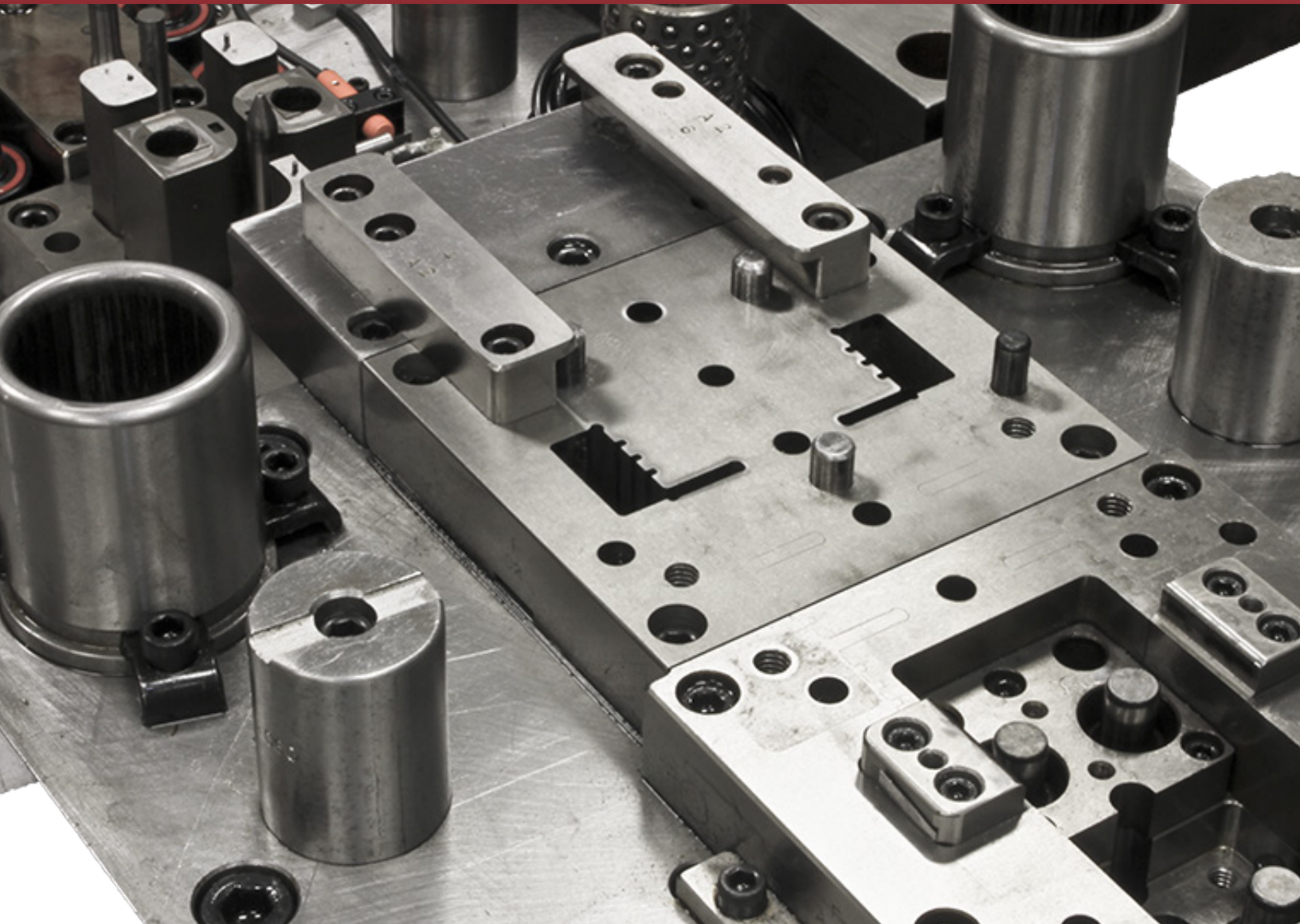


METAL STAMPING DESIGN GUIDE

Basic design concepts that improve quality and cut costs





Introduction

Metal stamping is a complex process that can include a number of metal forming processes – blanking, punching, bending, and piercing, to name a few.

All of these processes come with fundamentals that, when followed, allow manufacturers to produce the highest quality components for the lowest costs. It's also critical to understand these fundamentals so you can:

- Avoid costly errors
- Know what to expect in terms of costs and lead times
- Receive the highest quality components so your application performs at its best

The following guide illustrates best practices and formulas commonly employed in the metal stamping design process and includes tips to incorporate cost cutting considerations into your part.

Blanking

Blanking – cutting the rough, basic shape – is often the first step for a stamped metal part. The primary concerns at the blanking stage of the design are minimizing and avoiding burrs, which require secondary processing to remove and can drive up the cost of your part and extend lead time.

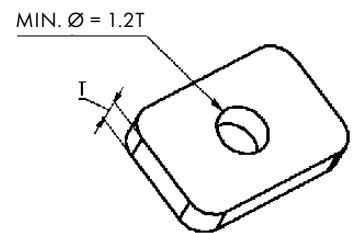
Piercing

Piercing is a common metal stamping process and ideal for forming holes and slots fully enclosed within the edges of a part. There are a number of factors to consider when designing stamped metal parts with holes to ensure proper manufacturing and functionality.

Hole Diameter

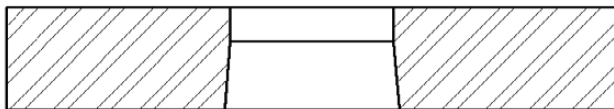
The minimum diameter of any hole should be 1.2x the material thickness or larger. Holes in materials with high tensile strength, such as stainless steel alloys, should be at least 2x wider than the material is thick.

Holes with smaller diameters require specialized punching or drilling processes, which themselves often require deburring. Specialized processes add to the cost of your part.



Hole Geometry/Taper

When a hole is punched in the stamping process it does not have a constant profile through the entire thickness of the part. The section view of a punched hole is shown below. The taper in the bottom side of the hole is caused by the punched hole diameter breaking through the material. The amount of taper in a punched hole is relative to the die clearance between the punch and the die used to punch the hole geometry.



PUNCHED HOLE



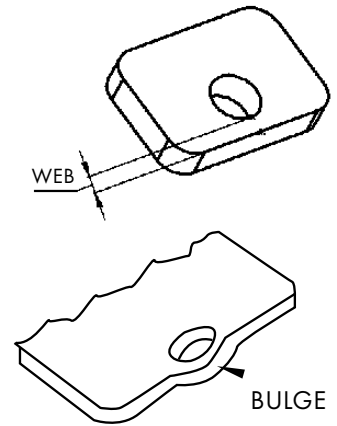
MACHINED HOLE

If a constant diameter is needed through the entire material thickness, a part must be drilled or machined as a secondary operation, resulting in much higher piece price.

Edge-to-Hole Spacing

The web – the space between a hole and the edge of a part – should be at least twice the distance as the material is thick. If edge-to-hole spacing is less than this distance, bulging along the edge will occur. If holes are required nearer to the edge than twice the material thickness, and bulge must be avoided, specialty drilling processes will be required.

For oblong slots that are less than 10 material thicknesses long, edge-to-hole spacing should be 2x material thickness; oblong slots longer than 10 material thicknesses require edge-to-hole spacing of at least 4x the material thickness to avoid bulging.



Bending

Metal stamping is a linear process wherein one step is performed after another, after another, and so on until a completed part is produced. The best stamped metal part designs take this step-by-step nature of the stamping process into consideration.

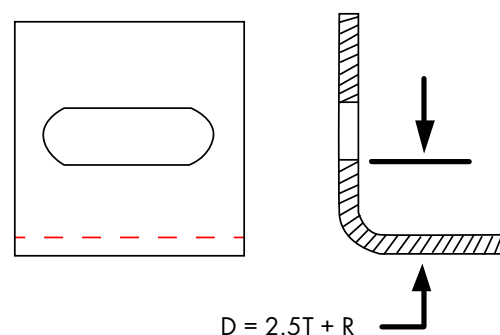
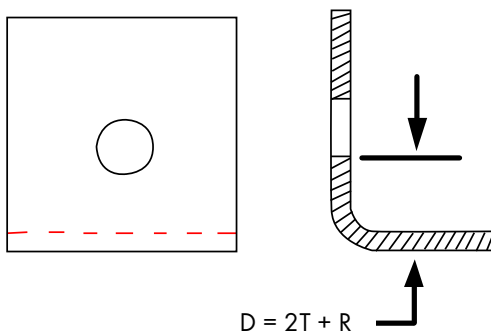
One of the more substantial forms, often performed toward the end of the progressive die stamping process, is bending. When designing bends into your stamped metal part, it is important to allow for enough material – make sure to design your part and its blank so that there is enough material to perform the bend.

Forming Bends Near Holes

When a bend is made too close to a hole, the hole becomes deformed.

For holes or slots less than 2.5mm in diameter or width, the minimum distance between a hole and the bend should be 2x the material thickness, plus the radius of the form.

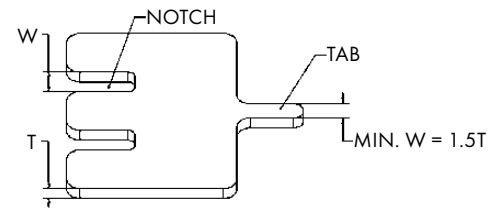
For holes greater than 2.5mm in diameter, the minimum distance should be 2.5x the material thickness plus the radius of the form.



Allowing less space than these standards will result in the hole distorting, bulging or stretching outside of its tolerances. Holes that must be closer to a form may require a post-stamping drilling processes, which can be very expensive.

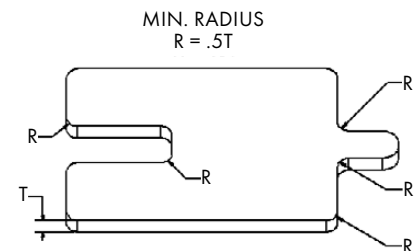
Notches and Tabs

Notches and tabs, as well as slots, should be designed with widths that are at least 1.5x the thickness of the material. If made any smaller, they can be difficult to create due to the force exerted on punches, causing them to break. In certain design situations, high precision stampers can accommodate smaller measurements, but be sure that they are absolutely necessary.



Corners

Every corner in your blank design should have a radius that is at least half of the material thickness. If your material is 1.5mm or less in thickness, corners can be very sharp.



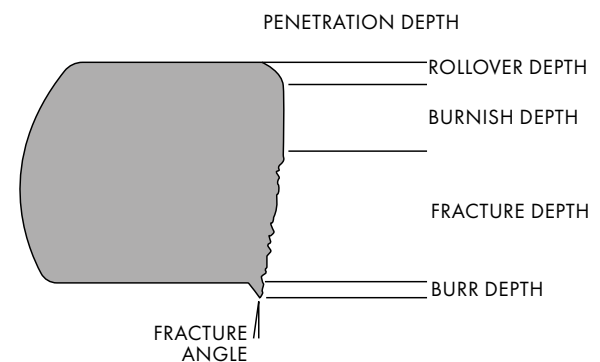
Burrs

Burr allowance is generally 10% of the thickness of the sheet material. To minimize instances and severity of burrs, avoid sharp corners and complex cutouts when possible. When such factors cannot be avoided, be sure to note burr direction in your design so they can be taken into account during stamping (See the “Stamped Edges” image below). Vibratory and tumble finishing can be performed to remove burrs when unavoidable.

Stamped Edges

When metal is cut or sheared, the applied downward stress on the material causes it to fracture as soon as the metal’s shear strength – the maximum strength a material withstands before separating at the cut location – is met. This action creates a burr on the bottom edge of the material, as shown in the diagram.

This “stamping burr” from the cutting process is critical for product designers to understand, as it creates an angle which can cause dimensional measurements to differ as compared to measuring from the burnished area. While this difference is very slight, it can come into play when parts are being held to very tight tolerances. The height of the burr, the fracture angle, and the depth of the fracture can vary depending on material type, punch to die clearance and tool sharpness.



Coining

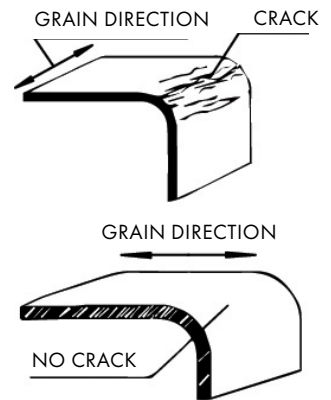
Coining the edges of a part is sometimes an option when trying to avoid costly secondary processes like vibratory deburring or edge grinding. During the coining process, edges of a stamped metal part are struck in such a manner as to flatten or break the burr; this can create a much smoother edge in the coined area of the part geometry. In addition to providing cost savings from secondary processes, coining can also add additional strength to localized areas of the part.

Plasticity and Grain Direction

Plasticity is the measure of permanent deformation a material undergoes when subjected to force.

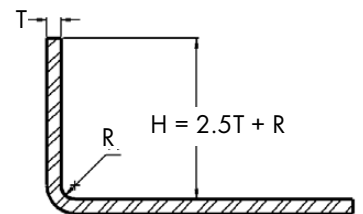
The more plasticity a metal has, the easier it is to form. High plasticity metals such as aluminum, brass, copper, and mild varieties of steel are formed easily and may require fewer considerations in regards to bends. The grain direction of these metals is not a major concern, and they generally require an inside bend height minimum of 20% less than higher strength metals (See bend height section below).

When bending high strength materials – tempered metals, stainless steels, and so on – grain direction becomes very important. Bends that go along the grain of a high strength metal are prone to cracking, which will make your part susceptible to failure. Thus, it becomes important to design bends in high strength material parts against the grain of the material.



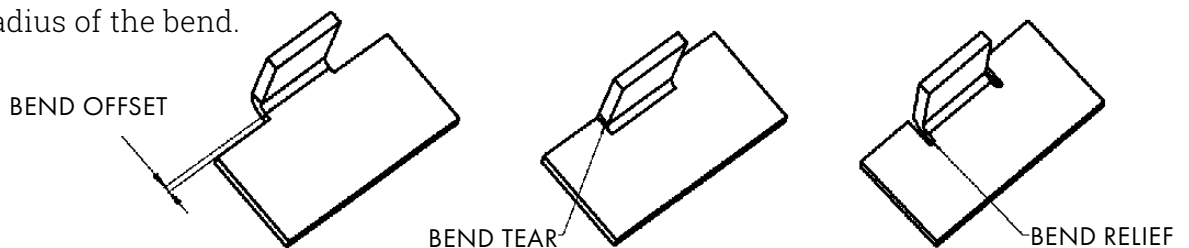
Bend Height

When adding a bend or form to a stamped metal part, the height of the bend becomes a design consideration. The thickness of the material comes into play when considering the overall height of the bend from the base material. The overall height of a bend has minimum requirements to be formed effectively and should be at minimum, 2.5x the thickness of the material + the radius of the bend. At times, the height of a bend may be less than $2.5T+R$ but can require costly secondary operations.



Bend Relief

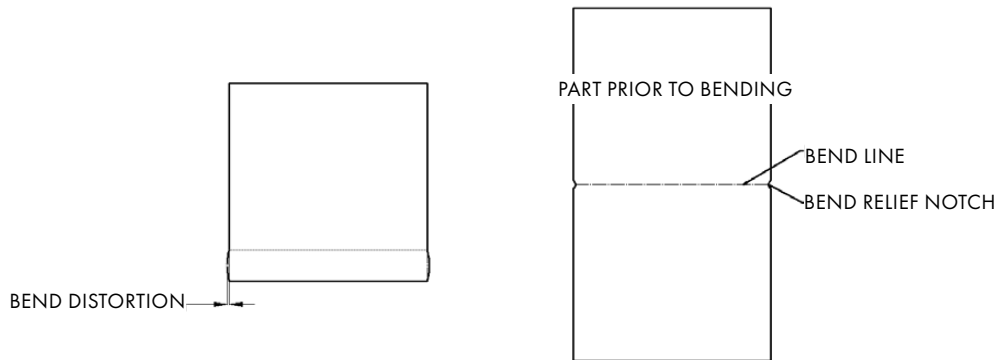
When a bend is made too close to an edge, the material on the base of the part along either side of the bent section may tear, drastically reducing the integrity of your part. Whenever possible, portions along an edge that are meant to be bent should be designed with an offset of extra material, at least as much as the radius of the bend.



When an offset cannot be included into a design, consider including bend relief notches in its stead. Bend relief notches are small notches located immediately adjacent to the portion of the part to be bent – they should be a minimum of twice as wide as the material is thick, and as long as the bend radius plus material thickness.

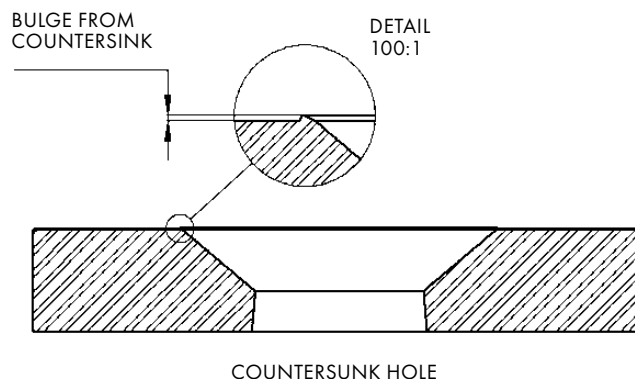
Bend Distortion/Bulge

An exaggerated example of bend distortion/bulge is pictured below. The bulging caused by this distortion can be as large as $\frac{1}{2}$ the material thickness. As material thickness increases and bend radius decreases the distortion/bulge becomes more severe. If bending distortion or bending bulge is unacceptable, the part can be relieved prior to bending with a relief notch on either side of the bend (See image below).



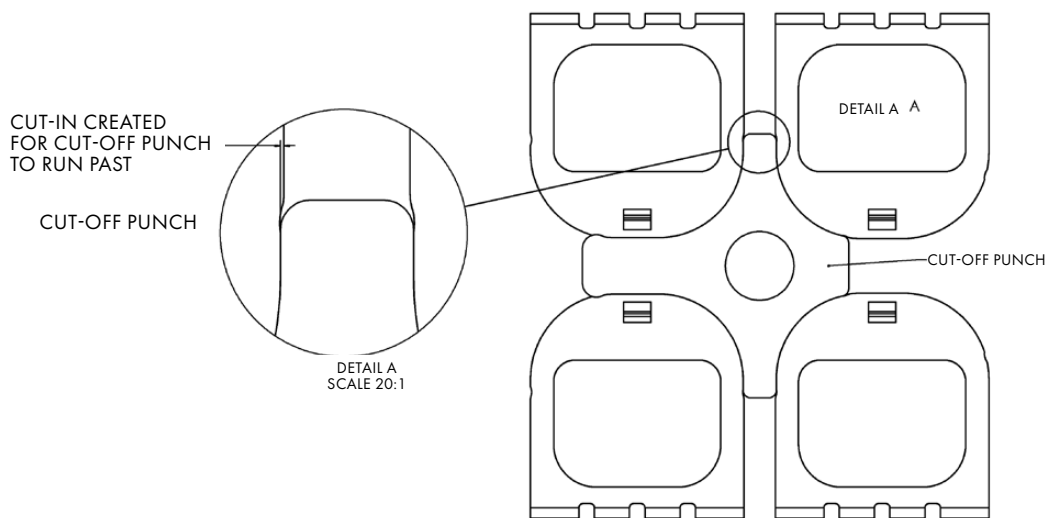
Countersinks

Countersinks are a common form utilized in stamped metal parts where the application calls for a variety of fasteners including flathead screws. There are several ways a countersink may be formed depending on the application or the quality necessary for the countersunk hole. One must keep in mind that in any countersinking operation a very small amount of material may rise above the surface of the material and create a very small "bulge" around the diameter of the hole (See image below).



Carrying Web and “Mismatch” Cut

During the design stage of a new progressive die, a conversation that should always happen with the customer is where the part will be carried in the strip as it is carried through the die. It is where these carrying webs exist that a “mismatch” cut will occur as the final part is cut free from the strip. In basic terms, a very slight cut-in or bump-out on the part is required and is typically about .005” deep. This feature is not necessary when utilizing compound or transfer type tooling but is required when utilizing progressive die tooling. At times, this mismatch cut can be hidden in a feature of the final part geometry that is acceptable to the customer and does not affect the function of the final part produced.





When to Consult the Experts

Industry leaders in metal stamping are equipped with design experts who can further scrutinize the manufacturing of a product and determine how to further minimize or eliminate unnecessary design costs. These manufacturers can also suggest materials or designs you may not be aware of that best meet your projects requirements in performance, aesthetics and overall cost. Even better, experts in metal stamping often provide capabilities in assembly, machining and finishing – giving you options to complete your project all in one stop.

At Engineering Specialties, Inc. (ESI), we provide a full staff of knowledgeable engineers and metal stamping capabilities to ensure the utmost precision on every project. In addition to utilizing progressive and four-slide stamping techniques, we also offer assembly and machining technologies to get your job completed quickly and at the lowest possible cost.

We're happy to take a look at your next project and see how we can help. [Visit our website](#) to learn more about our capabilities, or feel free to [contact us today](#).



About ESI

ESI has more than two decades of experience providing high quality stamped metal parts for a variety of industries around the world. Our [automated equipment](#) can handle thicknesses ranging from 1/8" to .005" (4 mm to .1 mm). We can produce high-volumes of more than 20 million complex parts – and we can do it without a single part per million defect. Our sophisticated solutions to common high-volume stamping problems guarantee that our customers receive the quality parts they need without delays.

To learn more about how we can provide for all of your stamping needs, [contact us](#) today.



452 Twin Lakes Rd
North Branford, CT 06471

www.esict.com

Phone: 800-486-5546

Fax: 800-486-2825

