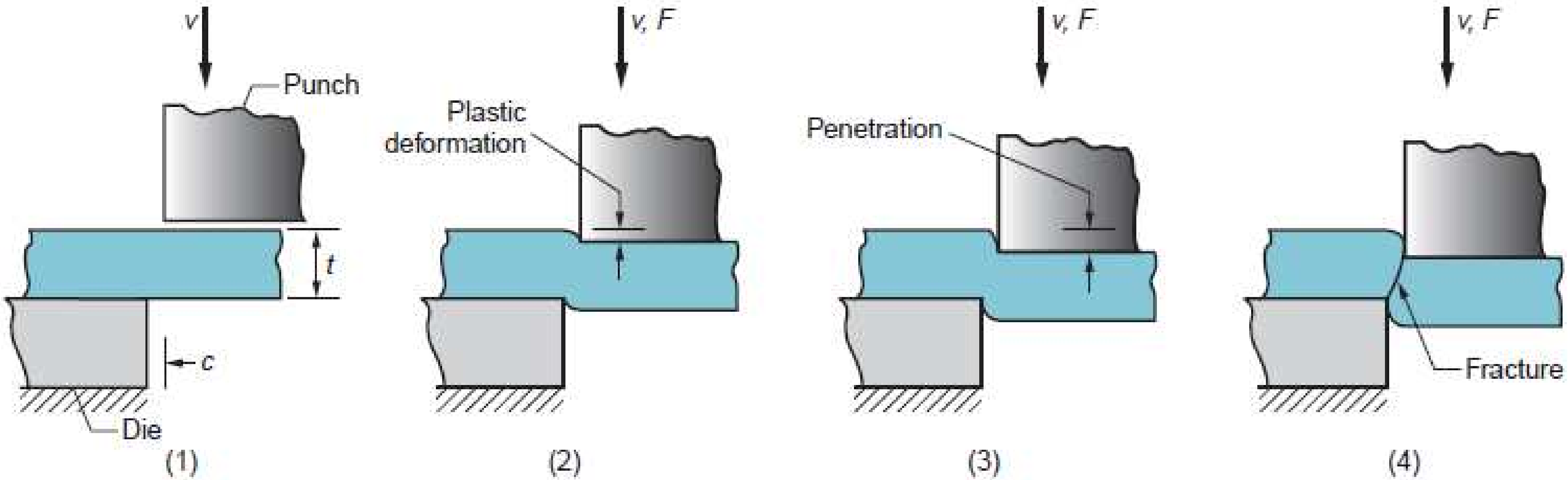
SHEET METALWORKING

SHEET METALWORKING

- Sheet metalworking includes cutting and forming operations performed on relatively thin sheets of metal.
- Typical sheet-metal thicknesses are between 0.4 mm and 6 mm. When thickness exceeds about 6 mm, the stock is usually referred to as plate rather than sheet.
- The most commonly used sheet metal is low carbon steel (0.06%–0.15% C typical). Its low cost and good formability, combined with sufficient strength for most product applications, make it ideal as a starting material.
- The commercial importance of sheet metalworking is significant. Consider the number of consumer and industrial products that include sheet or plate metal parts: automobile and truck bodies, airplanes, railway cars, locomotives, farm and construction equipment, appliances, office furniture, and more.

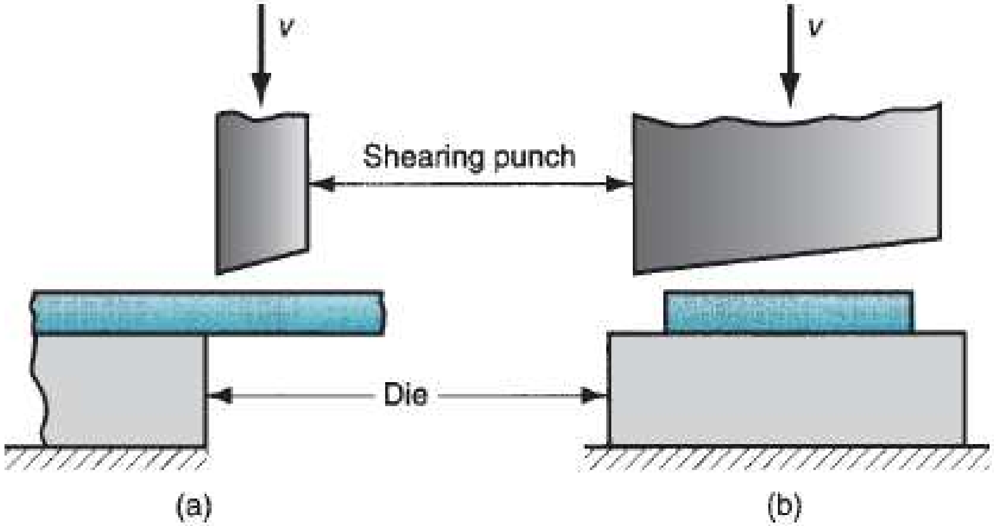
CUTTING OPERATIONS



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; (3) punch compresses and penetrates into work causing a smooth cut surface; and (4) fracture is initiated at the opposing cutting edges that separate the sheet. Symbols v and F indicate motion and applied force, respectively, t = stock thickness, c = clearance.

SHEARING, BLANKING, AND PUNCHING

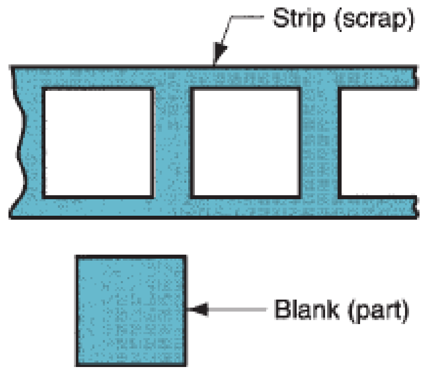
- Shearing is a sheet-metal cutting operation along a straight line between two cutting edges



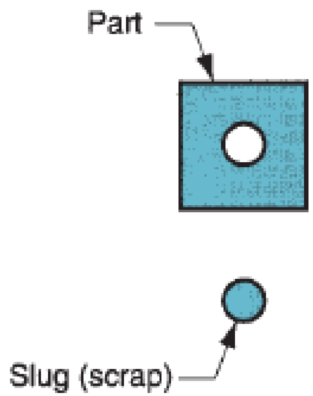
Shearing operation: (a) side view of the shearing operation; (b) front view of power shears equipped with inclined upper cutting blade. Symbol v indicates motion.

SHEARING, BLANKING, AND PUNCHING

- Blanking involves cutting of the sheet metal along a closed outline in a single step to separate the piece from the surrounding stock.



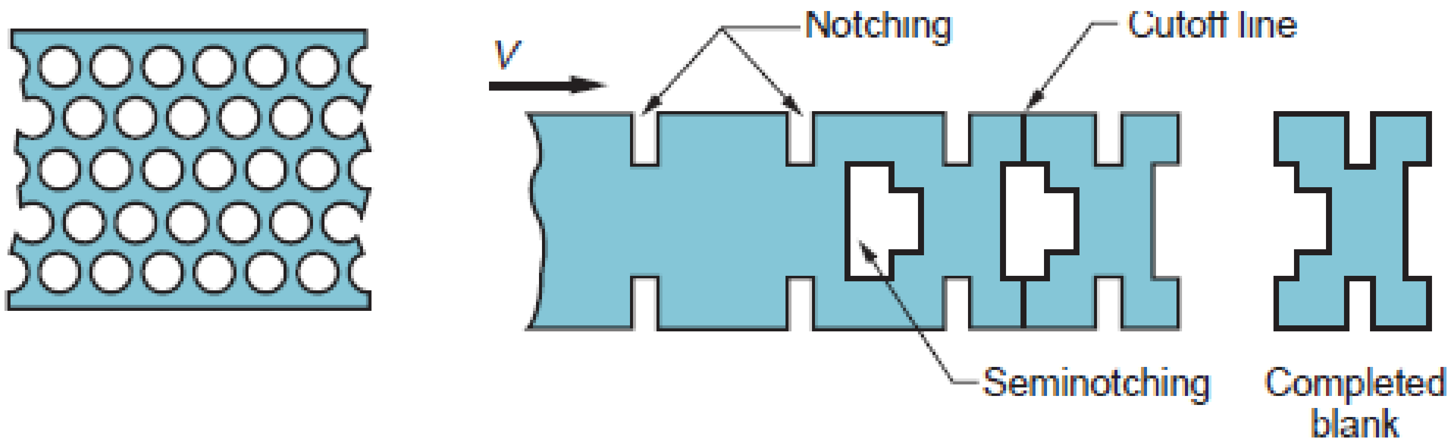
- Punching is similar to blanking except that it produces a hole, and the separated piece is scrap, called the slug.




PowerDirector

OTHER SHEET-METAL-CUTTING OPERATIONS

- **<u>Notching</u>** involves cutting out a portion of metal from the side of the sheet or strip.
- **<u>Seminotching</u>** removes a portion of metal from the interior of the sheet.

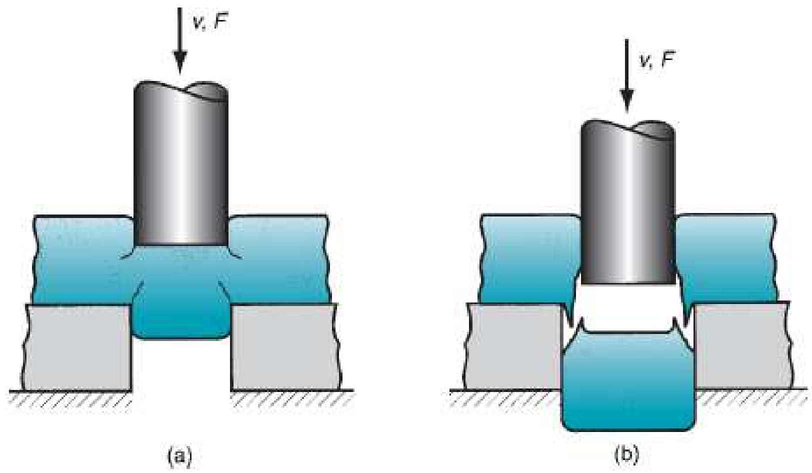


(b) (c)

(b) perforating, (c) notching and seminotching. Symbol v indicates motion of strip.

ENGINEERING ANALYSIS OF SHEET-METAL CUTTING

- **Clearance:** The clearance c in a shearing operation is the distance between the punch and die.
- Typical clearances in conventional pressworking range between 4% and 8% of the sheet-metal thickness t.
- The effect of improper clearances is illustrated in Figure.
- If the clearance is too small, then the fracture lines tend to pass each other, causing a double burnishing and larger cutting forces.
- If the clearance is too large, the metal becomes pinched between the cutting edges and an excessive burr results. In special operations requiring very straight edges, such as shaving and fine blanking, clearance is only about 1% of stock thickness.



ENGINEERING ANALYSIS OF SHEET-METAL CUTTING

- The correct clearance depends on sheet-metal type and thickness. The recommended clearance can be calculated by the following formula

$$c = A_c t \tag{20.1}$$

where c = clearance, mm (in); A_c = clearance allowance; and t = stock thickness, mm (in). The clearance allowance is determined according to type of metal. For convenience, metals are classified into three groups given in Table 20.1, with an associated allowance value for each group.

TABLE 20.1 Clearance allowance value for three sheet-metal groups.

Metal Group	A_c
1100S and 5052S aluminum alloys, all tempers	0.045
2024ST and 6061ST aluminum alloys; brass, all tempers; soft cold-rolled steel, soft stainless steel	0.060
Cold-rolled steel, half hard; stainless steel, half-hard and full-hard	0.075

ENGINEERING ANALYSIS OF SHEET-METAL CUTTING

These calculated clearance values can be applied to conventional blanking and hole-punching operations to determine the proper punch and die sizes. The die opening must always be larger than the punch size (obviously). Whether to add the clearance value to the die size or subtract it from the punch size depends on whether the part being cut out is a blank or a slug, as illustrated in Figure 20.6 for a circular part. Because of the geometry of the sheared edge, the outer dimension of the part cut out of the sheet will be larger than the hole size. Thus, punch and die sizes for a round blank of diameter D_b are determined as

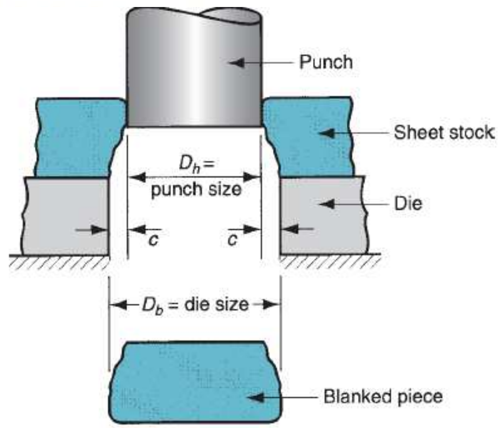
$$\text{Blanking punch diameter} = D_b - 2c$$

$$\text{Blanking die diameter} = D_b$$

Punch and die sizes for a round hole of diameter D_h are determined as:

$$\text{Hole punch diameter} = D_h$$

$$\text{Hole die diameter} = D_h + 2c$$



In order for the slug or blank to drop through the die, the die opening must have an ***angular clearance*** (see Figure 20.7) of 0.25° to 1.5° on each side.

ENGINEERING ANALYSIS OF SHEET-METAL CUTTING

- Cutting Forces Estimates of cutting force are important because this force determines the size (tonnage) of the press needed. Cutting force F in sheet metalworking can be determined by

$$F = StL \tag{20.4}$$

where S = shear strength of the sheet metal, MPa (lb/in^2); t = stock thickness, mm (in), and L = length of the cut edge, mm (in). In blanking, punching, slotting, and similar operations, L is the perimeter length of the blank or hole being cut. The minor effect of clearance in determining the value of L can be neglected. If shear strength is unknown, an alternative way of estimating the cutting force is to use the tensile strength:

$$F = 0.7(TS)tL \tag{20.5}$$

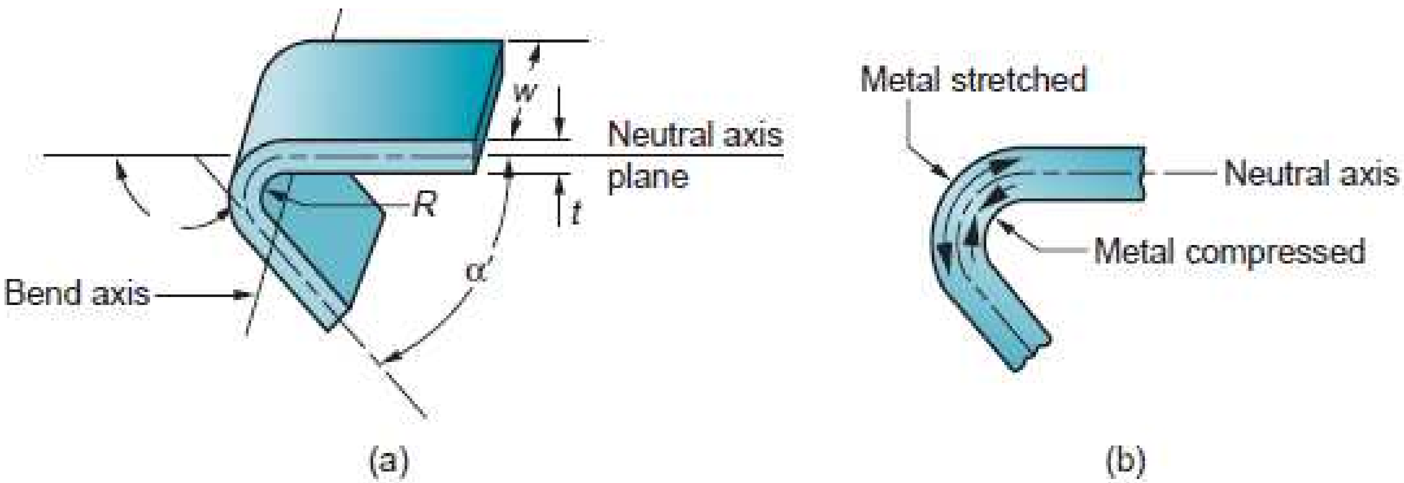
where TS = ultimate tensile strength MPa (lb/in^2).

Problem

- Around disk of 150-mm diameter is to be blanked from a strip of 3.2-mm, half-hard cold rolled steel whose shear strength = 310 MPa. Determine (a) the appropriate punch and die diameters, and (b) blanking force.

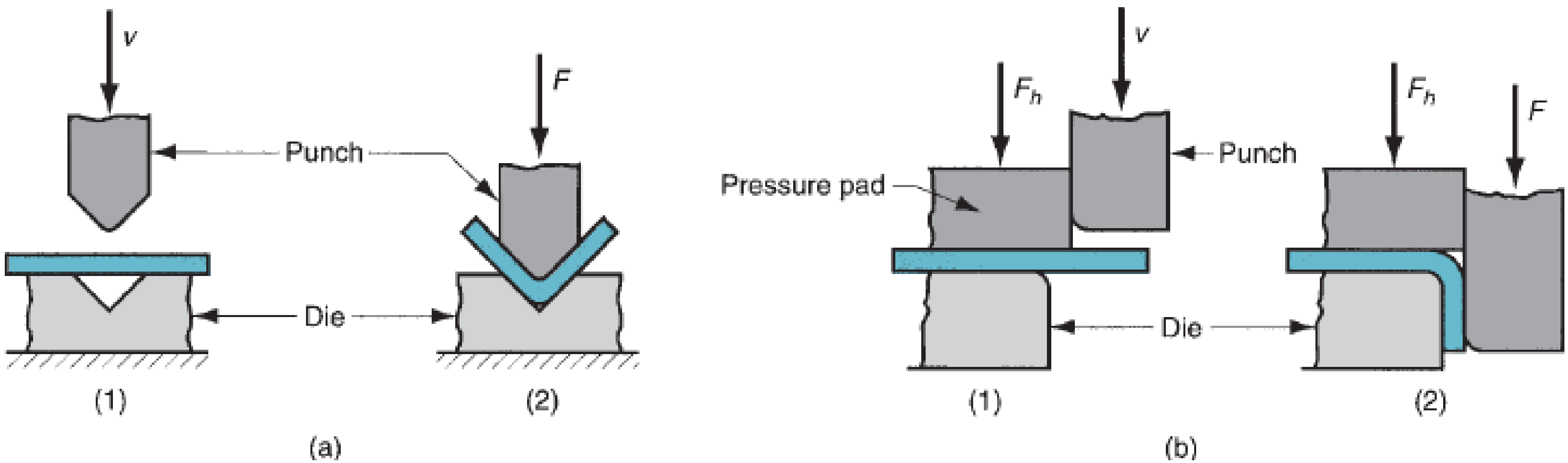
BENDING OPERATIONS

- Bending in sheet-metalwork is defined as the straining of the metal around a straight axis



(a) Bending of sheet metal; (b) both compression and tensile elongation of the metal occur in bending.

V-BENDING AND EDGE BENDING



Two common bending methods: (a) V-bending and (b) edge bending; (1) before and (2) after bending. Symbols: v = motion, F = applied bending force, Fh = blank.

ENGINEERING ANALYSIS OF BENDING

- **Bend Allowance:** If the bend radius is small relative to stock thickness, the metal tends to stretch during bending.

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

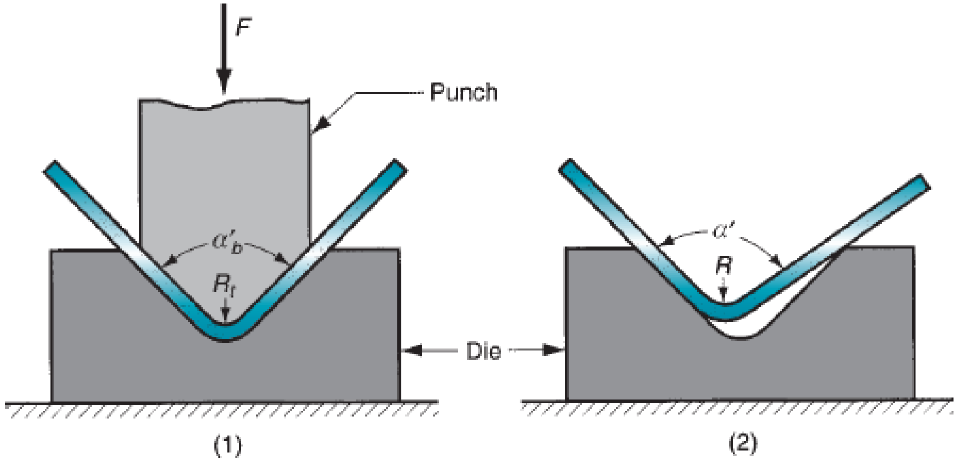
where A_b = bend allowance, mm(in); a = bend angle, degrees; R = bend radius, mm(in); t = stock thickness, mm(in); and K_{ba} is factor to estimate stretching. The following design values are recommended for K_{ba}: if <2t, K_{ba} = 0.33; and if R 2t, K_{ba} = 0.50.

ENGINEERING ANALYSIS OF BENDING

- **<u>Springback:</u>** When the bending pressure is removed at the end of the deformation operation, elastic energy remains in the bent part, causing it to recover partially toward its original shape

$$SB = \frac{\alpha' - \alpha'_t}{\alpha'_t}$$

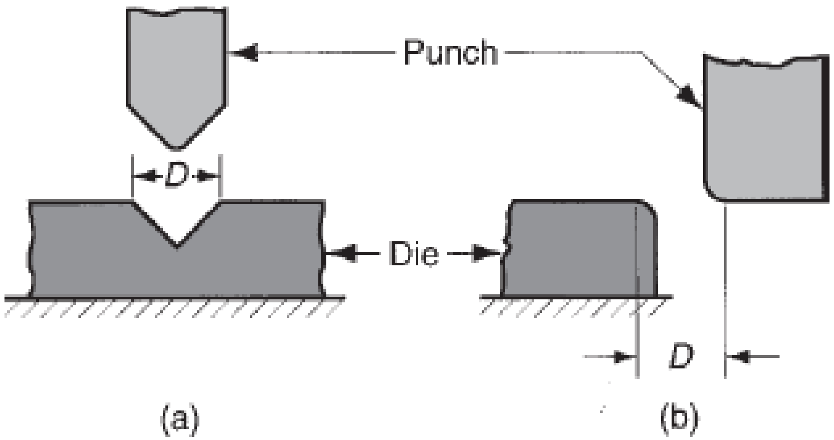
where SB = springback; α' = included angle of the sheet-metal part, degrees; and α'_t = included angle of the bending tool, degrees



Springback in bending shows itself as a decrease in bend angle and an increase in bend radius: (1) during the operation, the work is forced to take the Radius R_t and included angle α'_t = determined by the bending tool (punch in V-bending); (2) after the punch is removed, the work springs back to radius R and included angle α'. Symbol: F = applied bending force.

ENGINEERING ANALYSIS OF BENDING

- **Bending Force:** The force required to perform bending depends on the geometry of the punch-and-die and the strength, thickness, and length of the sheet metal.

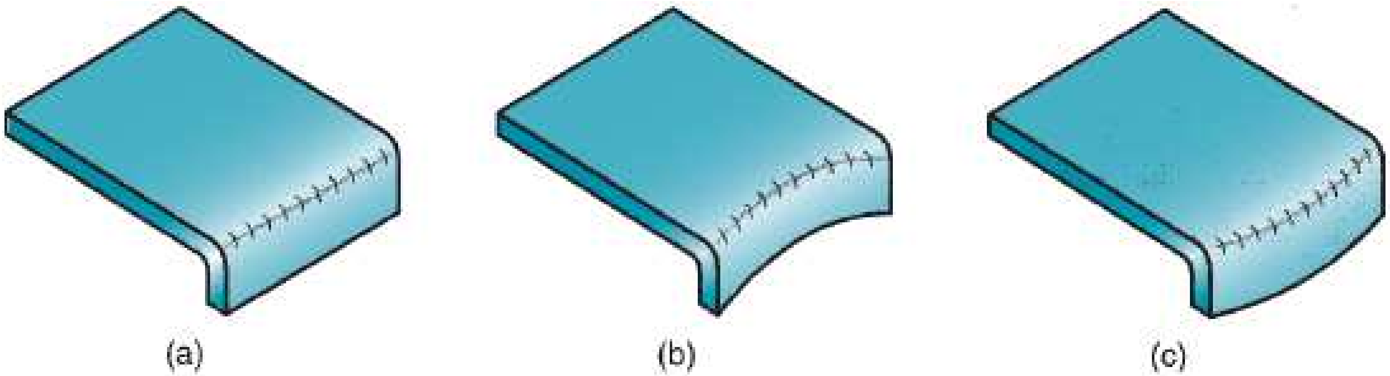


Die opening dimension D: (a) V-die, (b) wiping die.

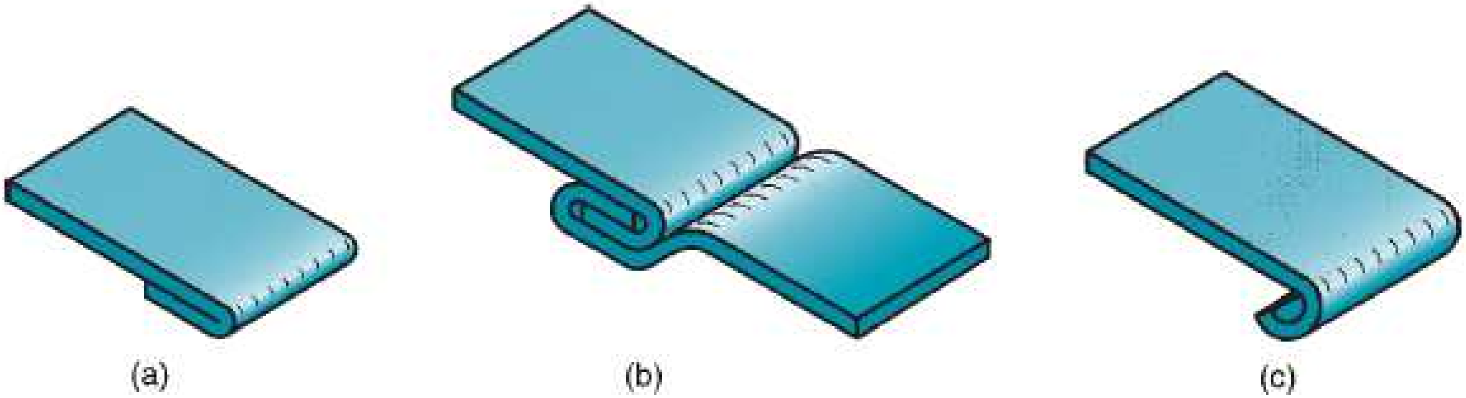
$$F = \frac{K_{bf}(TS)wt^2}{D}$$

Where F=bending force, N(lb); TS=tensile strength of the sheet metal, MPa (lb/in$_2$); w=width of part in the direction of the bend axis, mm (in); t ¼ stock thickness, mm (in); and D = die opening dimension mm (in). And Kbf is a constant that accounts for differences encountered in an actual bending process. Its value depends on type of bending: for V-bending, Kbf = 1.33; and for edge bending, Kbf = 0.33.

OTHER BENDING AND FORMING OPERATIONS

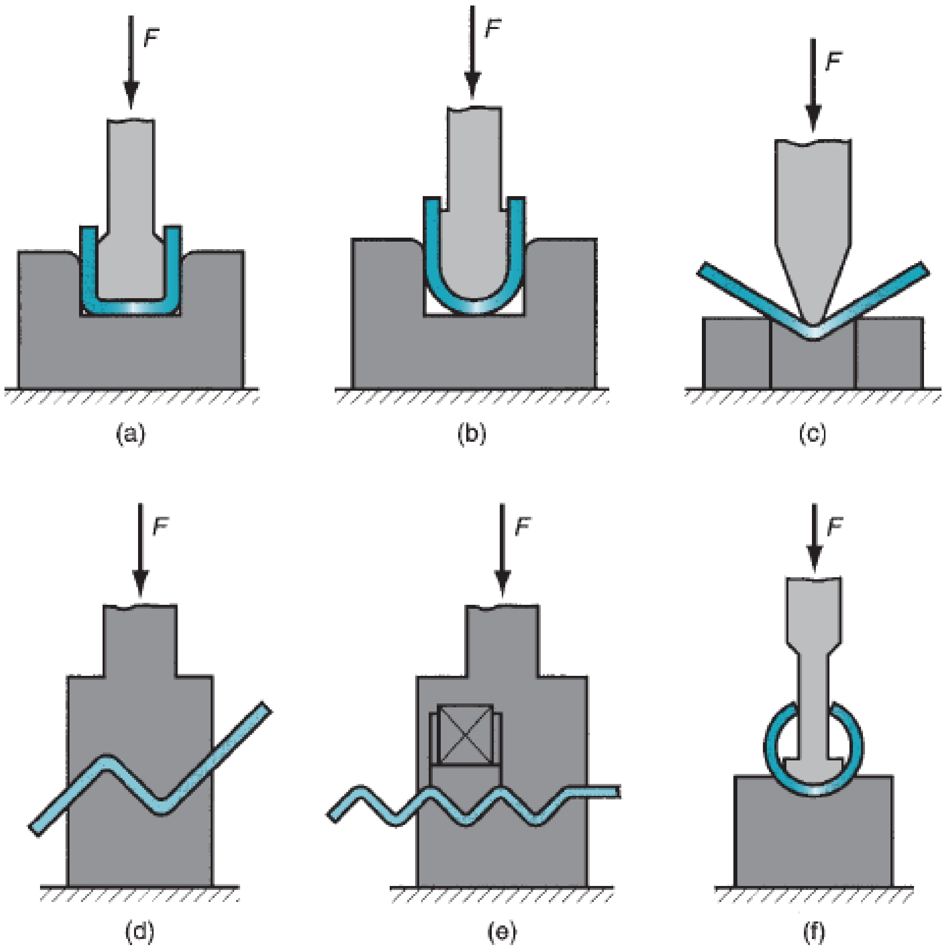


Flanging: (a) straight flanging, (b) stretch flanging, and(c) shrink flanging.



(a) Hemming, (b) seaming, and (c) curling.

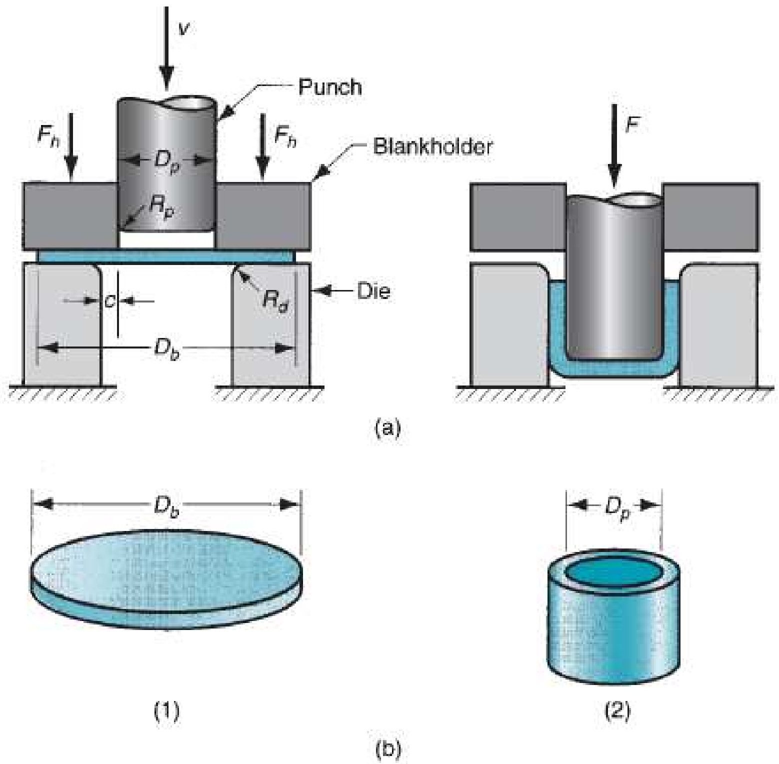
OTHER BENDING AND FORMING OPERATIONS



Miscellaneous bending operations: (a) channel bending, (b) U-bending, (c) air bending, (d) offset bending, (e) corrugating, and (f) tube forming. Symbol: F=applied force.

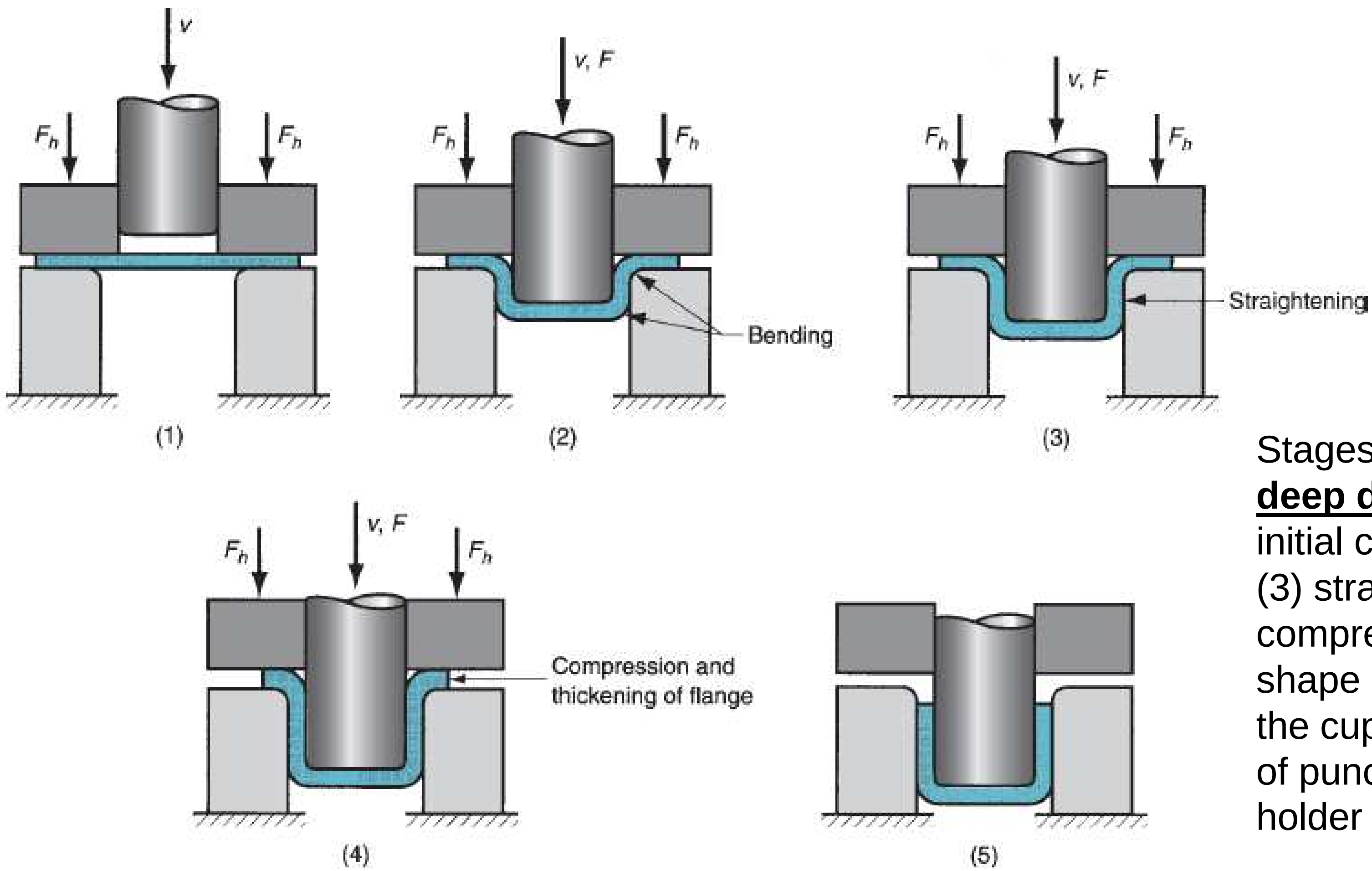
DRAWING

- Drawing is a sheet-metal-forming operation used to make cup-shaped, box-shaped, or other complex-curved and concave parts.



(a) Drawing of a cup-shaped part: (1) start of operation before punch contacts work, and (2) near end of stroke; and (b) corresponding workpart: (1) starting blank, and (2) drawn part. Symbols: c= clearance, D_b = blank diameter, D_p = punch diameter, R_d = die corner radius, R_p = punch corner radius, F = drawing force, F_h = holding force.

MECHANICS OF DRAWING



Stages in deformation of the work in **deep drawing**: (1) punch makes initial contact with work, (2) bending, (3) straightening, (4) friction and compression, and (5) final cup shape showing effects of thinning in the cup walls. Symbols: v = motion of punch, F = punch force, F_h = blank holder force.

ENGINEERING ANALYSIS OF DRAWING

- Measures of Drawing One of the measures of the severity of a deep drawing operation is the drawing ratio DR.

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

This is most easily defined for a cylindrical shape as the ratio of blank diameter D_b to punch diameter D_p

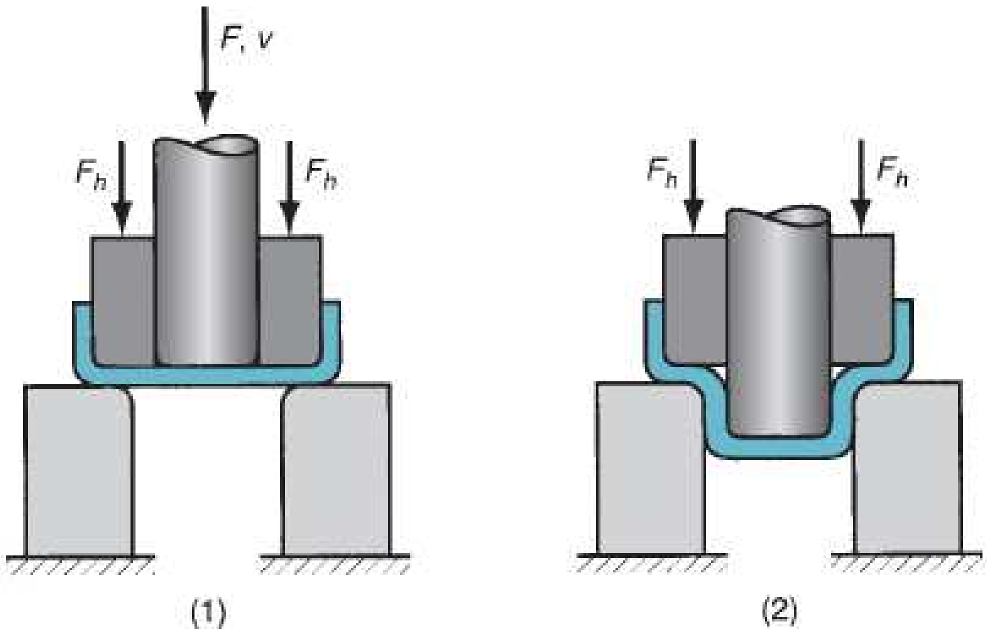
- Another way to characterize a given drawing operation is by the reduction r, where

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

It is very closely related to drawing ratio. Consistent with the previous limit on DR (DR ≤ 2.0), the value of reduction r should be less than 0.50.

OTHER DRAWING OPERATIONS

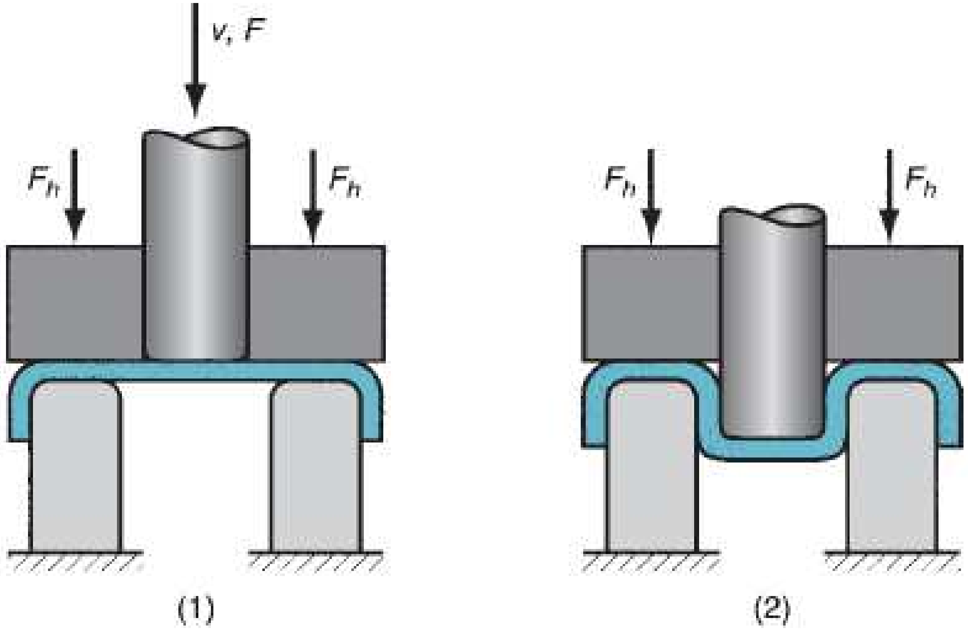
- Redrawing If the shape change required by the part design is too severe (drawing ratio is too high), complete forming of the part may require more than one drawing step.



Redrawing of a cup:
(1) start of redraw, and
(2)End of stroke. Symbols:
v = punch velocity, F = applied punch force, F_h = blankholder force.

OTHER DRAWING OPERATIONS

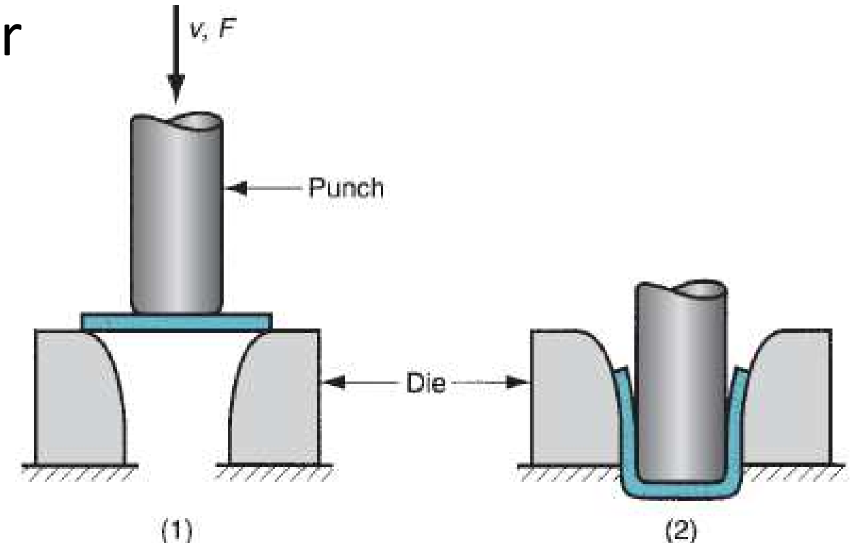
- A related operation is **reverse drawing**, in which a drawn part is positioned face down on the die so that the second drawing operation produces a configuration



Reverse drawing: (1) start and (2) completion. Symbols: v = punch velocity, F = applied punch force, F_h = blankholder force

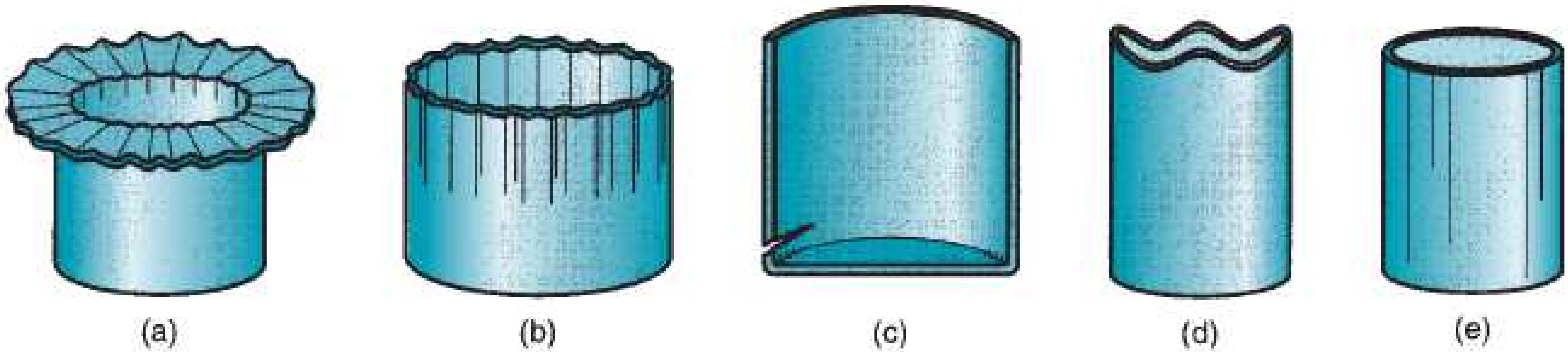
OTHER DRAWING OPERATIONS

- **Drawing Without a Blankholder** One of the primary functions of the blankholder is to prevent wrinkling of the flange while the cup is being drawn.
- The tendency for wrinkling is reduced as the thickness-to-diameter ratio of the blank increases. If the t=Db ratio is large enough, drawing can be accomplished without a blankholder



Drawing without a blankholder: (1) start of process, (2) end of stroke. Symbols v and F indicate motion and applied force, respectively.

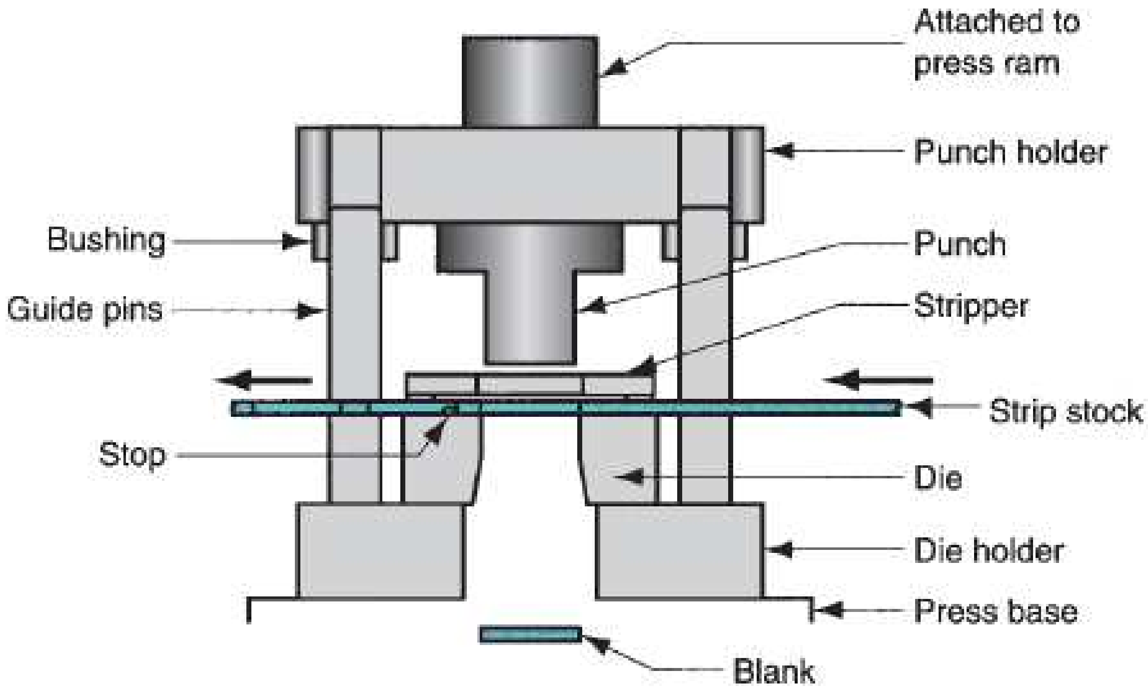
DEFECTS IN DRAWING



Common defects in drawn parts: (a) wrinkling can occur either in the flange or (b) in the wall, (c) tearing, (d) earing, and (e) surface scratches.

DIES AND PRESSES FOR SHEET-METAL PROCESSES

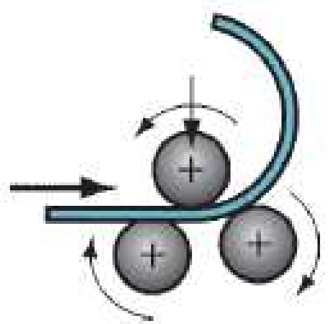
- **Dies**: Nearly all of the preceding pressworking operations are performed with conventional punch-and-die tooling. The tooling is referred to as a die.



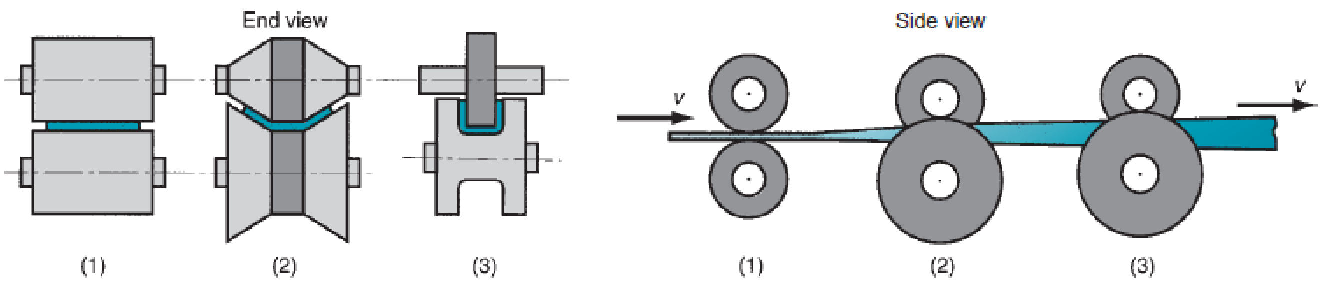
Components of a punch and die for a blanking operation.

ROLL BENDING AND ROLL FORMING

- The operations described in this section use rolls to form sheet metal. Roll bending is an operation in which (usually) large sheet-metal or plate-metal parts are formed into curved sections by means of rolls.
- Roll forming (also called contour roll forming) is a continuous bending process in which opposing rolls are used to produce long sections of formed shapes from coil or strip stock.



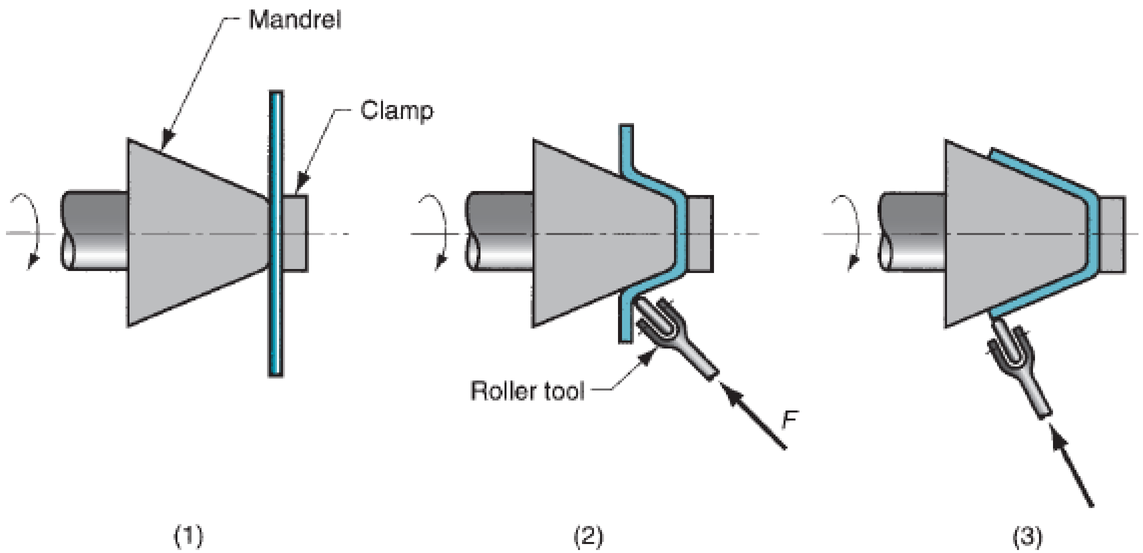
Roll bending.



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

SPINNING

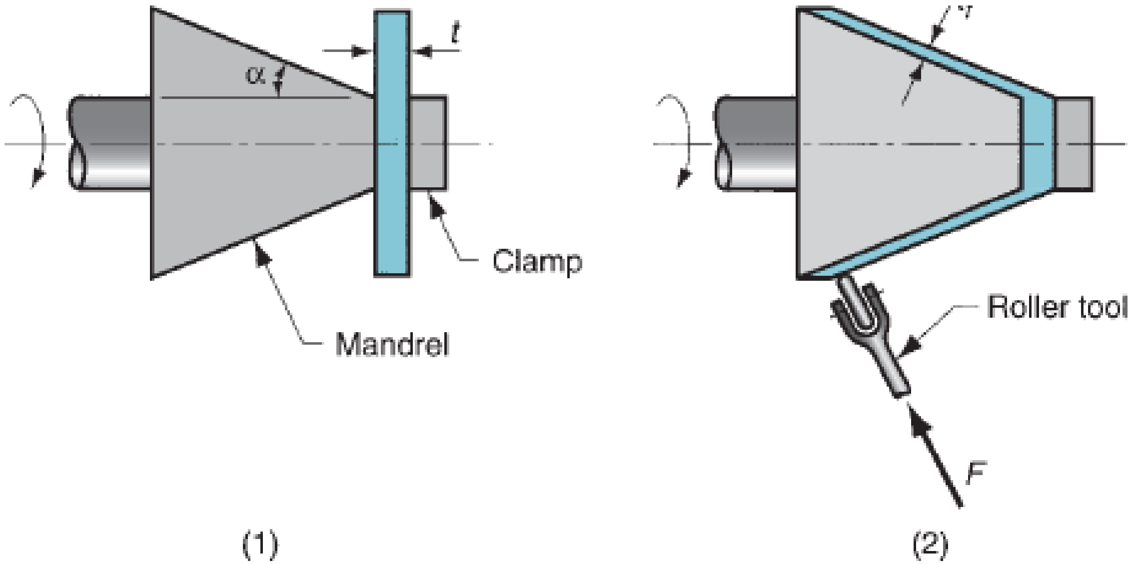
- Conventional spinning is the basic spinning operation.



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

SPINNING

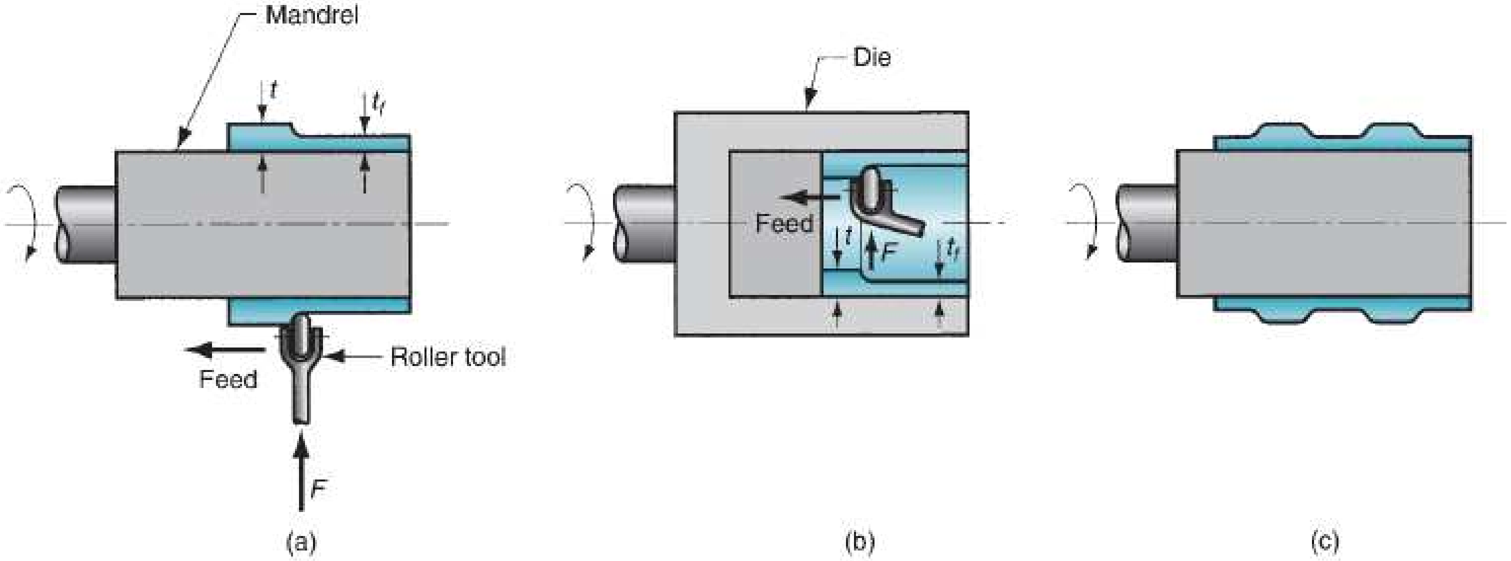
- **Shear Spinning** In shear spinning, the part is formed over the mandrel by a shear deformation process in which the outside diameter remains constant and the Wall thickness is therefore reduced



Shear spinning: (1) setup and (2) completion of process.

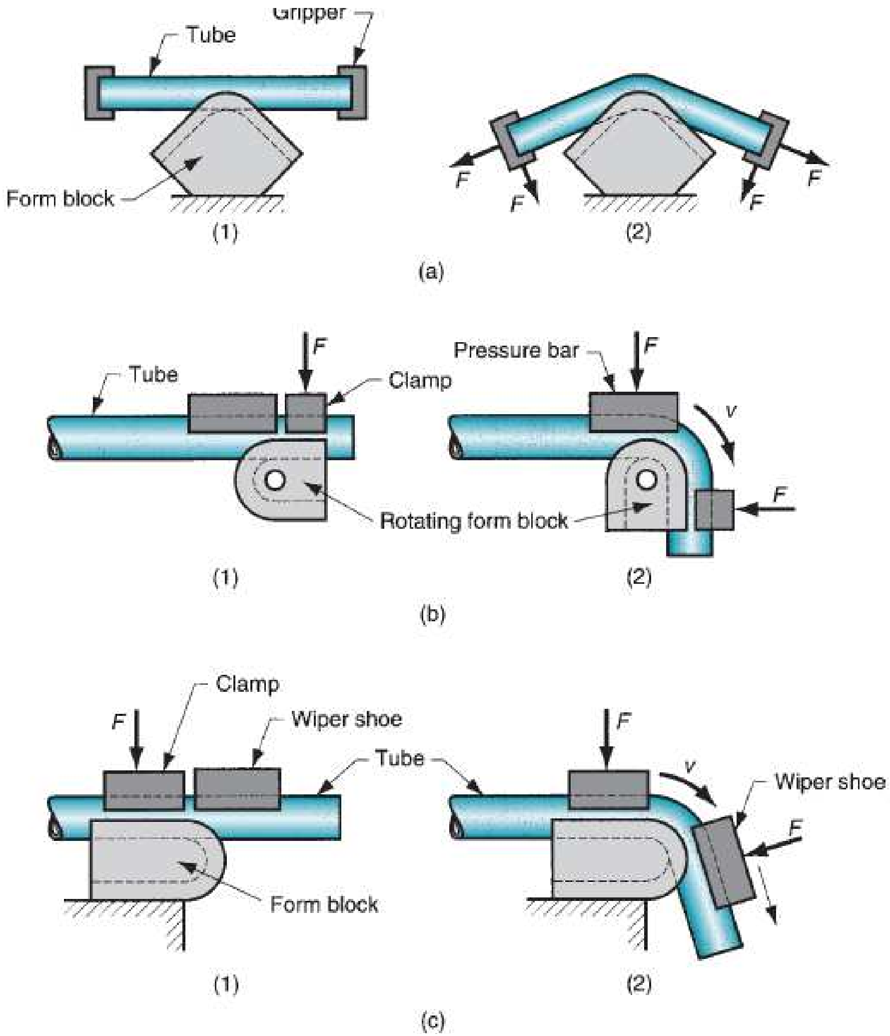
SPINNING

- Tube spinning is used to reduce the wall thickness and increase the length of a tube by means of a roller applied to the work over a cylindrical mandrel



Tube spinning: (a) external; (b) internal; and (c) profiling.

BENDING OF TUBE STOCK



Tube
bending methods:
(a) stretch bending,
(b) draw bending, and
(c) compression bending.
For each method: (1) start of process, and (2) during bending. Symbols v and F indicate motion and applied force, respectively.

REVIEW QUESTIONS

1. What are the differences between bulk deformation processes and sheet metal processes?
2. Identify the three basic types of sheet metalworking operations.
3. In conventional sheet metalworking operations, (a) what is the name of the tooling and (b) what is the name of the machine tool used in the operations?
4. What is the difference between a notching operation and a seminotching operation?
5. Describe each of the two types of sheet-metalbending operations: V-bending and edge bending.
6. For what is the bend allowance intended to compensate?
7. What is springback in sheet-metal bending?
8. Define drawing in the context of sheet metalworking.
9. What are some of the possible defects in drawn sheet-metal parts?
10. What is stretch forming?
11. Identify the principal components of a stamping die that performs blanking.
12. A power shears is used to cut soft cold-rolled steel that is 4.75 mm thick. At what clearance should the shears be set to yield an optimum cut?
13. A blanking operation is to be performed on 2.0-mm thick cold-rolled steel (half hard). The part is circular with diameter=75.0 mm. Determine the appropriate punch and die sizes for this operation.
14. The foreman in the pressworking section comes to you with the problem of a blanking operation that is producing parts with excessive burrs. (a) What are the possible reasons for the burrs? (b)What can be done to correct the condition?
15. A bending operation is to be performed on 4.0-mm thick cold-rolled steel sheet that is 25 mm wide and 100 mm long. The sheet is bent along the 25 mm direction, so that the bend is 25 mm long. The resulting sheet metal part has an acute angle of 30 and a bend radius of 6 mm. Determine (a) the bend allowance and (b) the length of the neutral axis of the part after the bend. (Hint: the length of the neutral axis before the bend ¼ 100.0 mm).
16. Extrusion is a fundamental shaping process. Describe it.
17. What is the difference between deep drawing and bar drawing?