Stamping Design Guidelines

Jonathan Zhang

Jonathan Zhang

Stamping Design Guideline

Stamping includes a variety of sheet-metal forming manufacturing processes using a machine press or stamping press, the processes including punching, blanking, embossing, bending, forming, drawing, flanging, and coining. This could be a single stage operation where every stroke of the press create the desired form on the sheet metal part, or could occur through a series of stages.

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. The following are some key factors that should take into consideration when selecting the material and specifying physical characteristics of the material.

Material Properties

There are many different ferrous and non-ferrous alloys available with different stock thicknesses and tolerances. The non-common alloys will be custom-produced by the steel mills, and they will only 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

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

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

The operations associated with stamping are blanking, piercing, forming, and drawing. These operations are done with dedicated tooling also known as *hard tooling*. This type of tooling is used in making high volume part of one design. By contrast, *soft tooling* is used in processes such as CNC turret presses, laser profilers and press brakes. All these operations can be done either at a single die station or multiple die stations — performing a progression of operations, known as a *progressive die*.

The Stamping Equipments

The stamping equipments can be categorized into two types:

- **Mechanical Presses** - Mechanical presses has a mechanical flywheel to store the energy, transfer it to the punch and to the work piece. They range in size from 20 tons up to 6000 tons. Strokes range from 5 to 500 mm (0.2 to 20 in) and speeds from 20 to 1500 strokes per minute. Mechanical presses are well suited for high-speed blanking, shallow drawing and for making precision parts.
- **Hydraulic Presses** - Hydraulic Presses utilize hydraulics system to deliver a controlled force. Tonnage can vary from 20 tons to 10,000 tons. Strokes can vary from 10 mm to 800 mm (0.4 to 32 in). *Hydraulic presses can deliver the full power at any point in the stroke, variable tonnage with overload protection, and adjustable stroke and speed.* Hydraulic presses are suitable for deep-drawing, compound die action as in blanking with forming or coining, low speed high tonnage blanking, and force type of forming rather than displacement type of forming.

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 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 *1/4 to 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.

Shearing

Shearing is a process for cutting sheet metal to size out of a larger stock such as roll stock. Shears are used as the preliminary step in preparing stock for the stamping processes, or smaller blanks for CNC presses.

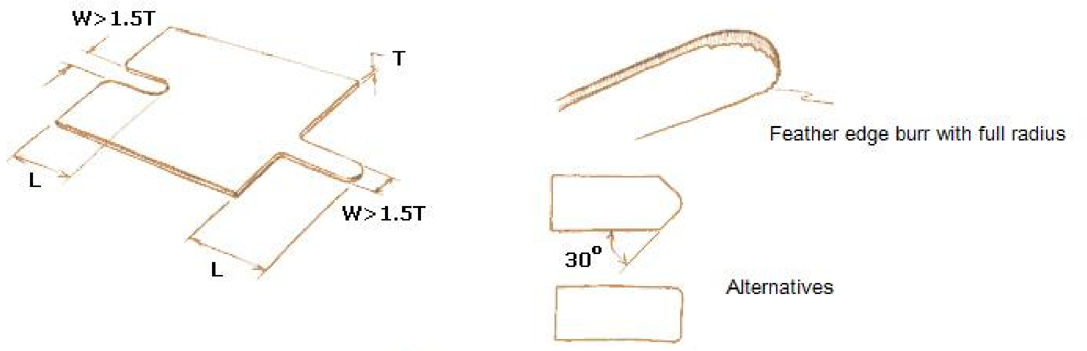
- Shearing material thickness range varies from 0.125 mm to 6.35 mm (0.005 to 0.250 in). The dimensional tolerance ranges from ±0.125 mm to ±1.5 mm (±0.005 to ±0.060 in).

Blanking

Blanking is cutting up a large sheet of stock into smaller pieces suitable for the next operation, such as drawing and forming. Often this is combined with piercing. Blanking can be as simple as a cookie cutter type die to produce prototype parts, or high speed dies that run at 1000+ strokes per minute, running coil stock which has been slit to a specified width.

For production parts, the final configuration of the drawn or formed shape needs to be established before the blank die can be built-since the blank size and the slit width size needs to be established precisely.

- Corners should have a minimum radius of 0.5 x material thickness or 0.4 mm (0.016in) whichever is greater. Sharper corners can be produced but at a greater die maintenance costs and more burrs.
- Slots or tabs widths should be greater than 1.5 X stock thickness. The length can be a maximum of 5 times slot/tab width.
- These rules can be violated at an increased tooling cost-- width as low as 1 X thickness and length as high as 7 X thickness can be achieved.
- On cutoffs, avoid full radiuses across the width of stock. A square cut-off is best. If a radius is necessary, then an angle-blended radius is best.



Notch and tab design

Burrs

Burrs, like plastic parting lines or flash on castings, are the normal by-products of the metal stamping process. Burrs are often not acceptable, usually for safety reasons, either for handling or for product safety (as burr cutting into insulation, or mechanical chafing). Another reason could be to improve surface appearance-discoloration from welding/brazing, oxidation, scale from heat treatment etc. The burr is a function of the clearance between the punch and the die, as well as the sharpness of the punch and the die. Shearing or 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 and hold down marks which are predictable, should be considered in the design of the end product. Burrs should be kept away from handling areas, preferably folded away, or in some obscure area. The same can be done with hold down marks too. Burrs can be removed by mass finishing processes or secondary operations, depending on the application.

De-burring

A normal burr from well-maintained tools is usually less than 10% of material thickness. If burrs are not acceptable, then de-burring process needs to be done. Typically de-burring will result in a rounded edge with a radius of 0.05 to 0.075 mm (0.002 to 0.003 in).

De-burring is done by tumbling parts in a barrel or a vibratory bowl, along with finishing media. Ceramic media is often used for steels. For softer materials, plastic media, walnut shells etc can be used. This type of de-burring is usually confined to unfinished materials. For materials that are already finished, such as plated or painted materials, bulk de-burring operations are not suitable, because the de-burring will remove the finish along with the burrs, thus, other forms of de-burring such as belt sanding or hand filing will have to be done with the associated higher costs.

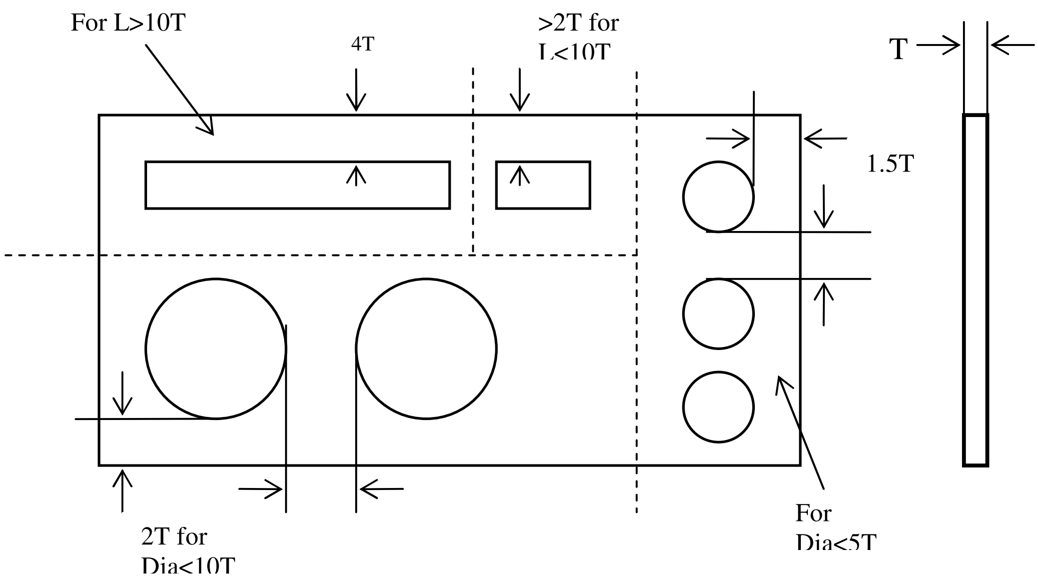
- De-burring can be avoided by considering the direction of the burrs in the design of the parts. If the burrs will be in a non-accessible area or will be folded later, then de-burring can be avoided.

Piercing

Piercing is the operation of cutting internal features (holes or slots) in stock. Piercing can also be combined with other operations such as lance and form (to make a small feature such as tab), extrusion (to make an extruded hole). All these operations can be combined with blanking. The holes that are pierced in a flat blank part should be done in the same operation to ensure hole-to-hole tolerance and part repeatability, and the measurement is usually at *+/-0.05mm (+/-0.002”)*.

However, when there are large numbers of holes, in a tight pitch - holes are closely spaced, there could be distortions, due to the high amount of tension (material stretching) on the upper surface, and compression on the bottom surface. This causes the material not to lay flat. It can be done by staggering holes in different operations.

- Minimum hole diameter should be at least 20% greater than stock thickness. In the case of stainless steels, it should be 2 times the material thickness.
- Minimum wall thickness (distance from the hole to the trim edge or hole to hole) should be at least 2 times stock material thickness.
- For non-round slots,
  - The minimum wall thickness should be 2 times thickness for short slot that is < 10 thicknesses long;
  - The minimum wall thickness should be 4 times thickness for long slot that is > 10 thicknesses long.
- Minimum hole (and short slot) to bend distance should be 2.5 × the stock thickness + bend radius.
- For long slots, the distance should be 4 × the stock thickness + bend radius.



In the case of holes that are pierced on different planes, as in a part with an offset form, the additional variables of bend and material spring-back must be considered.

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 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.

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 Considerations

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 the material thickness at a minimum.*

- The optimum clearance (total = per side × 2) should be from 20 to 25% of the stock material thickness. This can be increased to 30% in order to increase die life.
- Punch life can also be extended by sharpening whenever the punch edge becomes 0.125 mm (0.005 in) radius. Frequent sharpening extends the life of the tool, as well as cuts down on the punch force required.
- Sharpening process is done by removing 0.025 to 0.05 mm (0.001 to 0.002 in) of the material in one pass with a surface grinder, and repeated until the tool is sharp. If it is done frequently enough, only 0.125 to 0.25 mm (0.005 to 0.010 in) of the punch material is removed.
- Grinding should be done with the proper wheel for the tool steel. Consult with the abrasive manufacturer for proper choice of abrasive material, feeds and speeds, and coolant.
- After sharpening the edge is to be lightly stoned to remove grinding burrs and end up with a 0.025 to 0.05 mm (0.001 to 0.002 in) radius. This will reduce the chance of chipping.

Punching force calculation

Punching process can be done with shear or without shear.

Punching without shear - the entire punch surface strikes the material squarely, and the complete shear is done along the entire cutting edge of the punch at the same time.

- Punching Force = Punch Perimeter × Stock thickness × Material Shear Strength

  If Punch Diameter = 25 mm (1 in); Circumference = 78.54 mm (3.092 in); Thickness = 1.5 mm, (0.060 in); Material Shear Strength (Steel) = 0.345 kN/mm2 (25 tons/in2)

Punching Force = 78.54 × 1.5 × 0.345 (3.09 × 0.060 × 25) = *40.65 kN (4.64 tons)*
= *4.14 Metric Tons (4.64 US Tons)*

Punching with shear - This is the case where the punch surface penetrates the material in the middle, or at the corners, first, and as the punch descends the rest of the cutting edges contact the material and shear the material. The distance between the first contact point of the punch with the material, to when the whole punch starts cutting, is called the Shear Depth. Since the material is cut gradually (not at the same time initially), the tonnage requirement is reduced considerably.

The Punching Force calculated above is multiplied by a shear factor, which *ranges in value form 0.5 to 0.9 depending on the material, thickness, and shear depth.*

- For shear depths of 1.5 mm (0.060 in) the shear factor ranges from 0.5 (for 1.2 mm / 0.047 in stock) to 0.9 (for 6.25 mm / 0.25 in stock).
- For shear depth of 3 mm (0.120 in) the shear factor is 0.5.
- Punching Force = Punch Perimeter × Stock thickness × Shear Strength × Shear Factor.

  Since shear factor is about 0.5, the Punching Force is reduced by about 50%.For the same example above,
  Punching Force = 78.54 × 1.5 × 0.345 (3.09 × 0.060 × 25) × 0.5 (Shear Factor)
  = 40.65 kN (4.64 tons) × 0.5 = *2.07 Metric Tons (2.32 US Tons)*

3. Bending and Forming

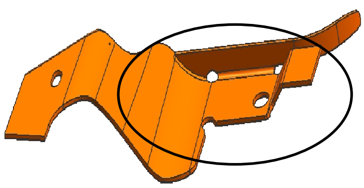
Most metal forming is a linear process. The work of perforating, forming and blanking is done by the up and down movement of the press equipment, and amazingly complex shapes can be generated using this process. Thus, a well designed stamping part should take the process and material into consideration.

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.

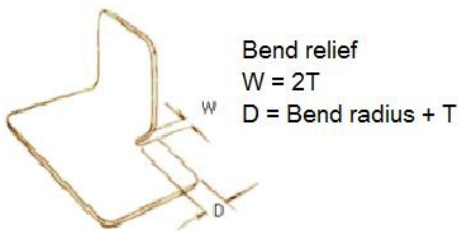
Bending

In general, anything up to 90 degrees can be done in one operation. Beyond that, a little more creativity may be needed. Forming a bend in this manner relies on a "leg" of material to be pushed or wiped up into position while the base material is held flat.

- On bends, the short leg (inside length) should be a minimum of 2.5 X stock thickness + radius.
- Bending using tight radiuses or in hard materials often results in burrs and fractures on the outside of the bends. These can be eliminated by using larger bend radiuses and by providing relief notches at the edges on the bend line.
- Bend relief notches should be provided = 2 X stock thicknesses in width (minimum 1.5mm / 0.060 in) and radius + stock thickness in length.
- Generally, bending perpendicular to rolling direction is easier than rolling parallel to the rolling direction. Bending parallel to the rolling direction can often lead to fracture in hard materials.
- Thus bending parallel to rolling direction is not recommended for cold rolled steel > Rb 70.
- And no bending is acceptable for cold rolled steel > Rb 85.
- Hot rolled steel can be bent parallel to the rolling direction.



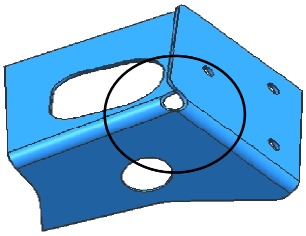
Relief notches



Forming 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. <u>*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.*</u>

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. <u>*The rule of thumb in stamping design is to leave a minimum of 2.5 times material thickness between trimmed or perforated features*</u>.

Also, the stretching and compression of a forming process can distort holes adjacent to a form or a bend. <u>*A hole should be designed at least 2.5 times of material thickness plus a bend radius away from the radius of a formed feature*</u>. 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 <u>*+/-1 degree*</u> 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 radius, punch speed and lubrication.

Drawing can be either shallow or deep depending on the amount of deformation. Shallow drawing is used to describe the process where the depth of draw is less than the smallest dimension of the opening; otherwise, it is considered deep drawing. Drawing leads to wrinkling and puckering at the edge where the sheet metal is clamped, which is removed by a trimming operation.

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.

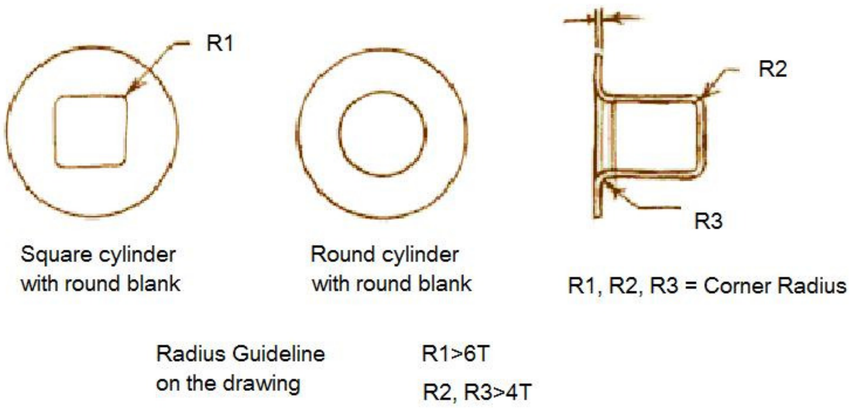
The blank should be cut from rolled strip material with a grain structure elongated across the blank in the direction of rolling. Since this cross-grain does not pull into a drawn shape evenly from all directions, great stresses are induced in the shell wall. Due to these uneven stresses, the drawn shell 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.



- Round shapes (cylinders) are easiest to draw. Square shapes can also be drawn if the inside and outside radiuses are at least 6 X stock thickness. Other shapes can be produced at the cost of complexity of tooling and part costs.
- The corner radiuses can be reduced further by successive drawing operation(s) with sufficient height for the draw.
- Perpendicularity can be held to ±1º, flatness can be held to 0.3%. This can be improved by performing extra operations.

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.

Graining

Cosmetics requirements of finished parts sometimes require graining. Graining is used to hide surface defects by creating uniform scratches using an abrasive belt sander for example. This results in an even surface appearance. Like all finishing operations, this is to be avoided since it adds extra costs to the product. The grinding grit can range from #100 for removal of gross defects, to about #180 for materials that need silk screening. The abrasives used are dependant on the material. Aluminum oxide is used for steels and silicon carbide is used for softer materials such as aluminum.

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.