Design of Sheet-metal Bending, Forming, Drawing Dies
Bending, Forming and Drawing:

**Figure 10-1**
Basic operations for shaping sheet metal: (a) bending, (b) forming, (c) drawing.

Figure 1: A schematic view of the combination of compressive and tensile forces at a bending zone.
Bending & Forming

**Bending:** Metal flow is uniform along the bend axis, with the inner surface of the bend in compression and the outer surface in tension.

**Forming:** Forming is similar to bending, except that the form or bend is along a curved axis instead of straight one. A part that is formed generally takes the shape of the punch or die. Metal flow is not as uniform as in bending because it may be localized to some extent, depending upon the shape of the workpiece.
Drawing: Drawing is the most complicated of the three sheet-metal shaping process. A drawing operation begins with a flat blank which is transformed into a cup or shell. The parent metal is subjected to severe plastic deformation. Shell forms produced may be cylindrical or rectangular with straight or tapered sides.
Bending methods

FIGURE 10-2
Methods of bending sheet metal: (a) V bending, (b) wiping die.
Bending methods

- **Bending methods**: two bending methods are used extensively in presses are wiping dies and V bending.
- **V bending**: V bending is accomplished by using a V block for a die and a wedge-shaped punch to force the metal into the die. The desirable width of the opening in the V is ordinarily at least 8 times the material thickness, up to about $\frac{5}{8}\text{in.}$ Plate. Die opening up to 10 to 12 times are used for forming heavier thickness of plate. The included angle of the V bend can be changed by varying the distance the punch forces the sheet metal into the V die.
Bending methods

**Wiping die:** A wiping die is shown in Fig. 10-2b. Here, the workpiece is clamped to a die block by a spring loaded or fluid-cylinder pressure pad, and the punch wipes extended material over one edge of the die. A bend radius is provided on the edge of the die block. A **radius or chamfer** is often provided on the leading edge of the punch to prevent the wiping action from being too severe.
Springback

Springback: Elastic stresses remaining in the bend area after being pressure is released will cause a slight decrease in the bend angle. Metal movement of this type is known as springback, and the magnitude of movement will vary according to material type, thickness, and hardness. A larger bend radius will also cause greater springback.

- Springback for low-carbon and soft nonferrous material is from 0 to 2°
- For 0.40 to 0.50 carbon steel and half–hard materials springback may vary from 3 to 5°
- Springback may be as high as 10 to 15° in harder materials.
Springback

\[ R_i \text{: Bend Radius Before Springback} \]
\[ R_f \text{: Bend Radius After Springback} \]
\[ \alpha_i \text{: Bending Angle Before Springback} \]
\[ \alpha_f \text{: Bending Angle After Springback} \]

\[ R_i < R_f \quad \alpha_i < \alpha_f \]
Springback Prevention

Prevention techniques:

1. To determine the necessary amount of *overbend* by trial and error method.

2. Coining (squeezing) the metal slightly at the corner in order to relieve elastic stresses. This is sometimes referred to as *corner setting*.
Springback Prevention

Prevention techniques:

3. Figure 10.5 shows various methods of *modifying the punch nose* for corner-setting operations. Corner setting verges on bottoming out, and *pressure is built up rapidly*; for this reason large contact areas should be avoided. Only a *small amount of squeezing* is necessary.
Bend Allowance

In bending operations the material near the bend radius is under *compression* while the material near the outside of the bend is under *tension*, as shown in Fig. 10.7. A *neutral plane exists* between the area under tension and the area under compression. If the material is uniform in section and its elastic limit is not exceeded, neutral plane will coincide with the centerline of the material.
Bend Allowance

When a blank or sheet is to be bent, it is necessary to consider the effect of stretching the metal at the outside of the bend. Since there is no stretch in the neutral plane, the length of the formed part along the neutral plane will be the correct length. The curved neutral plane of the bend area is called the bend allowance, as shown in Fig. 9-7.
Bend Allowance

U dies and channel dies: This type of tooling for bending, which may be considered a cross between V-bending and wiping dies, is so named because the workpieces produced in them bear resemblance to the letter U or a channel (see fig. 10.9). U dies are generally equipped with a pressure pad, as shown, which helps prevent the metal from bowing away from the flat face of the punch. This will not completely eliminate springback, and it may be necessary to bottom out against the pressure pad to prevent the legs of the channel from springing apart on the return stroke. This additional pressure will increase the press load by several tons but produces a part with the bottom of the channel only slightly hollow at its center.
Bending force

FIGURE 10-11
Determining $W$ for bending-pressure formula: (a) air bending, (b) channel bending and wiping dies.

Calculate the bending force in tons for a 45° bend in 24ST3 and the closing 8 times the metal thickness.
Forming dies

1. Solid form dies
2. Pad-type form dies
3. Curling dies
4. Embossing dies
5. Coining dies
6. Bulging dies
7. Assembly dies
Curling Dies

A curling die rolls a raw edge of sheet metal into a roll, or curl, as shown in the figure. The purpose is to strengthen the raw edge, provide a protective edge, and improve the appearance of the product. The curl is often applied over a wire ring for increased strength.
EMBOSSING

• Embossing dies 

Embossing is a shallow forming operation in which the workpiece material is stretched over a male die and caused to conform to the male-die surface by a mating female-die surface. In other words, the workpiece material is displaced between a male and female surface. The finished product will have a depressed detail on one side and a raised detail on the other. The major difference between embossing and forming is that the displaced pattern is much smaller and shallower. An embossed pattern may have more intricate detail than a formed pattern.
EMBOSSING

• Embossing is used to **stiffen and strengthen** a sheet metal part or to impart a raised or depressed design on the surface of the part.

• The circular grooves on the bottom of a sheet metal container are a good example of embossing for stiffness and strength. The **metal military button with insignia** raised on the surface is an example of embossing for detail.

• Embossing dies are limited to rather shallow detail. If the metal tears or wrinkles with a particular design, the use of a drawing quality or more ductile material may help. The problem areas within the die may also be toned down or made with less relief.
COINING

- Coining dies Coining is the process of pressing material in a die so that it flows into the spaces in the detail on the die face, as shown in Fig. 10-22. Coining differs from embossing in that in coining the metal flows, whereas in embossing the metal does not change in thickness to any great extent. Coining operations are generally performed cold, and for the most part all surfaces of the workpiece are confined or restrained.

- The metal slug to be coined should be nearly the size of the die cavity and contact the die surfaces over a large area because only a small amount of metal will flow during a single hit of the press. When it is necessary to redistribute a large amount of metal, a preliminary metal-redistribution process such as forging or extrusion should precede the coining.
COINING

- Coining has two major advantages: (1) ornate detail can be reproduced with excellent surface finish, and (2) very close tolerances can be held. Metal buttons, tableware, medallions, medals, coins, jewelry, and decoration items are examples of workpieces produced by coining when ornate detail is required.

- Drop hammers or knuckle-joint and hydraulic presses are generally used for coining operations because of the heavy forces required; however, coining may be done satisfactorily on any type of press providing it has the necessary capacity and precautions are taken to prevent overloading.
COINING

- Coining is a severe metal squeezing operation in which the flow of metal occurs only at the top layers of the material and not throughout the values.
- The operation is carried out in closed dies mainly for the purpose of producing fine details such as needed in minting coins, and medal or jewelry making.
- The blank is kept in the die cavity and pressures as high as five to six times the strength of material are applied.
- Depending upon the details required to be coined on the part, more than one coining operations may be used.
EMBOSSING and COINING

Embossing and Coining

- Embossing:
  - A metalworking operation used to create raised surfaces or lettering in sheet metal. There is theoretically no change in metal thickness during embossing

- Coining:
  - A metalworking operation used to create raised surfaces and imprints in metal. Coining is a relatively severe operation that creates variations in metal thickness.
Bulging dies

• **Bulging dies:** Bulging is an internal forming operation used to expand portions of a drawn shell or tube. The forming force is applied from inside the workpiece and is transmitted through a medium that will flow and not compress. The more common medium are rubber, urethane, heavy grease, oil, or water.

• A bulging die must be cut into sections that open, in order to remove the workpiece after bulging. Generally two halves are satisfactory. The die halves must be positively retained during the bulging operation and be equipped with quick acting clamps or latches. Bulging operations using liquid as a pressure-dispersing medium must be carefully made to keep leakage to a minimum.

• Rubber or urethane is preferred as a pressure dispersing medium because they are clean and easy to use.
Bulging dies
Bulging dies

FIGURE 10-23
Urethane bulging die: (a) split horizontally, (b) split vertically.
Bulging dies

FIGURE 10-24
Bulging die using liquid medium.
**Drawing operation**

Drawing is the name given to the production of cups, shells, boxes, and similar articles from metal blanks. A round blank is first cut from flat stock. The blank is then placed in the draw die, where the punch pushes the blank through the die. On the return stroke the cup is stripped from the punch by the counterbore in the bottom of the die. The top edge of the shell expands (springback) slightly in order to make this possible. Note that the amount of springback in the drawing has been exaggerated. The punch has an air vent to prevent a vacuum being formed when the part is stripped from the punch.

**FIGURE 10-27**

Methods of retaining metal in draw dies: (a) with rigid blank holder and (b) with spring pressure pad.
Metal flow during drawing

Wrinkle rather than compress, especially in thin sheets or with deeper draws. A blank holder is used to prevent the formation of wrinkles in this case. *There must be enough force on the blank holder to prevent formation of wrinkles* because after a wrinkle is started, the blank holder is raised from the surface of the metal and allows other wrinkles to start.
Draw radius on die

FIGURE 10-31

Example:

\[ d = 1.00 \quad h = 0.75 \quad T = 0.020 \]

\[ D = \sqrt{d^2 + 4h} = 2.00 \]

\[ D - d = 1 \]

\[ R = 0.119 \pm 0.005 \]
If the draw radius is too large, the metal will be released by the blank holder too soon and wrinkling will result. Too sharp a radius will hinder the normal flow of the metal and cause uneven thinning of the cup wall, with resultant tearing. The general rule is to make the draw radius 4 times the thickness. The draw radius may be increased to 6 to 8 times the metal thickness when drawing shallow cups of heavy-gage metals without a blank holder. The nomograph in Fig.10-30 gives a more exact method of determining draw-die radius, based on the hip of the blank diameter to the cup diameter.
Friction:
Material to be drawn:
The following principal factors affect the selection of grade and quality of low-carbon-steel sheets for deep drawing:
1. Severity of draw as determined by the amount of reduction and punch-nose radius
2. Thickness of sheet
3. Shape of part (round, rectangular, or conical)
4. Flange requirements
5. Ironing requirements
6. Desired finish
7. Grain size
8. Press speed
9. Availability of material
10. Cost
Percent reduction and depth of draw

Percent reduction and depth of draw: The percent reduction in drawing cylindrical shells is generally expressed in terms of the diameters of the blank $D$ and the drawn shell $d$, where $D$ equals the OD of the blank and $d$ equals the ID of the shell. This percentage provides an approximate value for the amount the work material is to be compressed. The drawability of a metal is often expressed as the percentage reduction from the blank diameter to the cup diameter. Percentage reduction is calculated from the formula

$$P = 100 \left(1 - \frac{d}{D}\right)$$

where $P =$ percentage reduction
$d =$ ID of drawn shell
$D =$ OD of blank
## Table 10-4  POSSIBLE NUMBER OF REDUCTIONS FOR A GIVEN RATIO OF SHELL HEIGHT TO DIAMETER

<table>
<thead>
<tr>
<th>Ratio ( h/d )</th>
<th>Number of reductions</th>
<th>Reduction %</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>First draw</td>
</tr>
<tr>
<td>Up to 0.75</td>
<td>1</td>
<td>40</td>
</tr>
<tr>
<td>0.75-1.5</td>
<td>2</td>
<td>40</td>
</tr>
<tr>
<td>1.5-3.0</td>
<td>3</td>
<td>40</td>
</tr>
<tr>
<td>3.0-4.5</td>
<td>4</td>
<td>40</td>
</tr>
</tbody>
</table>
Percent reduction and depth of draw

FIGURE 10-32
Chart for checking percentage reduction in drawing of cups. The inside diameter is ordinarily used for the cup diameter. (From ASM, "Metals Handbook," 8th ed., vol. 4, 1969.)
Drawing speed

**Drawing speed**: The velocity at which the punch penetrates the workpiece often has a definite effect on a drawing operation. Drawing speed is usually expressed in **linear feet per minute**. Low carbon is normally drawn from 30 to 50 fpm under normal conditions. Nonferrous work material is drawn from 150 to 200 fpm. Generally speaking, *the harder and less ductile the material, the slower the drawing speed must be*. When cracking or excessive thinning occurs, the drawing speed must be reduced.
Die clearance

Die clearance is the gap left between the punch and die to allow for the flow of the work material. Generally enough clearance is left to allow for thickening of the metal. This allowance ranges from 7 to 20 percent of the metal thickness, depending upon the type of operation and the metal. Table 10.5 shows draw clearance for various blank thickness.

<table>
<thead>
<tr>
<th>Blank thickness, in.</th>
<th>First draws</th>
<th>Redraws</th>
<th>Sizing draw*</th>
</tr>
</thead>
<tbody>
<tr>
<td>Up to 0.15</td>
<td>1.07t-1.09t</td>
<td>1.08t-1.1t</td>
<td>1.04t-1.05t</td>
</tr>
<tr>
<td>0.016-0.050</td>
<td>1.08t-1.1t</td>
<td>1.09t-1.12t</td>
<td>1.05t-1.06t</td>
</tr>
<tr>
<td>0.051-0.125</td>
<td>1.1t-1.12t</td>
<td>1.12t-1.14t</td>
<td>1.07t-1.09t</td>
</tr>
<tr>
<td>0.136 and up</td>
<td>1.12t-1.14t</td>
<td>1.15t-1.2t</td>
<td>1.08t-1.1t</td>
</tr>
</tbody>
</table>

* Used for straight-sided shells where diameter or wall thickness is important or where it is necessary to improve the surface finish in order to reduce finishing costs.
Trim allowance

The following equations can be used to calculate the blank size for cylindrical shells of relatively thin metal. The ratio of the shell diameter to the corner radius \( d/r \) can affect the blank diameter and should be taken into consideration.

\[
D = \begin{cases} 
\sqrt{d^2 + 4dh} & \text{when } d/r \text{ is 20 or more} \\
\sqrt{d^2 + 4dh - 0.5r} & \text{when } d/r \text{ is between 15 and 20} \\
\sqrt{d^2 + 4dh - r} & \text{when } d/r \text{ is between 10 and 15} \\
\sqrt{(d - 2r)^2 + 4d(h - r) + 2\pi r(d - 0.7r)} & \text{when } d/r \text{ is below 10}
\end{cases}
\]

where \( D = \) blank diameter
\( d = \) shell OD
\( h = \) shell height
\( r = \) corner radius of punch

These formulas give the theoretical blank size, which is only an approximation when applied to actual practice. Extra metal should be added to the formula blank diameter to provide for trimming, which is generally necessary on deeper draws to eliminate the uneven and irregular edge on the rim of the drawn cup. The extra material added to the blank diameter is referred to as trim allowance. The necessary trim allowance increases as the size of the drawn cup increases.