Stamping Design Guidelines

Jonathan Zhang
Stamping Design Guidelines

Metal Stamping is an economical way to form metal components with variety of characteristics including strength, durability, and wear resistance. Also they will have good conductive properties and stability. The purpose of this design guideline is to provide some basic design concepts which could optimize all the features that a metal stamping process offers.

1. Material Selection

Over-specifying a steel grade and blank thickness are major factors to drive up the cost of metal stamping. There are many choices of sheet and strip materials that will respond well to metal stamping and forming processes. However, the price and availability can vary in a wide range, so it has significant impact on the cost and delivery of production metal stampings. The following are some key factors that should take into consideration when selecting an alloy and specifying physical characteristics of the material.

**Material Properties**

There are many different ferrous and non-ferrous alloys available with different thicknesses and tolerances. The non-common alloys will only be custom-produced by the mills, and they will be available in the large quantities. It is possible to find someone who is using the material with the same specification for another application, but it would be a hit-or-miss, and it will have impact to the delivery schedules if missed.

The quality of steel products has been improved greatly in the recent years. Continuous casting yields a very consistent and homogenous alloy mix. Today’s metal materials are more ductile and much more consistent, so the savings can be found from stock warehoused alloys instead of the more specified materials.

**Steel Thickness**

Most common steel grades are offered in standard gage thicknesses and tolerances. These sizes are usually readily available as stock items and are generally the best choice when cost and delivery are a major factor.

Rolling mills work from master coils, and usually they would have a minimum order quantity somewhere in the truckload range. If the amount of material for your application is much less than the quantity that the steel mill required, steel warehouses would be an alternative way to search. The inventory of a steel warehouse may happen to have the material required to fall within the specified tolerance, but it would make the availability a variable from order to order, similar to the case of non-standard steel grade.
Custom made material is also available and can be purchased from the companies who are specializing in re-rolling smaller quantities, but the price of these special rolled materials can be significantly higher than steel mill or steel warehouse price.

**Gage Conversion Chart**

<table>
<thead>
<tr>
<th>Gage</th>
<th>Steel</th>
<th>Stainless Steel</th>
<th>Aluminum</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>0.179</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>8</td>
<td>0.164</td>
<td>0.172</td>
<td>-</td>
</tr>
<tr>
<td>9</td>
<td>0.150</td>
<td>0.156</td>
<td>-</td>
</tr>
<tr>
<td>10</td>
<td>0.135</td>
<td>0.141</td>
<td>-</td>
</tr>
<tr>
<td>11</td>
<td>0.120</td>
<td>0.125</td>
<td>-</td>
</tr>
<tr>
<td>12</td>
<td>0.105</td>
<td>0.109</td>
<td>-</td>
</tr>
<tr>
<td>13</td>
<td>0.090</td>
<td>0.094</td>
<td>0.072</td>
</tr>
<tr>
<td>14</td>
<td>0.075</td>
<td>0.078</td>
<td>0.064</td>
</tr>
<tr>
<td>15</td>
<td>0.067</td>
<td>0.070</td>
<td>0.057</td>
</tr>
<tr>
<td>16</td>
<td>0.060</td>
<td>0.063</td>
<td>0.051</td>
</tr>
<tr>
<td>17</td>
<td>0.054</td>
<td>0.056</td>
<td>0.045</td>
</tr>
<tr>
<td>18</td>
<td>0.048</td>
<td>0.050</td>
<td>0.040</td>
</tr>
<tr>
<td>19</td>
<td>0.042</td>
<td>0.044</td>
<td>0.036</td>
</tr>
<tr>
<td>20</td>
<td>0.036</td>
<td>0.038</td>
<td>0.032</td>
</tr>
<tr>
<td>21</td>
<td>0.033</td>
<td>0.034</td>
<td>0.028</td>
</tr>
<tr>
<td>22</td>
<td>0.030</td>
<td>0.031</td>
<td>0.025</td>
</tr>
<tr>
<td>23</td>
<td>0.027</td>
<td>0.028</td>
<td>0.023</td>
</tr>
<tr>
<td>24</td>
<td>0.024</td>
<td>0.025</td>
<td>0.020</td>
</tr>
<tr>
<td>25</td>
<td>0.021</td>
<td>0.022</td>
<td>0.018</td>
</tr>
<tr>
<td>26</td>
<td>0.018</td>
<td>0.019</td>
<td>0.017</td>
</tr>
<tr>
<td>27</td>
<td>0.016</td>
<td>0.017</td>
<td>0.014</td>
</tr>
<tr>
<td>28</td>
<td>0.015</td>
<td>0.016</td>
<td>-</td>
</tr>
<tr>
<td>29</td>
<td>0.014</td>
<td>0.014</td>
<td>-</td>
</tr>
<tr>
<td>30</td>
<td>0.012</td>
<td>0.013</td>
<td>-</td>
</tr>
<tr>
<td>31</td>
<td>-</td>
<td>0.011</td>
<td>-</td>
</tr>
</tbody>
</table>

**Flatness of the Coil Material**

Coiled strip as the raw material form is not flat. As material is unwound off the coil, it retains some of the curvature along its length, called *coil set*. In addition, the width of the strip usually has a slight arc to it. This is called *crossbow*. Coil set can be minimized or removed by material handling equipment at the beginning of the metal stamping process. In other hand, crossbow is much tougher to remove. And generally it will survive to affect the flatness of the finished stamping.
2. Stamping Processes - Blanking, Trimming, and Punching

*The Anatomy of a Die Cut*

A normal metal stamping process (creates a die cut) is to drive a sharpened tool steel punch through the sheet or strip material into a die cavity, where the slug or scrap is ejected. Cutting clearances between the punch and die are closely defined and specified in the die design stage, based on the requirement of the part. This stamping process produces a very predictable edge condition on the finished part.

During this process, several mini steps will happen:

- The punch starts out by trying to compress the material, producing a rolled or radius top edge;
- As the sharp punch begins to cut through the sheet metal, it shears the material, producing a straight, burnished wall, that usually is between \( \frac{1}{4} \) to \( \frac{1}{3} \) through the thickness;
- As the punch going downward, forces build up beyond the yield strength of the material, scrap breaking away in a line between the punch and die edges, and leaving burrs around the bottom edge.

*Burrs*

Burrs, like plastic parting lines or flash on castings, are the normal by-products of the metal stamping process. Blanking burrs are usually somewhat ragged, sharp, and uneven. They can vary in height as punch and die edges become dull, but generally, *a burr height up to 10% of material thickness can be expected*. Burrs can be dulled or removed by mass finishing processes or secondary operations, depending on the application.

*Define Dimension and Tolerance*

A normal piercing and blanking operation is extremely repeatable and a very close tolerance can be expected. *Size tolerance of .002” (0.05mm) can be held in most applications.*

*Punch and die clearances are normally around 8% to 10% of material thickness per side*, the bottom portion of the hole or trim will be tapered at the amount of die clearance. Therefore, when defining dimensions of the feature:

- The inside dimensions are normally measured at the shear, or smallest portion of the cut lines, disregarding the breakaway taper.
- The outside dimensions will be measured at the shear or largest portion of the cut lines, with the breakaway tapering smaller.

If the tapering breakaway cannot be tolerated in a particular application, a hole or an outside edge feature should be re-trimmed or “shaved” to produce a straight
edge. This must be specified, and will require an additional step or a secondary operation.

**Location Tolerance**

In most cases, the holes that are pierced in a flat blank part can be done in the same operation, and the measurement from a hole to a hole is repeatable within a close tolerance, usually at $\pm 0.002$” ($\pm 0.05$mm). The only exception is when holes are closely spaced ($<1-1/2 \times$ material thickness), and they must be pierced in separate stations or operations.

Gauging or feed accuracy will require more liberal tolerances. In the case of holes that are pierced on different planes, as in a part with an offset form, the added variables of bend and material spring-back must be considered and allowed for.

**Tooling Requirement**

The same compressive forces exerted on the material are shared by the tooling. A 1/2” (12.7mm) diameter punch perforating .062” (1.575mm) thick mild steel will require 2-1/2 tons of pressure behind it to push through. At 80 parts per minute line rate, this will place extreme impact forces on the body of the punch. The punch tooling can fail catastrophically if there is not enough cross-section area to support this force. To alleviate this extreme condition, it is best to design the perforations with a cross-section or diameter equal to material thickness at a minimum.
3. Bending and Forming

Most metal forming is a linear process. The work of perforating, forming and blanking is done by the up and down movement of the press equipment, and amazingly complex shapes can be generated using this process. Thus, a well designed stamping part should take the process and material into consideration.

As a general rule, the lower the alloy or temper is, the more formable the material will be. Tempers are rated in terms of how tightly they will bend without cracking and whether with the grain or across the grain. In addition, the harder a material is, the more it will “spring back” after forming. From a design standpoint, it means the extra work or over-bend must be induced in order to achieve the specified angle.

In general, anything up to 90 degrees can be done in one operation. Beyond that, a little more creativity may be needed. Forming in this manner relies on a “leg” of material to be pushed or wiped up into position while the base material is held flat. For that reason, the length of a formed leg should be at least 2-1/2 times the material thickness beyond a bend radius.

Distortion

The metal is displaced through the bend radius during the forming process. The material on the inside of the bend is compressed, while the material on the outside of the bend is stretched. On thicker materials and bends with relatively small inside bend radii (2 X material or less), there could be an overall thinning of the material through the bend.

In addition, because the metal is compressed on the inside of the bend, the excessive material is forced out towards either end of the bend radius, creating what is called bend bulges. If they are not acceptable, the blank must be contoured to compensate. A note such as “Bulging is not allowed in this area” should be added to the part drawing.

For the same reason, when two adjacent sides are folded up, as in forming a box, some relief is needed at the base of the bend to avoid “pinched” corners. Usually,
this would be in the form of a round hole placed at the convergence point of the sides. When a leg is formed up alongside a flat section of the part, consideration should be taken to the transition from form to flat. **The flat section should be trimmed back to the base of the bend radius. If the edge of a flat section must be flush with a formed leg, bend reliefs should be cut into the blank on either side of the leg.**

**Other Consideration**

As described above, the metal stamping process places compressive forces on the raw material. As the top edge is rolled into the cut, the bottom edge tends to turn slightly also. This distortion at the edges affects the flatness of the finished part. With thinner or milder materials, it will be minimal, but it will become severe with heavier stock or tougher materials, such as stainless steels and high-strength alloys. When flatness is critical, tooling can be designed to minimize distortion. In most cases, it will require extra stations or secondary operations.

For the same reason, perforated or trimmed features that are located too close to the material edge or too close to each other tend to roll the material in between the features, creating a distorted or thinned edge. **The rule of thumb in stamping design is to leave a minimum of 1-1/2 times material thickness between trimmed or perforated features.**

Also, the stretching and compression of a forming process can distort holes adjacent to a form or a bend. **A hole should be designed at least 2 times of material thickness away from the radius of a formed feature.** If this is not possible, the hole should be designed with sufficient clearance to allow for distortion.

**Dimensioning Forms**

Formed features are subject to a number of variables, including the material thickness, temper tolerances, angular tolerances on bends, and station-to-station inaccuracies in the process. Dimensions should always be given to the inside of a formed feature. An angular tolerance of $\pm 1$ degree or so should be allowed on any angle of bend. For this reason, a design feature that is placed at the outer end of a form should take the angular tolerance of the bend and the distance from the bend into consideration. Where a feature has multiple bends, tolerance stack-up should be analyzed and allowed for. Where the tolerances need to be tightly held, an additional qualifying operation may be required to meet this specification.
4. Deep Draw

The Process

Deep draw refers to the process of pulling a flat “blank” of material over a radius die edge and into a cavity, creating a closed bottom, round or irregularly shaped cup or cylinder. It should not be confused with stretch-forming process. The blank is actually forced into a plastic state as it is dragged over the die radius and down into the die. This process is done under a very controlled condition correlated with various calculations on blank holding pressure, punch and die radii, punch speed and lubrication.

Anatomy of a Deep Draw

Cupping and drawing are two stages in deep draw process. When the punch first contacts the blank, the nose of the punch initially embosses the material into the die. Some stretching occurs at this point and creates what is known as a “shock line”. This is a pronounced area of thinning around the radius at the bottom and goes up into the straight wall of the shell. Depending on the shape of the bottom, the material may still be near original thickness across the bottom face (flat bottom) or thinned out by a stretching action (spherical bottom).

As the blank is pulled into the die, the material at the circumference gathers and the wall progressively thickens. As the blank is pulled in to near shell diameter, the material thickens to as much as 10% over the original thickness. Clearance must be provided for this thickening to occur so that the material will not get bound up between punch and die.

In addition, the punch must be tapered so that the finished shell can be stripped off. Therefore, a drawn shell will taper from bottom to top. It is possible to minimize this through subsequent sizing operations, but not eliminate it entirely.

The blank should be cut from rolled strip material with a grain structure elongated across the blank in the direction of rolling. Since this cross-grain does not pull into a drawn shape evenly from all directions, great stresses are induced in the shell wall. Due to these uneven stresses, the drawn shell will not be perfectly round. A flange added to the top of the shell by design will minimize the unevenness, but the smaller the flange is, the less strength it has to keep the shell rounded.

Specifying a Drawn Shell

Thickness - Since the original blank is altered by the deep draw process, the wall thickness cannot be measured evenly and be specified in terms of mill tolerances. Depending on application, the gage of the blank material, the minimum wall thickness of the completed part, or the maximum wall thickness of the completed
part will be the three possible ways to define the thickness requirement. Wall thickness can be specified in more detail (in sections), but only after development work has been done with the draw process.

Dimension - Since the material is formed around the punch, the part (shell) is typically dimensioned to the inside wall (diameter), with taper allowed from bottom to top. Alternatively, the part can be dimensioned to the outside wall (diameter) with the maximum size found at the top, and tapering down to the bottom.

If a straight wall shell with no flange is required, the shell will be “pinch-trimmed” flush with the outside wall (diameter). Since the shell has a radius at the top, the remaining trimmed edge will have a partial radius from the inside, abruptly ending in a somewhat sharp outer edge.

Also, from clearance standpoint, since the die must have enough clearance to accept the shell, there will be a slight flare at the top of the shell. The bottom of a shell can be pierced out in a similar way to create a tubular part, but the same pinch-trim principles apply to the inside wall (diameter). If a straight, cut-off edge is desired, it would require a secondary machining or cut-off operation and should be specified on the part drawing.
5. Cosmetics

*Tool Marks*

The stamping tools will leave their marks on the finished piece when bending and shaping the metal in place, especially in thicker materials.

- A punch tool wiping by the material to form the shape will cause tool marks on the outside of the bend.
- Deep drawn parts will have shock lines near the bottom of the cup.
- Coining, swaging and embossing will leave the impression marks in the material surrounding the form.
- The face of the tooling which is used to form the part, and holes drilled for fasteners can leave marks on the part as well.

These tooling marks are a normal part of the metal forming process. However, when cosmetics are important, these marks can be minimized by the use of creative tooling techniques and careful die design.

*Handling*

Most of the metal stamping parts are automatically ejected from press equipment, moved through the manufacturing process in the large containers, mass finished and shipped in bulk form. They are subject to the dings and scratches due to this type of process.

It is helpful to understand the application and cosmetic requirements for the particular part. The cosmetic specifications should be described on the part drawing.