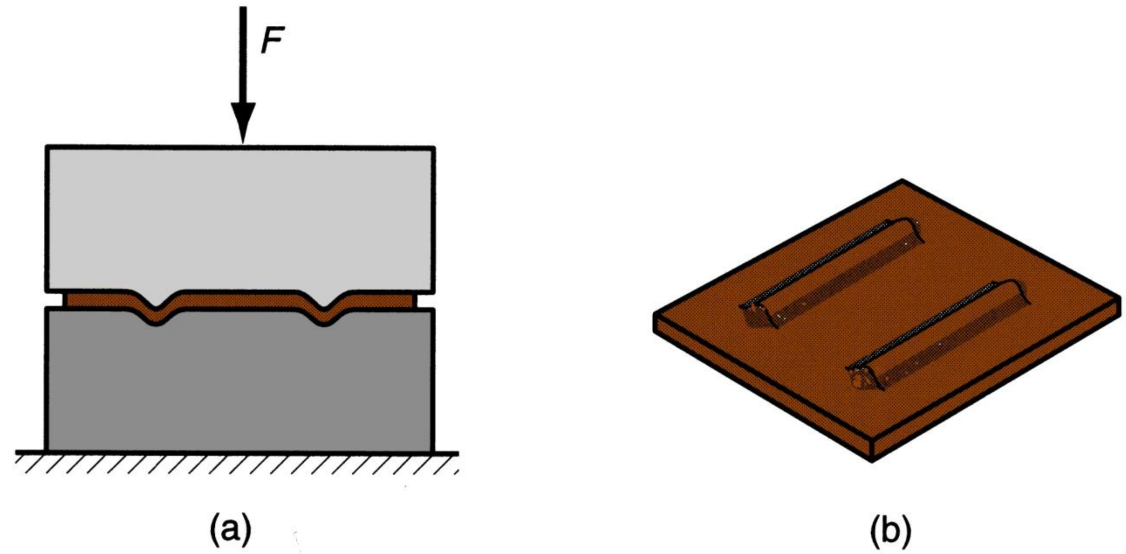


Figure 20.26 ‑ Embossing: (a) cross‑section of punch and die configuration during pressing; (b) finished part with embossed ribs

Guerin Process

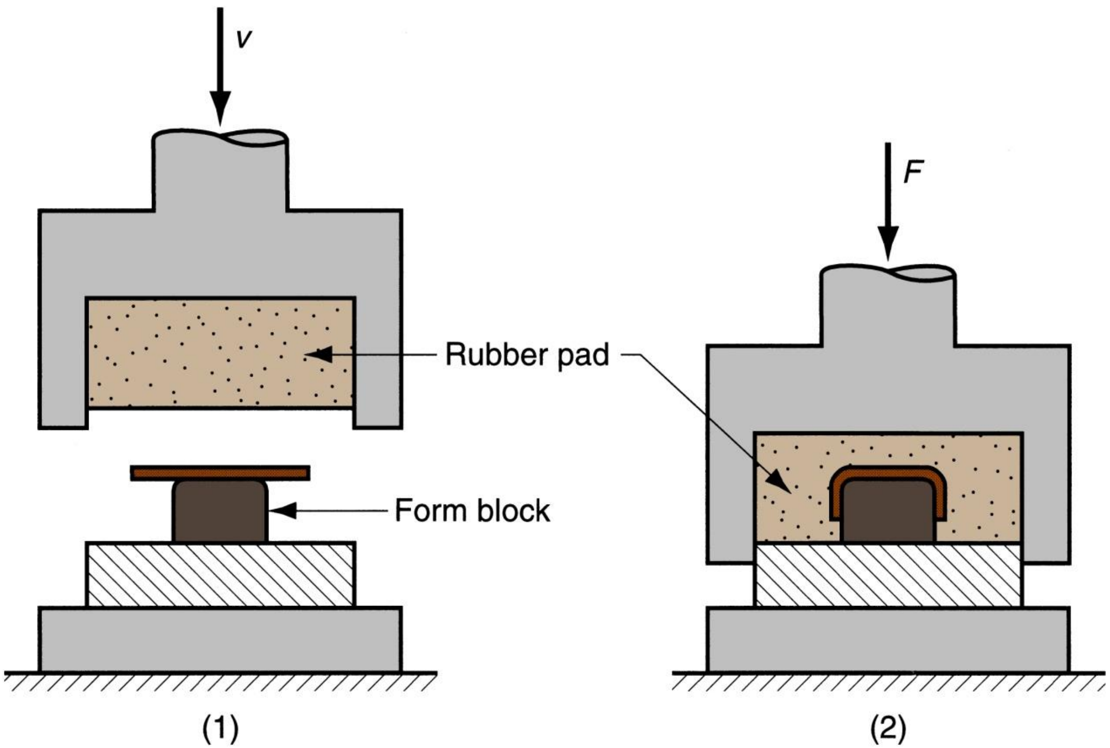


Figure 20.28 ‑ Guerin process: (1) before and (2) after

Symbols *v* and *F* indicate motion and applied force respectively

Advantages of Guerin Process

- Low tooling cost
- Form block can be made of wood, plastic, or other materials that are easy to shape
- Rubber pad can be used with different form blocks
- Process attractive in small quantity production

Dies for Sheet Metal Processes

Most pressworking operations performed with conventional *punch‑and‑die* tooling

- Custom‑designed for particular part
- The term *stamping die* sometimes used for high production dies

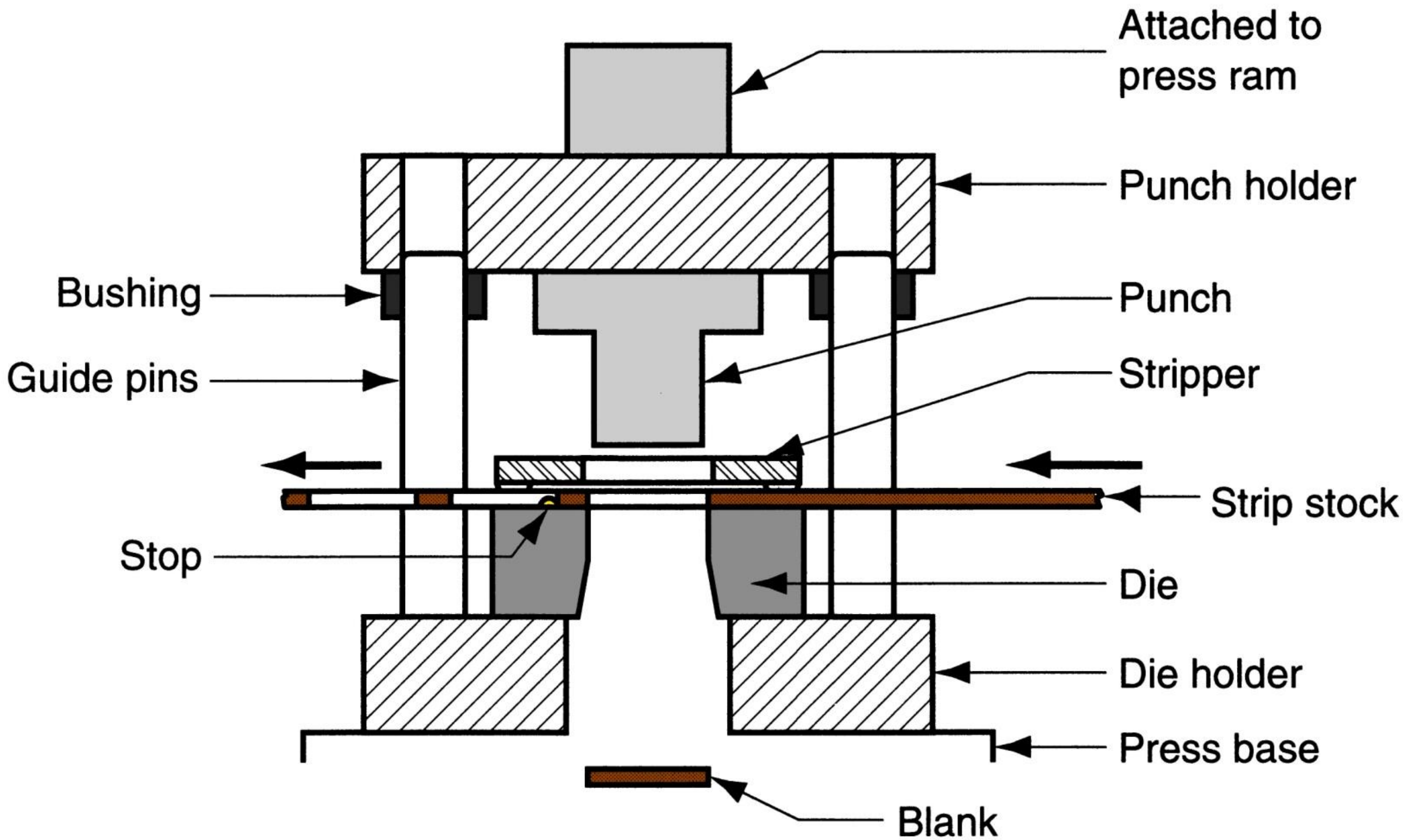


Figure 20.30 ‑ Components of a punch and die for a blanking operation

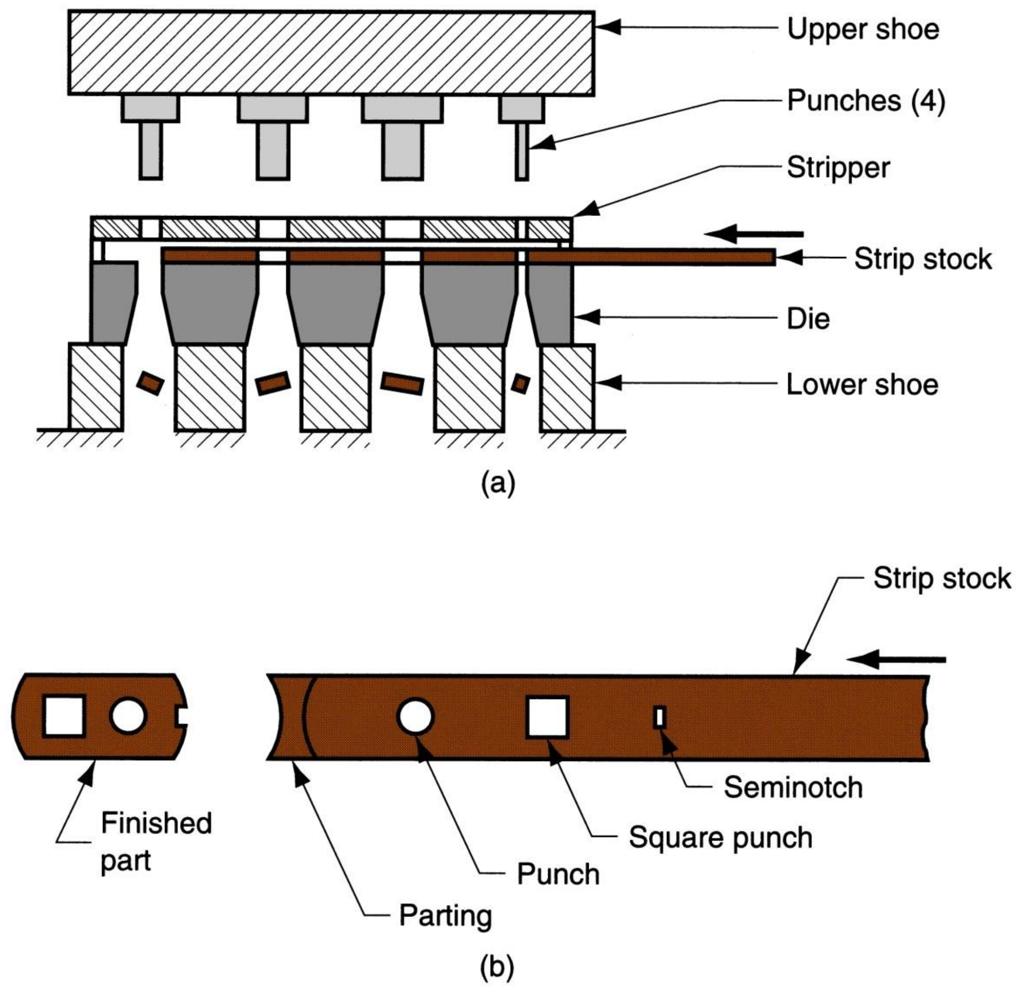


Figure 20.31 -

(a) Progressive die;

(b) associated strip development

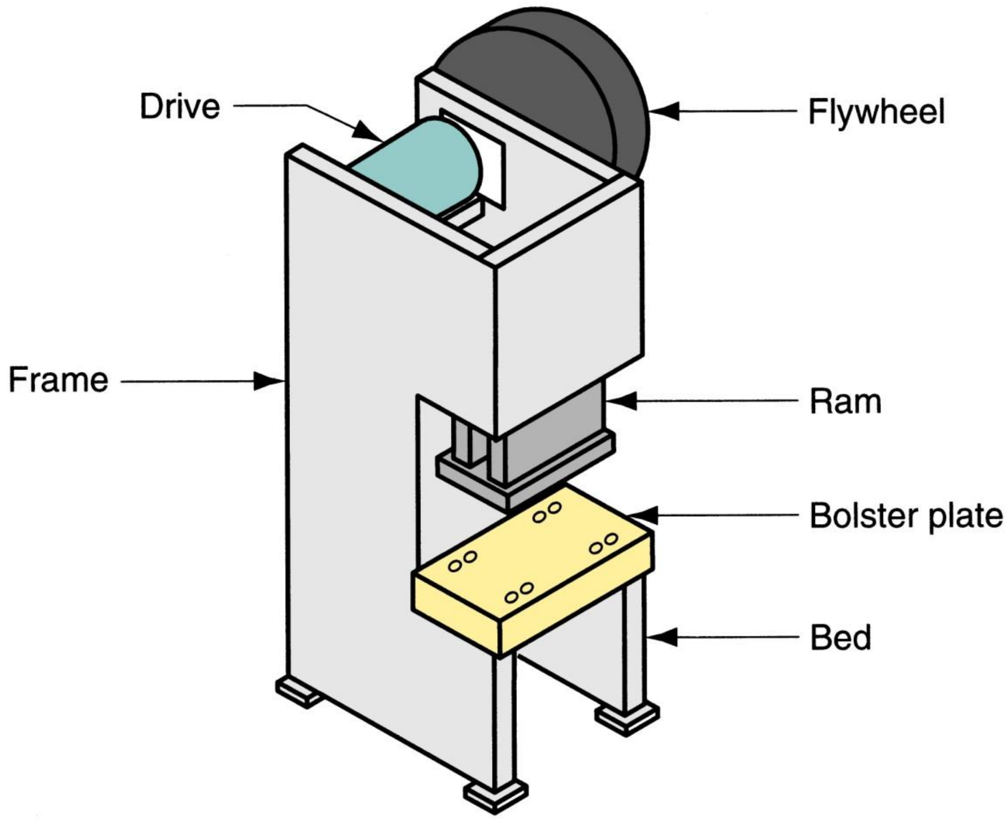


Figure 20.32 ‑ Components of a typical mechanical drive stamping press

Types of Stamping Press Frame

- *Gap frame* – configuration of the letter C and often referred to as a *C‑frame*
- *Straight‑sided frame* – box-like construction for higher tonnage

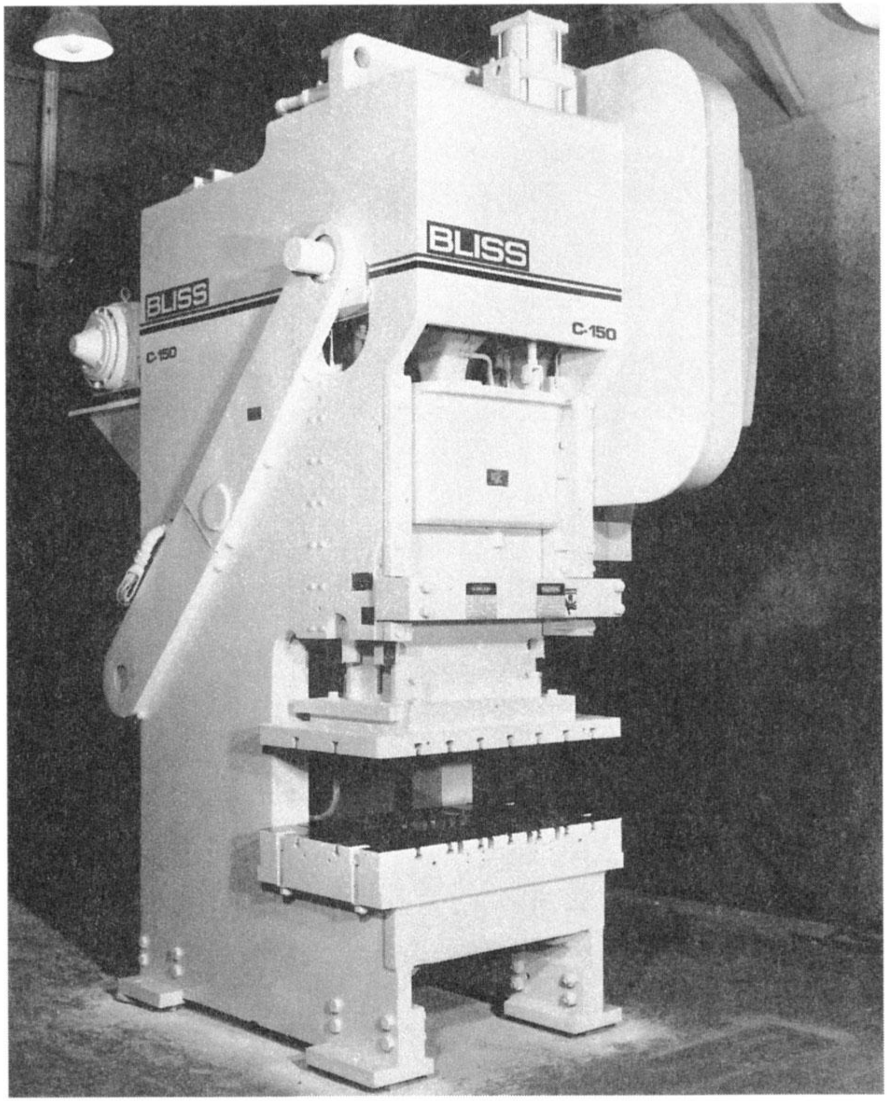


Figure 20.33 ‑ Gap frame press for sheet metalworking

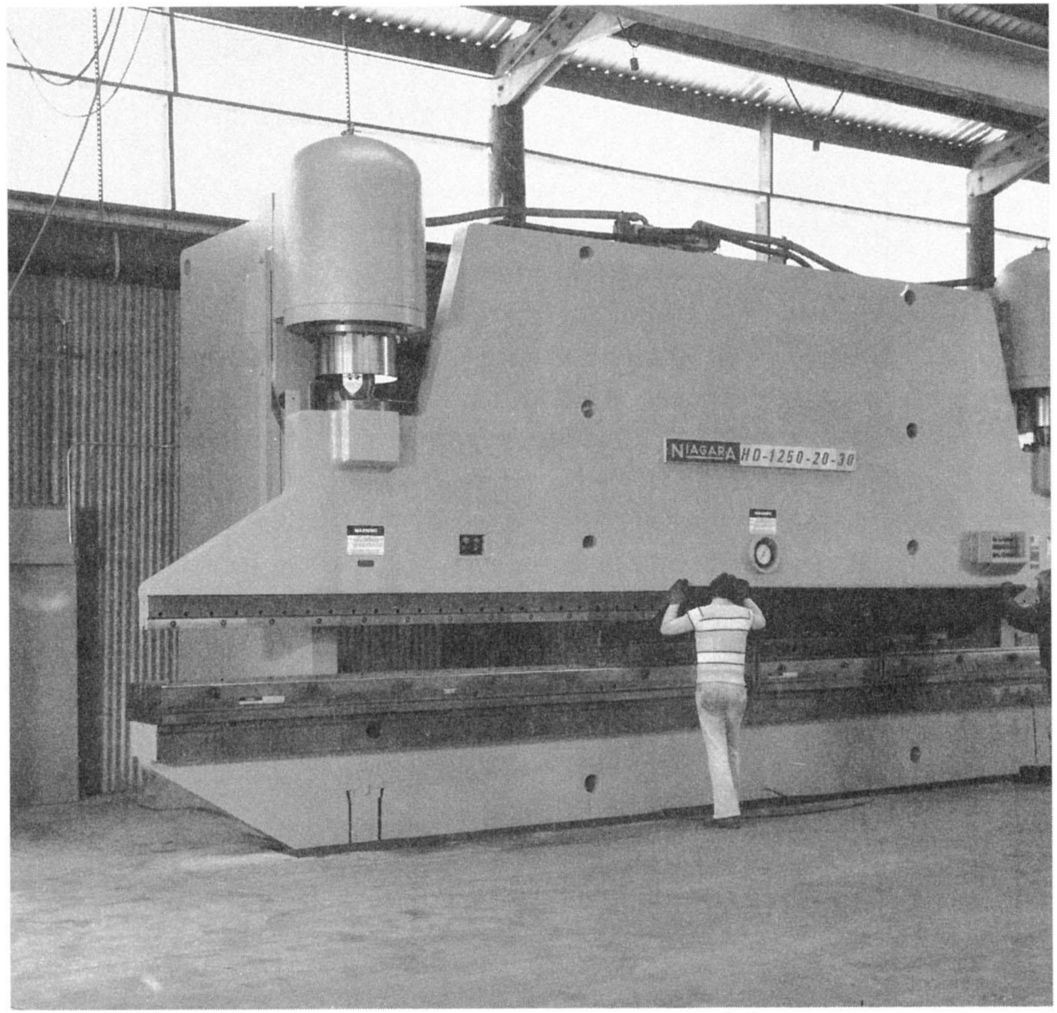
(photo courtesy of E. W. Bliss Company)

Capacity = 1350 kN (150 tons)

Figure 20.34 -

Press brake with bed width of 9.15 m (30 ft) and capacity of 11,200 kN (1250 tons); two workers are positioning plate stock for bending

(photo courtesy of Niagara Machine & Tool Works)



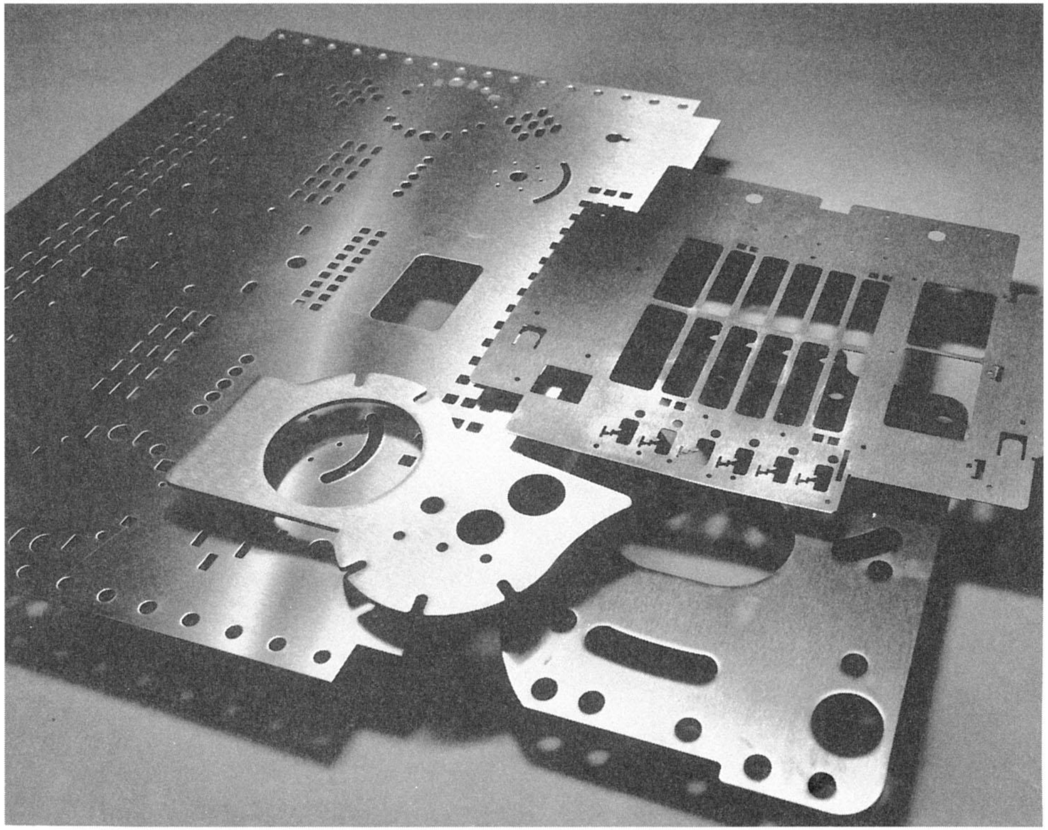


Figure 20.35 ‑ Several sheet metal parts produced on a turret press, showing variety of hole shapes possible

(photo courtesy of Strippet, Inc.)

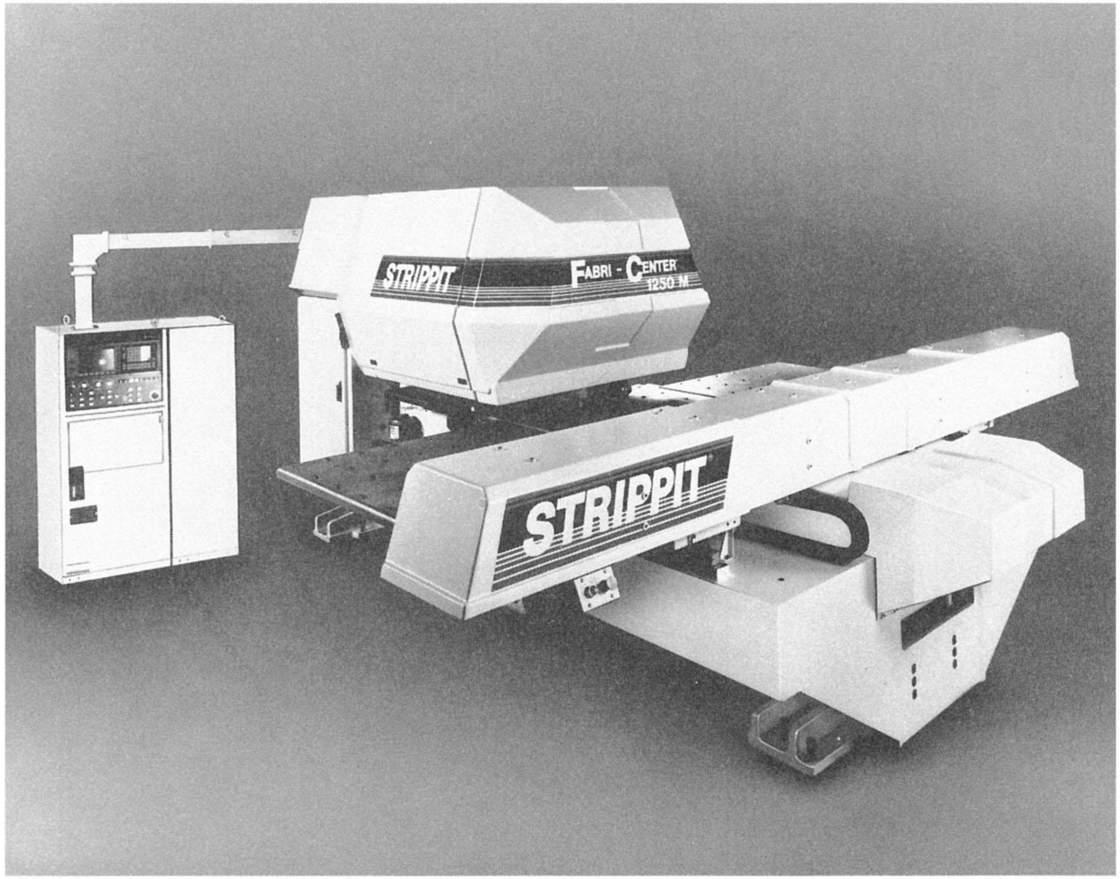


Figure 20.36 ‑ Computer numerical control turret press (photo courtesy of Strippet, Inc.)

Figure 20.37 ‑ Straight‑sided frame press (photo courtesy Greenerd Press & Machine Company, Inc.)

Power and Drive Systems

- Hydraulic presses - use a large piston and cylinder to drive the ram
  - Longer ram stroke than mechanical types
  - Suited to deep drawing
  - Slower than mechanical drives
- Mechanical presses – convert rotation of motor to linear motion of ram
  - High forces at bottom of stroke
  - Suited to blanking and punching

Sheet Metal Operations Not Performed on Presses

- Stretch forming
- Roll bending and forming
- Spinning
- High-energy-rate forming processes.

Stretch Forming

Sheet metal is stretched and simultaneously bent to achieve shape change

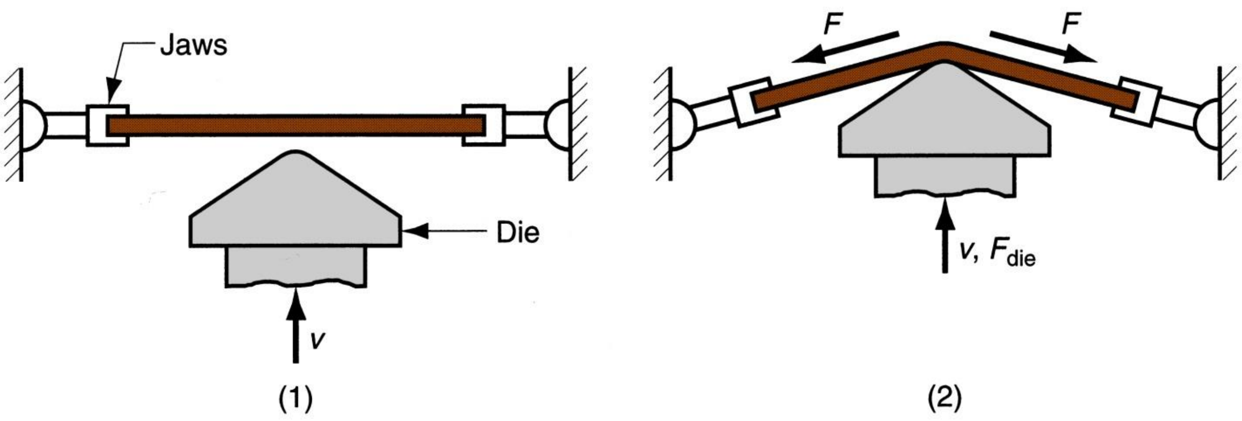


Figure 20.39 - Stretch forming: (1) start of process; (2) form die is pressed into the work with force F_{die}, causing it to be stretched and bent over the form. F = stretching force

Force Required in Stretch Forming

$$F = LtY_f$$

where F = stretching force; L = length of sheet in direction perpendicular to stretching; t = instantaneous stock thickness; and Y_f = flow stress of work metal

- Die force F_{die} can be determined by balancing vertical force components

Roll Bending

Large metal sheets and plates are formed into curved sections using rolls

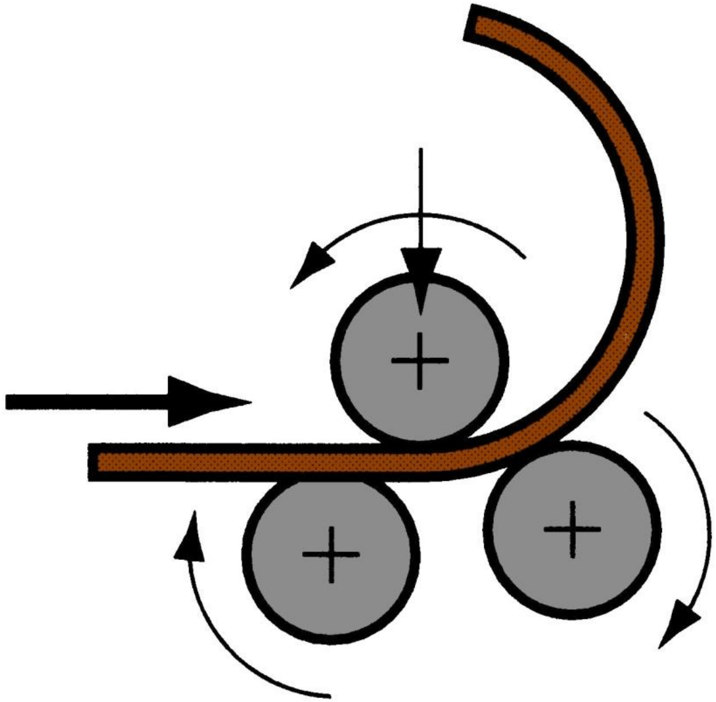


Figure 20.40 ‑ Roll bending

Roll Forming

Continuous bending process in which opposing rolls produce long sections of formed sha

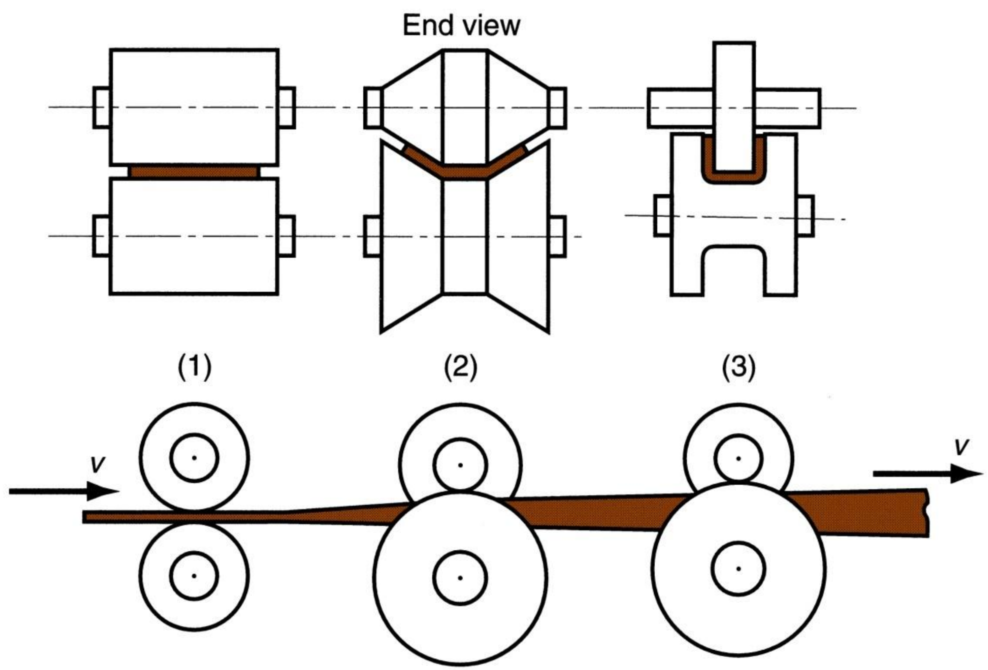


Figure 20.41 ‑ Roll forming of a continuous channel section:

(1) straight rolls

(2) partial form

(3) final form

Spinning

Metal forming process in which an axially symmetric part is gradually shaped over a rotating mandrel using a rounded tool or roller

- Three types:
  1. Conventional spinning
  2. Shear spinning
  3. Tube spinning

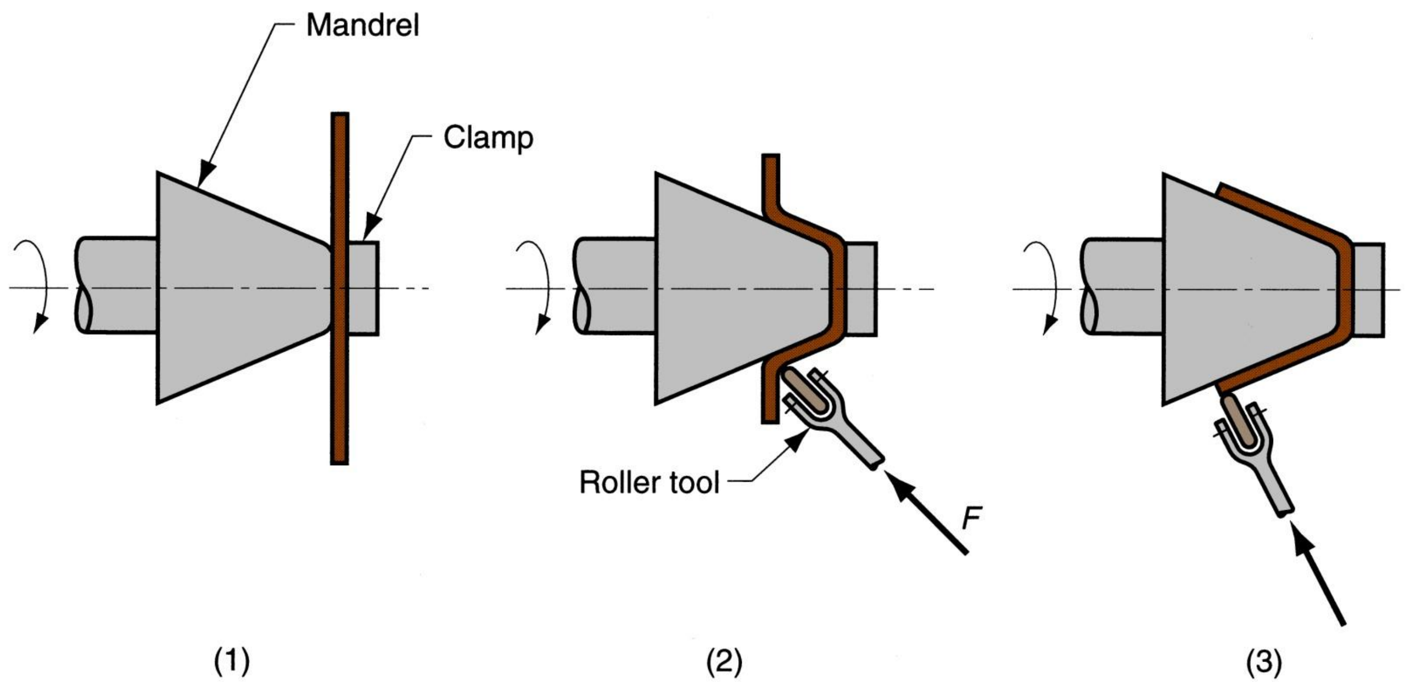


Figure 20.42 ‑ Conventional spinning: (1) setup at start of process; (2) during spinning; and (3) completion of process

High-Energy-Rate Forming (HERF)

Processes to form metals using large amounts of energy over a very short time

- HERF processes include:
  - Explosive forming
  - Electrohydraulic forming
  - Electromagnetic forming

Explosive Forming

Use of explosive charge to form sheet (or plate) metal into a die cavity

- Explosive charge causes a shock wave whose energy is transmitted to force part into cavity
- Applications: large parts, typical of aerospace industry

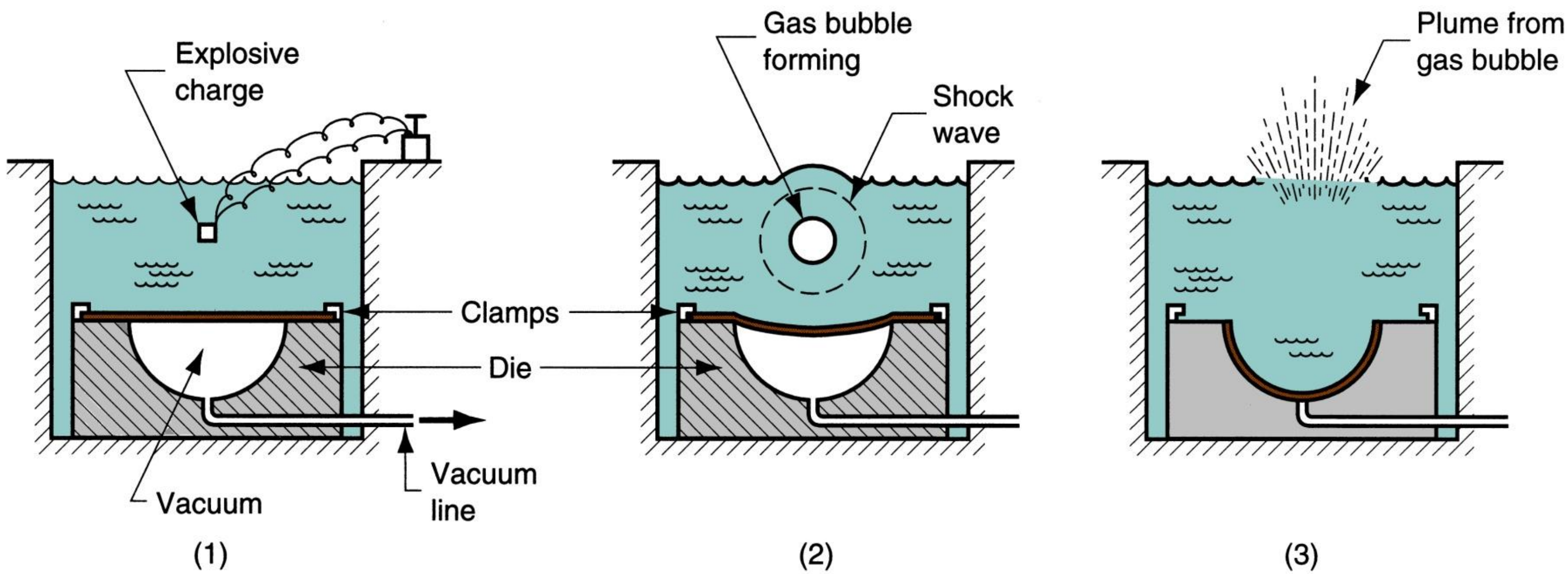


Figure 20.45 ‑ Explosive forming:
(1) setup, (2) explosive is detonated, and
(3) shock wave forms part and plume escapes water surface

Electromagnetic Forming

Sheet metal is deformed by mechanical force of an electromagnetic field induced in workpart by an energized coil

- Presently the most widely used HERF process
- Applications: tubular parts

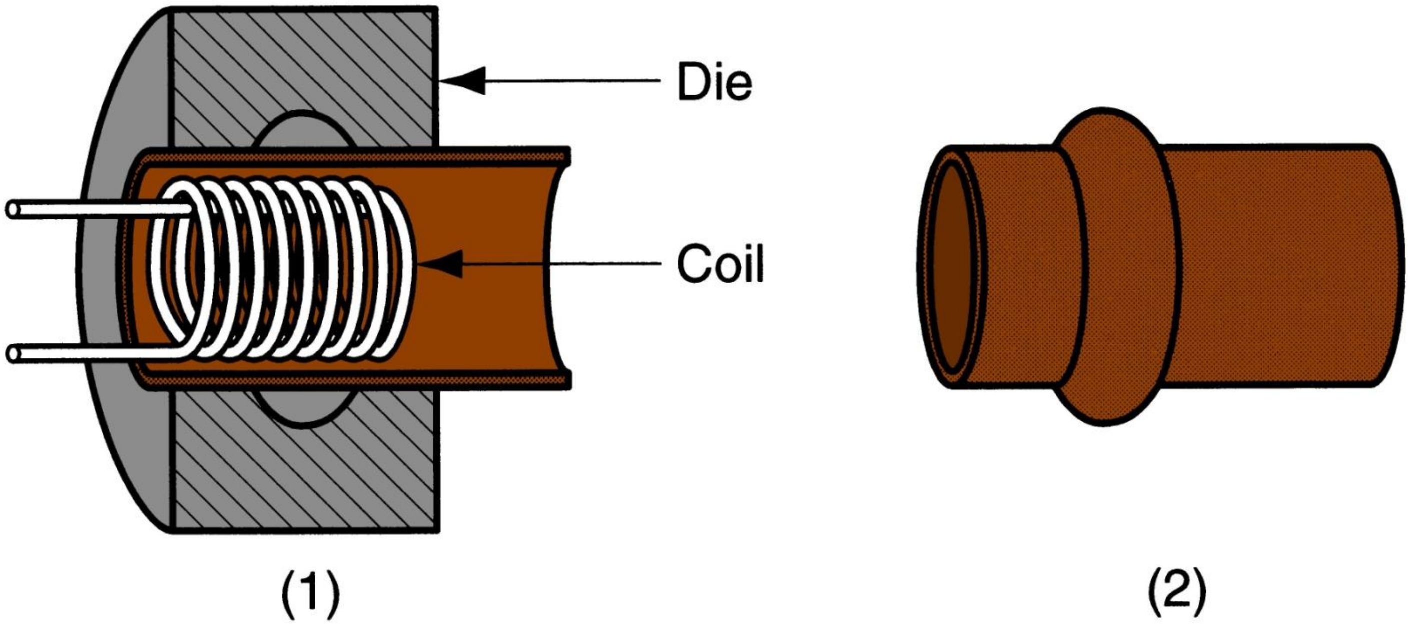


Figure 20.47 ‑ Electromagnetic forming: (1) setup in which coil is inserted into tubular workpart surrounded by die; (2) formed part