Metal stamping

Sheet metal working processes are cold working processes in which forming and related operations are performed. In sheet metal operation the surface area-to-volume ratio of the starting metal is high. This ratio distinguished the sheet metal operation from bulk deformation. Often it is called press working because the machines used to perform these operations are pressed. A part produced by sheet metal operation is often called stamping.

The tools used to perform stamping operation are a set of tools called punch (smaller members) the upper half and a die (larger member) the lower half. The sheet metal is placed in between the two die halves. When the two die halves are brought together the punch enters the die and part/s is/are produced. Punch has a shape that corresponds to that of a die but is smaller by an amount determined by the required punch die clearance. Again the punch die clearance is determined by the type and thickness of the material and the operation to be performed.

Stamping of sheet metal involves cutting or shearing, bending or forming, and drawing or deep-drawing operations. A detail of all the process has been given below.

Blanking or piercing: It involves cutting of the sheet metal along a close outline in a single step to separate the piece from the surrounding stock.

Forming: This operation produces one or more plane surfaces that are at an angle to the original or flat plane of the blank. Any change either small or big is classified as forming.

Drawing: Forming of a flat metal sheet into a hollow or concave shape, such as a cup, by stretching the metal is known as drawing.

Shaving: It is a secondary operation after blanking or piercing operation. It produces a smooth edge on the work piece instead of the breakaway edge. This is achieved by removing a small amount of stock from the edge of the part.

Trimming: It is similar to blanking but, it occurs after forming, drawing, or other operations when extra metal is left in the part for holding or locating purpose or as stock allowance. The removal of this extra stock is called trimming.
**Embossing:** It is used to create indentation in the sheet such as raised or depressed area with little or no change in material thickness. Making of nameplates and stiffening ribs are two applications.

**Coining:** It permits different designs to be imparted on either side of a blank. The blank is entirely captive in the die. Indentation results in thinning of the sheet metal and the raised sections result in thickening of the metal.

**Swaging:** This process uses an open die. The part is also squeezed into cavity but in contrast to coining, the excess material is not contained but allowed to flow at will.

**Characteristics and application of metal stamping**

If the parts are designed properly, then this can contribute significantly to design for manufacturability (DFM). Other parts can be reduced by incorporating diverse function to the selected parts. Springs and other flexible sections, snap-fit elements, folding tabs, and press-fixable designs can be incorporated and can eliminate the need for other parts, including separate screw fasteners or other fasteners. Projection welding and spot welding are easily provided. Such innovations can reduce the number of parts in an assembly and make the stamped part itself easy to assemble.

Perhaps the major characteristic of all stamped metal parts are of uniform wall thickness with few exception. The wall thickness ranges from a low of about 0.025 mm to about 20 mm, although pure bending or shearing operations are produced on even heavier stock. Most stamping, however, is performed in the range of about 1.3mm to 9.5mm stock thickness. The size of metal stampings ranges from the smallest parts used in wrist watches to large panels used in trucks or aircraft. The largest press brakes are as long as 9 m.

**Suitable materials for stamping**

Any material that can be produced in the form of sheet or strip can be press-worked except brittle nonmetallic materials like glass and similar brittle, very-high-hardness metals having hardness above R	extsubscript{c} 50.

Materials for stamping are classified in three groups: (1) Ferrous metals, (2) Nonferrous metals, (3) Nonmetallic.
**Ferrous Metals:**
Most widely used metal for general stamping applications is cold-rolled steel in sheet, strip, and coil form having a carbon content of between 0.05 and 0.20 %. The lower-carbon steels are the least expensive metal for stamping.

**Nonferrous Metals:**
Aluminum and copper alloys are the two principal nonferrous materials used for stamping. Other stampable nonferrous metals are nickel alloys, zinc, magnesium (when heated), titanium, and many less common metals.

**Nonmetallic Materials:**
Nonmetallic materials commonly blanked and pierced include fiber board, paper, leather, rubber, cork, wood and wood-based composition board, and various plastics, especially laminated thermosets. Except for glass-reinforced laminates, these materials are ideally suited for blanking with steel-rule dies and other short-run tooling. ABS plastic can be deep drawn. Some cup-shaped food containers of ABS are made by deep-drawing on metal-working presses.

**Design Recommendations**

**Stock utilization:** Stamping should be designed in such a way that materials can be used economically. Shapes that can be nested close together are better than those which must be more widely spaced on the stock material. Produce additional parts from the scrap for better utilization the material as shown in Figure M4.2.1 and Figure M4.2.2.

**Holes:**
Diameter of hole should not be less than the stock thickness shown in Figure M4.2.3 (top). The spacing between the holes should be a minimum of two times the stock thickness (T). But it is preferable to keep three times the diameter from the die strength point of view shown in Figure M4.2.3 (bottom).

The minimum distance from the edge of a hole to the adjacent edge of the blank should be at least stock thickness, but preferably it should be 1.5 to 2 times of that shown in Figure
M4.2.4. Too small spacing will cause the part to bulge in the edge area adjacent to the hole. The minimum distance between the lowest edge of the hole and the other surface should be 1.5 times stock thickness plus the radius of the bend, as illustrated in Figure M4.2.4.

Distortion of the hole will occur if this minimum distance is not observed. A nonfunctional window, either square or rectangular is pierced directly beneath the desired hole or holes, if the design requires the lowest edge of the hole to be closer than the recommended minimum. Round holes are recommended instead of square, rectangular or other holes, as the tooling cost of round holes are far below than other types.

**Figure M4.2.1:** Redesign of parts for better nesting of blanks to improved material utilization

**Figure M4.2.2:** Minor redesign allows a part to be blanked from scrap material
Figure M4.2.3: Design rules for size and spacing of holes

Figure M4.2.4: Pierced holes should not be located too close to the edge of the part.

**Sharp corners:**
Both internal and external sharp corners should be avoided as far as possible. Presence of sharp external corners in punches or dies leads to premature breakage, causing more pull-down, larger burrs, or rougher edges of the blanked part in the area of the corner. Further, sharp interior corners of punches and dies are the point of stress-concentration, and thereby leading to cracking and failure due to heat treatment or through usage. The minimum corner radius should not be less than one half of the stock thickness and never less than 0.8mm as shown in Figure M4.2.5.

Figure M4.2.5: Design rules for fillets and radii of blanked parts.
**Grain direction:**
In the design, strength requirement need to be considered with respect to the grain direction of the material. Often it is advised to indicate grain direction in the part drawing itself. For example, the grain direction of rolled sheet metal has a bearing on its strength and bending ability shown in Figure M4.2.6.

![Grain direction of rolled sheet metal](image)

**Figure M4.2.6:** Grain direction of rolled sheet metal has a bearing on its strength and bendability.

**Strip stock:**
When the part can have two parallel sides, often it is advisable to design a part so that it can be cut off from strip stock rather than being blanked by some expensive die. Figure M4.2.7 shows use of strip stock of the width of a part with a parting die utilizes material better than wider strip stock with full periphery blanking dies. Simple shear cutoff of strip stock provides the fullest utilization of raw materials. These are considered to be inexpensive and simple approach.
Figure M4.2.7: Use of strip stock of the width of the part with a parting die

Narrow sections:
Long, narrow projections need to be avoided as these projection cause die punch to be thin and fragile and also are subjected to distortion. It is recommended that long sections should not be narrower than 1.5 times the stock thickness (T) as shown in Figure M4.2.8.

Figure M4.2.8: Narrow projections and webs
Shaving allowances:
For providing a smooth edge, sometimes parts are shaved after blanking. In that case the recommended allowance for this operation is provided in Table M4.2.1.

Table M4.2.1: Per-Side Shaving Allowance for Steel. (Source: Design for Manufacturability Handbook by James G Bralla, 2nd Ed)

<table>
<thead>
<tr>
<th>Thickness (mm)</th>
<th>No. 1 temper</th>
<th>No. 5 temper</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>First shaving</td>
<td>Second shaving</td>
</tr>
<tr>
<td>1.2</td>
<td>0.08</td>
<td>0.04</td>
</tr>
<tr>
<td>1.6</td>
<td>0.10</td>
<td>0.05</td>
</tr>
<tr>
<td>2.0</td>
<td>0.13</td>
<td>0.06</td>
</tr>
<tr>
<td>2.4</td>
<td>0.15</td>
<td>0.08</td>
</tr>
<tr>
<td>2.8</td>
<td>0.18</td>
<td>0.09</td>
</tr>
<tr>
<td>3.2</td>
<td>0.23</td>
<td>0.11</td>
</tr>
</tbody>
</table>

Reinforcing ribs:
If the formed section of a part requires extra resistance to flexing then it is needed to providestiffening ribs as shown in Figure M4.2.9.

Figure M4.2.9: Stiffening ribs for a right-angle bend.

Screw threads:
After the stamping of sheet metal, usually parts are used for assembly. For such operation, screw fasteners are recommended as these are least expensive. To achieve sufficient thread length for providing proper tightening generally extruded or flanged holes are used. Design recommendations for screw threads are depicted in Figure M4.2.10 and Figure M4.2.11.
Figure M4.2.10: Design recommendations for screw threads in flat stock.

Figure M4.2.11: Design recommendations for screw threads in extruded holes.

Figure M4.2.12: Design recommendations for set-outs to replace separate rivets, pins etc.
**Set-outs**

Set-outs can be used to replace locators, rivets, cam followers, pins, etc. In order to avoid fracture in the stock material, it is recommended to limit height of set-outs to one-half the stock thickness and the punch should be slightly large in diameter than the set-out as shown in Figure M4.2.12. In a situation where the set-out is made hollow, the height can be made approximately 1.5 times the stock thickness.

**Burrs:**

It is important to observe the difference between the two sides of a blanked or sheared sheet so as to decide the face over sharp edge need to provided and which side rounded or pulled-down edge is to be made. The sharp edge should be handled properly to avoid the damage that cause to the other parts. Generally, curled, folded, or sharply bent edges should be provided so as to keep the burr side interior of the bend as shown in Figure M4.2.13.

![Figure M4.2.13](image_url): Blanking or piercing sequence.

**Formed parts:** For economical reason, the designer should recommend such shapes which can be produced by using the standard existing universal bending dies.

- The inside bend angle is preferred to be 90°.
- In channel forming, the recommended relationship between leg height and width is shown in the Figure M4.2.14. It must permit to use a single standard 90° bending tool.
- In order to prevent twisting and distortion, the width of the formed portion of the part recommended be at least 3 times stock thickness shown in the Figure M4.2.8.
Drawn parts:

Various design recommendation for drawing operations are given below:

- Since tapered wall shell and/or flanged shells are more expensive than the straight cylindrical shells, the usage of former is avoided.
- It is not required to specify both inside and outside diameters. Only one of these dimensions can be controlled because of variations in wall thickness.
- It is recommended to avoid sharp corners in the bottoms of drawn shapes. The recommended minimum radius is 4 times the stock thickness as shown in Figure M4.2.15.

In the case of rectangular boxes, corner radii at least 0.25 times the depth of draw is recommended.
**Counter sink and counter bores:**
These are considered to be expensive operations if quantities are limited and hence should be avoided unless it is really necessary as shown in Figure M4.2.16.

![Counter sink and counter bores](image)

**Figure M4.2.16:** Various countersinks and counter bores used with metal stampings.

**Recommended tolerances:**
In metal stamping processes large tolerances are provided. It depends on various factors such as:

- Design of the part
- Function of part
- Size of the part
- Press operations to be performed
- Material to be employed (kind and thickness)

Table M4.2.2 summarizes the recommendations for tolerances. Tolerances are added for the outside dimensions and subtracted for inside dimension.

**Table M4.2.2:** Recommended Dimensional Tolerances for Sheet-Metal Blanks Produced with Blanking Dies. (Source: Design for Manufacturability Handbook by James G Bralla, 2nd Ed)

<table>
<thead>
<tr>
<th>Material thickness, mm</th>
<th>Up to 75 mm</th>
<th>76 to 200 mm</th>
<th>200 to 600 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>To 1.5</td>
<td>0.08</td>
<td>.15</td>
<td>0.25</td>
</tr>
<tr>
<td>Over 1.5 to 3.0</td>
<td>0.15</td>
<td>0.25</td>
<td>0.40</td>
</tr>
<tr>
<td>Over 3.0 to 6.3</td>
<td>0.30</td>
<td>0.45</td>
<td>0.80</td>
</tr>
</tbody>
</table>