LARSON 1920

TOOL & STAMPING

Metal Stamping Design Guidelines

by Neil Fonger

90 Olive Street, Attleboro, MA 02703-3802 • Tel: 508-222-0897 Fax: 508-226-7407
www.larsontool.com

ISO 9001:2015 CERTIFIED

PMA PRECISION METALFORMING ASSOCIATION

Metal Stamping Design Guidelines

Metal Stamping provides an economical way to produce quantities of parts that can possess many qualities, including strength, durability, wear resistance, good conductive properties, and stability. In this paper, we are sharing some ideas that can help you design a part that optimizes all the features that the metal stamping process offers.

Material Selection

There are many sheet and strip materials to choose from that respond well to metal stamping and forming. However, price and availability can vary greatly and affect the cost and delivery of production metal stampings. There are factors that should be considered when selecting an alloy and specifying physical characteristics of that material.

Tolerancing

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 have minimum order quantities, somewhere in the truckload range. If the material required to produce a metal stamping order is much less than this quantity, a steel warehouse can search its inventory to find material that might happen to fall within the specified tolerance, but this makes availability a variable from order to order.

Custom material can be purchased from companies that specialize in re-rolling smaller quantities, but the cost can increase exponentially.

Chemistry

Over-specifying an alloy is one of the biggest factors in driving up the cost of a metal stamping. There are many different alloys produced—in ferrous and non-ferrous materials alike—but even more so than thickness tolerances, the less common alloys will be custom-produced by the mill and will be available in large quantities only. It is possible to find someone who is supplying the same product for another customer. Like with non-standard thicknesses, this would be hit or miss, and dependent on the larger customers' requirements and delivery schedules.

The quality of steel products has increased greatly in the last 20 years or so. Continuous casting yields a very consistent and homogenous alloy mix. From our experience, today's metals are more ductile and of a much more consistent quality. Taking advantage of the quality found in stock warehoused alloys can save on costs.

Blanking, Trimming, and Perforating

The Anatomy of a Hole

Normal metal stamping processes involve driving a sharpened tool steel punch through the sheet or strip material and into a die cavity where the slug or scrap is ejected. Cutting clearances between the punch and die are closely defined and specified. And the process produces a very predictable edge condition on the finished part. Basically, the punch starts out by trying to compress the material, producing a rolled or radiused top edge. As the sharp punch begins to cut through, it shears the material, producing a straight, burnished wall, usually between 1/4 to 1/3 of the thickness of the material. As forces build up beyond the strength of the material, it yields, breaking away in a line between the punch and die edges, and leaving a burr around the bottom edge.

Burrs

Burrs, like parting lines in plastics or flash on castings, are normal by-products of the metal stamping process. Blanking burrs are usually somewhat ragged, uneven, and sharp. They can vary in height as punch and die edges become dull, but generally, 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.

Hole Dimensioning and Tolerancing

Since 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 the amount of die clearance. Therefore, inside dimensions are normally measured at the shear, or smallest portion, disregarding the breakaway. Likewise, an outside dimension will be measured at the shear or largest portion, with the breakaway tapering smaller. If this breakaway cannot be tolerated in a particular application, a hole or outside edge feature can be re-trimmed or “shaved” to produce a straight edge. This must be specified, and will require an additional step or secondary operation. Normal piercing and blanking operations are extremely repeatable and very close tolerances can be expected. Size tolerances of .002” can be held in most applications.

Location

In most cases, holes pierced in a flat blank part can be done in the same operation, and location from hole to hole is repeatable within a close tolerance, usually +/-.002”. The only exception is when holes are closely spaced (<1-1/2 X material thickness) and must be pierced in separate stations or operations. Gauging or feed accuracy will require more liberal tolerances. In the case of holes pierced on different planes, as in a part with an offset form, the added variables of bend tolerances and material springback must be considered and allowed for.

Tooling Considerations

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

Bending and Forming

Most metal forming is a linear process. That is, the work of perforating, forming, and blanking is done by the up and down movement of press equipment. Amazingly complex shapes can be generated using this process, but a good metal stamping design will take the process and material into consideration.

As a general rule, the lower the alloy and temper, the more formable the material. Tempers are rated in terms of how tightly they will bend without cracking, and whether with the grain or across. In addition, the harder a material is, the more it will “spring back” when formed—and from a metal former’s standpoint, how much extra work or over-bend must be induced to achieve the specified angle. Generally, 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

As metal is formed, it is displaced through the bend radius. 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 times material or less), there could be some overall thinning of material through the bend. In addition, because material is compressed on the inside of the bend, the excess gets forced out 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 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.

Dimensioning Forms

Formed features are subject to a number of variables, including material thickness and temper tolerances, angular tolerances on bends, and station-to-station inaccuracies in the process. Dimensions should always be given to the inside of the formed feature. Angular tolerances of +/- 1 degree or so should be allowed on bends of any angle. For this reason, tolerancing of features 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 tolerances need to be tightly held, an additional qualifying operation may be required to meet this specification.

Deep Draw

The Process

Deep draw refers to the process of pulling a flat "blank" of material over a radiused die edge and into a cavity, producing a closed bottom, round, or irregularly shaped cup or cylinder. It should not be confused with stretch-forming. 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 calculated and very controlled conditions involving blankholding pressures, punch and die radii, punch speed, and lubrication.

Anatomy of a Deep Draw

The two stages of a draw are cupping and drawing. 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 produces what is known as a "shock line." This is a pronounced area of thinning around the radius at the bottom and just 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 used to produce a shell is 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, a drawn shell will not be perfectly round. A flange added to the top of the shell will minimize this, but the smaller the flange, the less strength it has to keep the shell round.

Specifying a Drawn Shell

Since the original blank is so altered by the deep draw process, the wall thickness cannot be specified in terms of mill tolerances. Depending on application, the three ways of specifying the thickness of material in a shell would be to call out the thickness of material to be used, the minimum wall thickness, or the maximum wall thickness. Wall thickness can be specified in more detail, but only after development work has been done with the draw process. Since the material is formed around the punch, shells are typically dimensioned to the inside diameter, with taper allowed from bottom to top. Alternatively, the shell can be dimensioned to the outside diameter with the maximum size found at the top, and tapering down to the bottom.

If a straight shell with no flange is required, the shell will be “pinch-trimmed” (trimmed flush with the outside 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, 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 manner to produce a tubular part, but the same pinch-trim principles apply to the inside 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.

Flatness

Raw material: coiled strip material by nature is not flat. As material is unwound off the coil, it retains some of that curve shape 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. But crossbow is much tougher to remove and generally survives to affect the flatness of the finished stamping.

Stamping Process

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

For the same reason, perforated or trimmed features that are placed too close to each other or the material edge tend to roll the material between, producing 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 forming can distort holes adjacent to a form or bend. Holes are best kept at least 2 times material thickness beyond the radius of a formed feature. If this is not possible, the hole should be designed with sufficient clearance to allow for distortion.

Cosmetics

Tool Marks

The forces required to bend and shape metal leave their marks on the finished piece, especially in thicker materials. A punch wiping by the material to form it 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 impressions in the material surrounding the form. Where the faces of the tooling are used to form the part, holes drilled for fasteners can leave marks on the part. 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 metal stampings are automatically ejected from press equipment, moved through the manufacturing process in the largest containers possible, mass finished, and shipped in bulk form. They are subject to the dings and scratches common to this type of process.

Submitting Designs

Given all the considerations discussed here, when submitting designs, it is most helpful to know what the application is and what the cosmetic requirements are. Whenever possible, cosmetic specifications should be described on the part drawing.

Larson Tool works collaboratively with companies of all sizes, providing guidance and cost-effective, high-quality results with everything from precision metal stamping and forming to deep draw stamping. The information provided here helps us create your parts—just the way you need them to look and perform.

At Larson Tool & Stamping Company, we believe in providing a positive experience for our customers—and we deliver—in customer service, engineering expertise, and metal stamping manufacturing. **Contact us now** and get the conversation started for your next metal stamping project.

Interested in Learning More?

Read about a deep draw stamping project involving a customer that encountered a possible problem after the first production run. There were concerns about the surface roughness of the inner cylinder failing. We were able to find a solution and satisfy the production needs of this customer, and continue to do so by delivering this part to specification—every time. **Read this case study now.**

LARSON TOOL & STAMPING 1920

About Larson Tool & Stamping Company

For over 100 years, **Larson Tool & Stamping Company** has been a valued supplier of precision metal stampings and assemblies to hundreds of U.S. companies. We offer a wide range of capabilities—including forming, stamping, deep drawing, assembly, brazing, painting, coining, and more. Larson delivers high-quality, cost-effective solutions.

Larson Tool works with our customers from the earliest stages of design to optimize your part design for the metal stamping process and determine the best materials, tooling, and process solutions for your product. We'll leverage our years of experience as a valued supplier of metal stampings and assemblies to help you with all your metal stamping needs.