19 Sheet Metalworking

Chapter Contents

Sheet metalworking includes cutting and forming operations performed on relatively thin sheets of metal. Typical sheet-metal thicknesses are between 0.4 mm (1/64 in) and 6 mm (1/4 in). When thickness exceeds about 6 mm, the stock is usually referred to as plate rather than sheet. The sheet or plate stock used in sheet metalworking is produced by flat rolling (Section 18.1). The most commonly used sheet metal is low carbon steel (0.06%–0.15% C is 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. Although these examples are conspicuous because they have sheet-metal exteriors, many of their internal components are also made of sheet or plate stock. Sheet-metal parts are generally characterized by high strength, good dimensional accuracy, good surface finish, and relatively low cost. For components that must be made in large quantities, economical mass-production operations can be designed to process the parts. Aluminum beverage cans are a prime example.

Sheet-metal processing is usually performed at room temperature (cold working). The exceptions are when the stock is thick, the metal is brittle, or the deformation is significant. These are usually cases of warm working rather than hot working.

Most sheet-metal operations are performed on machine tools called ***presses***. The term ***stamping press*** is used to distinguish these presses from forging and extrusion presses. The tooling that performs sheet metalwork is called a ***punch-and-die***; the term ***stamping die*** is also used. The sheet-metal products are called ***stampings***. To facilitate mass production, the sheet metal is often presented to the press as long strips or coils. Various types of punch-and-die tooling and stamping presses are described in Section 19.5. Final sections of the chapter cover various operations that do not utilize conventional punch-and-die tooling, and most of them are not performed on stamping presses.

The three major categories of sheet-metal processes are (1) cutting, (2) bending, and (3) drawing. Cutting is used to separate large sheets into smaller pieces, to cut out part perimeters, and to make holes in parts. Bending and drawing are used to form sheet-metal parts into their required shapes.

19.1 Cutting Operations

Cutting of sheet metal is accomplished by a shearing action between two sharp cutting edges. The shearing action is depicted in the four stop-action sketches of Figure 19.1, in which the upper cutting edge (the punch) sweeps down past a stationary lower cutting edge (the die). As the punch begins to push into the work, ***plastic deformation*** occurs in the surfaces of the sheet. As the punch moves downward, ***penetration*** occurs in which the punch compresses the sheet and cuts into the metal. This penetration zone is generally about one-third the thickness of the sheet. As the punch continues to travel into the work, ***fracture*** is initiated in the work at the two cutting edges. If the clearance between the punch and die is correct, the two fracture lines meet, resulting in a clean separation of the work into two pieces.

The sheared edges of the sheet have characteristic features as in Figure 19.2. At the top of the cut surface is a region called the ***rollover***. This corresponds to the depression made by the punch in the work prior to cutting. It is where initial plastic

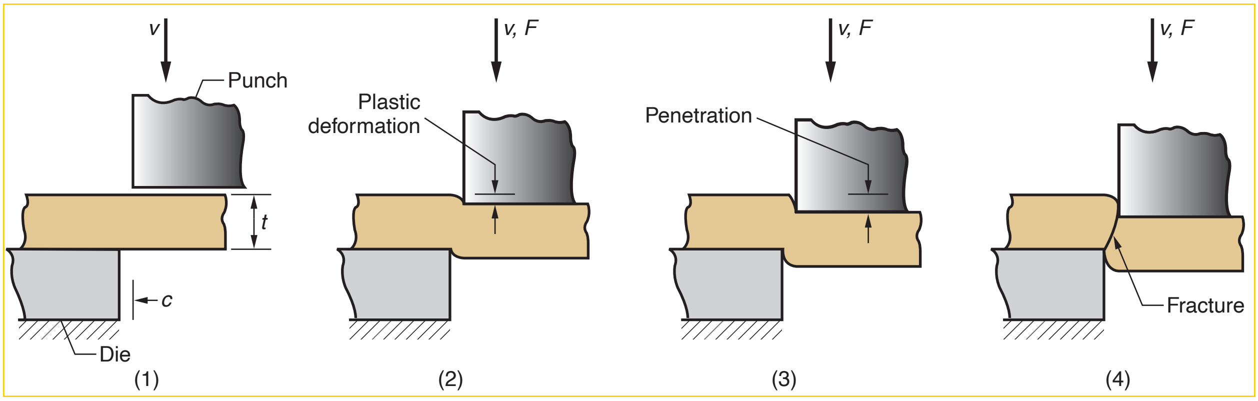

FIGURE 19.1 Shearing of sheet metal between two cutting edges: (1) just before the punch contacts work; (2) punch begins to push into work, causing plastic deformation; (3) punch compresses and penetrates into work causing a smooth cut surface; and (4) fracture is initiated at the opposing cutting edges that separate the sheet. Symbols v and F indicate motion and applied force, respectively, t = stock thickness, c = clearance.

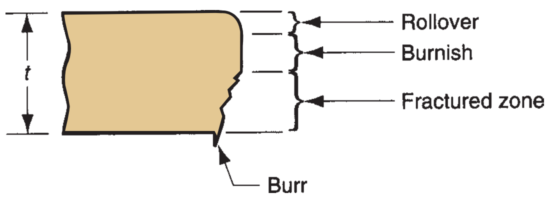

FIGURE 19.2 Characteristic sheared edges of the work.

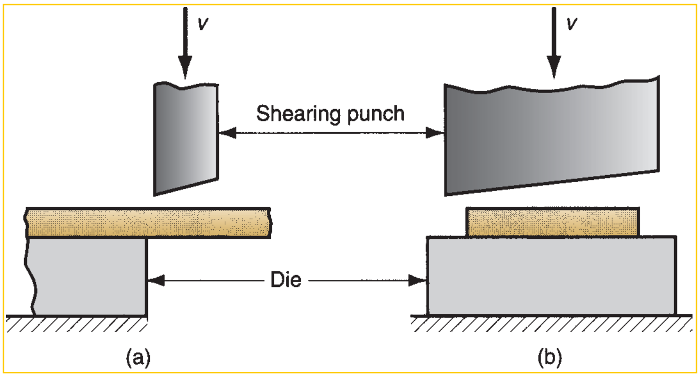

FIGURE 19.3 Shearing operation: (a) side view of the shearing operation; (b) front view of power shears equipped with inclined upper cutting blade. Symbol *v* indicates motion.

deformation occurred in the work. Just below the rollover is a relatively smooth region called the ***burnish***. This results from penetration of the punch into the work before fracture began. Beneath the burnish is the ***fractured zone***, a relatively rough surface of the cut edge where continued downward movement of the punch caused fracture of the metal. Finally, at the bottom of the edge is a ***burr***, a sharp corner on the edge caused by elongation of the metal during final separation of the two pieces.

19.1.1 SHEARING, BLANKING, AND PUNCHING

The three most important operations in pressworking that cut metal by the shearing mechanism just described are shearing, blanking, and punching.

Shearing is a sheet-metal cutting operation along a straight line between two cutting edges, as shown in Figure 19.3(a). Shearing is typically used to cut large sheets into smaller sections for subsequent pressworking operations. It is performed on a machine called a ***power shears***, or ***squaring shears***. The upper blade of the power shears is often inclined, as shown in Figure 19.3(b), to reduce the required cutting force.

Blanking involves cutting of the sheet metal along a closed outline in a single step to separate the piece from the surrounding stock, as in Figure 19.4(a). The part that is cut out is the desired product in the operation and is called the ***blank***. ***Punching*** is similar to blanking except that it produces a hole, and the separated piece is scrap, called the ***slug***. The remaining stock is the desired part. The distinction is illustrated in Figure 19.4(b).

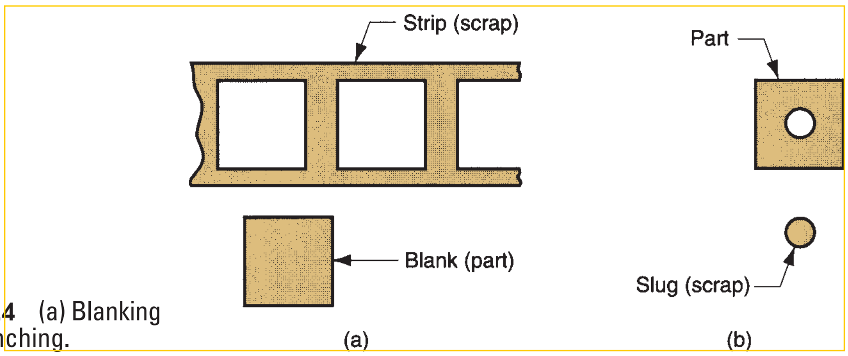

FIGURE 19.4 (a) Blanking and (b) punching.

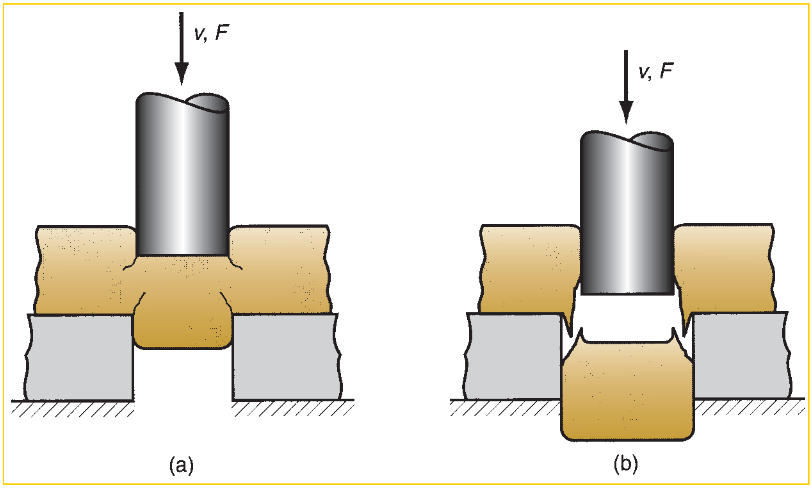

FIGURE 19.5 Effect of clearance: (a) clearance too small causes less-than-optimal fracture and excessive forces; and (b) clearance too large causes oversized burr. Symbols v and F indicate motion and applied force, respectively.

19.1.2 ENGINEERING ANALYSIS OF SHEET-METAL CUTTING

Process parameters in sheet-metal cutting are clearance between punch and die, stock thickness, type of metal and its strength, and length of the cut. This section defines these parameters and some of the relationships among them.

Clearance The clearance c in a shearing operation is the distance between the punch and die, as shown in Figure 19.1(a). Typical clearances in conventional press-working range between 4% and 8% of the sheet-metal thickness t. The effect of improper clearances is illustrated in Figure 19.5. 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 (Section 19.1.3), clearance is only about 1% of stock thickness.

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

$$c = A_c t \tag{19.1}$$

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 19.1, with an associated allowance value for each group.

TABLE • 19.1 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

Compiled from [3].

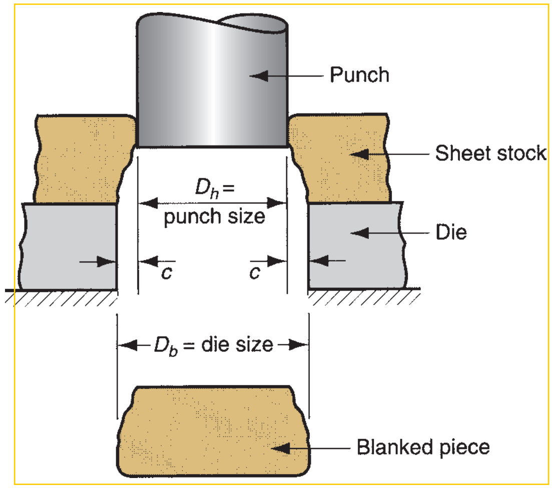

FIGURE 19.6 Die size determines blank size D_b; punch size determines hole size D_h; $c =$ clearance.

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 19.6 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

$$\text{Blanking punch diameter} = D_b - 2c \tag{19.2a}$$

$$\text{Blanking die diameter} = D_b \tag{19.2b}$$

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

$$\text{Hole punch diameter} = D_h \tag{19.3a}$$

$$\text{Hole die diameter} = D_h + 2c \tag{19.3b}$$

In order for the slug or blank to drop through the die, the die opening must have an ***angular clearance*** (see Figure 19.7) of $0.25°$ to $1.5°$ on each side.

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

$$F = StL \tag{19.4}$$

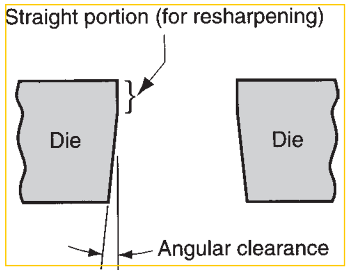

FIGURE 19.7 Angular clearance.

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:

$$F = 0.7(TS)tL \tag{19.5}$$

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

These equations for estimating cutting force assume that the entire cut along the sheared edge length L is made at the same time. In this case the cutting force will be a maximum. It is possible to reduce the maximum force by using an angled cutting edge on the punch or die, as in Figure 19.3(b). The angle (called the ***shear angle***), spreads the cut over time and reduces the force experienced at any one moment. However, the total energy required in the operation is the same, whether it is concentrated into a brief moment or distributed over a longer time period.

Example 19.1 Blanking clearance and force

A round 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.

Solution: (a) From Table 19.1, the clearance allowance for half-hard cold-rolled steel is A_c = 0.075. Accordingly,

$$c = 0.075(3.2 \text{ mm}) = \mathbf{0.24\ mm}$$

The blank is to have a diameter = 150 mm, and die size determines blank size. Therefore,

$$\text{Die opening diameter} = \mathbf{150.00\ mm}$$
$$\text{Punch diameter} = 150 - 2\,(0.24) = \mathbf{149.52\ mm}$$

(b) To determine the blanking force, assume that the entire perimeter of the part is blanked at one time. The length of the cut edge is

$$L = \pi D_b = 150\pi = 471.2 \text{ mm}$$

and the force is

$$F = 310(471.2)(3.2) = \mathbf{467{,}469\ N}\ (\sim 53 \text{ tons})$$

19.1.3 OTHER SHEET-METAL-CUTTING OPERATIONS

In addition to shearing, blanking, and punching, there are several other cutting operations in pressworking. The cutting mechanism in each case involves the same shearing action discussed above.

Cutoff and Parting Cutoff is a shearing operation in which blanks are separated from a sheet-metal strip by cutting the opposite sides of the part in sequence, as shown in Figure 19.8(a). With each cut, a new part is produced. The features of

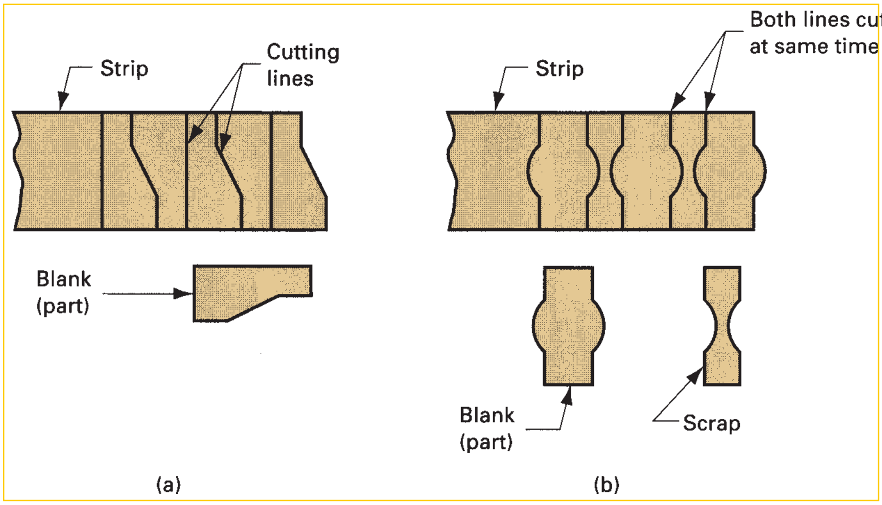

FIGURE 19.8 (a) Cutoff and (b) parting.

a cutoff operation that distinguish it from a conventional shearing operation are (1) the cut edges are not necessarily straight, and (2) the blanks can be nested on the strip in such a way that scrap is avoided.

Parting involves cutting a sheet-metal strip by a punch with two cutting edges that match the opposite sides of the blank, as shown in Figure 19.8(b). This might be required because the part outline has an irregular shape that precludes perfect nesting of the blanks on the strip. Parting is less efficient than cutoff in the sense that it results in some wasted material.

Slotting, Perforating, and Notching Slotting is the term sometimes used for a punching operation that cuts out an elongated or rectangular hole, as pictured in Figure 19.9(a). ***Perforating*** involves the simultaneous punching of a pattern of holes in sheet metal, as in Figure 19.9(b). The hole pattern is usually for decorative purposes, or to allow passage of light, gas, or fluid.

To obtain the desired outline of a blank, portions of the sheet metal are often removed by notching and seminotching. ***Notching*** involves cutting out a portion of metal from the side of the sheet or strip. ***Seminotching*** removes a portion of metal from the interior of the sheet. These operations are depicted in Figure 19.9(c). Seminotching might seem to the reader to be the same as a punching or slotting operation. The difference is that the metal removed by seminotching creates part of the blank outline, while punching and slotting create holes in the blank.

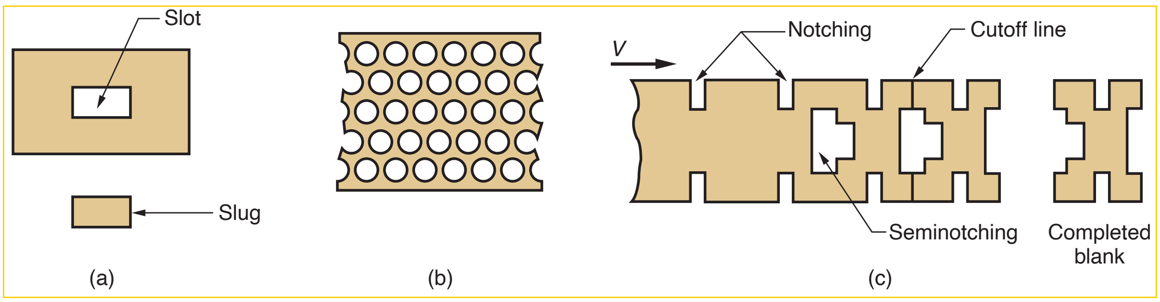

FIGURE 19.9 (a) Slotting, (b) perforating, (c) notching and seminotching. Symbol *v* indicates motion of strip.

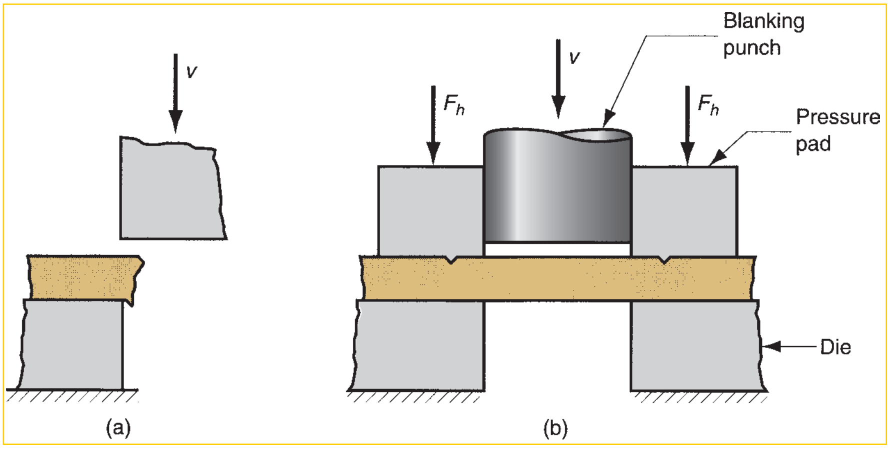

FIGURE 19.10 (a) Shaving, and (b) fine blanking. Symbols: v = motion of punch, F_h = blank holding force.

Trimming, Shaving, and Fine Blanking Trimming is a cutting operation performed on a formed part to remove excess metal and establish size. The term has the same basic meaning here as in forging (Section 18.4). A typical example in sheet metalwork is trimming the upper portion of a deep drawn cup to leave the desired dimensions on the cup.

Shaving is a shearing operation performed with very small clearance to obtain accurate dimensions and cut edges that are smooth and straight, as pictured in Figure 19.10(a). Shaving is typically performed as a secondary or finishing operation on parts that have been previously cut.

Fine blanking is a shearing operation used to blank sheet-metal parts with close tolerances and smooth, straight edges in one step, as illustrated in Figure 19.10(b). At the start of the cycle, a pressure pad with a V-shaped projection applies a holding force F_h against the work adjacent to the punch in order to compress the metal and prevent distortion. The punch then descends with a slower-than-normal velocity and smaller clearances to provide the desired dimensions and cut edges. The process is usually reserved for relatively small stock thicknesses.

19.2 Bending Operations

Bending in sheet-metal work is defined as the straining of the metal around a straight axis, as in Figure 19.11. During the bending operation, the metal on the inside of the neutral plane is compressed, while the metal on the outside of the neutral plane is stretched. These strain conditions can be seen in Figure 19.11(b). The metal is plastically deformed

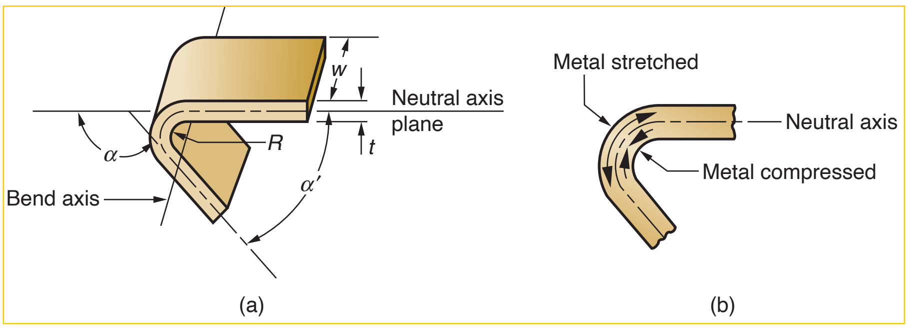

FIGURE 19.11 (a) Bending of sheet metal; (b) both compression and tensile elongation of the metal occur in bending.

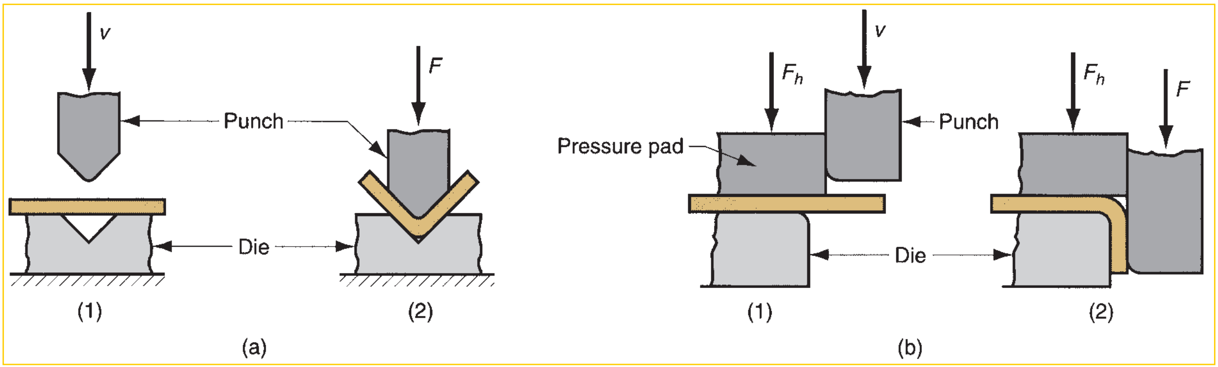

FIGURE 19.12 Two common bending methods: (a) V-bending and (b) edge bending; (1) before and (2) after bending. Symbols: $v =$ motion, $F =$ applied bending force, $F_h =$ blank

so that the bend takes a permanent set upon removal of the stresses that caused it. Bending produces little or no change in the thickness of the sheet metal.

19.2.1 V-BENDING AND EDGE BENDING

Bending operations are performed using punch and die tooling. The two common bending methods and associated tooling are V-bending, performed with a V-die; and edge bending, performed with a wiping die. These methods are illustrated in Figure 19.12.

In ***V-bending***, the sheet metal is bent between a V-shaped punch and die. Included angles ranging from very obtuse to very acute can be made with V-dies. V-bending is generally used for low-production operations. It is often performed on a press brake (Section 19.5.2), and the associated V-dies are relatively simple and inexpensive.

Edge bending involves cantilever loading of the sheet metal. A pressure pad is used to apply a force F_h to hold the base of the part against the die, while the punch forces the part to yield and bend over the edge of the die. In the setup shown in Figure 19.12(b), edge bending is limited to bends of $90°$ or less. More complicated wiping dies can be designed for bend angles greater than $90°$. Because of the pressure pad, wiping dies are more complicated and costly than V-dies and are generally used for high-production work.

19.2.2 ENGINEERING ANALYSIS OF BENDING

Some of the important terms in sheet-metal bending are identified in Figure 19.11. The metal of thickness t is bent through an angle called the bend angle α. This results in a sheet-metal part with an included angle α', where $\alpha + \alpha' = 180°$. The bend radius R is normally specified on the inside of the part, rather than at the neutral axis, and is determined by the radius on the tooling used to perform the operation. The bend is made over the width of the workpiece w.

Bend Allowance If the bend radius is small relative to stock thickness, the metal tends to stretch during bending. It is important to be able to estimate the amount of stretching that occurs, if any, so that the final part length will match the specified dimension. The problem is to determine the length of the neutral axis before bending to account for stretching of the final bent section. This length is called the ***bend allowance***, and it can be estimated as follows:

$$A_b = 2\pi \frac{\alpha}{360}(R + K_{ba}t) \tag{19.6}$$

where A_b = bend allowance, mm (in); α = 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} [3]: if $R < 2t$, $K_{ba} = 0.33$; and if $R \geq 2t$, $K_{ba} = 0.50$. The values of K_{ba} predict that stretching occurs only if bend radius is small relative to sheet thickness.

Springback 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. This elastic recovery is called ***springback***, defined as the increase in included angle of the bent part relative to the included angle of the forming tool after the tool is removed. This is illustrated in Figure 19.13 and is expressed:

$$SB = \frac{\alpha' - \alpha'_b}{\alpha'_b} \tag{19.7}$$

where SB = springback; α' = included angle of the sheet-metal part, degrees; and α'_b = included angle of the bending tool, degrees. Although not as obvious, an increase in the bend radius also occurs due to elastic recovery. The amount of springback increases with modulus of elasticity E and yield strength Y of the work metal.

Compensation for springback can be accomplished by several methods. Two common methods are overbending and bottoming. In ***overbending***, the punch angle and radius are fabricated slightly smaller than the specified angle on the final part so that the sheet metal springs back to the desired value. ***Bottoming*** involves squeezing the part at the end of the stroke, thus plastically deforming it in the bend region.

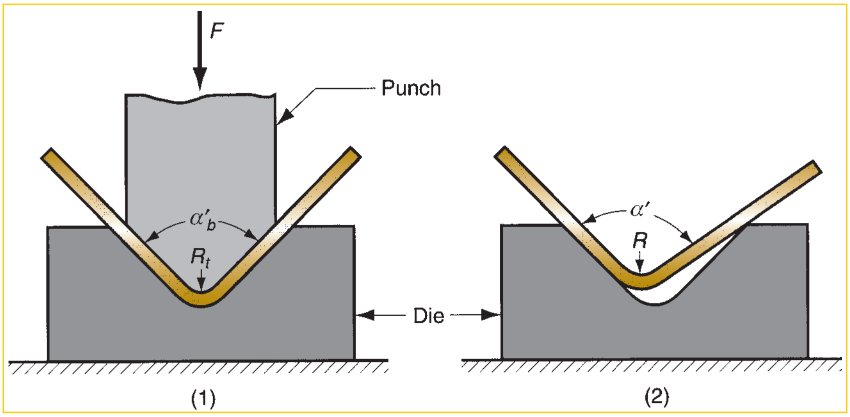

FIGURE 19.13 Springback in bending shows itself as a decrease in bend angle and an increase in bend radius: (1) during the operation, the work is forced to take the radius R_t and included angle α'_b = determined by the bending tool (punch in V-bending); (2) after the punch is removed, the work springs back to radius R and included angle α'. Symbol: F = applied bending force.

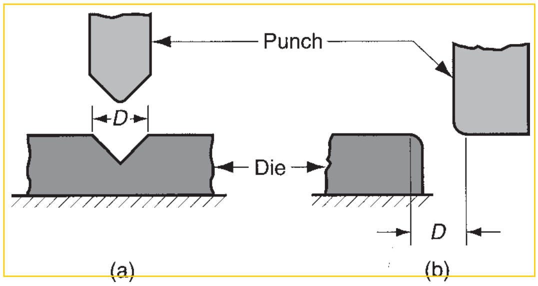

FIGURE 19.14 Die opening dimension D: (a) V-die, (b) wiping die.

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. The maximum bending force can be estimated by means of the following equation:

$$F = \frac{K_{bf}(TS)\,wt^2}{D} \tag{19.8}$$

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); $t =$ stock thickness, mm (in); and $D =$ die opening dimension as defined in Figure 19.14, mm (in). Equation (19.8) is based on bending of a simple beam in mechanics, and K_{bf} is a constant that accounts for differences encountered in an actual bending process. Its value depends on type of bending: for V-bending, $K_{bf} = 1.33$; and for edge bending, $K_{bf} = 0.33$.

Example 19.2 Sheet-metal bending

A sheet-metal blank is to be bent as shown in Figure 19.15. The metal has a modulus of elasticity $= 205\ (10^3)$ MPa, yield strength $= 275$ MPa, and tensile strength $= 450$ MPa. Determine (a) the starting blank size and (b) the bending force if a V-die is used with a die opening dimension $= 25$ mm.

Solution: (a) The starting blank $= 44.5$ mm wide. Its length $= 38 + A_b + 25$ (mm). For the included angle $\alpha' = 120°$, the bend angle $\alpha = 60°$. The value of K_{ba} in Equation (19.6) $= 0.33$ since $R/t = 4.75/3.2 = 1.48$ (less than 2.0).

$$A_b = 2\pi\frac{60}{360}(4.75 + 0.33 \times 3.2) = 6.08 \text{ mm}$$

Length of the blank is therefore $38 + 6.08 + 25 = \mathbf{69.08\ mm}$.

(b) Force is obtained from Equation (19.8) using $K_{bf} = 1.33$.

$$F = \frac{1.33(450)(44.5)(3.2)^2}{25} = \mathbf{10{,}909\ N}$$

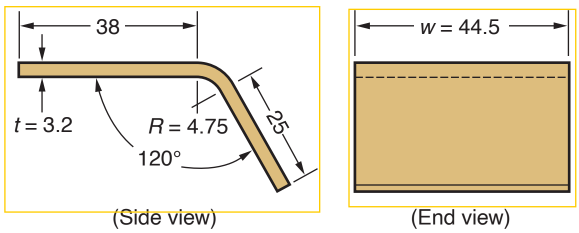

FIGURE 19.15 Sheet-metal part of Example 19.2 (dimensions in mm).

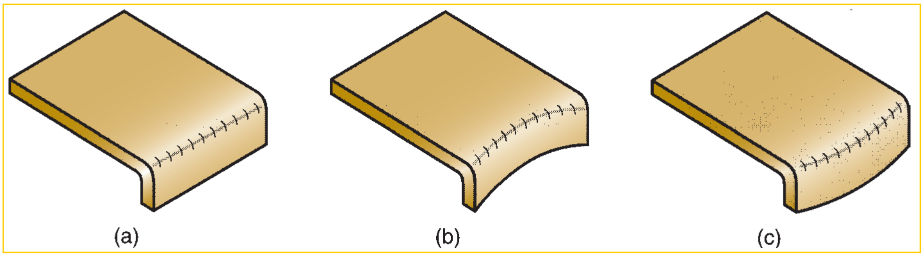

FIGURE 19.16 Flanging: (a) straight flanging, (b) stretch flanging, and (c) shrink flanging.

19.2.3 OTHER BENDING AND FORMING OPERATIONS

Some sheet-metal operations involve bending over a curved axis rather than a straight axis, or they have other features that differentiate them from the bending operations described above.

Flanging, Hemming, Seaming, and Curling Flanging is a bending operation in which the edge of a sheet-metal part is bent at a 90° angle (usually) to form a rim or flange. It is often used to strengthen or stiffen sheet metal. The flange can be formed over a straight bend axis, as illustrated in Figure 19.16(a), or it can involve some stretching or shrinking of the metal, as in (b) and (c).

Hemming involves bending the edge of the sheet over on itself, in more than one bending step. This is often done to eliminate the sharp edge on the piece, to increase stiffness, and to improve appearance. ***Seaming*** is a related operation in which two sheet-metal edges are assembled. Hemming and seaming are illustrated in Figure 19.17(a) and (b).

Curling, also called ***beading***, forms the edges of the part into a roll or curl, as in Figure 19.17(c). As in hemming, it is done for purposes of safety, strength, and aesthetics. Examples of products in which curling is used include hinges, pots and pans, and pocket-watch cases. These examples show that curling can be performed over straight or curved bend axes.

Miscellaneous Bending Operations Various other bending operations are depicted in Figure 19.18 to illustrate the variety of shapes that can be bent. Most of these operations are performed in relatively simple dies similar to V-dies.

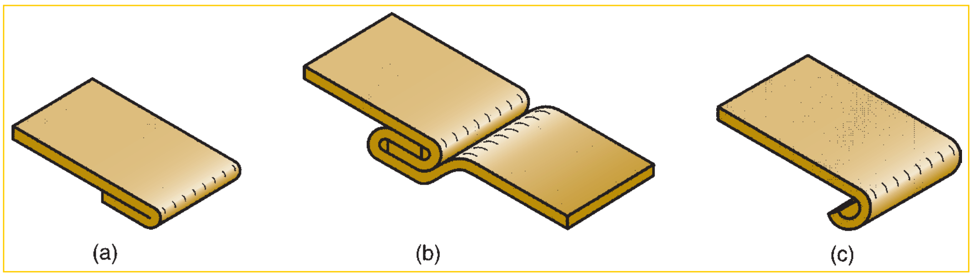

FIGURE 19.17 (a) Hemming, (b) seaming, and (c) curling.

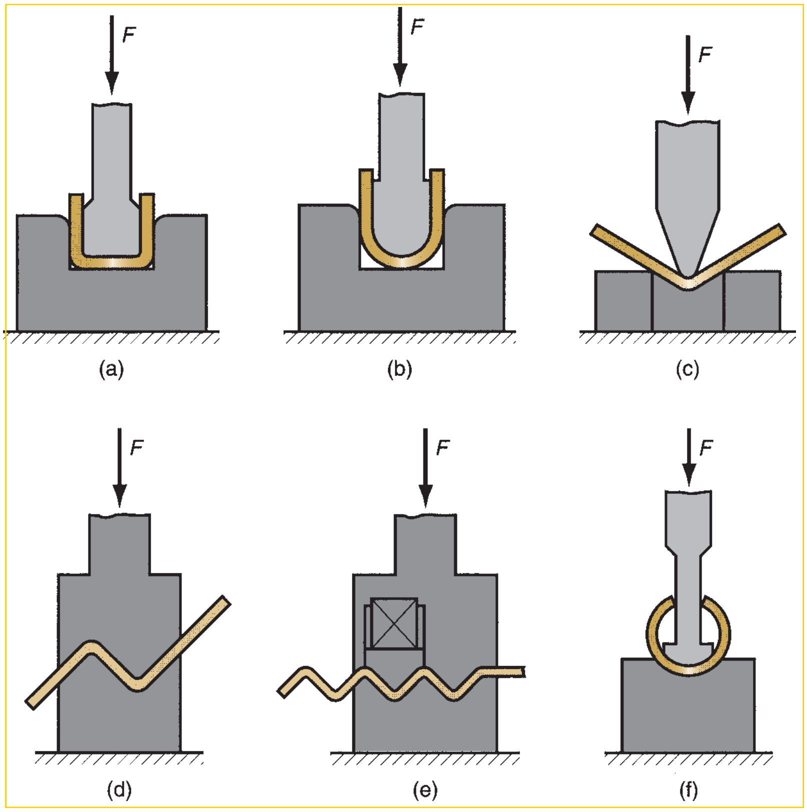

FIGURE 19.18 Miscellaneous bending operations: (a) channel bending, (b) U-bending, (c) air bending, (d) offset bending, (e) corrugating, and (f) tube forming. Symbol: $F =$ applied force.

3 Drawing

Drawing is a sheet-metal-forming operation used to make cup-shaped, box-shaped, or other complex-curved and concave parts. It is performed by placing a piece of sheet metal over a die cavity and then pushing the metal into the opening with a punch, as in Figure 19.19. The blank must usually be held down flat against the die by a blankholder. Common parts made by drawing include beverage cans, ammunition shells, sinks, cooking pots, and automobile body panels.

19.3.1 MECHANICS OF DRAWING

Drawing of a cup-shaped part is the basic drawing operation, with dimensions and parameters as pictured in Figure 19.19. A blank of diameter D_b is drawn into a die cavity by means of a punch with diameter D_p. The punch and die must have corner radii, given by R_p and R_d. If the punch and die were to have sharp corners (R_p and $R_d = 0$), a hole-punching operation (and not a very good one) would be accomplished rather than a drawing operation. The sides of the punch and die are separated by a clearance c. This clearance in drawing is about 10% greater than the stock thickness:

$$c = 1.1\,t \tag{19.9}$$

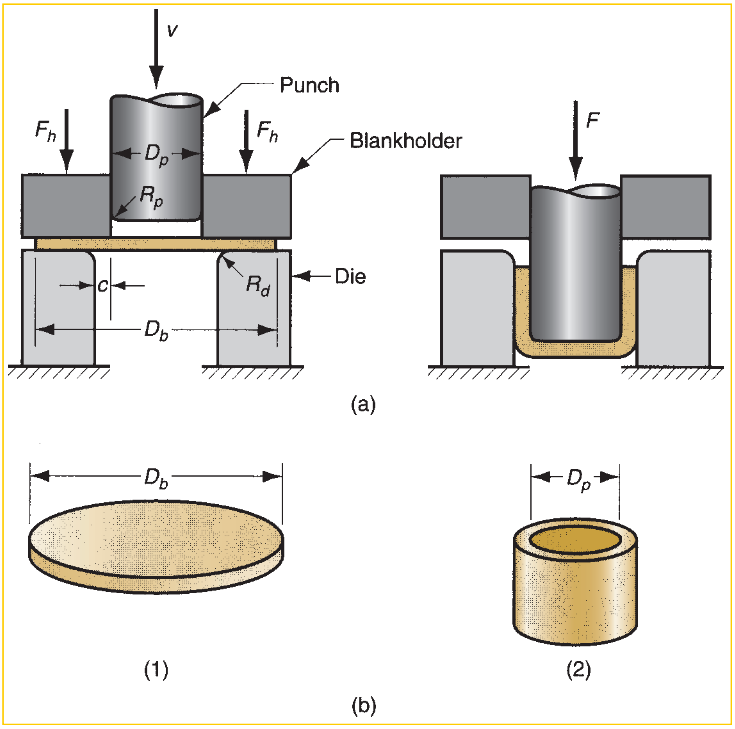

FIGURE 19.19 (a) Drawing of a cup-shaped part: (1) start of operation before punch contacts work, and (2) near end of stroke; and (b) corresponding work part: (1) starting blank, and (2) drawn part. Symbols: c = clearance, D_b = blank diameter, D_p = punch diameter, R_d = die corner radius, R_p = punch corner radius, F = drawing force, F_h = holding force.

The punch applies a downward force F to accomplish the deformation of the metal, and a downward holding force F_h is applied by the blankholder, as shown in the sketch.

As the punch proceeds downward toward its final bottom position, the work experiences a complex sequence of stresses and strains as it is gradually formed into the shape defined by the punch and die cavity. The stages in the deformation process are illustrated in Figure 19.20. As the punch first begins to push into the work, the metal is subjected to a ***bending*** operation. The sheet is simply bent over the corner of the punch and the corner of the die, as in Figure 19.20(2). The outside perimeter of the blank moves in toward the center in this first stage, but only slightly.

As the punch moves further down, a ***straightening*** action occurs in the metal that was previously bent over the die radius, as in Figure 19.20(3). The metal at the bottom of the cup, as well as along the punch radius, has been moved downward with the punch, but the metal that was bent over the die radius must now be straightened in order to be pulled into the clearance to form the wall of the cylinder. At the same time, more metal must be added to replace that being used in the cylinder wall. This new metal comes from the outside edge of the blank. The metal in the outer portions of the blank is pulled or ***drawn*** toward the die opening to resupply the previously bent and straightened metal now forming the cylinder wall. This type of metal flow through a constricted space gives the drawing process its name.

During this stage of the process, friction and compression play important roles in the flange of the blank. In order for the material in the flange to move toward the die opening, ***friction*** between the sheet metal and the surfaces of the blankholder and the die must be overcome. Initially, static friction is involved until the metal starts to slide; then, after metal flow begins, dynamic friction governs the process.

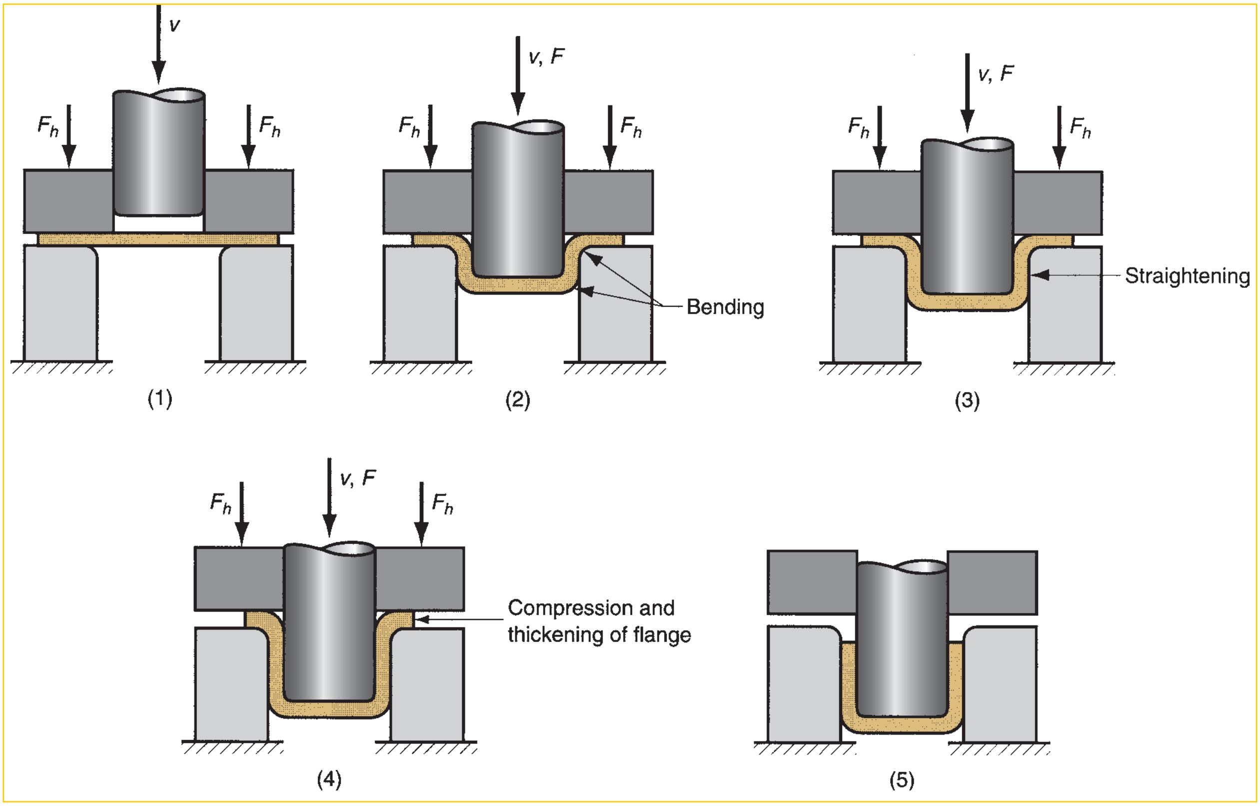

FIGURE 19.20 Stages in deformation of the work in deep drawing: (1) punch just before contact with work, (2) bending, (3) straightening, (4) friction and compression, and (5) final cup shape showing effects of thinning in the cup walls. Symbols: $v =$ motion of punch, $F =$ punch force, $F_h =$ blankholder force.

The magnitude of the holding force applied by the blankholder, as well as the friction conditions at the two interfaces, are determining factors in the success of this aspect of the drawing operation. Lubricants or drawing compounds are generally used to reduce friction forces. In addition to friction, ***compression*** is also occurring in the outer edge of the blank. As the metal in this portion of the blank is drawn toward the center, the outer perimeter becomes smaller. Because the volume of metal remains constant, the metal is squeezed and becomes thicker as the perimeter is reduced. This often results in wrinkling of the remaining flange of the blank, especially when thin sheet metal is drawn, or when the blankholder force is too low. It is a condition which cannot be corrected once it has occurred. The friction and compression effects are illustrated in Figure 19.20(4).

The holding force applied by the blankholder is now seen to be a critical factor in deep drawing. If it is too small, wrinkling occurs. If it is too large, it prevents the metal from flowing properly toward the die cavity, resulting in stretching and possible tearing of the sheet metal. Determining the proper holding force involves a delicate balance between these opposing factors.

Progressive downward motion of the punch results in a continuation of the metal flow caused by drawing and compression. In addition, some ***thinning*** of the cylinder wall occurs, as in Figure 19.20(5). The force being applied by the punch is opposed by the metal in the form of deformation and friction in the operation. A portion of the deformation involves stretching and thinning of the metal as it is pulled over the edge of the die opening. Up to 25% thinning of the side wall may occur in a successful drawing operation, mostly near the base of the cup.

19.3.2 ENGINEERING ANALYSIS OF DRAWING

It is important to assess the limitations on the amount of drawing that can be accomplished. This is often guided by simple measures that can be readily calculated for a given operation. In addition, drawing force and holding force are important process variables. Finally, the starting blank size must be determined.

Measures of Drawing One of the measures of the severity of a deep drawing operation is the ***drawing ratio*** DR. This is most easily defined for a cylindrical shape as the ratio of blank diameter D_b to punch diameter D_p. In equation form,

$$DR = \frac{D_b}{D_p} \tag{19.10}$$

The drawing ratio provides an indication, albeit a crude one, of the severity of a given drawing operation. The greater the ratio, the more severe the operation. An approximate upper limit on the drawing ratio is a value of 2.0. The actual limiting value for a given operation depends on punch and die corner radii (R_p and R_d), friction conditions, depth of draw, and characteristics of the sheet metal (e.g., ductility, degree of directionality of strength properties in the metal).

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

$$r = \frac{D_b - D_p}{D_b} \tag{19.11}$$

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.

A third measure in deep drawing is the ***thickness-to-diameter ratio*** t/D_b (thickness of the starting blank t divided by the blank diameter D_b). Often expressed as a percent, it is desirable for the t/D_b ratio to be greater than 1%. As t/D_b decreases, tendency for wrinkling (Section 19.3.4) increases.

In cases where these limits on drawing ratio, reduction, and t/D_b ratio are exceeded by the design of the drawn part, the blank must be drawn in two or more steps, sometimes with annealing between the steps.

Example 19.3
Cup drawing

A drawing operation is used to form a cylindrical cup with inside diameter = 75 mm and height = 50 mm. The starting blank size = 138 mm and the stock thickness = 2.4 mm. Based on these data, is the operation feasible?

Solution: To assess feasibility, the drawing ratio, reduction, and thickness-to-diameter ratio is determined.

$$DR = 138/75 = 1.84$$
$$r = (138 - 75)/138 = 0.4565 = 45.65\%$$
$$t/D_b = 2.4/138 = 0.017 = 1.7\%$$

According to these measures, the drawing operation is feasible. The drawing ratio is less than 2.0, the reduction is less than 50%, and the t/D_b ratio is greater than 1%. These are general guidelines frequently used to indicate technical feasibility.

Forces The ***drawing force*** required to perform a given operation can be estimated roughly by the formula:

$$F = \pi D_p t\,(TS)\left(\frac{D_b}{D_p} - 0.7\right) \tag{19.12}$$

where $F =$ drawing force, N (lb); $t =$ original blank thickness, mm (in); $TS =$ tensile strength, MPa (lb/in^2); and D_b and D_p are the starting blank diameter and punch diameter, respectively, mm (in). The constant 0.7 is a correction factor to account for friction. Equation (19.12) estimates the maximum force in the operation. The drawing force varies throughout the downward movement of the punch, usually reaching its maximum value at about one-third the length of the punch stroke.

The ***holding force*** is an important factor in a drawing operation. As a rough approximation, the holding pressure can be set at a value $= 0.015$ of the yield strength of the sheet metal [9]. This value is then multiplied by that portion of the starting area of the blank that is to be held by the blankholder. In equation form,

$$F_h = 0.015Y\pi\{D_b^2 - (D_p + 2.2t + 2R_d)^2\} \tag{19.13}$$

where $F_h =$ holding force in drawing, N (lb); $Y =$ yield strength of the sheet metal, MPa (lb/in^2); $t =$ starting stock thickness, mm (in); $R_d =$ die corner radius, mm (in); and the other terms have been previously defined. The holding force is usually about one-third the drawing force [10].

Example 19.4
Forces in deep drawing

For the drawing operation of Example 19.3, determine (a) drawing force and (b) holding force, given that the tensile strength of the sheet metal (low-carbon steel) $= 300$ MPa and yield strength $= 175$ MPa. The die corner radius $=$ 6 mm.

Solution: (a) Maximum drawing force is given by Equation (19.12):

$$F = \pi(75)(2.4)(300)\left(\frac{138}{75} - 0.7\right) = \mathbf{193{,}396\ N}$$

(b) Holding force is estimated by Equation (19.13):

$$F_h = 0.015(175)\,p(138^2 - (75 + 2.2 \times 2.4 + 2 \times 6)^2) = \mathbf{86{,}824\ N}$$

Blank Size Determination For the final dimensions to be achieved on the cylindrical drawn shape, the correct starting blank diameter is needed. It must be large enough to supply sufficient metal to complete the cup. Yet if there is too much material, unnecessary waste will result. For drawn shapes other than cylindrical cups, the same problem of estimating the starting blank size exists, only the shape of the blank may be other than round.

The following is a reasonable method for estimating the starting blank diameter in a deep drawing operation that produces a round part (e.g., cylindrical cup and more complex shapes so long as they are axisymmetric). Because the volume of the final product is the same as that of the starting sheet-metal blank, then the blank diameter can be calculated by setting the initial blank volume equal to the final volume of the product and solving for diameter D_b. To facilitate the calculation, it is often assumed that negligible thinning of the part wall occurs.

19.3.3 OTHER DRAWING OPERATIONS

The discussion has focused on a conventional cup-drawing operation that produces a simple cylindrical shape in a single step and uses a blankholder to facilitate the process. Some of the variations of this basic operation are considered in this section.

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. The second drawing step, and any further drawing steps if needed, are referred to as ***redrawing***. A redrawing operation is illustrated in Figure 19.21.

When the part design indicates a drawing ratio that is too large to form the part in a single step, the following is a general guide to the amount of reduction that can be taken in each drawing operation [10]: For the first draw, the maximum reduction of the starting blank should be 40% to 45%; for the second draw (first redraw), the maximum reduction should be 30%; and for the third draw (second redraw), the maximum reduction should be 16%.

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 such as that shown in Figure 19.22. Although it may seem that reverse drawing would produce a more severe deformation than redrawing, it is actually easier on the metal. The reason is that the sheet metal is bent in the same direction at the outside and inside corners of the die in reverse drawing; while in redrawing the metal is bent in the opposite directions at the two corners. Because of this difference, the metal experiences less strain hardening in reverse drawing and the drawing force is lower.

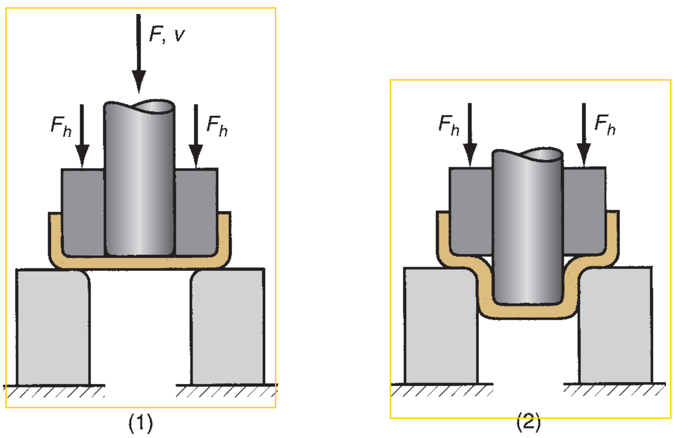

FIGURE 19.21 Redrawing of a cup: (1) start of redraw, and (2) end of stroke. Symbols: v = punch velocity, F = applied punch force, F_h = blankholder force.

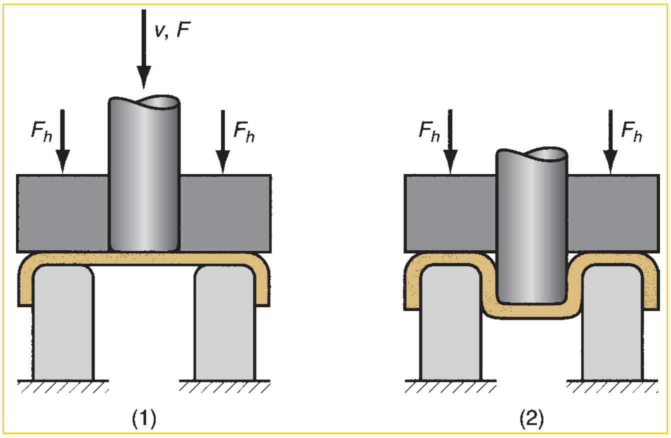

FIGURE 19.22 Reverse drawing: (1) start and (2) completion. Symbols: v = punch velocity, F = applied punch force, F_h = blankholder force.

Drawing of Shapes Other than Cylindrical Cups Many products require drawing of shapes other than cylindrical cups. The variety of drawn shapes include square or rectangular boxes (as in sinks), stepped cups, cones, cups with spherical rather than flat bases, and irregular curved forms (as in automobile body panels). Each shape presents unique technical problems in drawing. Eary and Reed [2] provide a detailed discussion of the drawing of these kinds of shapes.

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/D_b ratio is large enough, drawing can be accomplished without a blankholder, as in Figure 19.23. The limiting condition for drawing without a blankholder can be estimated from the following [5]:

$$D_b - D_p < 5t \tag{19.14}$$

The draw die must have the shape of a funnel or cone to permit the material to be drawn properly into the die cavity. When drawing without a blankholder is feasible, it has the advantages of lower cost tooling and a simpler press, because the need to separately control the movements of the blankholder and punch can be avoided.

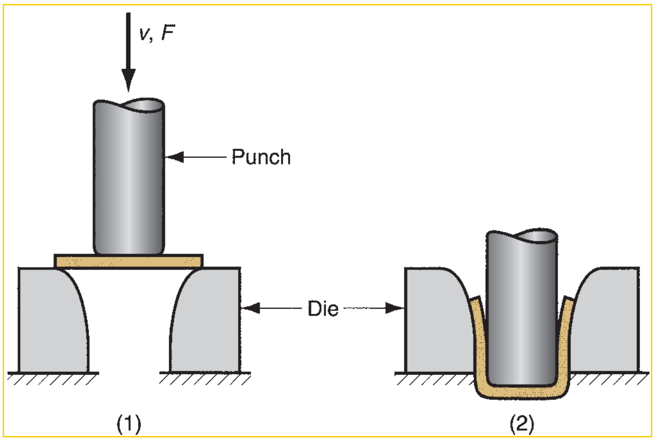

FIGURE 19.23 Drawing without a blankholder: (1) start of process, (2) end of stroke. Symbols v and F indicate motion and applied force, respectively.

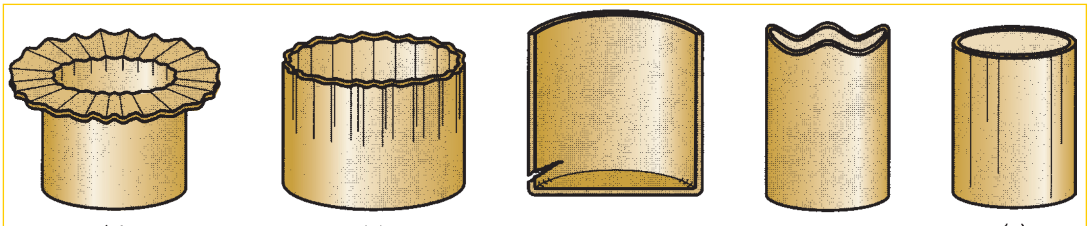

FIGURE 19.24 Common defects in drawn parts: (a) wrinkling can occur either in the flange or (b) in the wall, (c) tearing, (d) earing, and (e) surface scratches.

19.3.4 DEFECTS IN DRAWING

Sheet-metal drawing is a more complex operation than cutting or bending, and more things can go wrong. A number of defects can occur in a drawn product, some of which have already been alluded to. Following is a list of common defects, with sketches in Figure 19.24:

(a) ***Wrinkling in the flange***. Wrinkling in a drawn part consists of a series of ridges that form radially in the undrawn flange of the work part due to compressive buckling.

(b) ***Wrinkling in the wall***. If and when the wrinkled flange is drawn into the cup, these ridges appear in the vertical wall.

(c) ***Tearing***. Tearing is an open crack in the vertical wall, usually near the base of the drawn cup, due to high tensile stresses that cause thinning and failure of the metal at this location. This type of failure can also occur as the metal is pulled over a sharp die corner.

(d) ***Earing***. This is the formation of irregularities (called ***ears***) in the upper edge of a deep drawn cup, caused by anisotropy in the sheet metal. If the material is perfectly isotropic, ears do not form.

(e) ***Surface scratches***. Surface scratches can occur on the drawn part if the punch and die are not smooth or if lubrication is insufficient.

19.4 Other Sheet-Metal-Forming Operations

In addition to bending and drawing, several other sheet-metal forming operations can be accomplished on conventional presses. These are classified as (1) operations performed with metal tooling and (2) operations performed with flexible rubber tooling.

19.4.1 OPERATIONS PERFORMED WITH METAL TOOLING

Operations performed with metal tooling include (1) ironing, (2) coining and embossing, (3) lancing, and (4) twisting.

Ironing In deep drawing the flange is compressed by the squeezing action of the blank perimeter seeking a smaller circumference as it is drawn toward the die opening. Because of this compression, the sheet metal near the outer edge of the blank

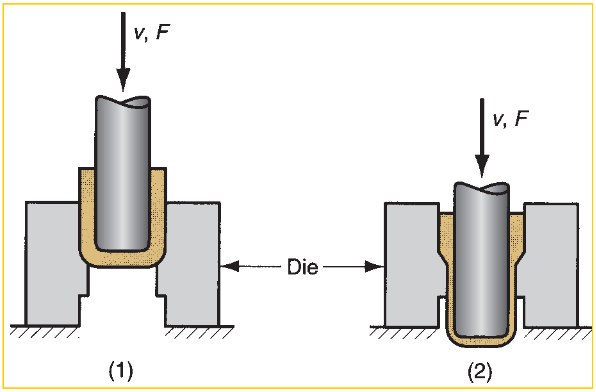

FIGURE 19.25 Ironing to achieve a more uniform wall thickness in a drawn cup: (1) start of process; (2) during process. Note thinning and elongation of walls. Symbols *v* and *F* indicate motion and applied force, respectively.

becomes thicker as it moves inward. If the thickness of this stock is greater than the clearance between the punch and die, it will be squeezed to the size of the clearance, a process known as ***ironing***.

Sometimes ironing is performed as a separate step that follows drawing. This case is illustrated in Figure 19.25. Ironing makes the cylindrical cup more uniform in wall thickness. The drawn part is therefore longer and more efficient in terms of material usage. Beverage cans and artillery shells, two very high-production items, include ironing among their processing steps to achieve economy in material usage.

Coining and Embossing Coining is a bulk deformation operation discussed in the previous chapter. It is frequently used in sheet-metal work to form indentations and raised sections in the part. The indentations result in thinning of the sheet metal, and the raised sections result in thickening of the metal.

Embossing is a forming operation used to create indentations in the sheet, such as raised (or indented) lettering or strengthening ribs, as depicted in Figure 19.26. Some stretching and thinning of the metal are involved. This operation may seem similar to coining. However, embossing dies possess matching cavity contours, the punch containing the positive contour and the die containing the negative; whereas coining dies may have quite different cavities in the two die halves, thus causing more significant metal deformation than embossing.

Lancing Lancing is a combined cutting and bending or cutting and forming operation performed in one step to partially separate the metal from the sheet. Several examples are shown in Figure 19.27. Among other applications, lancing is used to make louvers in sheet-metal air vents for heating and air conditioning systems in buildings.

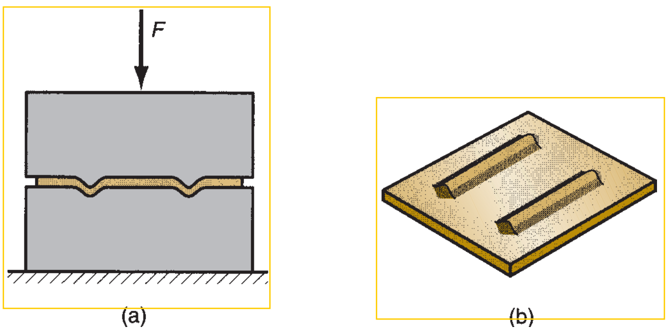

FIGURE 19.26 Embossing: (a) cross section of punch and die configuration during pressing; (b) finished part with embossed ribs.

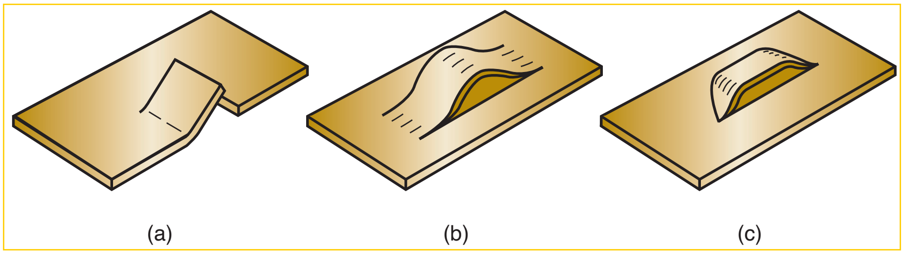

FIGURE 19.27 Lancing in several forms: (a) cutting and bending; (b) and (c) two types of cutting and forming.

Twisting Twisting subjects the sheet metal to a torsion loading rather than a bending load, thus causing a twist in the sheet over its length. This type of operation has limited applications. It is used to make such products as fan and propeller blades. It can be performed in a conventional punch and die which has been designed to deform the part in the required twist shape.

19.4.2 RUBBER FORMING PROCESSES

The two operations discussed in this article are performed on conventional presses, but the tooling is unusual in that it uses a flexible element (made of rubber or similar material) to effect the forming operation. The operations are (1) the Guerin process, and (2) hydroforming.

Guerin Process The ***Guerin process*** uses a thick rubber pad (or other flexible material) to form sheet metal over a positive form block, as in Figure 19.28. The rubber pad is confined in a steel container. As the ram descends, the rubber gradually surrounds the sheet, applying pressure to deform it to the shape of the form block. It is limited to relatively shallow forms, because the pressures developed by the rubber—up to about 10 MPa (1500 lb/in^2)—are not sufficient to prevent wrinkling in deeper formed parts.

The advantage of the Guerin process is the relatively low cost of the tooling. The form block can be made of wood, plastic, or other materials that are easy to shape, and the rubber pad can be used with different form blocks. These factors

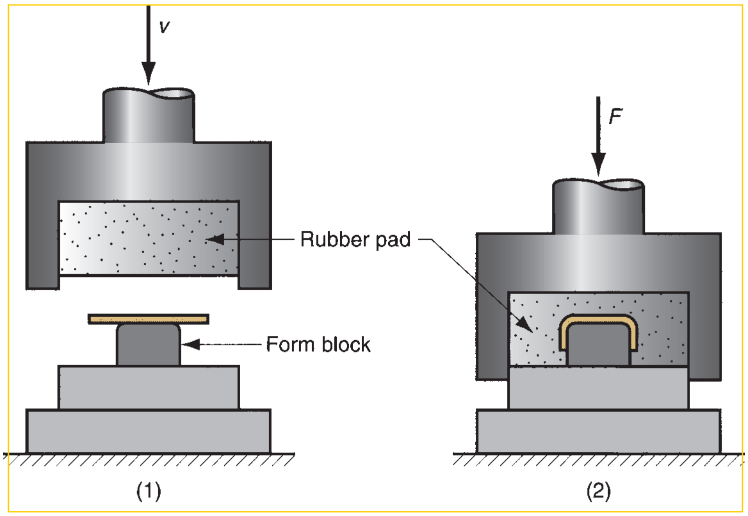

FIGURE 19.28 Guerin process: (1) before and (2) after. Symbols *v* and *F* indicate motion and applied force respectively.

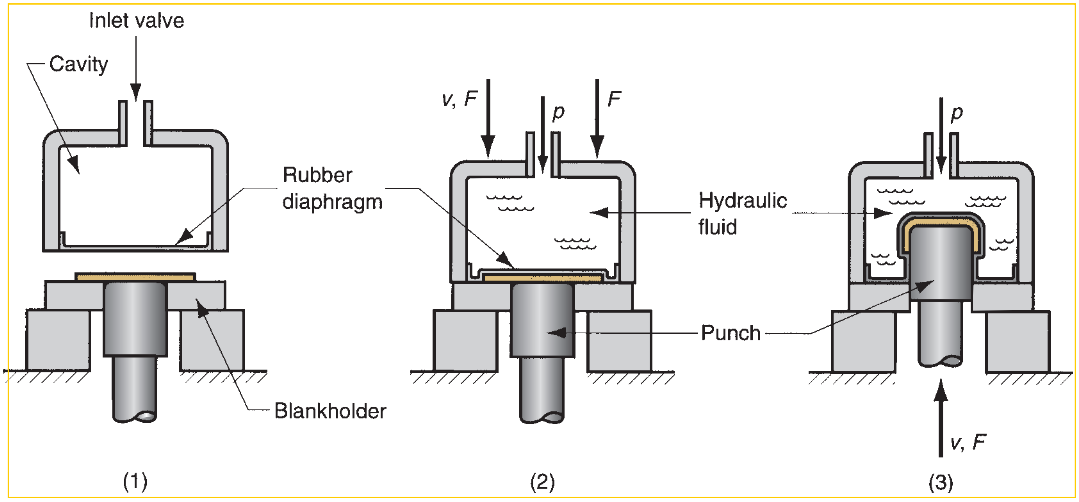

FIGURE 19.29 Hydroform process: (1) start-up, no fluid in cavity; (2) press closed, cavity pressurized with hydraulic fluid; (3) punch pressed into work to form part. Symbols: $v =$ velocity, $F =$ applied force, $p =$ hydraulic pressure.

make rubber forming attractive in small-quantity production, such as the aircraft industry, where the process was developed.

Hydroforming Hydroforming is similar to the Guerin process; the difference is that it substitutes a rubber diaphragm filled with hydraulic fluid in place of the thick rubber pad, as illustrated in Figure 19.29. This allows the pressure that forms the work part to be increased—to around 100 MPa (15,000 lb/in^2)—thus preventing wrinkling in deep formed parts. In fact, deeper draws can be achieved with the hydroform process than with conventional deep drawing. This is because the uniform pressure in hydroforming forces the work to contact the punch throughout its length, thus increasing friction and reducing the tensile stresses that cause tearing at the base of the drawn cup.

19.5 Dies and Presses for Sheet-Metal Processes

This section examines the punch-and-die tooling and production equipment used in conventional sheet-metal processing.

19.5.1 DIES

Nearly all of the preceding pressworking operations are performed with conventional punch-and-die tooling. The tooling is referred to as a ***die***. It is custom-designed for the particular part to be produced. The term ***stamping die*** is sometimes used for high-production dies. Typical materials for stamping dies are tool steel types D, A, O, and S (Table 6.5).

Components of a Stamping Die The components of a stamping die to perform a simple blanking operation are illustrated in Figure 19.30. The working components are the ***punch*** and ***die***, which perform the cutting operation. They are attached to

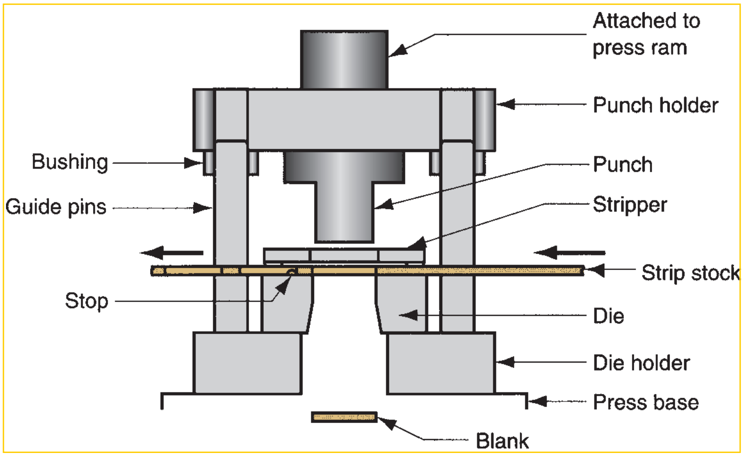

FIGURE 19.30 Components of a punch and die for a blanking operation.

the upper and lower portions of the ***die set***, respectively called the ***punch holder*** (or ***upper shoe***) and ***die holder*** (***lower shoe***). The die set also includes guide pins and bushings to ensure proper alignment between the punch and die during the stamping operation. The die holder is attached to the base of the press, and the punch holder is attached to the ram. Actuation of the ram accomplishes the pressworking operation.

In addition to these components, a die used for blanking or hole-punching must include a means of preventing the sheet metal from sticking to the punch when it is retracted upward after the operation. The newly created hole in the stock is the same size as the punch, and it tends to cling to the punch on its withdrawal. The device in the die that strips the sheet metal from the punch is called a ***stripper***. It is often a simple plate attached to the die as in Figure 19.30, with a hole slightly larger than the punch diameter.

For dies that process strips or coils of sheet metal, a device is required to stop the sheet metal as it advances through the die between press cycles. That device is called (try to guess) a ***stop***. Stops range from simple solid pins located in the path of the strip to block its forward motion, to more complex mechanisms synchronized to rise and retract with the actuation of the press. The simpler stop is shown in Figure 19.30.

There are other components in pressworking dies, but the preceding description provides an introduction to the terminology.

Types of Stamping Dies Aside from differences in stamping dies related to the operations they perform (e.g., cutting, bending, drawing), other differences deal with the number of separate operations to be performed in each press actuation and how they are accomplished.

The type of die considered above performs a single blanking operation with each stroke of the press and is called a ***simple die***. Other dies that perform a single operation include V-dies (Section 19.2.1). More complicated pressworking dies include compound dies, combination dies, and progressive dies. A ***compound die*** performs two operations at a single station, such as blanking and punching, or blanking and drawing [2]. A good example is a compound die that blanks and punches a washer. A ***combination die*** is less common; it performs two operations at two different stations in the die. Examples of applications include blanking two different parts (e.g., right-hand and left-hand parts), or blanking and then bending the same part [2].

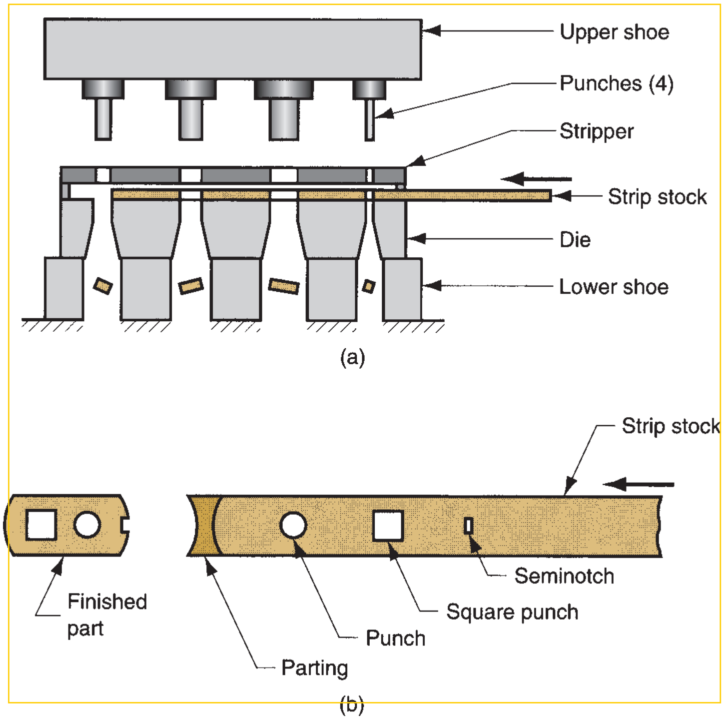

FIGURE 19.31 (a) Progressive die and (b) associated strip development.

A ***progressive die*** performs two or more operations on a sheet-metal coil at two or more stations with each press stroke. The part is fabricated progressively. The coil is fed from one station to the next and different operations (e.g., punching, notching, bending, and blanking) are performed at each station. When the part exits the final station it has been completed and separated (cut) from the remaining coil. Design of a progressive die begins with the layout of the part on the strip or coil and the determination of which operations are to be performed at each station. The result of this procedure is called the ***strip development***. A progressive die and associated strip development are illustrated in Figure 19.31. Progressive dies can have a dozen or more stations. They are the most complicated and most costly stamping dies, economically justified only for complex parts requiring multiple operations at high-production rates.

19.5.2 PRESSES

A press used for sheet metalworking is a machine tool with a stationary ***bed*** and a powered ***ram*** (or ***slide***) that can be driven toward and away from the bed to perform various cutting and forming operations. A typical press, with principal components labeled, is diagrammed in Figure 19.32. The relative positions of the bed and ram are established by the ***frame***, and the ram is driven by mechanical or hydraulic power. When a die is mounted in the press, the punch holder is attached to the ram, and the die holder is attached to a ***bolster plate*** of the press bed.

Presses are available in a variety of capacities, power systems, and frame types. The capacity of a press is its ability to deliver the required force and energy to

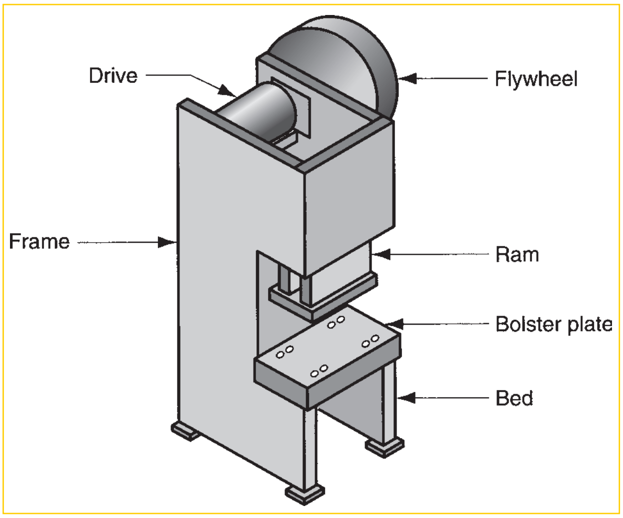

FIGURE 19.32 Components of a typical (mechanical drive) stamping press.

accomplish the stamping operation. This is determined by the physical size of the press and by its power system. The power system refers to whether mechanical or hydraulic power is used and the type of drive used to transmit the power to the ram. Production rate is another important aspect of capacity. Type of frame refers to the physical construction of the press. There are two frame types in common use: gap frame and straight-sided frame.

Gap Frame Presses The ***gap frame*** has the general configuration of the letter C and is often referred to as a ***C-frame***. Gap frame presses provide good access to the die, and they are usually open in the back to permit convenient ejection of stampings or scrap. The principal types of gap frame press are (a) solid gap frame, (b) adjustable bed, (c) open back inclinable, (d) press brake, and (e) turret press.

The ***solid gap frame*** (sometimes called simply a ***gap press***) has one-piece construction, as shown in Figure 19.32. Presses with this frame are rigid, yet the C-shape allows convenient access from the sides for feeding strip or coil stock. They are available in a range of sizes, with capacities up to around 9000 kN (1000 tons). The model shown in Figure 19.33 has a capacity of 1350 kN (150 tons). The ***adjustable bed frame*** press is a variation of the gap frame, in which an adjustable bed is added to accommodate various die sizes. The adjustment feature results in some sacrifice of tonnage capacity. The ***open-back inclinable*** press has a C-frame assembled to a base in such a way that the frame can be tilted back to various angles so that the stampings fall through the rear opening by gravity. Capacities of OBI presses range between one ton and around 2250 kN (250 tons). They can be operated at high speeds—up to around 1000 strokes per minute.

The ***press brake*** is a gap frame press with a very wide bed, as shown in Figure 19.34. This allows a number of separate dies (simple V-bending dies are typical) to be set up in the bed, so that small quantities of stampings can be made economically. These low quantities of parts, sometimes requiring multiple bends at different angles, necessitate a manual operation. For a part requiring a series of bends, the operator moves the starting piece of sheet metal through the desired sequence of bending dies, actuating the press at each die, to complete the work needed.

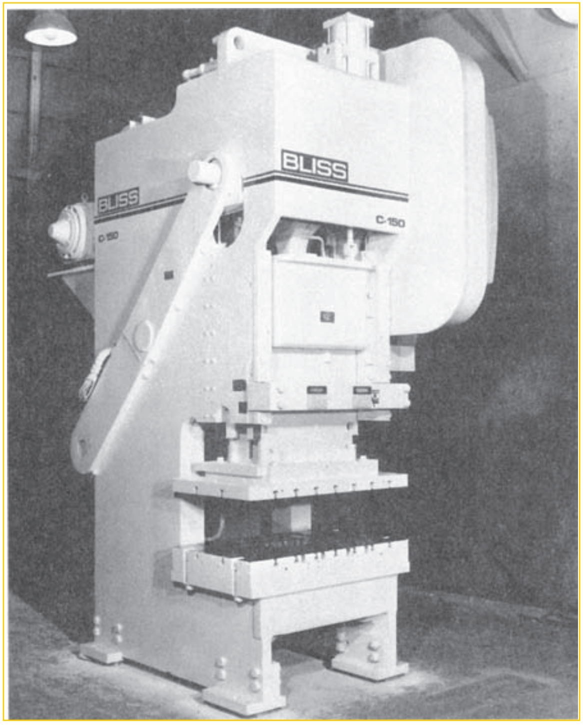

FIGURE 19.33 Gap frame press for sheet metalworking. (Photo courtesy of BCN Technology Services). Capacity $= 1350$ kN (150 tons).

FIGURE 19.34 Press brake. (Photo courtesy of Strippit, Inc.)

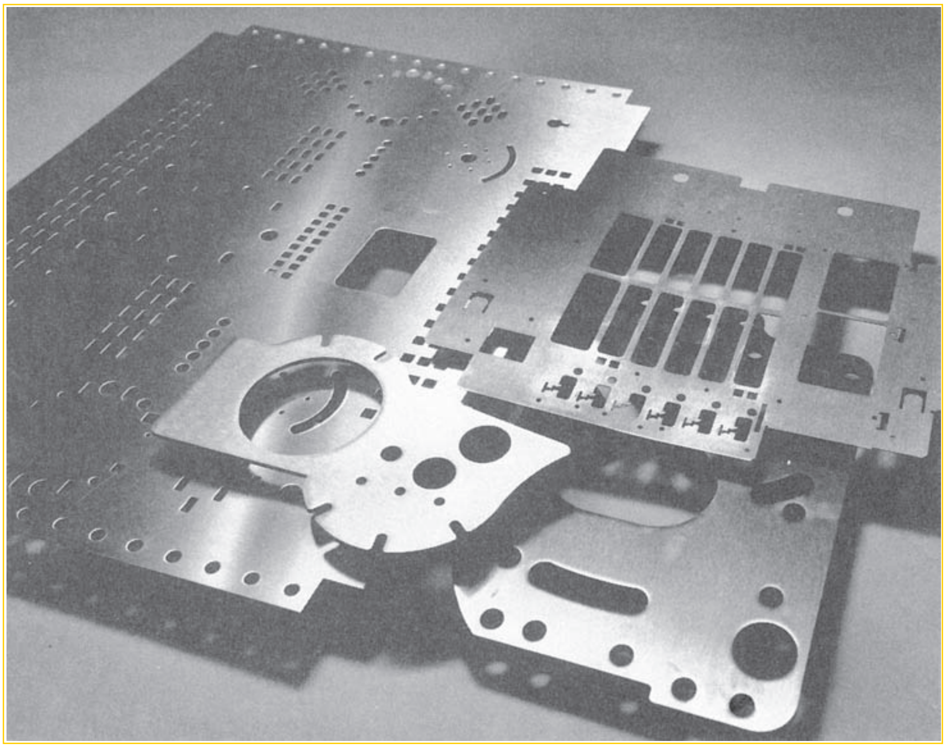

FIGURE 19.35 Several sheet-metal parts produced on a turret press, showing variety of possible hole shapes. (Photo courtesy of Strippit, Inc.)

Whereas press brakes are well adapted to bending operations, ***turret presses*** are suited to situations in which a sequence of punching, notching, and related cutting operations must be accomplished on sheet-metal parts, as in Figure 19.35. Turret presses have a C-frame, although this construction is not obvious in Figure 19.36. The conventional ram and punch is replaced by a turret containing many punches of different sizes and shapes. The turret works by indexing (rotating) to the position holding the punch to perform the required operation. Beneath the punch turret is a corresponding die turret that positions the die opening for each punch. Between the punch and die is the sheet-metal blank, held by an x–y positioning system that operates by computer numerical control (Section 37.3). The blank is moved to the required coordinate position for each cutting operation.

Straight-sided Frame Presses For jobs requiring high tonnage, press frames with greater structural rigidity are needed. Straight-sided presses have full sides, giving it a box-like appearance as in Figure 19.37. This construction increases the strength and stiffness of the frame. As a result, capacities up to 35,000 kN (4000 tons) are available in straight-sided presses for sheet metalwork. Large presses of this frame type are used for forging (Section 18.3).

In all of these presses, gap frame and straight-sided frame, the size is closely correlated to tonnage capacity. Larger presses are built to withstand higher forces in pressworking. Press size is also related to the speed at which it can operate. Smaller presses are generally capable of higher production rates than larger presses.

Power and Drive Systems Power systems on presses are either hydraulic or mechanical. ***Hydraulic presses*** use a large piston and cylinder to drive the ram. This

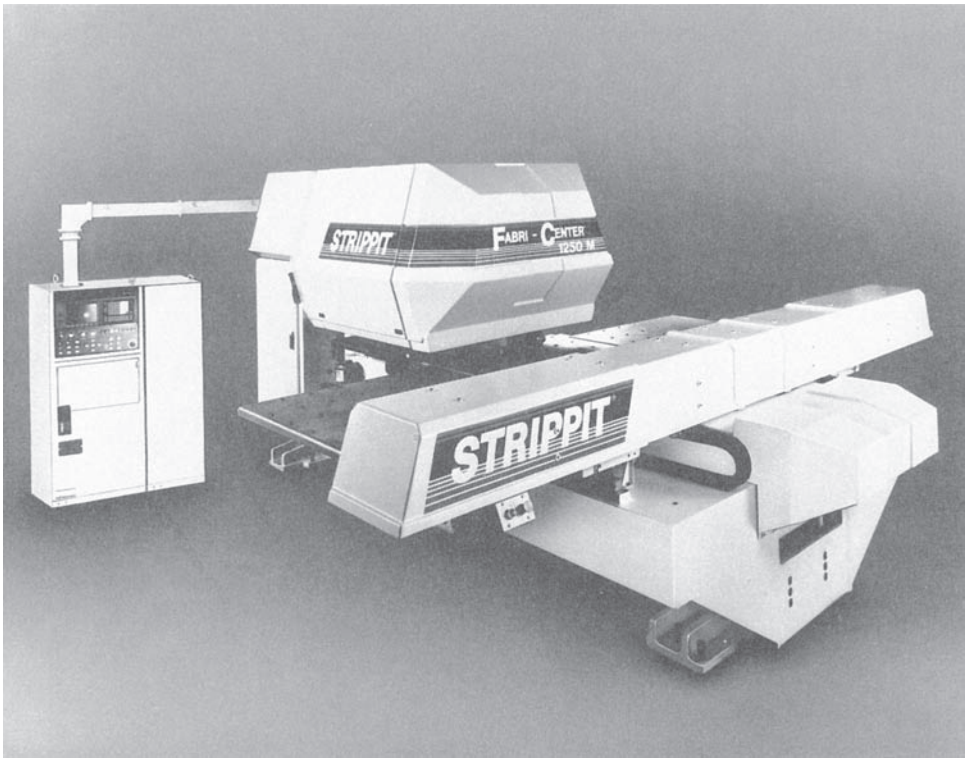

FIGURE 19.36 Computer numerical control turret press. (Photo courtesy of Strippit, Inc.)

FIGURE 19.37 Straight-sided frame press. (Photo courtesy of Greenerd Press & Machine Company, Inc.)

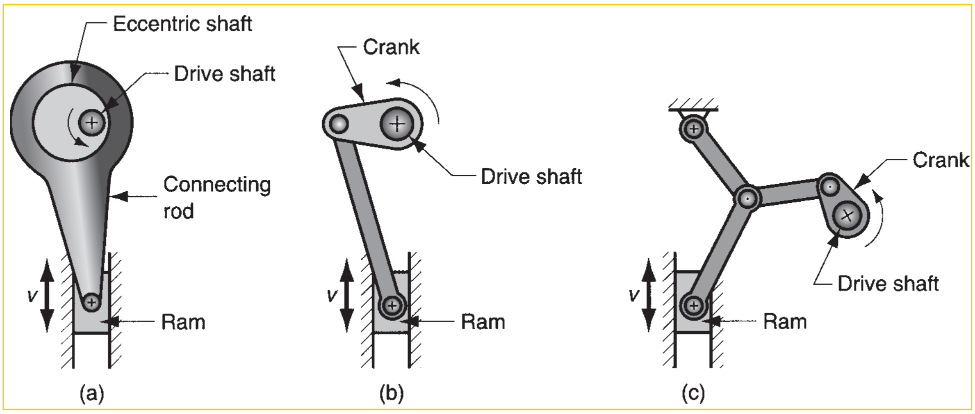

FIGURE 19.38 Types of drives for sheet-metal presses: (a) eccentric, (b) crankshaft, and (c) knuckle joint.

power system typically provides longer ram strokes than mechanical drives and can develop the full tonnage force throughout the entire stroke. However, it is slower. Its application for sheet metal is normally limited to deep drawing and other forming operations where these load-stroke characteristics are advantageous. These presses are available with one or more independently operated slides, called single action (single slide), double action (two slides), and so on. Double-action presses are useful in deep drawing operations where it is required to separately control the punch force and the blankholder force.

There are several types of drive mechanisms used on ***mechanical presses***. These include eccentric, crankshaft, and knuckle joint, illustrated in Figure 19.38. They convert the rotational motion of a drive motor into the linear motion of the ram. A ***flywheel*** is used to store the energy of the drive motor for use in the stamping operation. Mechanical presses using these drives achieve very high forces at the bottom of their strokes, and are therefore quite suited to blanking and punching operations. The knuckle joint delivers very high force when it bottoms, and is therefore often used in coining operations.

19.6 Sheet-Metal Operations Not Performed on Presses

A number of sheet-metal operations are not performed on conventional stamping presses. This section examines several of these processes: (1) stretch forming, (2) roll bending and forming, (3) spinning, and (4) high-energy-rate forming processes.

19.6.1 STRETCH FORMING

Stretch forming is a sheet-metal deformation process in which the sheet metal is intentionally stretched and simultaneously bent in order to achieve shape change. The process is illustrated in Figure 19.39 for a relatively simple and gradual bend. The work part is gripped by one or more jaws on each end and then stretched and bent over a positive die containing the desired form. The metal is stressed in tension to a level above its yield point. When the tension loading is released, the metal has been plastically deformed. The combination of stretching and bending results in relatively little springback in the part. An estimate of the force required in stretch

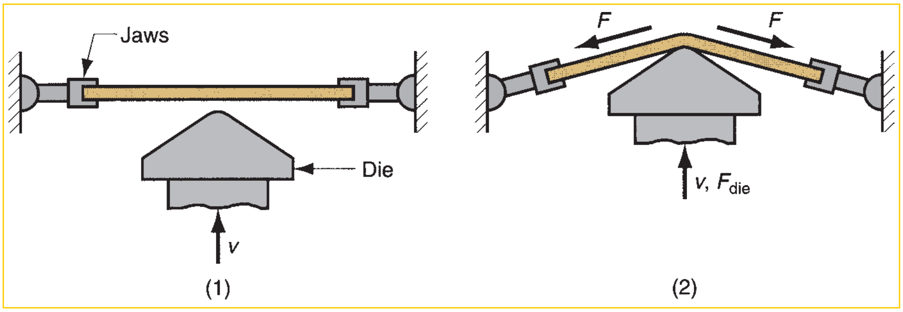

FIGURE 19.39 Stretch forming: (1) start of process; (2) form die is pressed into the work with force F_{die}, causing it to be stretched and bent over the form. $F =$ stretching force.

forming can be obtained by multiplying the cross-sectional area of the sheet in the direction of pulling by the flow stress of the metal. In equation form,

$$F = LtY_f \tag{19.15}$$

where $F =$ stretching force, N (lb); $L =$ length of the sheet in the direction perpendicular to stretching, mm (in); $t =$ instantaneous stock thickness, mm (in); and $Y_f =$ flow stress of the work metal, MPa (lb/in^2). The die force F_{die} shown in the figure can be determined by balancing vertical force components.

More complex contours than that shown in the figure are possible by stretch forming, but there are limitations on how sharp the curves in the sheet can be. Stretch forming is widely used in the aircraft and aerospace industries to economically produce large sheet-metal parts in the low quantities characteristic of those industries.

19.6.2 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. One possible arrangement of the rolls is pictured in Figure 19.40. As the sheet passes between the rolls, the rolls are brought toward each other to a configuration that achieves the desired radius of curvature on the work. Components for large storage tanks and pressure vessels are fabricated by roll bending. The operation can also be used to bend structural shapes, railroad rails, and tubes.

A related operation is ***roll straightening*** in which nonflat sheets (or other cross-sectional forms) are straightened by passing them between a series of rolls. The rolls subject the work to a sequence of decreasing small bends in opposite directions, thus causing it to be straight at the exit.

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. Several pairs of rolls are usually required to progressively

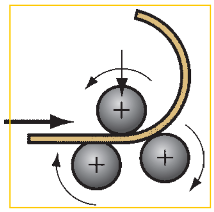

FIGURE 19.40 Roll bending.

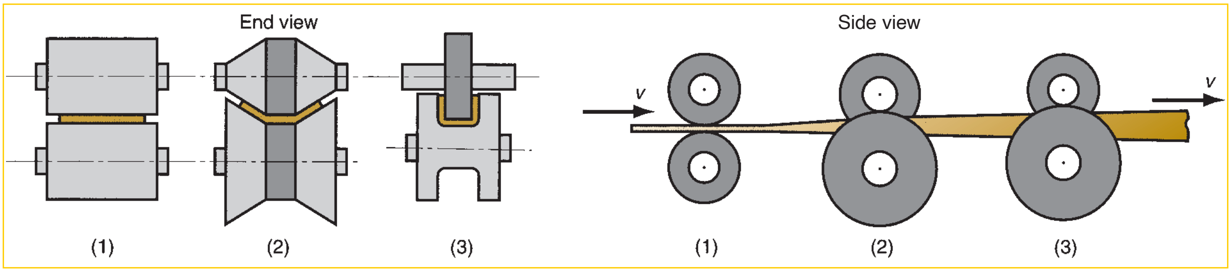

FIGURE 19.41 Roll forming of a continuous channel section: (1) straight rolls, (2) partial form, and (3) final form.

accomplish the bending of the stock into the desired shape. The process is illustrated in Figure 19.41 for a U-shaped section. Products made by roll forming include channels, gutters, metal siding sections (for homes), pipes and tubing with seams, and various structural sections. Although roll forming has the general appearance of a rolling operation (and the tooling certainly looks similar), the difference is that roll forming involves bending rather than compressing the work.

19.6.3 SPINNING

Spinning is a metal-forming process in which an axially symmetric part is gradually shaped over a mandrel or form by means of a rounded tool or roller. The tool or roller applies a very localized pressure (almost a point contact) to deform the work by axial and radial motions over the surface of the part. Basic geometric shapes typically produced by spinning include cups, cones, hemispheres, and tubes. There are three types of spinning operations: (1) conventional spinning, (2) shear spinning, and (3) tube spinning.

Conventional Spinning Conventional spinning is the basic spinning operation. As illustrated in Figure 19.42, a sheet-metal disk is held against the end of a rotating mandrel of the desired inside shape of the final part, while the tool or roller deforms the metal against the mandrel. In some cases, the starting work part is other than a flat disk. The process requires a series of steps, as indicated in the figure, to complete

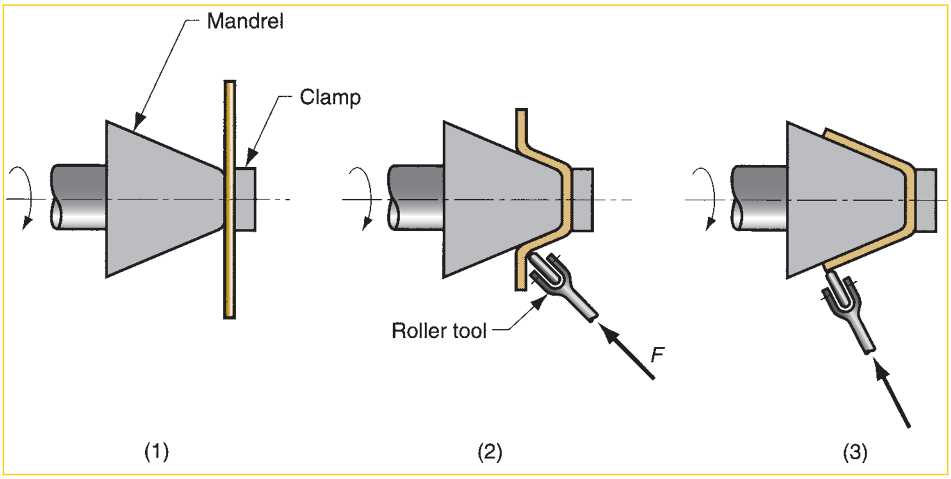

FIGURE 19.42 Conventional spinning: (1) setup at start of process; (2) during spinning; and (3) completion of process.

the shaping of the part. The tool position is controlled either by a human operator, using a fixed fulcrum to achieve the required leverage, or by an automatic method such as numerical control. These alternatives are ***manual spinning*** and ***power spinning***. Power spinning has the capability to apply higher forces to the operation, resulting in faster cycle times and greater work size capacity. It also achieves better process control than manual spinning.

Conventional spinning bends the metal around a moving circular axis to conform to the outside surface of the axisymmetric mandrel. The thickness of the metal therefore remains unchanged (more or less) relative to the starting disk thickness. The diameter of the disk must therefore be somewhat larger than the diameter of the resulting part. The required starting diameter can be figured by assuming constant volume, before and after spinning.

Applications of conventional spinning include production of conical and curved shapes in low quantities. Very large diameter parts—up to 5 m (15 ft) or more—can be made by spinning. Alternative sheet-metal processes would require excessively high die costs. The form mandrel in spinning can be made of wood or other soft materials that are easy to shape. It is therefore a low-cost tool compared to the punch and die required for deep drawing, which might be a substitute process for some parts.

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, as in Figure 19.43. This shear straining (and consequent thinning of the metal) distinguishes this process from the bending action in conventional spinning. Several other names have been used for shear spinning, including ***flow turning***, ***shear forming***, and ***spin forging***. The process has been applied in the aerospace industry to form large parts such as rocket nose cones.

For the simple conical shape in the figure, the resulting thickness of the spun wall can be readily determined by the sine law relationship:

$$t_f = t \sin a \tag{19.16}$$

where $t_f =$ the final thickness of the wall after spinning, $t =$ the starting thickness of the disk, and $a =$ the mandrel angle (actually the half angle). Thinning is sometimes quantified by the spinning reduction r:

$$r = \frac{t - t_f}{t} \tag{19.17}$$

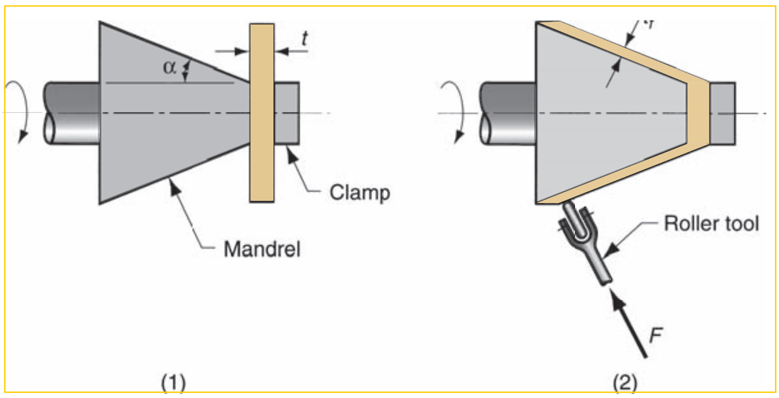

FIGURE 19.43 Shear spinning: (1) setup, and (2) completion of process.

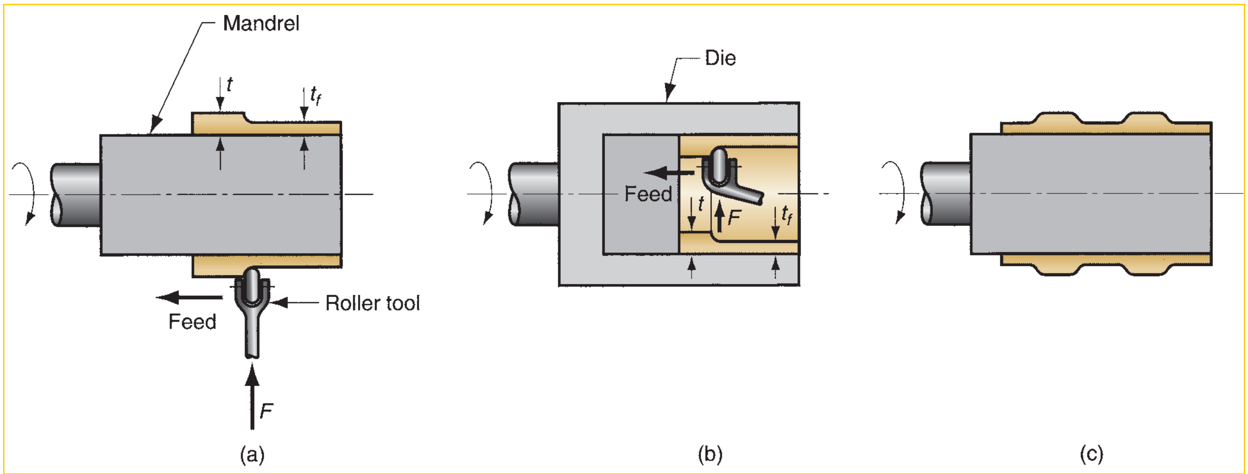

FIGURE 19.44 Tube spinning: (a) external, (b) internal, and (c) profiling.

There are limits to the amount of thinning that the metal will endure in a spinning operation before fracture occurs. The maximum reduction correlates well with reduction of area in a tension test [9].

Tube 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, as in Figure 19.44. Tube spinning is similar to shear spinning except that the starting workpiece is a tube rather than a flat disk. The operation can be performed by applying the roller against the work externally (using a cylindrical mandrel on the inside of the tube) or internally (using a die to surround the tube). It is also possible to form profiles in the walls of the cylinder, as in Figure 19.44(c), by controlling the path of the roller as it moves tangentially along the wall.

Spinning reduction for a tube-spinning operation that produces a wall of uniform thickness can be determined as in shear spinning by Equation (19.17).

19.6.4 HIGH-ENERGY-RATE FORMING

Several processes have been developed to form metals using large amounts of energy applied in a very short time. Owing to this feature, these operations are called ***high-energy-rate forming*** (HERF) processes. They include explosive forming, electrohydraulic forming, and electromagnetic forming.

Explosive Forming Explosive forming involves the use of an explosive charge to form sheet (or plate) metal into a die cavity. One method of implementing the process is illustrated in Figure 19.45. The work part is clamped and sealed over the die, and a vacuum is created in the cavity beneath. The apparatus is then placed in a large vessel of water. An explosive charge is placed in the water at a certain distance above the work. Detonation of the charge results in a shock wave whose energy is transmitted by the water to cause rapid forming of the part into the cavity. The size of the explosive charge and the distance at which it is placed above the part are largely a matter of art and experience. Explosive forming is reserved for large parts, typical of the aerospace industry.

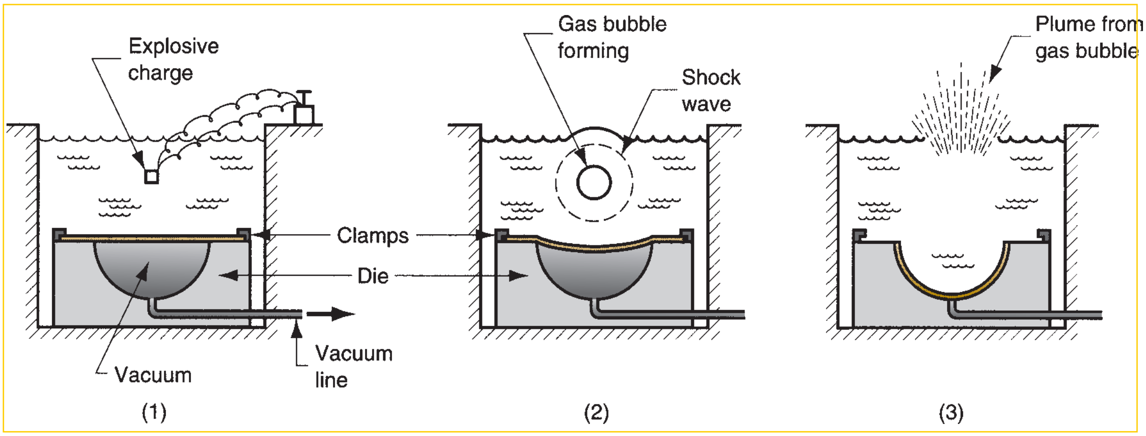

FIGURE 19.45 Explosive forming: (1) setup, (2) explosive is detonated, and (3) shock wave forms part and plume escapes water surface.

Electrohydraulic Forming Electrohydraulic forming is a HERF process in which a shock wave to deform the work into a die cavity is generated by the discharge of electrical energy between two electrodes submerged in a transmission fluid (water). Owing to its principle of operation, this process is also called ***electric discharge forming***. The setup for the process is illustrated in Figure 19.46. Electrical energy is accumulated in large capacitors and then released to the electrodes. Electrohydraulic forming is similar to explosive forming. The difference is in the method of generating the energy and the smaller amounts of energy that are released. This limits electrohydraulic forming to much smaller part sizes.

Electromagnetic Forming Electromagnetic forming, also called ***magnetic pulse forming***, is a process in which sheet metal is deformed by the mechanical force of an electromagnetic field induced in the work part by an energized coil. The coil, energized by a capacitor, produces a magnetic field. This generates eddy currents in the work that produce their own magnetic field. The induced field opposes the primary field, producing a mechanical force that deforms the part into the surrounding cavity. Developed in the 1960s, electromagnetic forming is the most widely used HERF process [10]. It is typically used to form tubular parts, as illustrated in Figure 19.47.

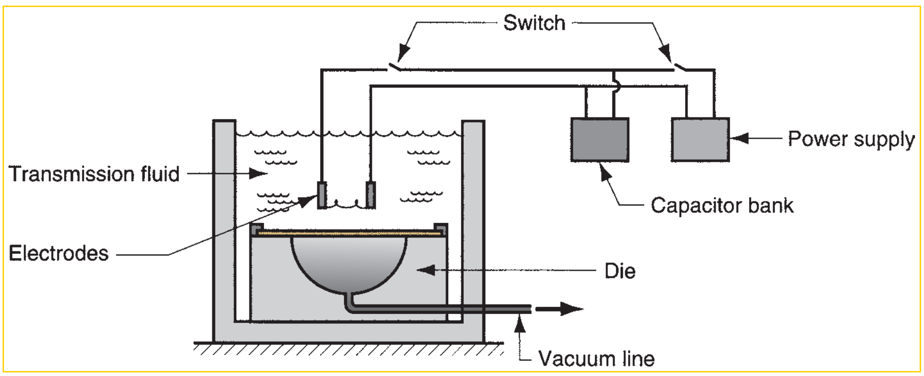

FIGURE 19.46 Electrohydraulic forming setup.

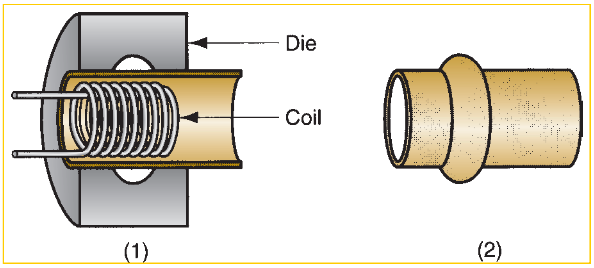

FIGURE 19.47 Electromagnetic forming: (1) setup in which coil is inserted into tubular work part surrounded by die; (2) formed part.

9.7 Bending of Tube Stock

Several methods of producing tubes and pipes are discussed in the previous chapter, and tube spinning is described in Section 19.6.3. This section examines methods by which tubes are bent and otherwise formed. Bending of tube stock is more difficult than sheet stock because a tube tends to collapse and fold when attempts are made to bend it. Special flexible mandrels are usually inserted into the tube prior to bending to support the walls during the operation.

Some of the terms in tube bending are defined in Figure 19.48. The radius of the bend R is defined with respect to the centerline of the tube. When the tube is bent, the wall on the inside of the bend is in compression, and the wall at the outside is in tension. These stress conditions cause thinning and elongation of the outer wall and thickening and shortening of the inner wall. As a result, there is a tendency for the inner and outer walls to be forced toward each other to cause the cross section of the tube to flatten. Because of this flattening tendency, the minimum bend radius R that the tube can be bent is about 1.5 times the diameter D when a mandrel is used and 3.0 times D when no mandrel is used [10]. The exact value depends on the wall factor WF, which is the diameter D divided by wall thickness t. Higher values of WF increase the minimum bend radius; that is, tube bending is more difficult for thin walls. Ductility of the work material is also an important factor in the process.

Several methods to bend tubes (and similar sections) are illustrated in Figure 19.49. ***Stretch bending*** is accomplished by pulling and bending the tube around a fixed form block, as in Figure 19.49(a). ***Draw bending*** is performed by clamping the tube against a form block, and then pulling the tube through the bend by

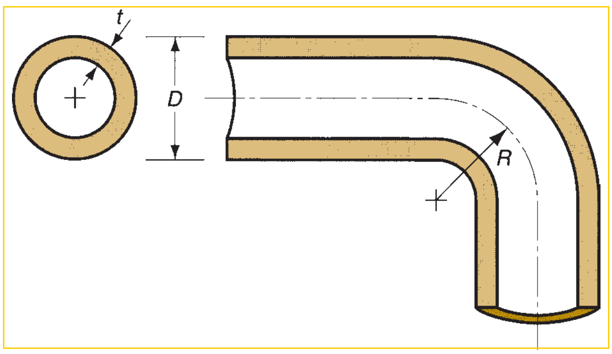

FIGURE 19.48 Dimensions and terms for a bent tube: D = outside diameter of tube, R = bend radius, t = wall thickness.

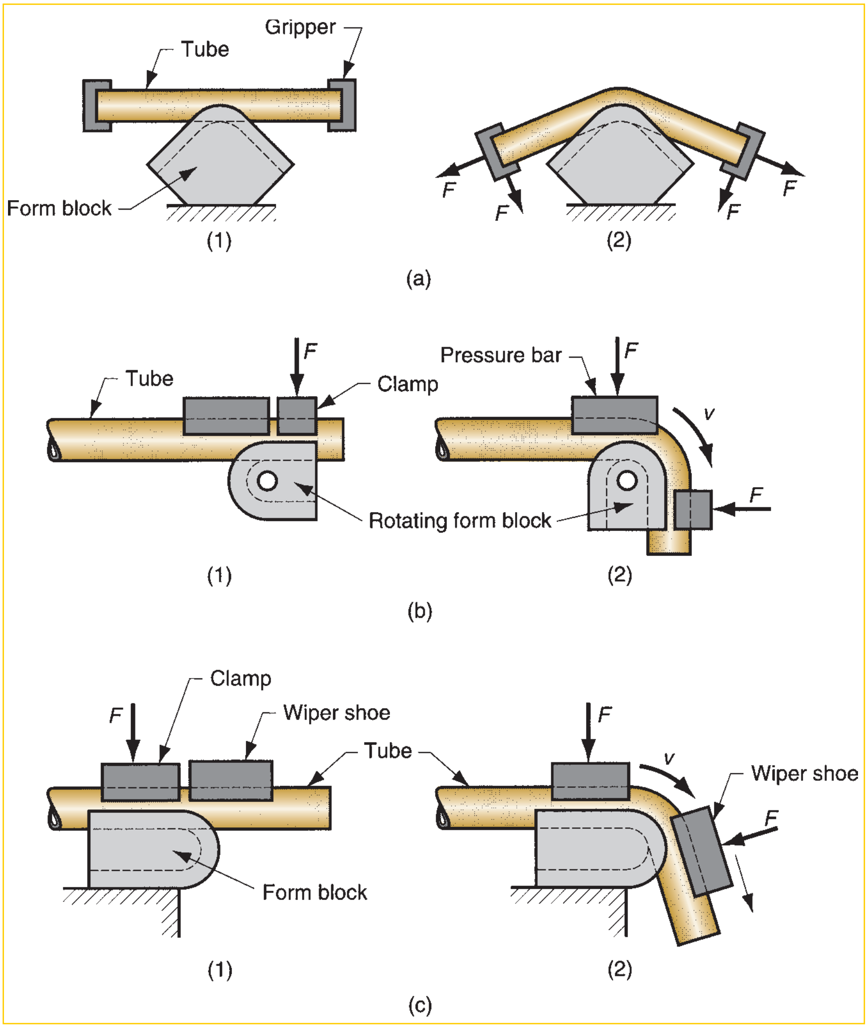

FIGURE 19.49 Tube bending methods: (a) stretch bending, (b) draw bending, and (c) compression bending. For each method: (1) start of process, and (2) during bending. Symbols *v* and *F* indicate motion and applied force.

rotating the block as in (b). A pressure bar is used to support the work as it is being bent. In ***compression bending***, a wiper shoe is used to wrap the tube around the contour of a fixed form block, as in (c). ***Roll bending*** (Section 19.6.2), generally associated with the forming of sheet stock, is also used for bending tubes and other cross sections.

References

[1] ***ASM Handbook***, Vol. 14B: ***Metalworking: Sheet Forming***, ASM International, Materials Park, Ohio, 2006.

[2] Eary, D. F., and Reed, E. A. ***Techniques of Pressworking Sheet Metal***, 2nd ed. Prentice-Hall, Inc., Englewood Cliffs, New Jersey, 1974.

[3] Hoffman, E. G. (ed.). ***Fundamentals of Tool Design***, 2nd ed. Society of Manufacturing Engineers, Dearborn, Michigan, 1984.

[4] Hosford, W. F., and Caddell, R. M. ***Metal Forming: Mechanics and Metallurgy***, 3rd ed. Cambridge University Press, Cambridge, UK, 2007.