Production Engineering II

2.2 Sheet Metal Working
Sheet metal forming

• Sheet metal working includes cutting and forming operations performed on relatively thin sheet of metal.

• Typical sheet-metal thickness are 0.4 and 6mm, when thickness exceeds 6mm the stock is referred to as plate rather than sheet.

• The sheet or plate which used for sheet metal working are produced by rolling.
Parts made by sheet and plate metal:
• Automobile bodies, airplanes, railway cars, locomotives, farm and construction equipment, appliances, office furniture and etc.

**Advantages of sheet metal working:**
✓ High strength, good dimensional accuracy, good surface finish, relatively low cost.
✓ For components that must be made in large quantities, economical mass production can be designed.
Most sheet metal processing is performed at room temperature (cold working), except when the stock is thick, the metal is brittle, or the deformation is significant it uses warm or hot working.

Most sheet metal operations are performed on machine tools called presses. The term stamping press is used to distinguish this presses from forging & extrusion presses.

The tooling that performs sheet metal work is called a punch-and die. To facilitate mass production, the sheet metal is often presented to the press as long stripes or coils.
Sheet metal working

✓ Sheet metals are categorized into three major processes:

1. cutting, 2. Bending, and 3. drawing

1. Cutting operations

• Used to separate large sheets into smaller pieces, to cut out part perimeters, and to make holes in parts.

• Cutting of sheet metal is accomplished by a shearing action between two sharp cutting edges.
Cutting operation
• The shearing action is depicted in the four stop-action:

Figure.1.1 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.

**NB:** if the clearance (c) between the punch and die is correct, the two fracture lines meet, resulting in a clean separation of the work into two pieces.
Sheet metal cutting

The sheared edges of the sheet have characteristic features:

• **Rollover**: this corresponds to the depression made by the punch in the work prior to cutting. It is where initial plastic deformation occurred in the work.

• **Burnish**: this results from penetration of the punch into the workpiece before fracture begins.

• **Fractured zone**: a relatively rough surface of the cut edge continued downward movement of the punch caused fracture of metal.

• **Burr**: a sharp corner on the edge caused by elongation of the metal during final separation of the two pieces.
Cutting

• There are three principal operations in press working that cut metal by the shearing mechanism: shearing, blanking, and punching.

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

✓ Shearing is typically used to cut large sheets into smaller sections for subsequent press working operations.

✓ It is performed on a machine called a power shears, or squaring shears. The upper blade of the power shears is often inclined to reduce the required cutting force.
Figure 1.2 shearing operation: (a) side view of shearing operation; and (b) front view of power shears equipped with inclined upper cutting blade.

b. **Blanking**: involves cutting of the sheet metal along a closed outline in a single step to separate the piece from the surrounding stock. The party that is cut out is the desired product in the operation and is called the blank.

c. **Punching**: is similar to blanking except that the separated piece is scrap, called the slag. The remaining stock is the desired part.
Engineering analysis of sheet metal cutting

- Important parameters in sheet metal cutting are Clearance between punch and die, stock thickness, type of metal and its strength, and length of the cut.
**Engineering analysis of sheet metal cutting**

- Clearance in a shearing operation is the distance between the punch and die. Typical clearance in conventional press working range between 4% and 8% of the sheet metal thickness.
- If the clearance is too small, then fracture lines tend to pass each other, causing a double burnishing and larger cutting force.
- If the clearance is too large, the metal becomes pinched between the cutting edges and an excessive burr results.

✓ In special operations requiring very straight edges, such as shaving and fine blanking, clearance is only about 1% of stock thickness.

Figure.1.4 (a) too small clearance, and (b) too large clearance.
Engineering analysis of sheet metal cutting

- The recommended clearance depends on sheet metal type and thickness, it can be calculated:

\[ c = A_C t \]

Where, \( c \) = clearance, mm; \( A_C \) = clearance allowance; and
\( t \) = stock thickness

- The clearance allowance is determined according to the type of metal.
- These calculated clearance values can be applied to conventional blanking and hole-punching operations to determine the proper punch and die sizes.
Engineering analysis of sheet-metal cutting

<table>
<thead>
<tr>
<th>Metal Group</th>
<th>$A_c$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1100S and 5052S aluminum alloys, all tempers.</td>
<td>0.045</td>
</tr>
<tr>
<td>2024ST and 6061ST aluminum alloys; brass, all tempers;</td>
<td>0.060</td>
</tr>
<tr>
<td>soft cold-rolled steel, soft stainless steel.</td>
<td></td>
</tr>
<tr>
<td>Cold-rolled steel, half hard; stainless steel, half-hard and full-hard.</td>
<td>0.075</td>
</tr>
</tbody>
</table>

Table.1 clearance allowance value for three sheet-metal groups
• The die opening must be always be larger than the punch size.

✓ Punch and die sizes for a round blank of diameter $D_b$ are determined as:

   blanking punch diameter = $D_b - 2C$
   blanking die diameter = $D_b$

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

   hole punch diameter = $D_h$
   hole die diameter = $D_h + 2C$
Figure 1.5  

Die size determines blank size ($D_b$); Punch size determines hole size ($D_h$) and

Angular clearance- allows slug or blank to drop through die. Typical values of 0.25° to 1.5° on each side.
Cutting force: used to determine the size (tonnage) of the press needed and can be calculated as follows:

\[ F = S \cdot t \cdot L \]

Where, \( F \)- cutting force; \( S \)-shear strength of the sheet metal, MPa; \( t \)-stock thickness; and \( L \)-length of the cut edge, mm.

- In blanking, punching, slotting, and similar operations, \( L \) is the perimeter length of the blank or hole being cut.
- If shear strength is unknown, an alternative way of estimating the cutting force is to use the tensile strength, as follows:

\[ F = 0.7 \cdot TS \cdot t \cdot L \]

Where, \( TS \) – ultimate tensile strength, MPa.
Other sheet-metal cutting operations

a. **Cut off**: is a shearing operation in which blanks are separated from a sheet-metal strip by cutting the opposite sides of the part in sequence.

b. **Parting**: cutting a sheet-metal strip by a punch with two cutting edges that match the opposite sides of the blank.

c. **Shaving**: performed with very small clearance to obtain accurate dimensions and cut edges that are smooth and straight.

d. **Fine blanking**: used to blank sheet-metal parts with close tolerances and smooth, straight edges in one step.
Figure 1.7 (a) shaving; and (b) fine blanking
2. Bending operations

• The straining of the metal around a straight axis to take a permanent bend.

• During operation, the metal on the inside of the neutral plane is compressed, while the metal on the outside of the neutral plane is stretched.

Figure 2.1  (a) bending of sheet-metal; and (b) both compressive and tensile elongation of the metal occur in bending.
Types of sheet metal bending

a. **v-bending** - performed by compressing the sheet metal between a matching V-shaped punch and die.
   - For low production
   - Performed on press brake
   - V-dies are simple and inexpensive

b. **Edge-bending** - performed by compressing the sheet metal between two flat dies and using a punch to bend an extended portion of the sheet over the lower die corner.
   - For high production
   - Pressure pad required
   - Dies are more complex and costly
Types of bending

Figure 2.2 two common bending methods: (a) v-bending; and (b) edge bending
Engineering analysis of bending

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

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

where \( BA \) = bending allowance; \( A \) = bend angle; \( R \) = bend radius; \( t \) = stock thickness; and \( K_{ba} \) = stretch factor.

If \( R < 2t \), \( K_{ba} = 0.33 \)
If \( R \geq 2t \), \( K_{ba} = 0.50 \)
springback

• The Increase in included angle of bent part relative to included angle of forming tool after tool is removed.

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

\[
SB = \frac{\alpha' - \alpha'_t}{\alpha'_t}
\]

Where, \( SB \) = springback; \( \alpha' \) = included angle of the sheet metal part; and \( \alpha'_{t} \) = included angle of the bending tool, degrees.
Figure 2.3  Springback in bending is seen as a decrease in bend angle and an increase in bend radius: (1) during bending, the work is forced to take radius $R_t$ and included angle $\alpha_t'$ of the bending tool, (2) after punch is removed, the work springs back to radius $R$ and angle $\alpha'$, $F =$ applied bending force.
Bending analysis

**Bending force:** the force required to perform bending depends on the geometry of the punch and die, strength, thickness, and length of the sheet metal. Maximum bending force estimated as follows:

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

where \( F \) = bending force; \( TS \) = tensile strength of sheet metal; \( w \) = part width in direction of bend axis; and \( t \) = stock thickness. For V- bending, \( K_{bf} = 1.33 \); for edge bending, \( K_{bf} = 0.33 \), \( D \): Die Opening Dimension
Die opening dimension

Figure 2.4 Die opening dimension $D$: (a) V-die, (b) wiping die.
Various bending operations are depicted in figure to illustrate the variety of shapes that can be bent. Most of these operations are performed in relative simple dies similar to V-dies.
other bending operations

- **Flanging**: bending edges of sheet metal to 90 degrees.
- **Hemming**: folding the edges of sheet metal over itself.
- **Seaming**: joining two edges of sheet metal by hemming.
- **Curling (beading)**: bending the periphery of sheet metal into a cavity of a die to improve appearance and eliminate exposed edges.
3. Drawing

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

• Performed by placing a piece of sheet metal over a die cavity and then pushing the metal into the opening with a punch.

• Products: beverage cans, ammunition shells, automobile body panels

• Also known as deep drawing (to distinguish it from wire and bar drawing)
Figure 3.1  (a) Drawing of cup-shaped part: (1) before punch contacts work, (2) near end of stroke; (b) workpart: (1) starting blank, (2) drawn part.
Mechanics of drawing

• Sides of punch and die separated by a clearance $c$ given by:

\[ c = 1.1 \, t \]

where $t = \text{stock thickness}$

• In other words, clearance is about 10% greater than stock thickness
Engineering analysis of drawing

Test of drawing feasibility:

- Drawing ratio
- Reduction
- Thickness to diameter ratio.

Drawing and reduction ratio are easily defined for cylindrical shapes.

\[
DR = \frac{D_b}{D_p}
\]

where \(D_b\) = blank diameter; and \(D_p\) = punch diameter

The greater the ratio, the more severe the drawing operation.

Upper limit: \(DR \leq 2.0\)
Engineering analysis of drawing

\[ r = \frac{D_b - D_p}{D_b} \]

Where, \( r \) = reduction ratio
Value of \( r \) should be less than 0.50

**Thickness to Diameter Ratio:**

Thickness of starting blank divided by blank diameter
- Desirable for \( t/D_b \) ratio to be greater than 1%
- As \( t/D_b \) decreases, tendency for wrinkling increases
These are guidelines for Feasibility testing
Engineering analysis of drawing

Blank size determination

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

The drawing force required to perform a given operation can be estimated:

\[ F = \pi D_p t \left( TS \left( \frac{D_b}{D_p} - 0.7 \right) \right) \]

Where, \( F \) - drawing force; \( t \) = original blank thickness; \( TS \) = tensile strength; and \( D_b \) and \( D_p \) are the starting blank diameter and punch. 0.7 is the correction factor for friction.

Holding force is an important factor in drawing operation, and can be estimated:

\[ F_h = 0.015 Y \pi \left( D_b^2 - (D_p + 2t + 2R_d)^2 \right) \]

Where, \( F_h \) = holding force; \( Y \) = yield strength of sheet metal; \( t \) = starting stock thickness; \( R_d \) = die corner radius.  
✓ Holding force is usually about one-third of the drawing force.
Other drawing types

**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.

**Reverse drawing:** drawn part is positioned face down on the die so that the second drawing operation will be performed.

![Figure 3.2](image)

**Drawing without blank holder:** to prevent wrinkling of the flange while the cup is being drawn.

✓ If the thickness to diameter ratio is large enough, drawing can be accomplished without a blankholder.

\[ Db - Dp < 5t \]

**Advantage of drawing without blank holder**

lower tooling cost, uses simpler press, and the need to control the movement of the blankholder and punch can be avoided.
Defects in drawing

- Wrinkling in the flange - small holding force
- Wrinkling in the wall - insufficient holding force,
- Tearing - high stress and sharp die radius
- Earing - anisotropy of the material
- Surface scratches - die or punch not having a smooth surface, insufficient lubrication.

Figure 3.3 common defects in drawn parts: (a) wrinkling in the flange, (b) wrinkling in the wall, (c) tearing (d) earing, and (e) surface scratches
Other sheet metal forming operations

Other sheet metal forming operations performed on conventional presses:

• Operations performed with metal tooling
• Operations performed with flexible rubber tooling

a. Operations performed on metal tooling includes:
   1. Ironing
   2. Coining
   3. Lancing
   4. twisting
1. **Ironing**: a continuous thinning process and often accompanies deep drawing, i.e., thinning of the wall of a cylindrical cup by passing it through an ironing die. Products: beverage cans, artillery shells,

2. **Coining**: used in sheet metal work to form indentations and raised sections in the part.

3. **Embossing**: Used to create indentations in sheet, such as raised (or indented) lettering or strengthening ribs.

   Embossing dies possess matching cavity contours, the punch containing the positive contour and 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.
Operations performed on metal tooling

Figure 3.4 Ironing to achieve a more uniform wall thickness in a drawn cup: (1) start of process; (2) during process

Figure 3.5 Embossing: (a) cross-section of punch and die configuration during pressing; (b) finished part with embossed ribs
4. **lancing**: a combined cutting and bending or cutting and forming operation performed in one step to partially separate the metal from the sheet.

➢ Lancing used to make louvers in sheet metal for venting of heat from the interiors of electrical cabinets.

![Lancing in several forms](image)

**Figure.3.6 lancing in several forms**: (a) cutting and bending; (b) and (c) types of cutting and forming
b. Rubber forming process

The operations performed on conventional presses and the tooling are flexible element (made of rubber or similar materials) to effect the forming operation.

1. Guerin process: uses a thick rubber pad (other flexible material) to form sheet metal over a positive form block.

**Advantage of Guerin process**

- low cost of tooling
- form block can be made of wood, plastic, or other materials that are easy to shape
- The rubber pad can be used with different from blocks.
- Process attractive in small quantity production e.g. aircraft industry
2. **Hydroforming**: is similar to Guerin process, the difference is that it substitutes a rubber diaphragm filled with hydraulic fluid in place of the thick rubber pad.

- Allows the working pressure to be around 100MPa, whereas in Guerin 10MPa. This prevents wrinkling in deep drawing.
Dies and presses for sheet metal process

Most press working operations performed with conventional punch-and-die tooling

- The term stamping die sometimes used for high production dies.

![Diagram of a punch and die for a blanking operation](image)

Figure 3.9 components of a punch and die for a blanking operation
Types of stamping dies

- **Simple die**: performs a single blanking operation with each stroke of the press.

- **Compound die**: performs two operations at a single station, such as blanking and punching, or blanking and drawing. E.g. a compound die that blanks and punches a washer.

- **Combination die**: performs two operations at two different stations in the die. E.g. blanking two different parts (e.g. right and left hand parts), or blanking and then bending the same part.

- **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 from the remaining coil.
**Progressive die:**
- can have a dozen or more stations,
- the most complicated and most costly stamping dies,
- economically justified only for complex parts requiring multiple operations at high-production rates.

![Diagram of a progressive die with labels](image)

**Figure.3.10 (a) progressive die and (b) associated strip development**
Figure.3.11  progressive blanking and piercing of flat washer

Procedure:
The first punch is to make the hole of the washer. The washer is then blanked from the strip. The punch A is piercing the hole for the next washer.
Presses

• A press used for sheet metal working is a machine tool with a stationary bed and a powered ram that can be driven toward and away from the bed to perform various cutting and forming operations.

• The relative positions of the bed and ram are established by the frame, and the ram is driven by mechanical or hydraulic power.

• The **capacity of a press** is its ability to deliver the required force and energy to accomplish the stamping operation.

• 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.
There are two types of presses: gap frame and straight-sided frame press.
Types of stamping press frame

i. **Gap frame presses:**
   - has the general configuration of the letter C and is often called **C-frame**.
   - Provide good access to the die, and they are usually open in the back to permit convenient ejection of stampings or scrap.
   - Types of gap frame press: (a) solid gap frame, (b) adjustable bed, (c) open back inclinable, (d) press brake, and (e) turret press.

✓ **Solid gap frame** - the frame is rigid or one piece.
✓ **Adjustable bed** - has an adjustable bed to accommodate various die sizes.
✓ **Open back inclinable** - the frame can be tilted to various angles.
✓ **Press brake** - has a very wide bed, allows a number of separate dies to be set in the bed.
✓ **Turret press** - suited to situations in which a sequence of punching, notching, and related cutting operations.
Figure 3.13 Gap frame press for sheet metalworking; capacity = 1350 kN (150 tons)

Figure 3.14 Press brake; bed width = 9.15 m (30 ft) and capacity = 11,200 kN (1250 tons)
ii. Straight-sided frame presses

- Have full sides, giving it a box like appearance.
- Its construction increases the strength and stiffness of the frame.
- Large presses of this frame type are used for forging.
- Its capacity is up to 35,000KN (4000 tons).
Sheet metal operations not performed on presses

1. **Stretch forming**: Sheet metal is stretched and simultaneously bent to achieve shape change.

![Diagram of stretch forming process]

Figure 3.16 Stretch forming: (1) start of process; (2) form die is pressed into the work with force $F_{\text{die}}$, causing it to be stretched and bent over the form. $F$ = stretching force.

**Force required for Stretch forming can be computed from**: $F = LtY_f$

Where,
- $L$ = length of sheet in direction perpendicular to stretching;
- $t$ = instantaneous stock thickness;
- $Y_f$ = flow stress of work metal

Die force $F_{\text{die}}$ can be determined by balancing vertical force components.
2. **Roll bending**: Large metal sheets and plates are formed into curved sections using rolls.

![Figure 3.17 roll bending](image1)

![Figure 3.18 roll forming of a continuous channel section: (1) straight rolls, (2) partial form, and (3) final form.](image2)

3. **Roll forming**: Continuous bending process in which opposing rolls produce long sections of formed shapes from coil or strip stock.
4. **Spinning**: Metal forming process in which an axially symmetric part is gradually shaped over a rotating mandrel using a rounded tool or roller.

- There are three types of spinning:
  1. Conventional spinning
  2. Shear spinning
  3. Tube spinning

![Diagram of conventional spinning](image)

**Figure.3.19** Conventional spinning: (1) setup at start of process; (2) during spinning; and (3) completion of process.
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.

Terms in tube bending:
✓ the radius of the bend is defined with respect to the centerline of the tube.
✓ The minimum bend radius $R$ that the tube can be bend is about 1.5 times the diameter when mandrel is used. If mandrel is not used it becomes 3 times the diameter.
• The following are some of the methods of tube bending:
  a. **stretch bending**- accomplished by pulling and bending the tube around a fixed form blocks.
  b. **draw bending**- performed by clamping the tube against a form block, and then pulling the tube through the bend by rotating the block.
  c. **compression bending**- a wiper shoe is used to wrap the tube around the contour of a fixed form block.

Figure 3.20 tube bending methods: (a) stretch bending, (b) draw bending, and (c) compression bending. For each method: (1) start of process, and (2) during bending.