MANAGING COST DRIVERS FOR Precision metal manufacturing



Managing Cost Drivers for Precision Metal Manufacturing H&S Manufacturing

This eBook was designed to share knowledge and stimulate discussion on various design characteristics and how they affect the cost of fabricated precision sheet metal and machined parts. We will cover several topics, including fabricated parts manufactured by cutting (punch/laser), forming, milling, finishes, and cosmetics.

In some cases, we will distinguish between 'sheet metal' and 'machined' parts and note those differences. Sheet metal typically refers to sheet materials that range in thickness from 0.020 inches (0.5 mm) to 0.250 inches (6.4 mm). Thinner materials are often called shims, while thicker material is referred to as plate or ingot.

At H&S Manufacturing, we typically work with thicknesses from 0.020 inches (0.5 mm) to 0.125 inches (3.2 mm). These thicknesses are typical but may vary depending on the requirement. We cut and form these sheets through various methods to produce the desired part. The precision aspect of this industry has to do with the dimensional accuracy of the finished parts. For sheet metal, the sheet thickness of the raw material is typically within 5% of nominal. Brand new sheet metal equipment typically holds dimensions to within 0.005 inches (0.13 mm), and bends within ½ degree. While machining the raw material, tolerances are typically of no concern. With the exception of machining castings and extrusions, most surfaces are cut. Therefore, tolerances of the raw material have no bearing on the finished part dimension, but only on how the finished dimension is achieved.

Machining typically refers to the removal of material using some type of cutter to produce the features of the part. Standard stock material shapes include plate, bar, round, tube, pipe, or extruded material. H&S focuses on milling, which involves feeding the raw material or part past a rotating cutter with cutting edges on its sides, end, or both. In contrast, turning rotates the raw material or part around its axis and applies a cutting tool parallel to the axis to create a cylinder, or at right angles to the axis to create a face.

Documentation

Clear legible documentation is imperative. In most manufacturing environments, the large majority of print users are not formally trained or educated on print interpretation. While most experienced sheet metal mechanics and machinists are very proficient at reading prints, not all operators and laborers are.

In manufacturing plants all over the world, some workers may have less than a high school education, almost all are trained on the job, and many have a different primary language than the home country of the manufacturer. Preparing the print in a way that requires a senior level, higher wage, more experienced person to understand its meaning means that the manufacturer accounts for that financially.

The ease at which the information can be deciphered reflects on the quality and efficiency of the manufacturing processes used to produce the part. In short, understand the target audience or user of the documentation. Here are a few items that should be considered when preparing a print:

Revision Control

A revision level on each print ensures that the customer and manufacturer are on the same page. Revision levels should be updated regardless of how minor the change is to avoid confusion later on.





Title Blocks

A title block should include the part description, part number, and revision. Other items regularly listed in the title block include CAD file name, engineering approvals, tolerances, and dates.

Geometric Dimensioning and Tolerance (GD&T,

GD&T can be a good thing if used properly and drafted correctly. It is an excellent method of showing important relationships between bends, edges, and other design features. It also helps inspection by detailing the critical dimensions necessary for functionality. There are cautions and considerations that should be taken when using this dimensioning scheme:

GD&T requires greater effort from the designer, though it usually yields more cost-effective designs with more efficiently producible parts than over-tolerancing parts.





GD&T requires highly skilled employees to produce and inspect.

GD&T often requires the use of more sophisticated inspection equipment than standard-tolerance parts.



However, in cases where a requirement exists for tighter tolerances on certain features, then using GD&T can focus additional efforts to the specific dimensions necessary rather than wasting time on over-tolerancing all dimensions of the part.

customerservice@hsmfg.com





Location of Zero Datum

When dimensioning a drawing, account for the usual inspection methods used in manufacturing. For example, most sheet metal gets measured with handheld calipers during the production process. The location of the zero datum can result in more calculations necessary to measure the part. Adding calculations reduces the efficiency of the process and increases the likelihood of errors.

5

While handheld calipers are also used to measure some dimensions on machined parts, most are measured using a height gauge on a flat inspection table. Though slower, this method is necessary for tight-tolerance machined parts and allows for easier location of the zero datum to measure dimensions compared with hand calipers.



Holes or Features | Formed and Flat

Customers frequently ask the H&S team how close to a bend a hole can be located. A hole or cut out of any shape can be located right in the middle of a bend, but keep in mind that the shape will get distorted during the forming/bending process. Generally speaking, formed features cannot be located in the middle of the bend lest they get flattened or misshapen during the forming/bending process. Normal bending procedures are enabled when the edge of a hole or feature is at least two material thicknesses away from the start of the bend. This rule holds for most formed features. However, depending on the direction of the form and the die combinations necessary to form the part relief, modifications may be necessary to the brake tooling to clear the formed feature. Unformed features and holes closer than two material thicknesses frequently require the use of backup material. Incorporating the backup material adds additional manufacturing time and may still result in disfigured holes or features and process variations.





Flange Size

Another common question concerns flange sizes. In material thicknesses ranging from 0.020 inches (0.5 mm) to 0.125 inches (3.2 mm), a good general rule for inside minimum flange length is three times the material thickness plus the inside bend radius. Coining with thinner materials can yield a smaller radius but requires additional considerations such as reduced material strength, more pronounced brake marks, and more. Of note, the aerospace sector forbids coining due to the strength reduction.

Bend Radius

The appropriate minimum bend radius is another hot topic. As a general rule for minimum bend radius, the inner radius should be at least one material thickness. Thicker material and sometimes more brittle material requires a larger bend radius to avoid cracking.





Standardization

Bend Relief

Bend reliefs are cutouts or holes that prevent tearing or cracking of material as the part is formed. Forming without tearing or cracking the material will improve the accuracy and consistency of the part during production. Reliefs are typically specified as a shop option and are not dimensioned or get marked as 'no inspection.' The minimum bend reliefs should be one material thickness plus the inside bend radius.

Standardization represents a major cost consideration in metal part design. In terms of production cost, boring is better. Using standardized design features will lower the overall cost of the production run. Conversely, if the design is constantly pushing everything to the limit and each feature is unique, then production costs inflate significantly.

When possible, use the same piece of hardware throughout a design. The more we buy the less we pay. The less we pay, the lower your costs. When selecting items from catalogs or websites, understand which items are standard or off the shelf, versus which ones are special or custom. Words like special and custom are always buzz words for higher cost and the seller wants you to use them because they make more money on them. Using the same radius call out wherever possible on a machined part means less set up time, less tool changes, more efficient run time, and lower production costs.

Designers know their designs better than anyone else. Most manufacturers or vendors see many designs or prints every day. Designers play a vital role in communicating about similar designs or pointing out consistent features across a family of designs.

Tolerances

Back in the day, build-to-order sheet metal tolerances were +/- 0.06 inches (1.5 mm), primarily limited by the accuracy of the equipment. Today, normal production tolerances are around 15% of that, though it's important to keep tolerance ranges reasonable and practical for the application. The most significant improvement has been in the machinery. Some brand new, modern high-speed equipment can position within 0.004 inches (0.10 mm) and repeat within 0.002 inches (0.05 mm).

Materials and methods continue to play a role in the limits of precision. They greatly influence practical considerations for tolerances between holes (hole-to-hole), between bends (bend-to-bend), and so forth. When necessary, dedicated tooling and special processing can achieve even tighter tolerances. It is a mistake to simply dimension all mating parts expecting +/- 0.005 inches (0.127 mm) accuracy, however. Over-tolerancing forces additional labor in sorting and inspection and may result in excessive scrap. Too-tight tolerances ultimately result in higher costs and lower productivity.

Hole Sizes

The most common method for producing holes involves mechanically punching a tool through sheet metal into a die tool to rip out a slug and form a hole in the sheet metal. The size and shape of the punch and die tooling determine the size and shape of the hole. The minimum size hole or feature produced this way must be at least 1½ the material thickness. The die tool must be slightly larger than the punch to minimize tooling wear and reduce the required punching pressure.

This difference in the punch and die tool size is called clearance or die clearance. The die clearance is generally about 10% of the material thickness. For example, if the material is 0.100inch thick aluminum and the punch diameter is 1.000 inch, the die diameter would be 1.010 inches. The size of the hole on the punch side will be the same size as the punch tool and the size of the hole on the die side will be the same size as the die tool. This is referred to as blow out. Except for tooling wear, there is very little variation from one hole to the next.







Hole Sizes continued...

Most fabricators look to the engineers and draftsmen to give a tolerance range that allows the use of existing tooling. When that is not possible, a capital investment in new tooling is required. Generally speaking, +/- 0.003 inches (0.08mm) is a reasonable hole size tolerance for punching holes. Keep in mind, however, that we are measuring what will pass through the hole, not the rim size of the hole itself.

If required, tighter tolerances can be held on holes by milling them instead of punching them. This method of production is ideal when the blow out described above is unacceptable. This is the preferred method for many machined parts. For sheet metal parts, this method is less efficient and adds cost to each part. Standard hole size tolerances for machining applications are +/- 0.003 inches (0.08 mm). Though there are many variables to consider, milled holes can achieve tolerances within +/- 0.0005 inches (0.013 mm). Some of the considerations include material, hole depth, hole location (length of tool needed), thru or blind hole, and more.





Hole-to-hole (same surface)

The accuracy of the distance from one hole to another hole primarily depends upon the machinery. Most modern equipment will hold better than +/- 0.005 inches (0.13 mm) However, each feature punched introduces stress into the sheet metal. If the part has many closely spaced holes, features such as perforated patterns, or formed features such as counter sinks or lances, punching may cause the sheet to warp or distort. This distortion may result in unwanted variations between holes and features. A standard tolerance for hole-to-hole would be +/- 0.010 inches (0.25 mm), especially in areas where a large percentage of material is being removed. Use tighter tolerances, as low as +/- 0.005 (0.12 mm), only when absolutely necessary. In machining, the amount of material being removed is also a large contributing factor in hole-to-hole tolerances. As heat builds in the part and cutting instrument, material moves and causes variation. Machining processes make this very controllable and allow for much tighter tolerances, though tighter tolerances will always require more labor for sorting and inspection. Also, reducing feeding and speed to achieve tighter tolerances means less efficient production, which adds cost to the part. Standard hole-to-hole tolerances for machining applications are +/- 0.005 inches (0.12 mm).

10

Hole-to-edge (same surface)

In most modern sheet metal fabrication, the profile (or edges) of the part are generally punched or laser cut, just like any hole. Therefore, the same considerations for hole-to-hole apply. When producing features by punching very near to an edge (less than one material thickness), the edge may push out from the stress of punching. In the case of cut outs parallel to an edge, the remaining web can roll or twist. Edge migration introduces variables in the accuracy of the hole or feature location to the edge. Standard hole-to-edge tolerance is +/- 0.010 inches (0.25 mm), especially in areas where holes or features are less than the minimum distance recommended from the edge. Use tighter tolerances within +/- 0.005 inches (0.12 mm) only when absolutely necessary. Additional considerations should be given to the intended use of the hole or feature. For hardware insertion, most hardware manufactures publish a minimum edge distance to minimize distortion. Hole-to-hole tolerance considerations for machined parts also dictate hole-to-edge tolerance. Standard hole-to-hole tolerances for machining applications are +/- 0.005 inches (0.12 mm).



customerservice@hsmfg.com

www.hsmfg.com

Hole-to-bend (dimensions across one bend)

If sheet metal is ready for forming on the press brake, features and parts have already been punched on a CNC turret press and/or cut using a laser, as well as line sanded or tumbled to remove burrs. The deburring process can remove as much as 0.003 inches when cosmetic appearance is a priority. Precision press brakes will position and repeat within the +/- 0.002-inch range. Skilled brake operators can load the parts for forming consistently from bend-to-bend. Nevertheless, consideration must be given to several factors, including:



A +/- 0.015-inch tolerance for hole-to-bend is functionally reasonable for most applications. Resort to +/- 0.010-inch tolerances only when absolutely necessary.



Bend-to-bend (dimensions across multiple bends)

All of the considerations of hole-to-bend apply here as well, compounded by the fact that multiple material surfaces and thicknesses are involved. Those variations will be multiplied by factors of at least two, and sometimes many more depending on the number of bends across the dimension. Standard tolerance should allow +/- 0.020 inches (0.50 mm) bend-to-bend. Use tighter tolerances down to +/- 0.010 inches (0.25 mm) only when absolutely necessary.



Tolerance Summary

Gauge Tolerance

Gauge tolerance is the tolerance of the devices used to measure or inspect the products being manufactured, which must be considered when applying tolerance. Most sheet metal with standard tolerance is inspected on the manufacturing floor using handheld calipers. A well-calibrated set of calipers will have a tolerance of +/- 0.001 inches (0.025 mm), whereas a well-maintained, top-of-the-line height gauge used for inspecting machined parts will have a tolerance of +/- 0.0005 inches (0.13 mm).

Whereas brand new machinery and proper tooling will repeat within 0.004 inches (0.10 mm), it is a mistake to simply dimension all mating parts expecting +/- 0.005-inch (0.13 mm) accuracy. Such overkill forces additional labor in sorting and inspection, creates excess scrap, and lessens manufacturing efficiency. Over-tolerancing results in higher cost and lower productivity. Correct-tolerance parts still offer excellent fit and function, with the added benefit of production efficiency.

When dimensioning machined parts, tighter tolerances are typical. One should still consider the cost impact of the tolerances designed into the part and remember the impact that tolerance overkill can have on the efficiency and productivity of the manufacturing process. For both cost and manufacturing efficiency, consider the sum of the variables involved affecting tolerance, such as:

Machine Capabilities

Material limitations and variations

Accumulation of variations from multiple processes

Gauge tolerance

Tolerance should be viewed in terms of what is required and not what can be produced. Given enough time and money, almost anything can be produced.





Cosmetics

Cosmetic call outs can cause the cost of metal parts to skyrocket. Metal parts can be categorized in several ways:

External (in full view at all times in final product)

External/internal (partial view, or in full view until fully assembled at customer site)

Internal (out of customer view)

Structural/no cosmetic call out required

Along with similar designations, most companies have developed their own inspection criteria. Most are based on a viewing time and distance with a maximum number of recognizable flaws allowed within a specified time and at a specified distance. These specifications typically state the criteria is primarily used to train inspection personnel and can be used in determining acceptance or rejection decisions.

By its very nature, the metal fabrication process—from the handling of raw materials, the equipment used, and the methods and processes used in fabrication—creates opportunities for flaws and defects in material surfaces. Fabricators and machine shops have been producing products that meet customer requirements for years. In some cases, flaws get covered up by the finish of the part, such as paint or powder coating. In other cases, however, these flaws get accentuated by the finish of the part, such as shiny nickel or chem film.



Graining is required on many higher cosmetic finish call outs. The graining process does provide a more aesthetically pleasing finish, but it does have limitations. Graining or sanding is done early in the process, leaving many post processes that can and do cause flaws. Also, machined parts or sheet metal parts that have some forming operations performed on the punch press cannot run through automatic sanding machines. Those parts need special jigs to carry the parts through the sanding machine or must undergo hand graining.

The higher the cosmetic call out, the more inspection time required, and the more scrap produced. Most cosmetic specifications differ only slightly from one class to another. For example, there is very little variation that can be achieved in metal processes to produce a part with only two flaws versus four flaws. More stringent criteria simply means more inspection and possibly more fall out.

Cosmetic call outs are necessary to convey to the manufacturers the desired results. Take caution and consider the actual use of the part to determine if and when the extra money and effort spent to achieve a higher cosmetic finish is really necessary.



About H&S Manufacturing

H&S Manufacturing Co. has been a premier metal fabricator since our inception in 1968. With capabilities spanning precision sheet metal fabrication, machining, assembly, and testing, H&S Manufacturing Co. uses state-of-the-art manufacturing solutions, sophisticated software capabilities, and skilled personnel to ensure the highest quality results for every customer. With priorities that include customer satisfaction, on-time delivery, and cost reduction, we strive to build true partnerships with our customers. To see how we can support your project, please don't hesitate to contact us or request a quote today.





CONTACT US

RESOURCE LIBRARY

H&S Manufacturing Co.

2913 Singleton Street Rowlett, TX 75088

Contact Us

Phone: 972.475.4747 Fax: 972.475.6155 Email: customerservice@hsmfg.com Website: www.hsmfg.com

Follow us