TENERE

CUSTOM MECHANICAL SOLUTIONS
TENERE.COM

A STARTER GUIDE TO SHEET METAL FABRICATION

An introduction to basic sheet metal fabrication terms and definitions

WHAT'S COVERED?

- Sheet Metal Fabrication Defined
- Glossary of Terms
- Sheet Metal Forming and Stamping
- Types of Sheet Metal Stamping Molds
- Joining Methods
- Sheet Metal Automation
- Designing for Sheet Metal Fabrication

INTRODUCTION

Whether you're liking a photo on social media or making the switch to a solar powered home, sheet metal is everywhere. And because its uses are so varied - from the servers that run social media platforms to in-home intelligent energy storage systems - we've put together a guide to help you understand the ins and outs of sheet metal fabrication.

Common Materials: Aluminum, Brass, Bronze, Copper, Nickel, Steel, Tin

Sheet Metal Fabrication Defined

Sheet metal fabrication is the process of transforming sheet metal into specific shapes, usually by bending, punching, or cutting. Metal sheets of various gauges can be manipulated into nearly any shape or size.

Depending on the process and the engineering required, sheet metal fabrication often involves varying levels of human interaction, but all involve some form of heavy machinery and equipment.

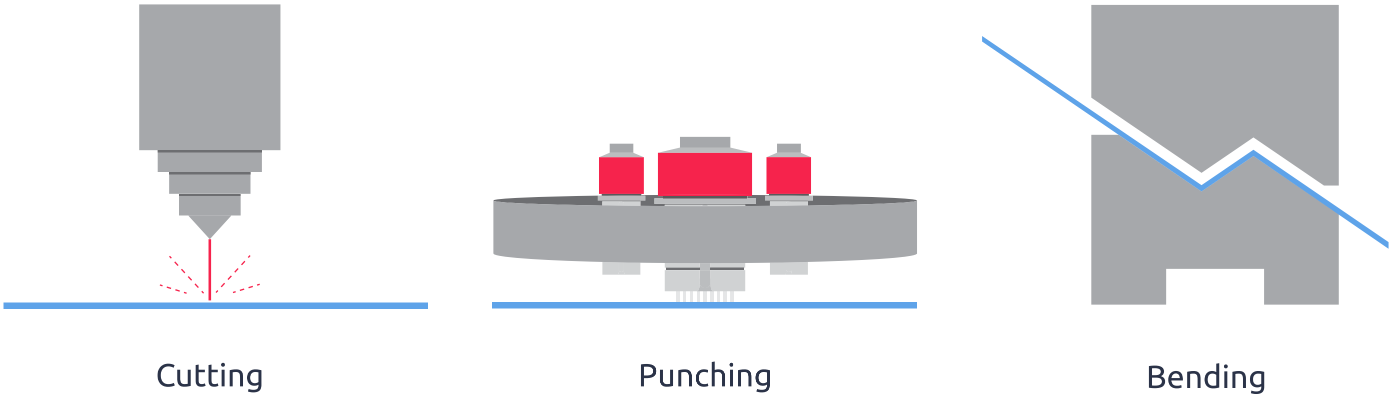

FIGURE 1

GLOSSARY

ASSEMBLY - the action of putting together individual or partially assembled units to build a complete product.

BASE METAL - the sheet of metal that is to be cut, bent, or punched.

BENDING - the process of applying pressure to specific areas of the base metal to achieve a desired shape.

BLANKING - removing a piece from the entirety of the base metal, where the piece is kept and the remaining metal around it is discarded. Like a cookie-cutter. Blanking typically occurs prior to bending but can happen while forming, depending on the machine.

CAD - short for "computer-aided design," engineers use CAD software to design 2D and 3D models.

COINING - a different design is created on each side of the metal or on just the top layer and not throughout - usually used for fine detail.

CUTTING - the use of blades, torches, lasers, and other means to remove pieces of sheet metal.

DIE (PRESS BRAKE) - the cavity below a piece of sheet metal that allows the punch to pierce the metal.

EMBOSSING - one side of the material is raised while the other is depressed.

FORMING - the process of cutting, punching, or bending sheet metal to create a desired shape.

HARD TOOLING - sheet metal stamping

LANCING - a slice or slit made in a piece of metal to free it up without separating it from the main piece.

LASER CUTTING - an extremely precise method of cutting that uses a concentrated beam of light. Also used by evil geniuses.

MACHINING/MILLING - the controlled removal of material using a cutting tool or lathe.

NESTING - strategically fitting multiple parts on a single sheet of metal to reduce waste. Commonly arranged automatically by nesting software.

PEMMING - the installation of threaded hardware (usually a specific set of threaded fasteners) to allow for assembly; includes nuts, studs, and standoffs.

PLASMA CUTTING - the use of concentrated ionized gas (plasma) to melt away portions of sheet metal. Less precise than laser cutting.

POWDER COATING - a dry powder surface coating that's applied to final metal pieces. When heated, it forms a bond with the metal, ensuring a lasting finish and color.

PRESS BRAKE - a machine that forms predetermined angles, or bends, by squeezing a sheet of metal between a matching punch and a die.

PUNCH (PRESS BRAKE) - the tool above a die that uses compression to create a bend or pierce a piece of sheet metal.

RIVETING - the installation of a mechanical fastener, or rivet, through a hole in the material to hold two or more pieces of sheet metal together.

SET-UP TIME - the amount of time it takes to set up the proper dies and punches for a job. This time varies depending on the complexity of the part and machine.

SHEARING - a form of cutting in which downward force is applied to sheet metal, causing it to break cleanly.

SOFT TOOLING - the sheet metal fabrication processes that include laser cutting, turret punches, and press brake forming.

STAMPING - accomplished using steel dies - progressive and stage - that are designed to stamp, bend, and form sheet metal.

TONNAGE - the amount of pressure applied by a press.

TURRET - a type of punch press used to punch sheet metal. Named after the rotating turret head above the metal that rotates to change tools.

WELDING - the joining together of two or more pieces of metal by melting or soldering them together. Types include TIG, MIG, projection, and spot welding.

METAL FORMING AND STAMPING

Press Brakes

A press brake squeezes a single sheet of metal between two plates or dies to bend the metal to predefined specifications. The force (or tonnage) needed to bend the metal is often determined by the type of metal, gauge, and geometry of the bend. Press brakes offer two types of dies: standard and custom.

Standard dies produce simple bends based on straight-forward geometry. But when products call for a more creative approach—like 3D shapes—a custom die must be developed.

Turrets and Lasers

When it comes to cutting sheet metal, turret punches and laser cutters are common options.

Turret punches hold a variety of standard tools to punch shapes into sheet metal. The sheet is laid flat and clamped into the turret, which is programmed to move the sheet metal across the machine bed and make a series of punches to achieve the desired part. The rotating turret changes the tool based on which feature it needs to punch.

Laser cutting utilizes laser technology to cut

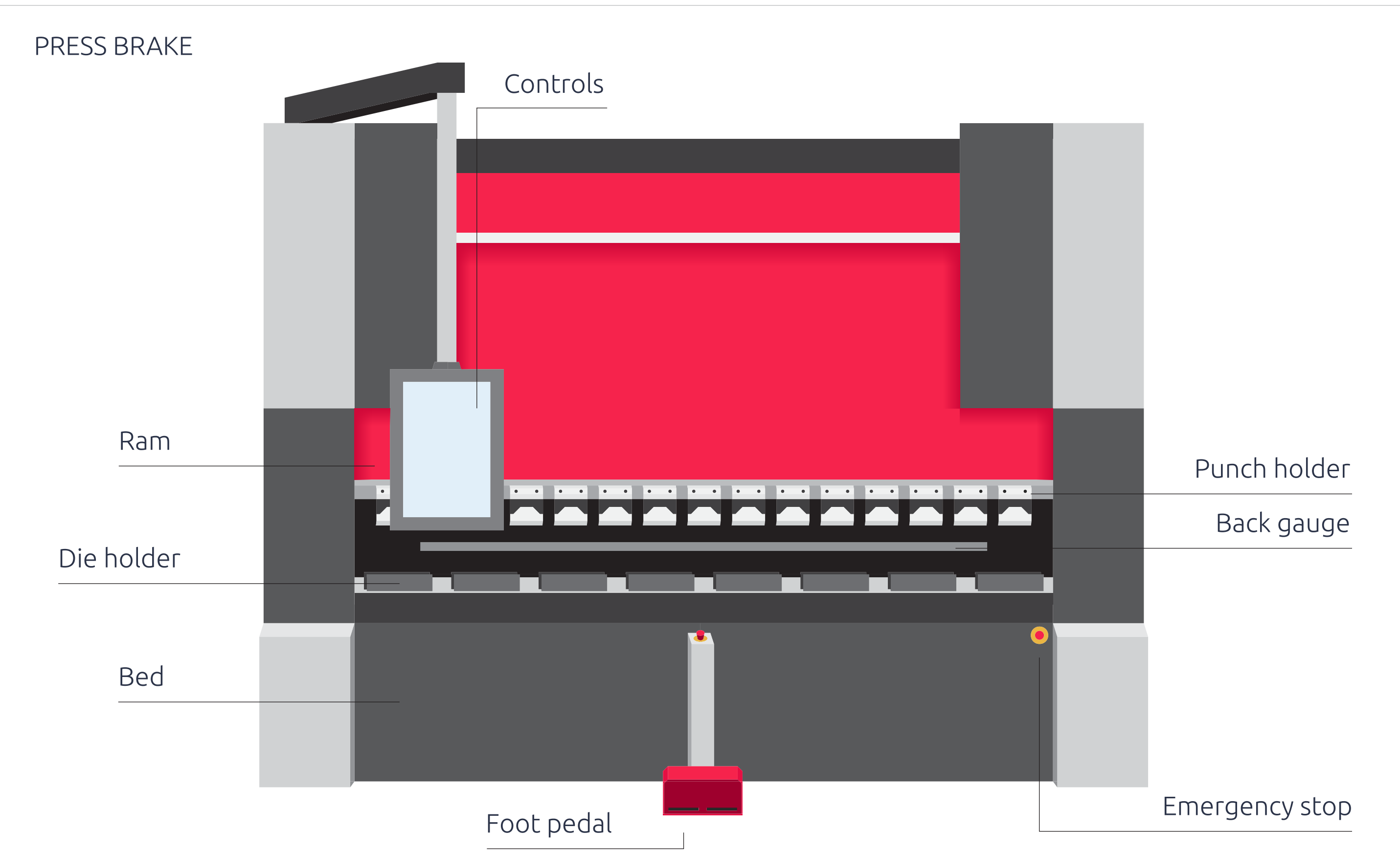

FIGURE 2

through sheet metal. The sheet is placed into the machine and the laser cutting head moves to make the desired cuts to the part, as programmed. Laser cutting offers geometry flexibility since the cut options aren't limited to a standard tool.

Where laser cutters can cut thicker metal types, turret punches have the capability to work with more types of metals. Turrets also have more capabilities such as coining and bending, which reduces the manual labor required to complete the part.

Metal Stamping

Metal stamping is great for producing multiple, uniform metal parts at scale. Blank or coiled sheet metal is fed into a press where the metal is shaped by a tool and die. Common operations are embossing, blanking, coining, punching, and flanging.

Soft Tool vs. Hard Tool

Soft tool and hard tool sheet metal fabrication allow for flexibility and scalability. Soft tooling, or metal forming, uses lasers, turrets, and brake presses which allows a product to hit the ground running with minimal to zero tooling investment. Soft tooling also allows for design flexibility needed during the design, prototype, and production ramp of a product.

Once the product's design is established, it's important to start looking for additional cost improvements. Hard tooling, or metal stamping, is a great option for high volume parts, decreasing the overall cost of the part long-term and increasing part consistency. Parts are produced much quicker and require less labor, which can be highly beneficial as a product becomes a staple in the market and demand increases.

Most sheet metal assemblies are a mix of soft and hard tool parts.

TYPES OF METAL STAMPING MOLDS

Progressive Dies

A progressive die is made up of a series of progressions in a process where, when advanced through the mold, each piece is further bent or punched until complete. As the press moves up, the metal strip advances through the machine, one station at a time. The final station cuts the completed piece free from the strip of metal. The finished pieces and excess metal are automatically separated.

Stage Dies

Unlike progressive dies, a stage die works on one part at a time. The part is moved in and out of the die either by human hand or by a robotic process. Stage die types include compound, pierce, forming, ratchet, and coining.

They're the best option when part design does not allow for carrying points or when form points conflict with cutting, stretching, or forming requirements.

PROGRESSIVE DIE

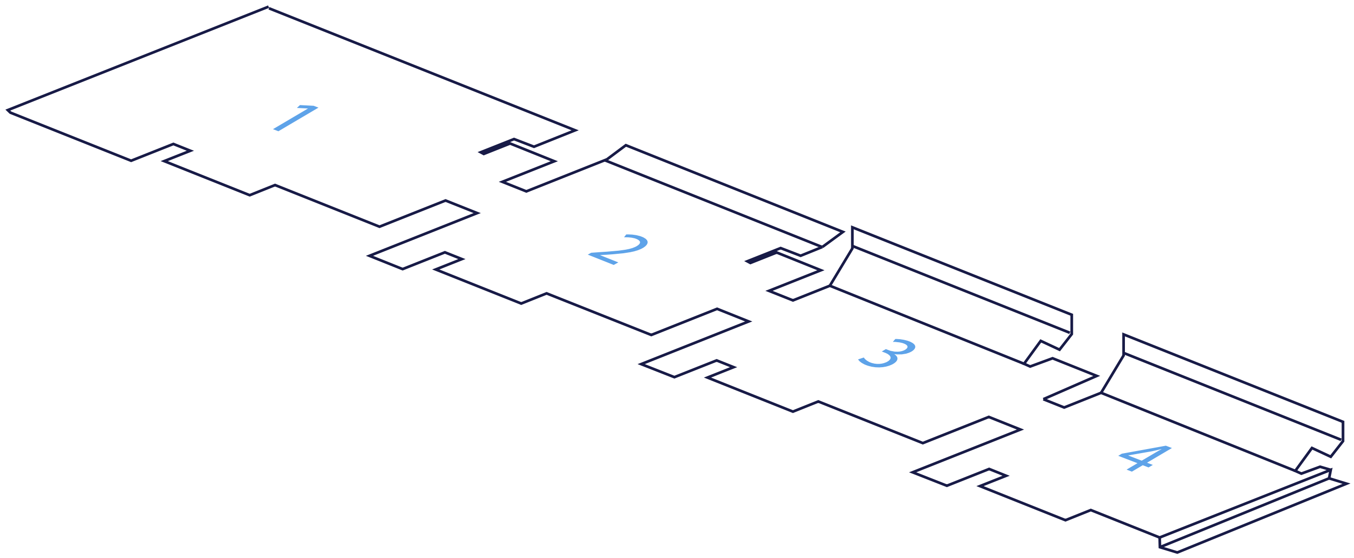

FIGURE 3

JOINING METHODS

Sheet metal fabrication isn't just about cutting and bending. In cases where multiple metal parts need to be joined, welding and riveting become important operations.

Welding

When dealing with tight tolerances and thin wall products, the heat produced by welding has a direct effect on metal quality. As a rule, heat should be minimized whenever possible. While welding is the most costly joining method, it is often the only viable process in areas that do not allow for overlap or assembly holes.

The most common types of welding are:

MIG - MIG stands for "metal inert gas." It creates an electric arc between a piece of wire and the sheet metal it's touching, causing both to melt and join together. This produces a strong weld bead.

TIG - TIG stands for "tungsten inert gas." It is commonly used to weld thin pieces of stainless steel or aluminum. While it's more difficult to master than MIG welding, it does produce stronger welds than MIG welding.

Robotic - Unlike MIG or TIG welding which require human experts, robotic welding automates the entire process. Most robots are used for resistance spot welding and arc welding. If the robot is performing MIG or TIG welding, human assistance is required to prepare the welding materials.

Spot – Two pieces of metal are placed under pressure between two copper alloy tips which apply an electrical current, welding the pieces together. Welds are flush to the surface and no fluxes or fillers are required, allowing for a clean appearance.

Riveting

Rivets are used to fasten two or more pieces of metal together and can be purchased in steel, stainless steel, and aluminum. The advantages of riveting over welding include their ability to fasten dissimilar materials together or their ability to fasten areas that are not accessible to other processes.

Rivets can be drilled out and re-installed if disassembly should be necessary for any reason.

There are three basic types of rivets.

Solid Rivets - These rivets are inserted using a special rivet machine or standard hardware insertion machine. They're made up of a shaft and head that become deformed during insertion. Solid rivets are very strong but require access to both sides of the sheet metal, as well as an overlapping joint.

Semi-Tubular Rivets - These rivets are also composed of a shaft and a head, the difference being in the partial hole at the tip. This small hole reduces the amount of force required to install. Similar to a solid rivet, access to both sides of the sheet metal and an overlapping joint are required.

Blind Rivets - These rivets have an integrated mandrel running through them. When pulled out by a riveter, this mandrel compresses the back of the rivet to flatten it against the metal, securing two pieces of metal together. The riveter only needs access to one side of the part, (which is where it gets the name "blind") making this is a convenient riveting process. With proper material preparation, blind rivets can be flush on both sides.

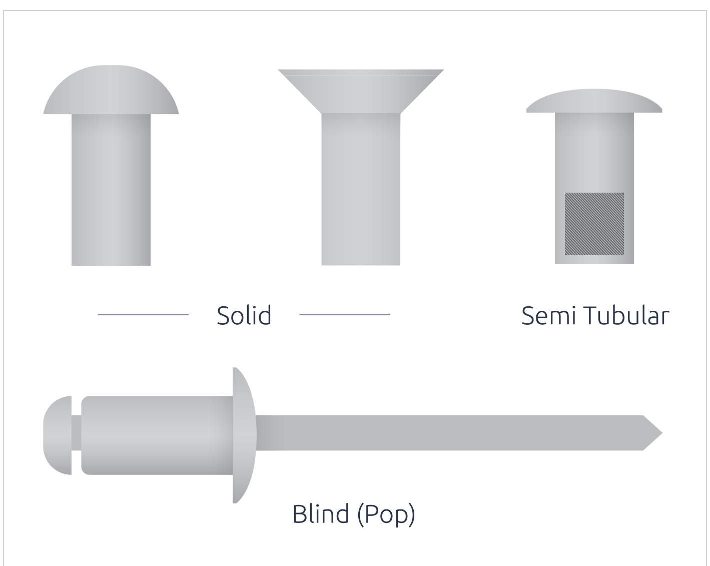

FIGURE 4

SHEET METAL AUTOMATION

Automating sheet metal fabrication operations like bending, cutting, or welding can simplify processes and speed up production timelines simply because they remove human error and fatigue. Here's a quick look at two of the more popular ways to automate the sheet metal fabrication process.

Robotic Operators

Robotic operations improve productivity as well as quality. That's because robotic operators can perform with uniform precision every time, reducing potential ergonomic and safety hazards to human operators. For example, some programs require extremely repetitive motion that can injure a person over time. Robotic operators eliminate that risk.

Robotics can also be quickly reprogrammed to accommodate last-minute engineering changes, small batch runs, and line change-overs.

Machine Robot - Machine manufacturers are now incorporating robotics directly into their technology. These robots are built specifically for each machine and can perform duties based on how the machine is programmed.

Press brakes are a good example of a manufacturing process that benefits greatly from robotics. The robot is programmed to move sheet metal across the press brake to achieve each desired bend in the program.

Another example is a robot used with a stage die; the robot is programmed to recognize when to move a part to the stamping press, wait for the stamping to take place, and remove the part when done.

Robotic Arm (Collaborative) - Robotic arms do not communicate with a machine and are programmed separately. They work well for pick-and-place operations where they are programmed to move parts from one location to another. These robots are often paired with a human counterpart, making them collaborative.

Machine Automation

Automatic Tool Changing - Whether working with press brakes or turrets, automatic tool changers (ATCs) are fast, reliable, and accurate. Once it receives a command, the changer mechanism will remove an old tool and replace it with a new one. Manual set-up time for a press brake can take 1-4 hours, depending on the number of bends, where ATCs can reduce that time to 20-30 minutes, no matter how complex the program.

Material Load/Unload Systems - Turret presses can now be purchased with automated sheet loading and unloading so the process can continue seamlessly without having to manually feed sheet metal onto the turret. This increases productivity and reduces potential ergonomic hazards.

DESIGNING FOR SHEET METAL FABRICATION

IIt is important to work with a design for manufacturability (DFM) engineer who has expertise in sheet metal fabrication during the prototype phase before manufacturing begins.

Here are a few helpful design tips.

Excessive Forming - Incorporating cuts or bends that do not have a functional purpose can create added costs. Excessive forming can also make the part impossible to bend.

Critical Dimensions – call out information not available on models—datum planes, tolerances (block and critical), material type, finish

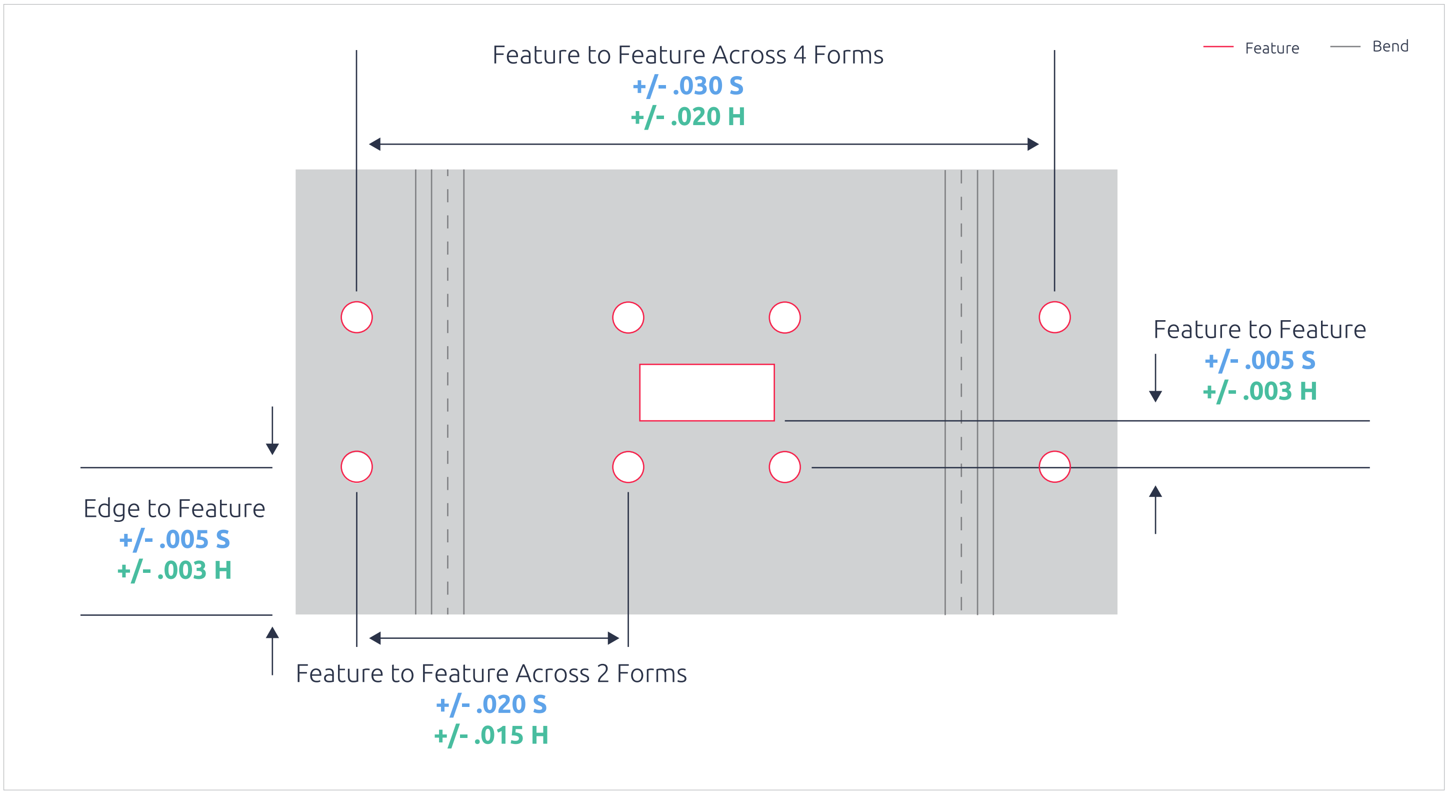

FIGURE 5

requirements, hardware specifications, hole tapping, welding requirements, surface requirements, and edge requirements, just to name a few.

Standard Tolerances - Although the machinery and tooling will repeat within .004", it is a mistake to simply engineer all mating parts, to be within +/-.005". This excess forces additional labor in sorting and inspection.

Tolerances that are too tight result in higher costs and lower productivity. Correct tolerance will still produce excellent fit and function, with the added benefit of manufacturing efficiency.

Hole Sizes +/- .003" - The size and shape of the punch and die tooling determine the size and shape of the hole. A minimum hole or relief size is determined by the thickness of stock to be used. For best results, the punched feature can be no less than the material being punched. The die tool is slightly larger than the punch to minimize tooling wear and to reduce the pressure required to punch the hole. Generally speaking, 10% of the material thickness is used for most applications.

For example, if the material is .100" aluminum and the punch diameter is 1.000", the die diameter would be 1.010". The size of the hole on the punch side will be the same size as the punch tool. The size of the hole on the die side will be the same size as the die tool. Except for tooling wear, there is very little variation from one hole to the next. Generally speaking, +/-.003" is a reasonable and functional tolerance.

Hole to Hole +/- .005" - Accuracy of the distance from one hole to another is dependent primarily upon the machinery used to process the sheet. Some equipment will hold better than +/-.005" with little difficulty.

However, all holes and features punched through the sheet can introduce stress into the sheet metal. If the part has a closely spaced perforated pattern or formed features such as dimples or counter sinks, the result can cause the sheet to warp and distort. This can cause unwanted variation between holes or features. If this condition exists, a greater tolerance should be applied to certain areas surrounding this characteristic.

Hole to Edge +/- .010" - Part profiles are punched just like any other feature, except when using a machine with shearing capabilities. These dimensions should be considered the same as hole-to-hole.

When punching close to an edge (less than double the material thickness), the edge can be pushed out by the stress of punching the metal. This edge movement can introduce variables in the accuracy of the hole location in relation to the edge. There are techniques to minimize this problem, but

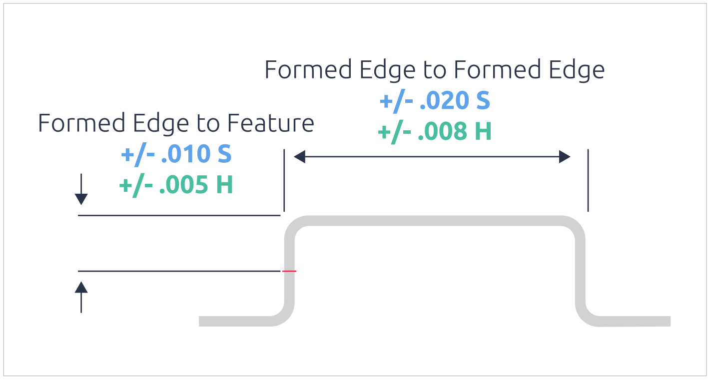

FIGURE 6

whenever possible, engineers should allow +/-.010" hole-to-edge. Tolerances of +/-.005" should be used only when absolutely necessary.

Hole to Bend +/- .015" - Several factors have been introduced leading up to this stage in the fabrication process. Features and parts have been punched on a CNC turret press, line sanded or tumbled to remove burrs, and is now being formed on a press brake. The deburring process may remove .003" when cosmetic appearance is a priority.

Precision press brakes will position and repeat within the +/- .002" range. Skilled brake operators are able to load the parts for forming consistently from bend to bend.

Nevertheless, consideration must be given to the natural variation in material thickness (5% of nominal thickness), the +/-.005" from the turret press, the effects of cosmetic sanding, and the variation introduced by the press brake. A tolerance of +/-.015" hole-to-bend is functionally reasonable for most applications. Resort to +/-.010" only when absolutely necessary.

Bend to Bend +/- .020" - Considering the variables that affect hole-to-bend tolerances, now multiple material surfaces and thickness are introduced. Whenever possible, engineers should allow +/-.020" bend-to-bend. Resort to +/.010" only when absolutely necessary. Increased tolerances need to be applied if going across multiple bends.

SALES@TENERE.COM
715.294.1577