SHEET METALWORKING

1. Cutting Operations
2. Bending Operations
3. Drawing
4. Other Sheet Metal Forming Operations
5. Dies and Presses for Sheet Metal Processes
6. Sheet Metal Operations Not Performed on Presses
7. 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
- Economical mass production for large quantities

Sheet Metalworking Terminology

- Punch-and-die - tooling to perform cutting, bending, and drawing
- Stamping press - machine tool that performs most sheet metal operations
- Stampings - sheet metal products

Basic Types of Sheet Metal Processes

1. Cutting
   - Shearing to separate large sheets
   - Blanking to cut part perimeters out of sheet metal
   - Punching to make holes in sheet metal
2. Bending
   - Straining sheet around a straight axis
3. Drawing
   - Forming of sheet into convex or concave shapes

Sheet Metal Cutting

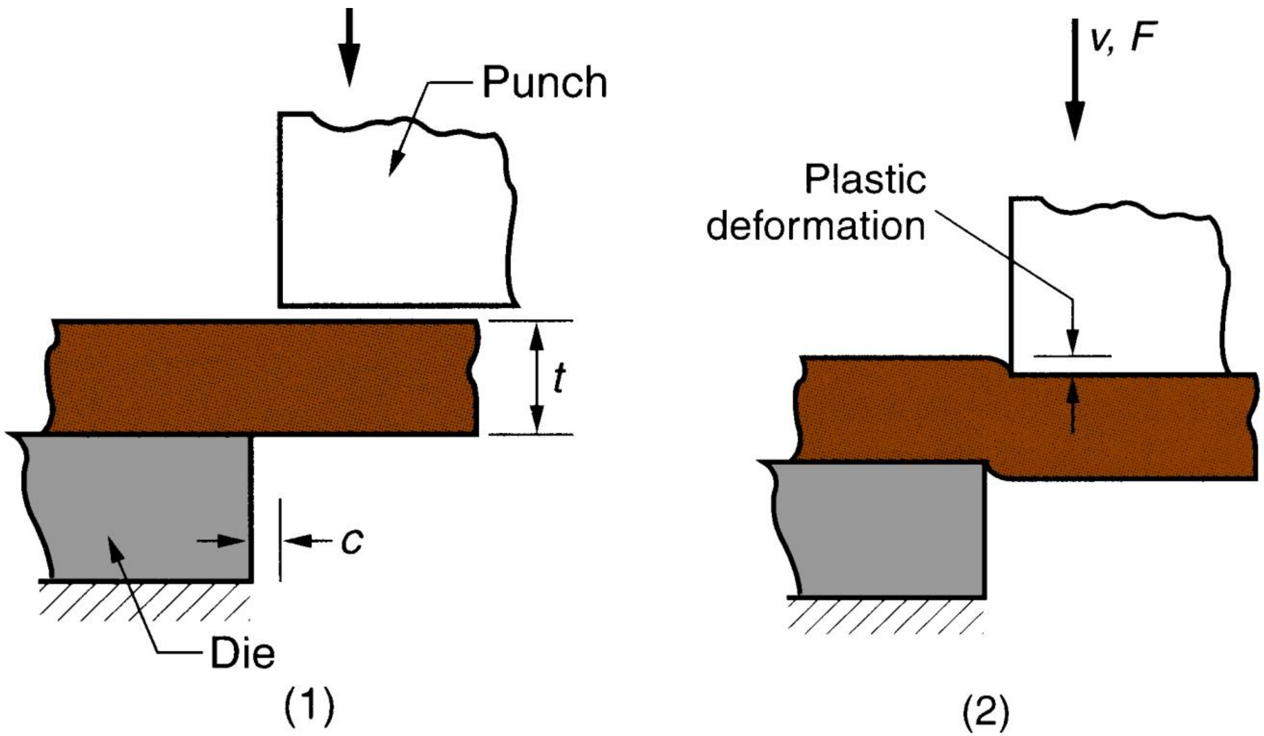


Figure 20.1 Shearing of sheet metal between two cutting edges: (1) just before the punch contacts work; (2) punch begins to push into work, causing plastic deformation;

Sheet Metal Cutting

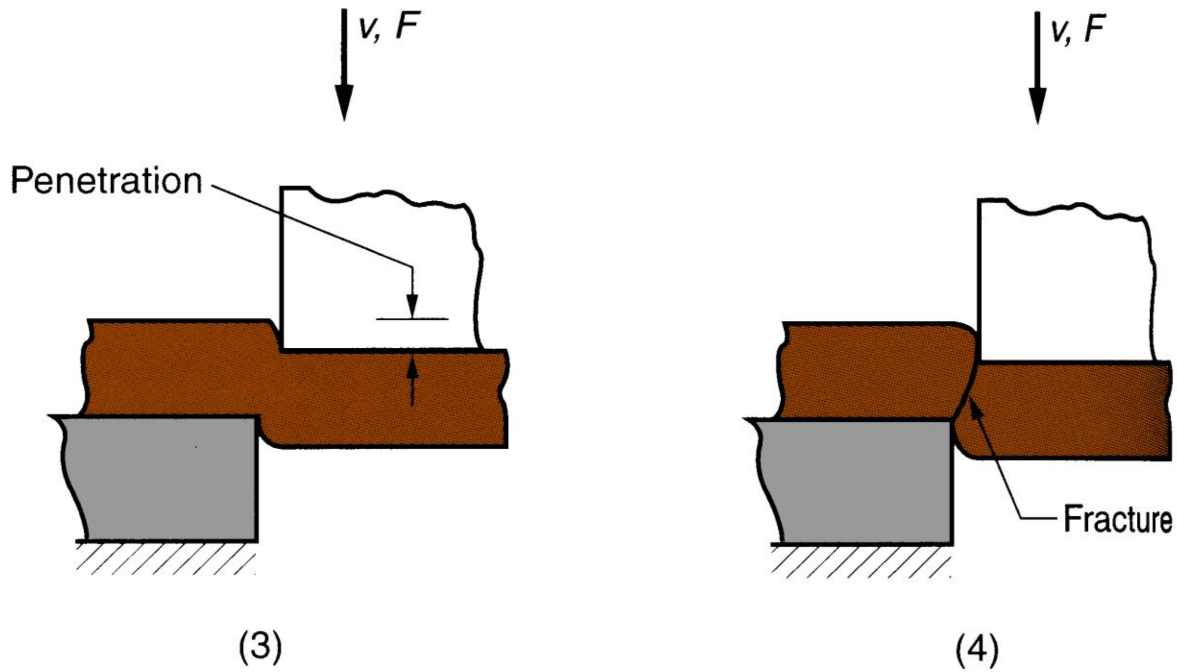


Figure 20.1 Shearing of sheet metal between two cutting edges: (3) punch compresses and penetrates into work causing a smooth cut surface; (4) fracture is initiated at the opposing cutting edges which separates the sheet.

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

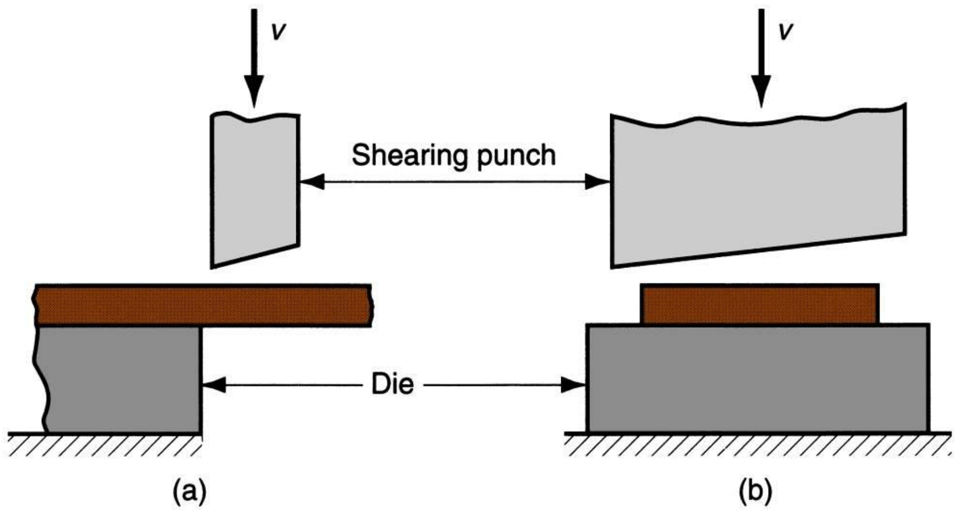


Figure 20.3 Shearing operation: (a) side view of the shearing operation; (b) front view of power shears equipped with inclined upper cutting blade.

Blanking and Punching

- Blanking - sheet metal cutting to separate piece (called a *blank*) from surrounding stock
- Punching - similar to blanking except cut piece is scrap, called a *slug*

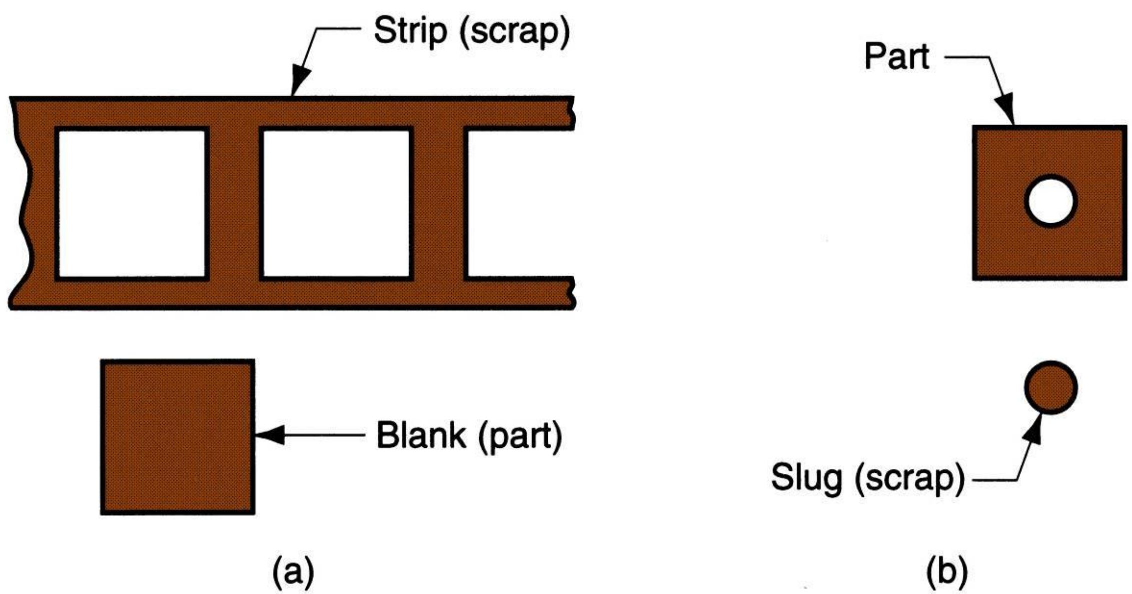


Figure 20.4 (a) Blanking and (b) punching.

Clearance in Sheet Metal Cutting

Distance between punch cutting edge and die cutting edge

- 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 is calculated by:

  $$c = at$$

  where c = clearance; a = allowance; and t = stock thickness
- Allowance a is determined according to type of metal

Sheet Metal Groups Allowances

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 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

Punch and Die Sizes

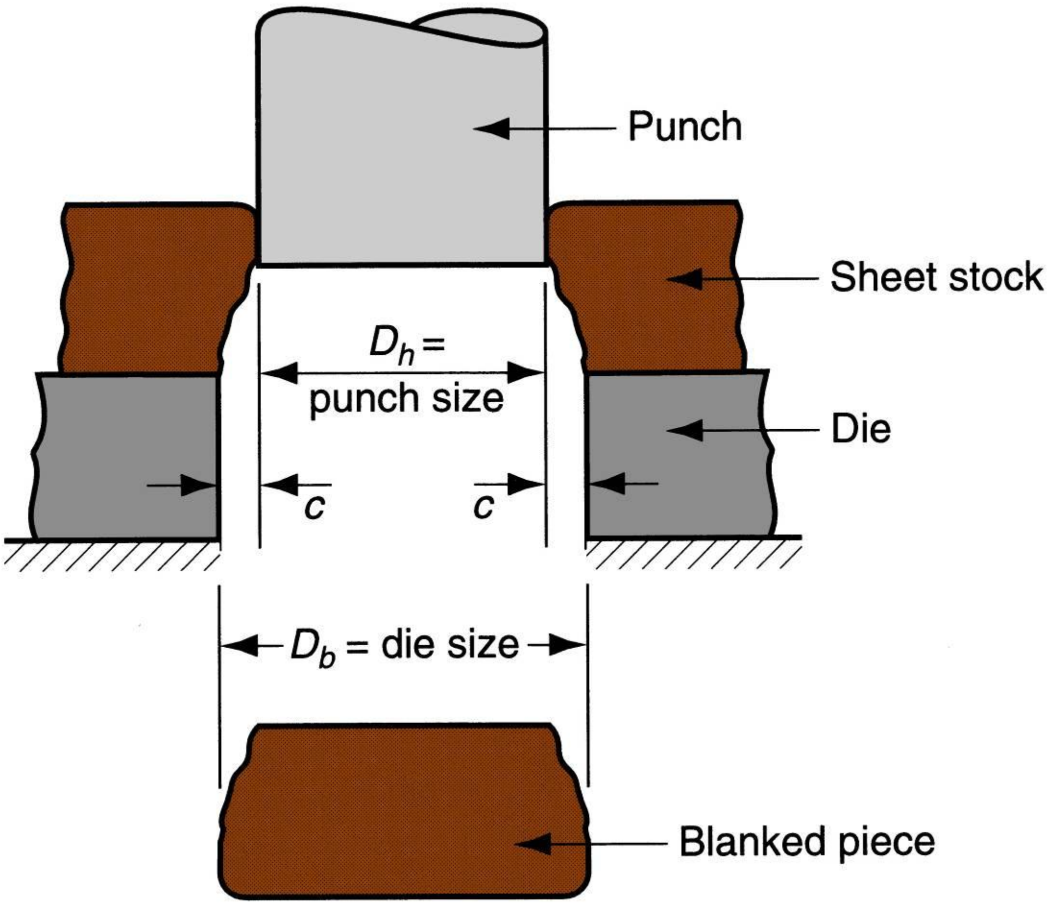


Figure 20.6 Die size determines blank size D_b; punch size determines hole size D_h.; c = clearance

Angular Clearance

Purpose: allows slug or blank to drop through die

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

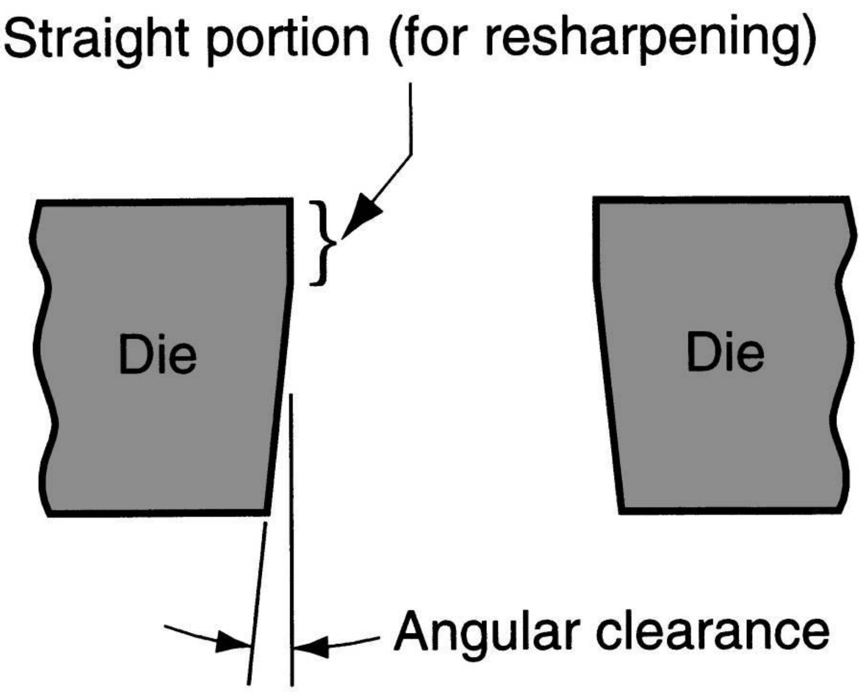


Figure 20.7 Angular clearance.

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

Sheet Metal Bending

Straining sheetmetal around a straight axis to take a permanent bend

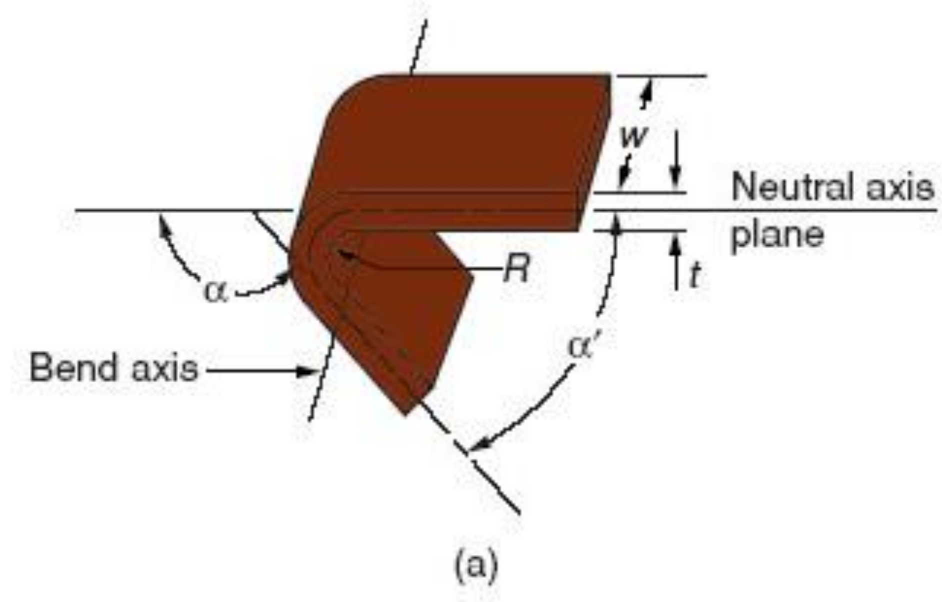


Figure 20.11 (a) Bending of sheet metal

Sheet Metal Bending

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

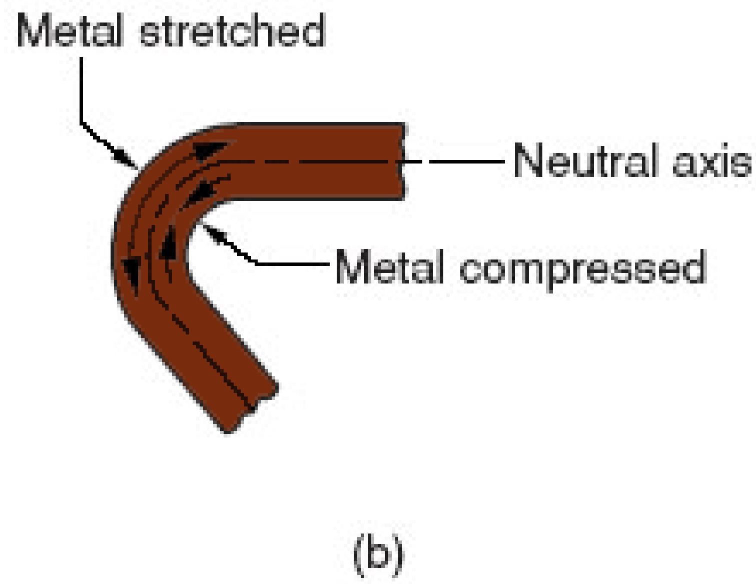


Figure 20.11 (b) both compression and tensile elongation of the metal occur in bending.

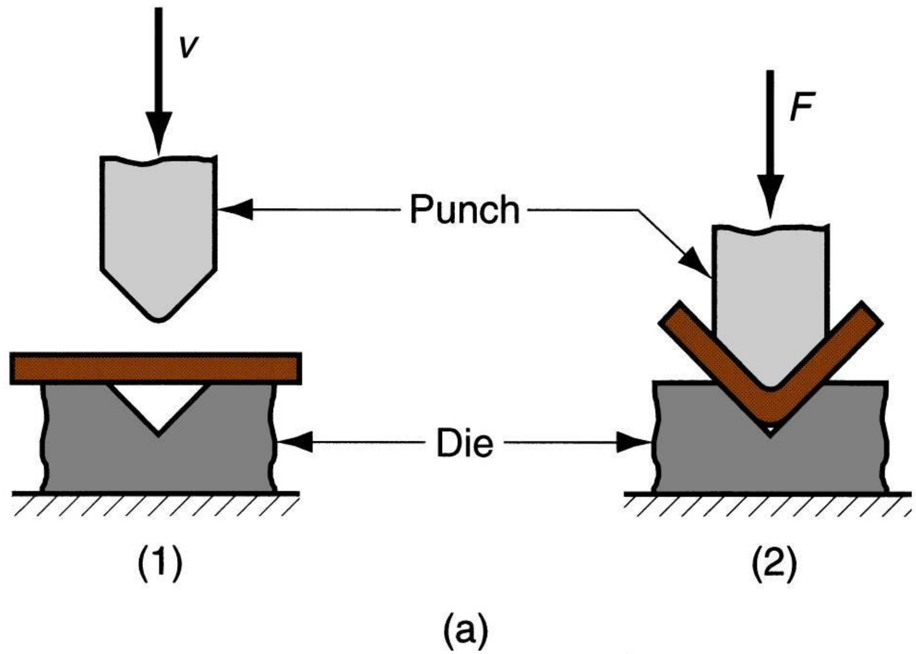
Types of Sheet Metal Bending

- 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

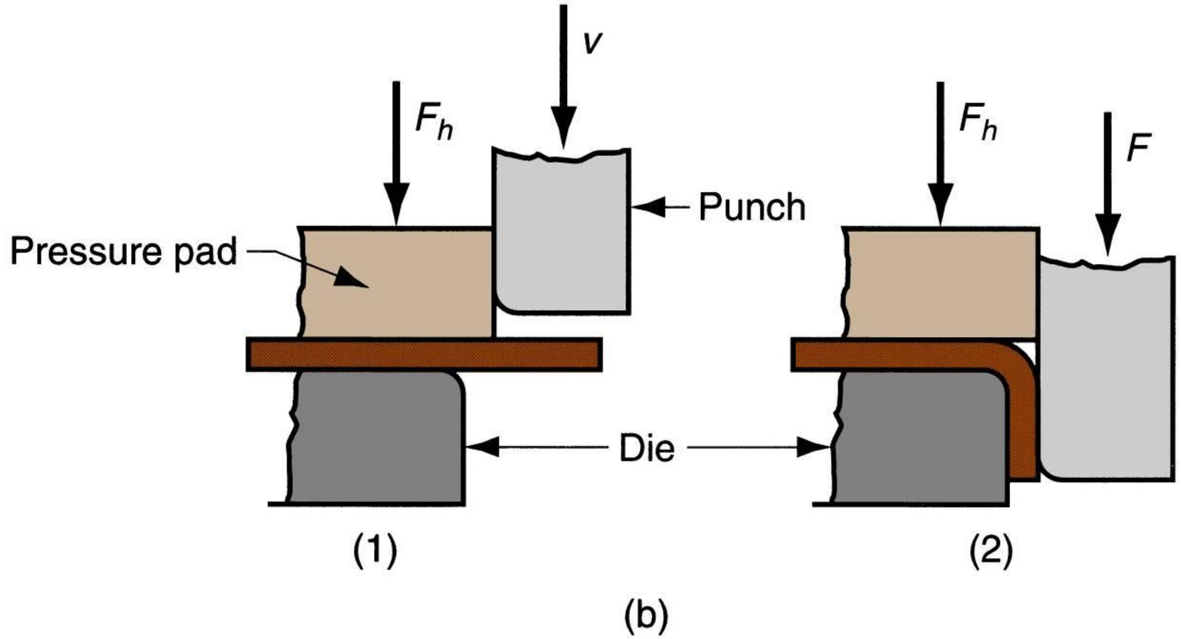


Figure 20.12 (b) edge bending.

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 final part length = specified dimension
- Problem: to determine the length of neutral axis of the part before bending

Bend Allowance Formula

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

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

- If $R < 2t$, $K_{ba} = 0.33$
- If $R \geq 2t$, $K_{ba} = 0.50$

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

Springback

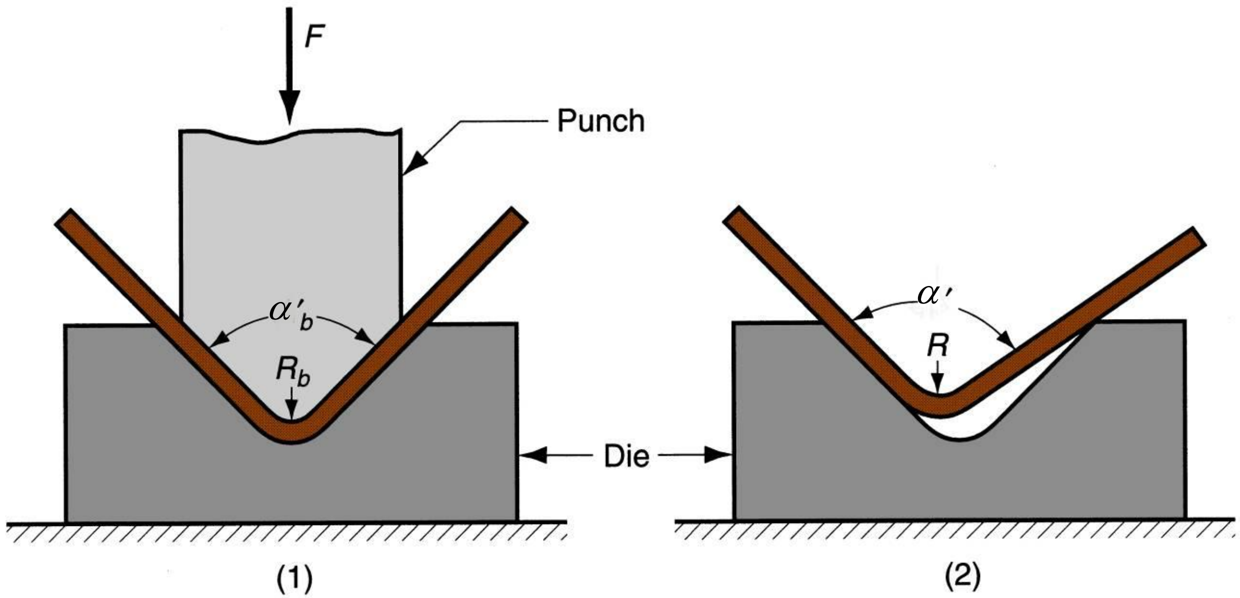


Figure 20.13 Springback in bending is seen as a decrease in bend angle and an increase in bend radius: (1) during bending, the work is forced to take radius R_b and included angle α_b' of the bending tool, (2) after punch is removed, the work springs back to radius R and angle α'.

Bending Force

Maximum bending force estimated as follows:

$$F = \frac{K_{bf} TSwt^2}{D}$$

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

Die Opening Dimension

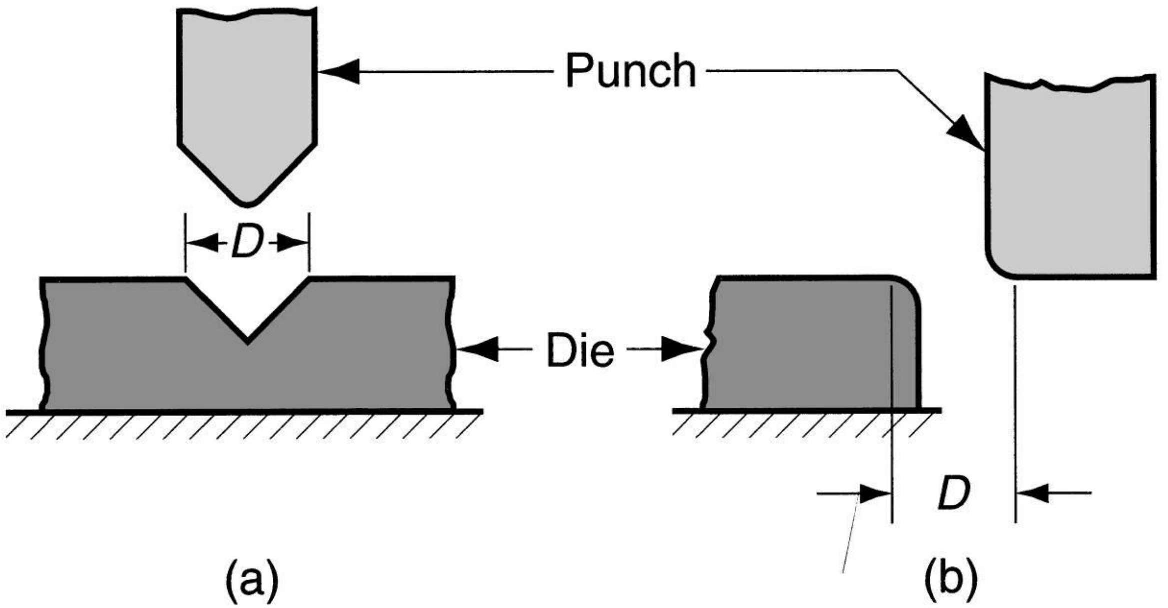


Figure 20.14 Die opening dimension D: (a) V-die, (b) wiping die.

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
- Also known as *deep drawing* (to distinguish it from wire and bar drawing)

Drawing

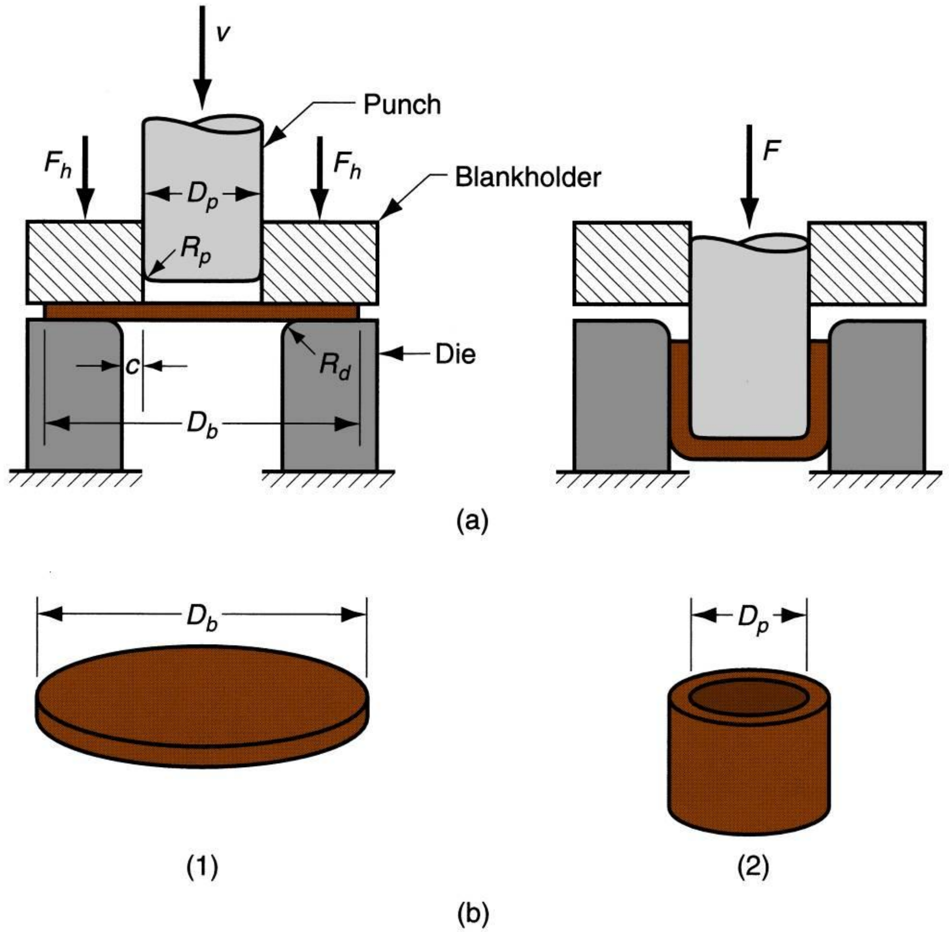


Figure 20.19 (a) Drawing of cup-shaped part: (1) before punch contacts work, (2) near end of stroke; (b) workpart: (1) starting blank, (2) drawn part.

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 is about 10% greater than stock thickness

Tests of Drawing Feasibility

- Drawing ratio
- Reduction
- Thickness-to-diameter ratio

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: $DR \leq 2.0$

Reduction *r*

- 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 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

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

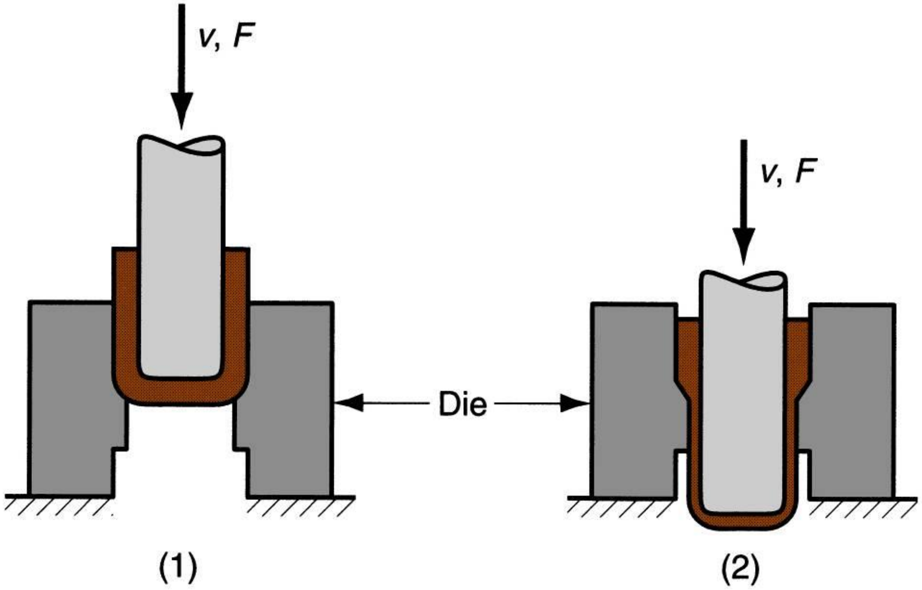


Figure 20.25 Ironing to achieve more uniform wall thickness in a drawn cup: (1) start of process; (2) during process. Note thinning and elongation of walls.

Embossing

Creates indentations in sheet, such as raised (or indented) lettering or strengthening ribs

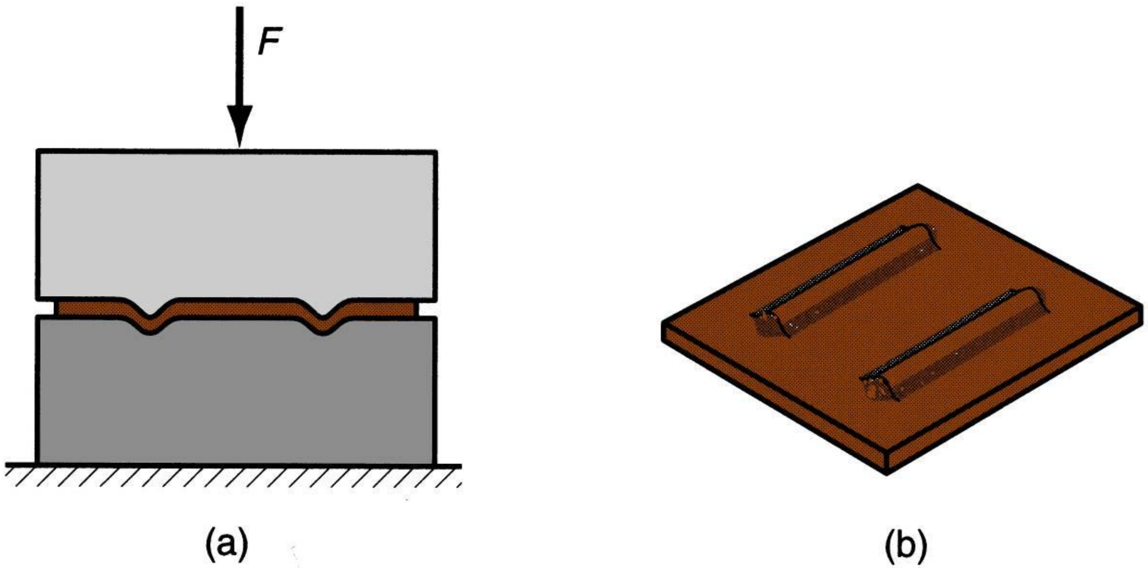


Figure 20.26 Embossing: (a) cross-section of punch and die configuration during pressing; (b) finished part with embossed ribs.

Guerin Process

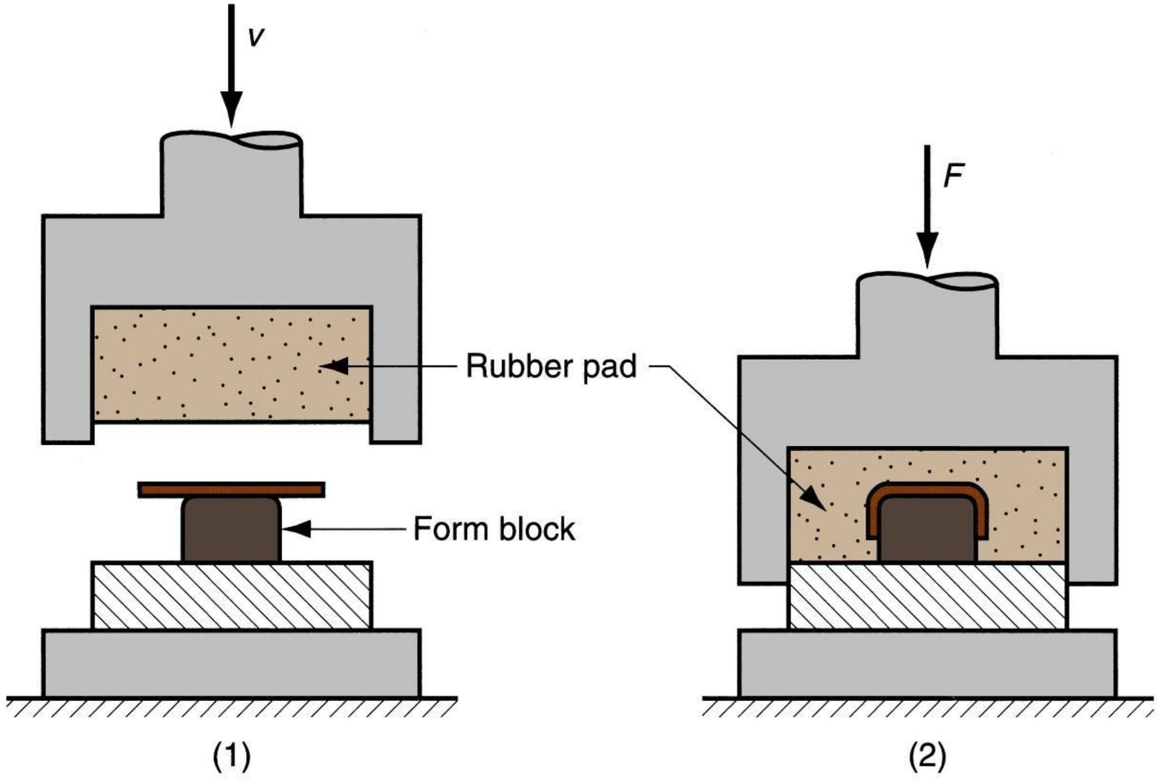


Figure 20.28 Guerin process: (1) before and (2) after. Symbols *v* and *F* indicate motion and applied force respectively.

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

Punch and Die Components

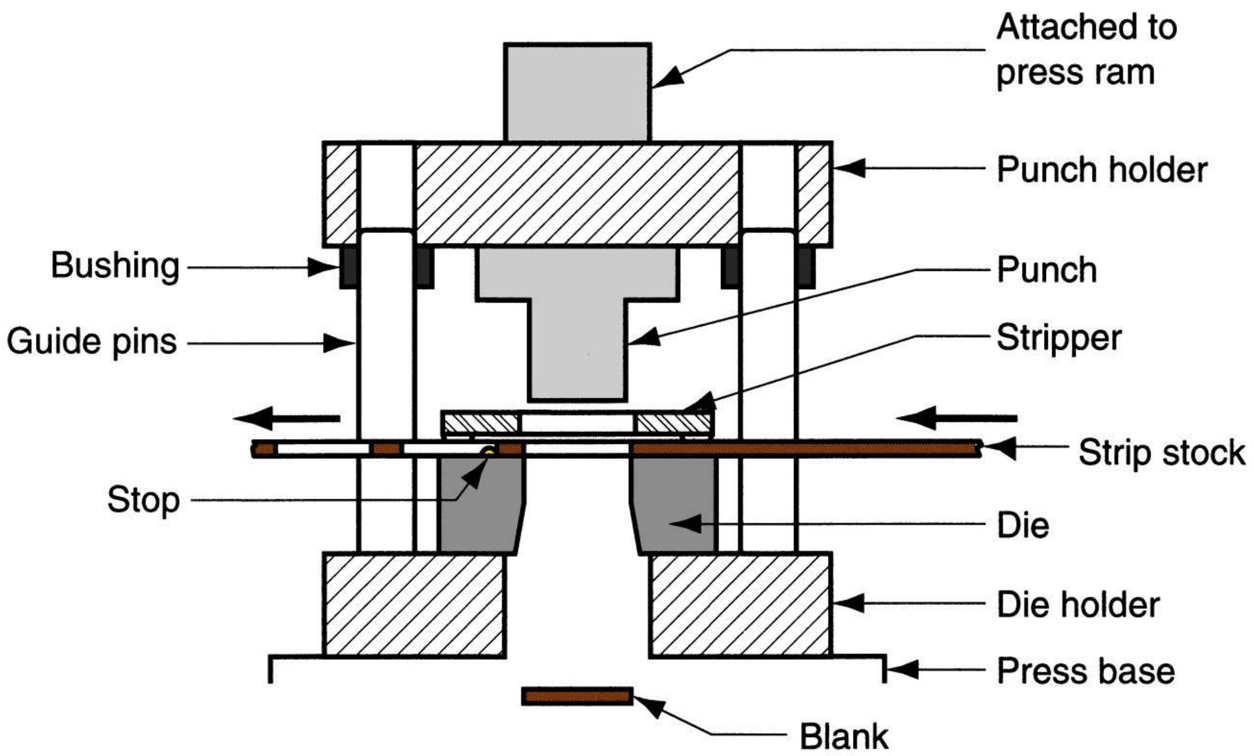


Figure 20.30 Components of a punch and die for a blanking operation.

Progressive Die

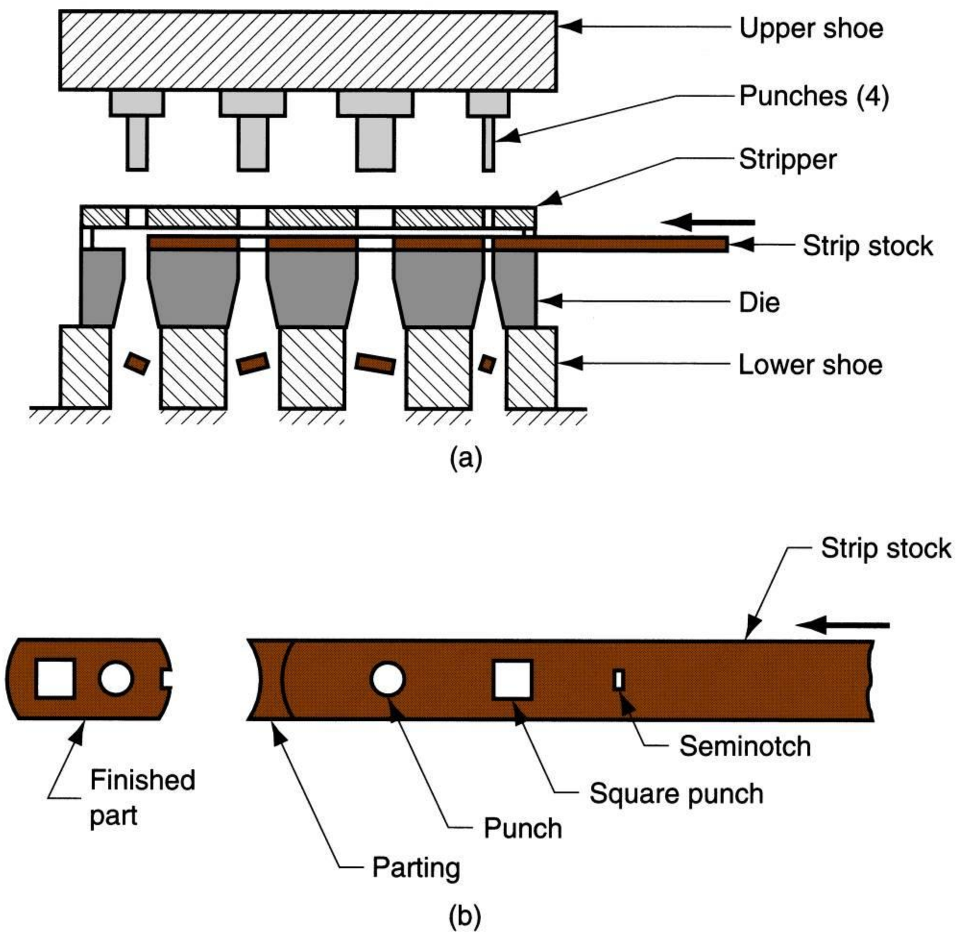


Figure 20.31 (a) Progressive die; (b) associated strip development

Stamping Press

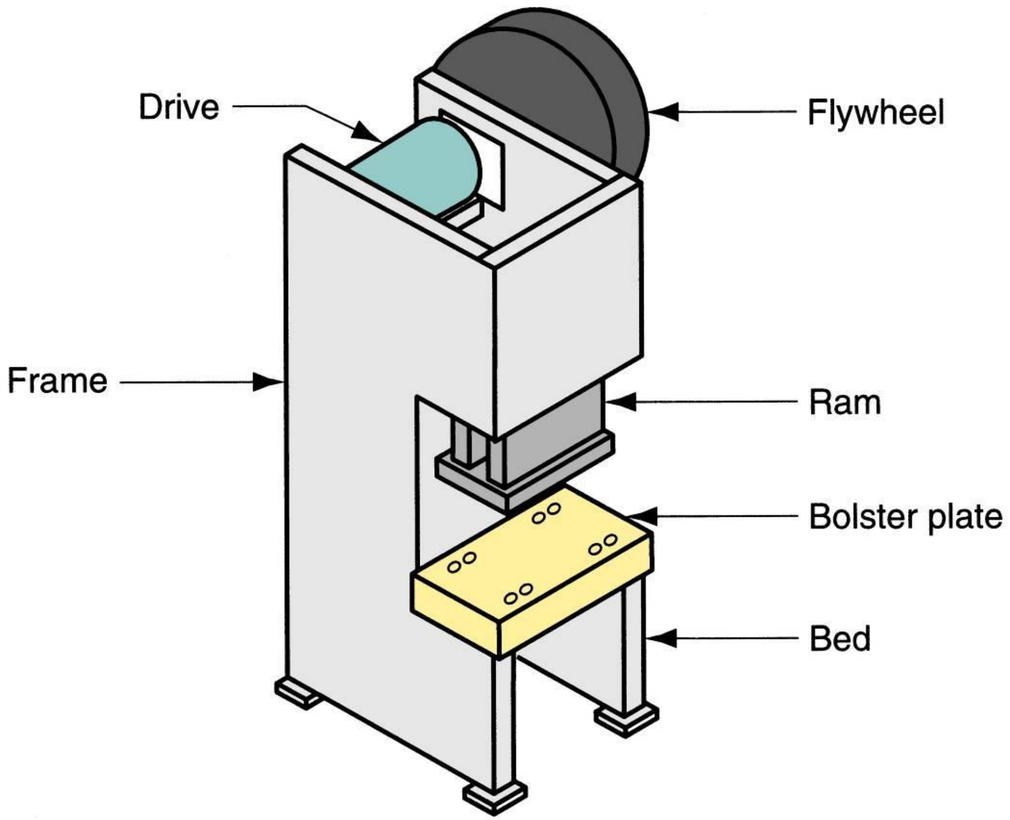


Figure 20.32 Components of a typical mechanical drive stamping press

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 (ohoto courtesy of E. W. Bliss Co.); capacity = 1350 kN (150 tons)

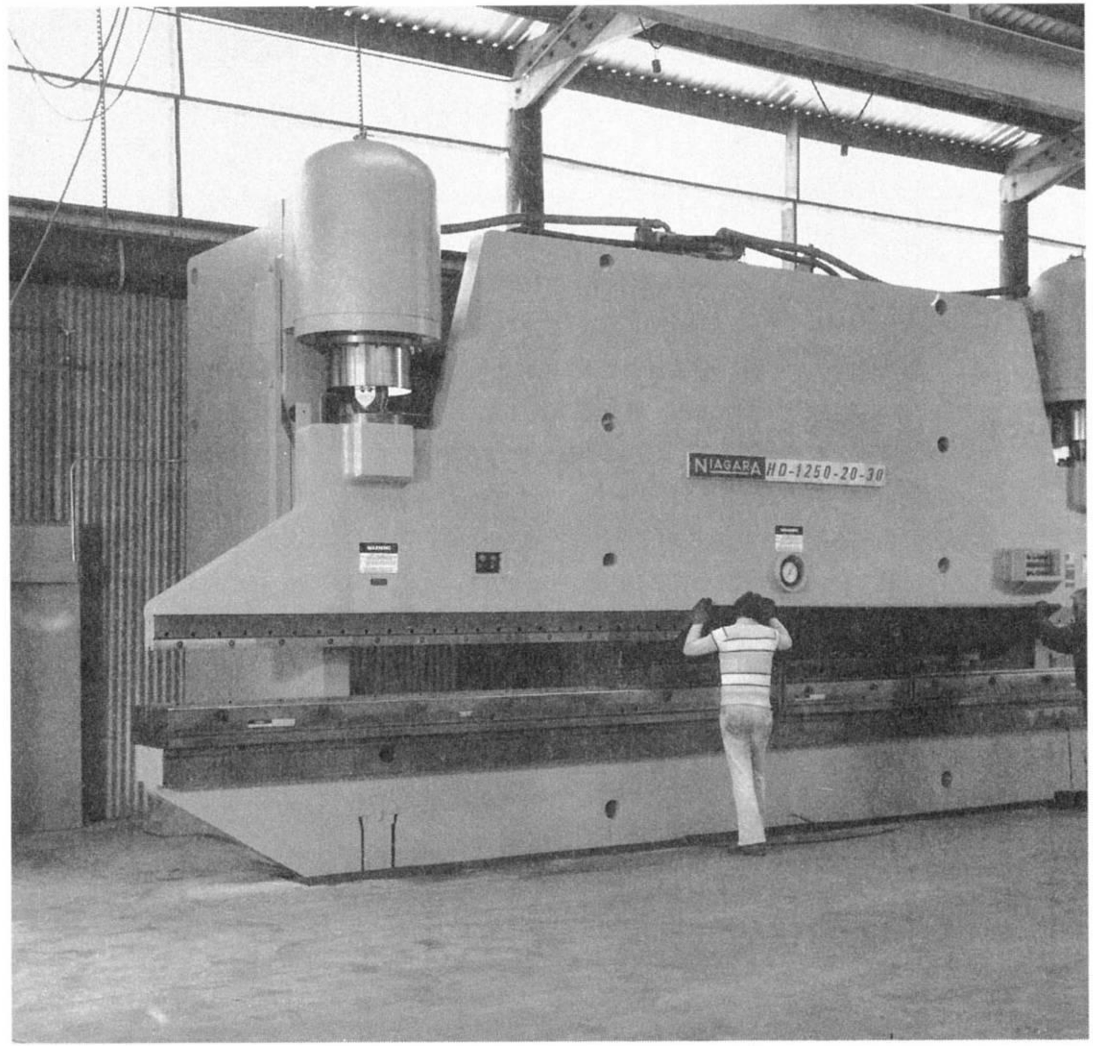


Figure 20.34 Press brake (photo courtesy of Niagara Machine & Tool Works); bed width = 9.15 m (30 ft) and capacity = 11,200 kN (1250 tons).

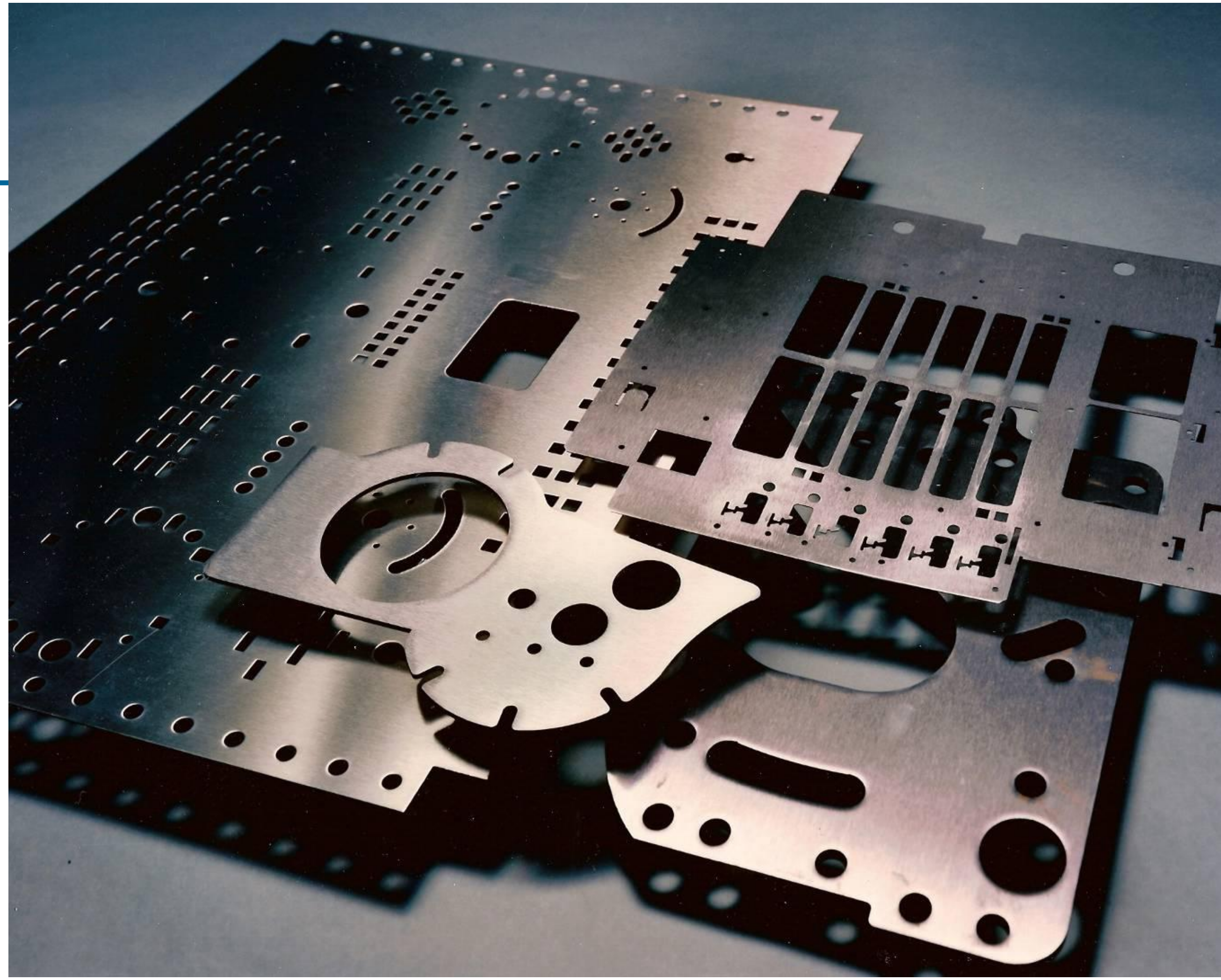


Figure 20.35 Sheet metal parts produced on a turret press, showing variety of hole shapes possible (photo courtesy of Strippet Inc.).

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

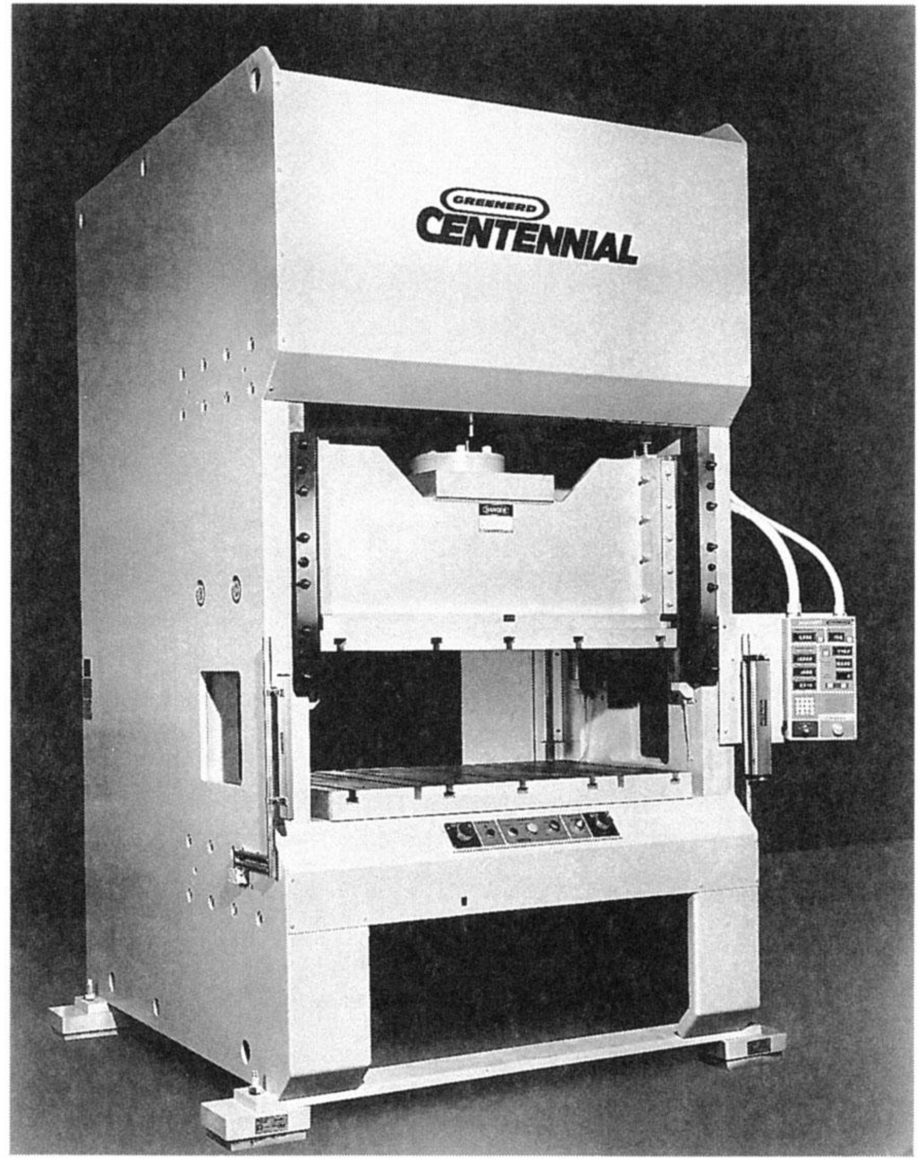


Figure 20.37

Straight-sided frame press (photo courtesy of 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

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

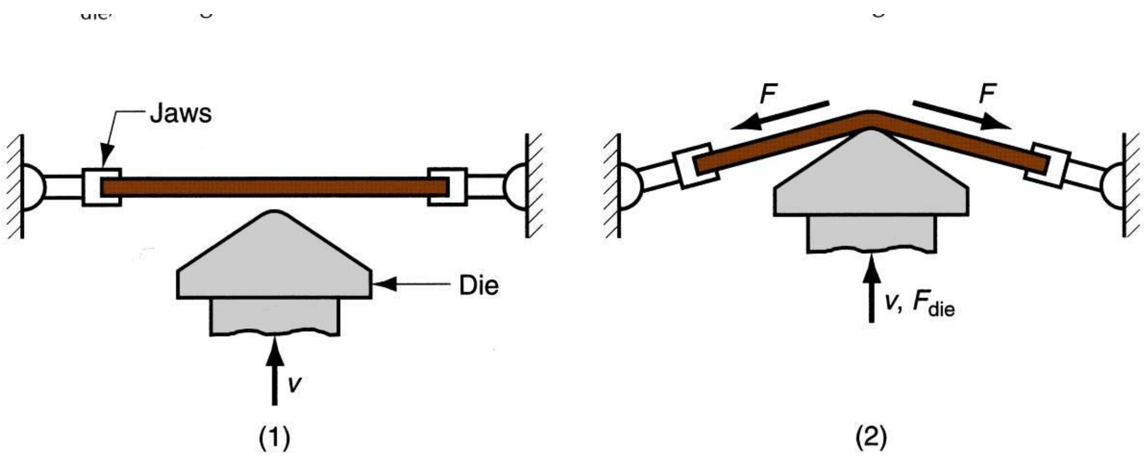


Figure 20.39 Stretch forming: (1) start of process; (2) form die is pressed into the work with force F_{die}, causing it to be stretched and bent over the form. F = stretching force.

Force Required in Stretch Forming

$$F = LtY_f$$

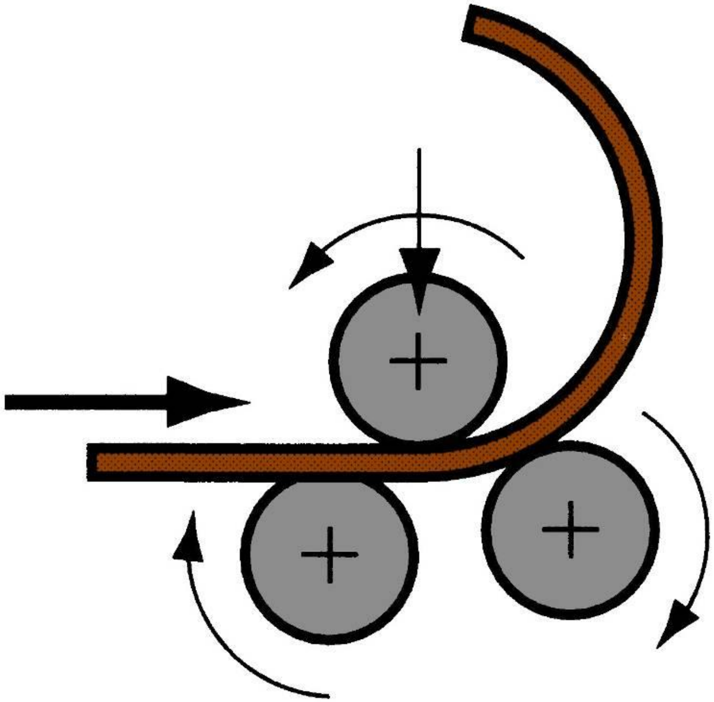
where F = stretching force; L = length of sheet in direction perpendicular to stretching; t = instantaneous stock thickness; and Y_f = flow stress of work metal

- 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 shapes from coil or strip stock

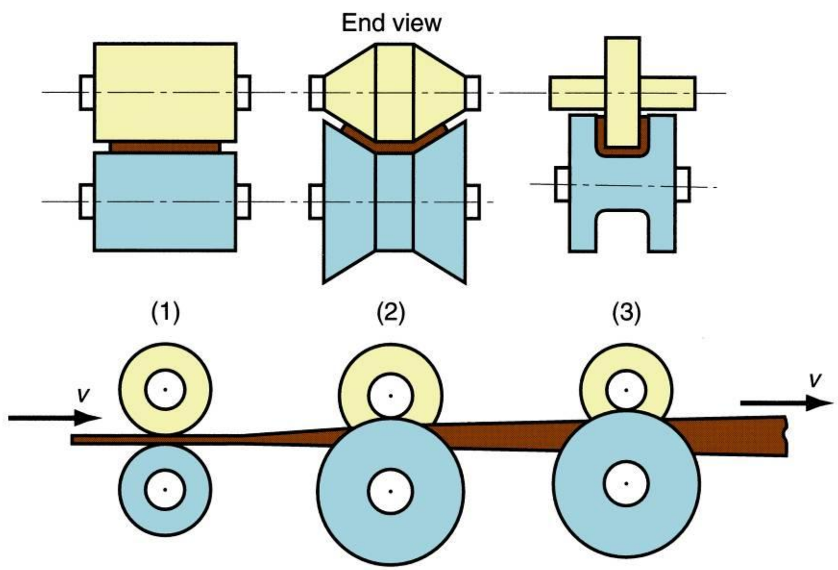


Figure 20.41 Roll forming of a continuous channel section: (1) straight rolls, (2) partial form, (3) final form.

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

Conventional Spinning

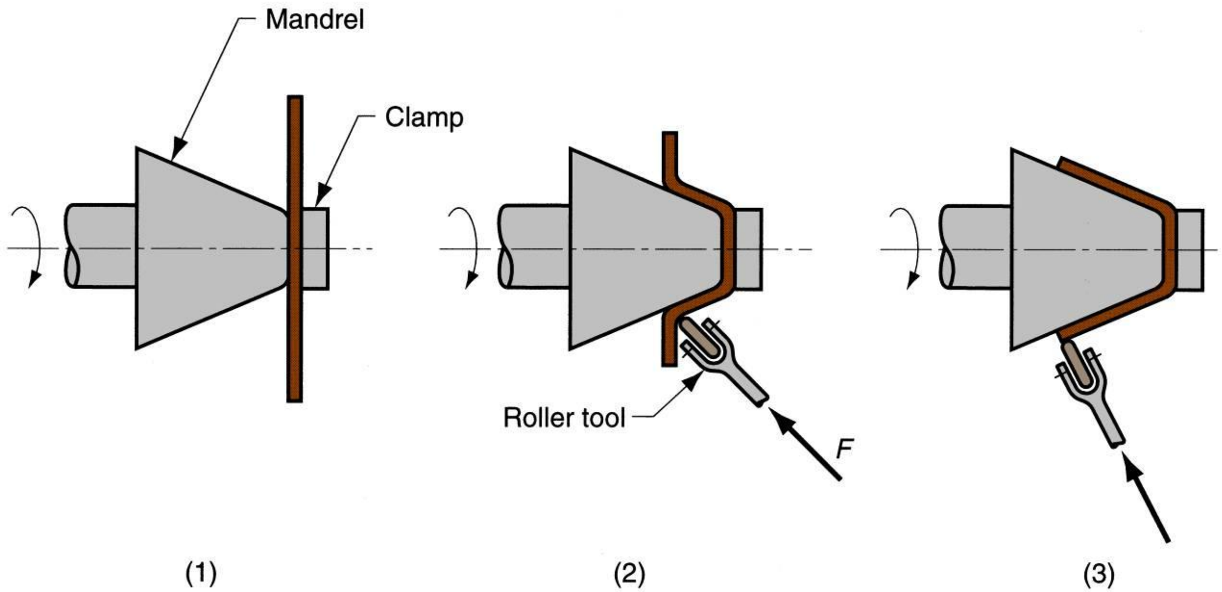


Figure 20.42 Conventional spinning: (1) setup at start of process; (2) during spinning; and (3) completion of process.

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

Explosive Forming

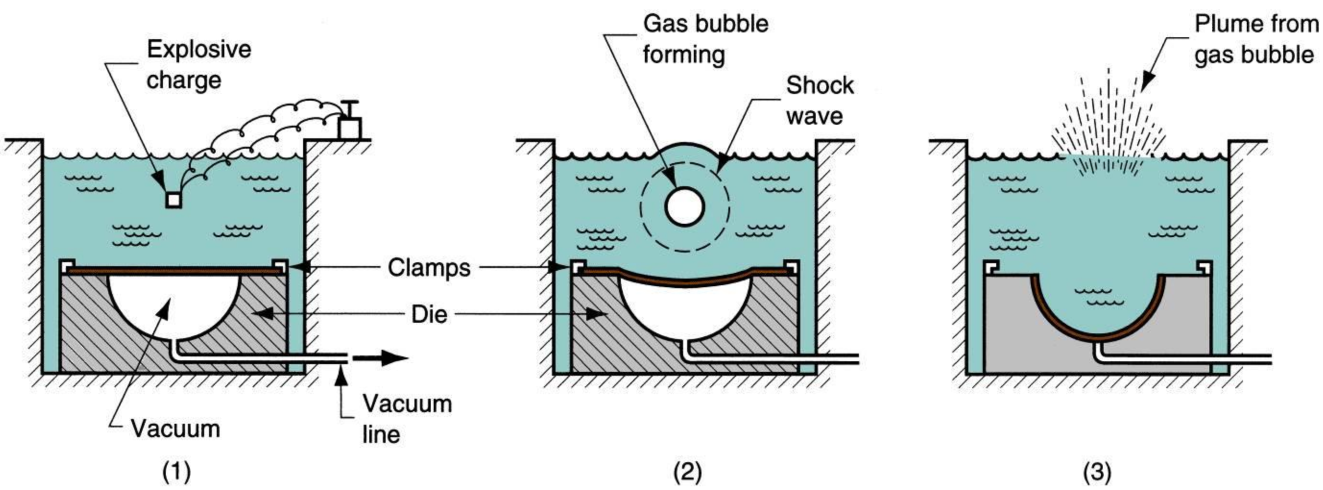


Figure 20.45 Explosive forming: (1) setup, (2) explosive is detonated, and (3) shock wave forms part and plume escapes water surface.

Electromagnetic Forming

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

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

Electromagnetic Forming

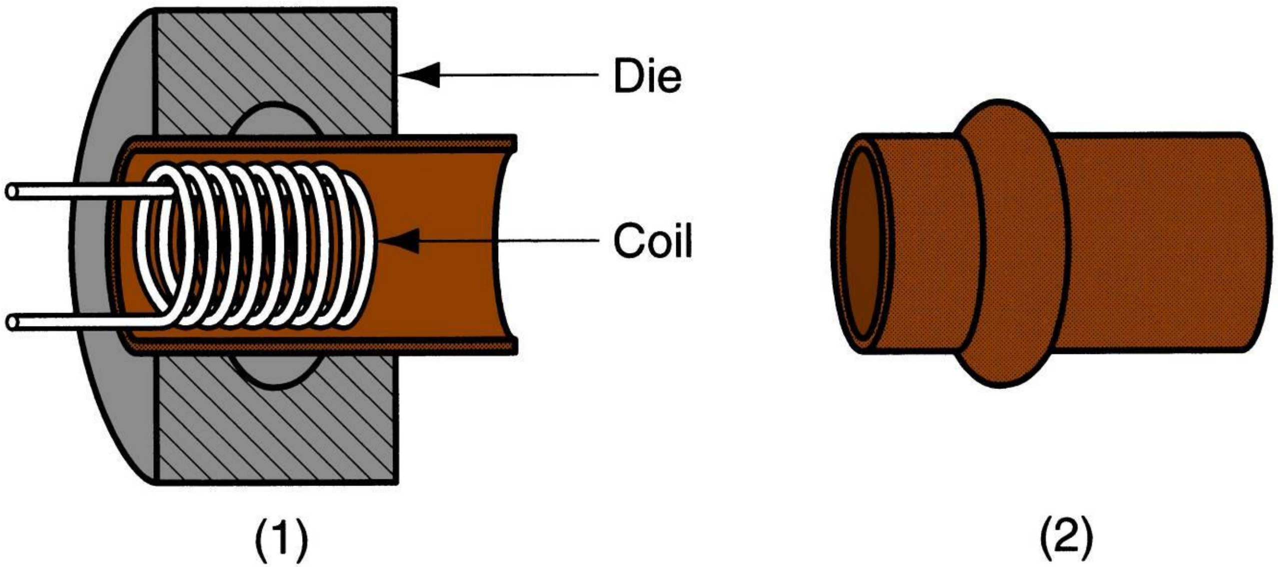


Figure 20.47 Electromagnetic forming: (1) setup in which coil is inserted into tubular workpart surrounded by die; (2) formed part.