CADREX
MANUFACTURING SOLUTIONS

CONCEPT TO SCALE
CADREX.COM

DESIGNING FOR SHEET METAL FABRICATION

A Guide to Standard Tolerances

It 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 and a guide to standard tolerances.

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.

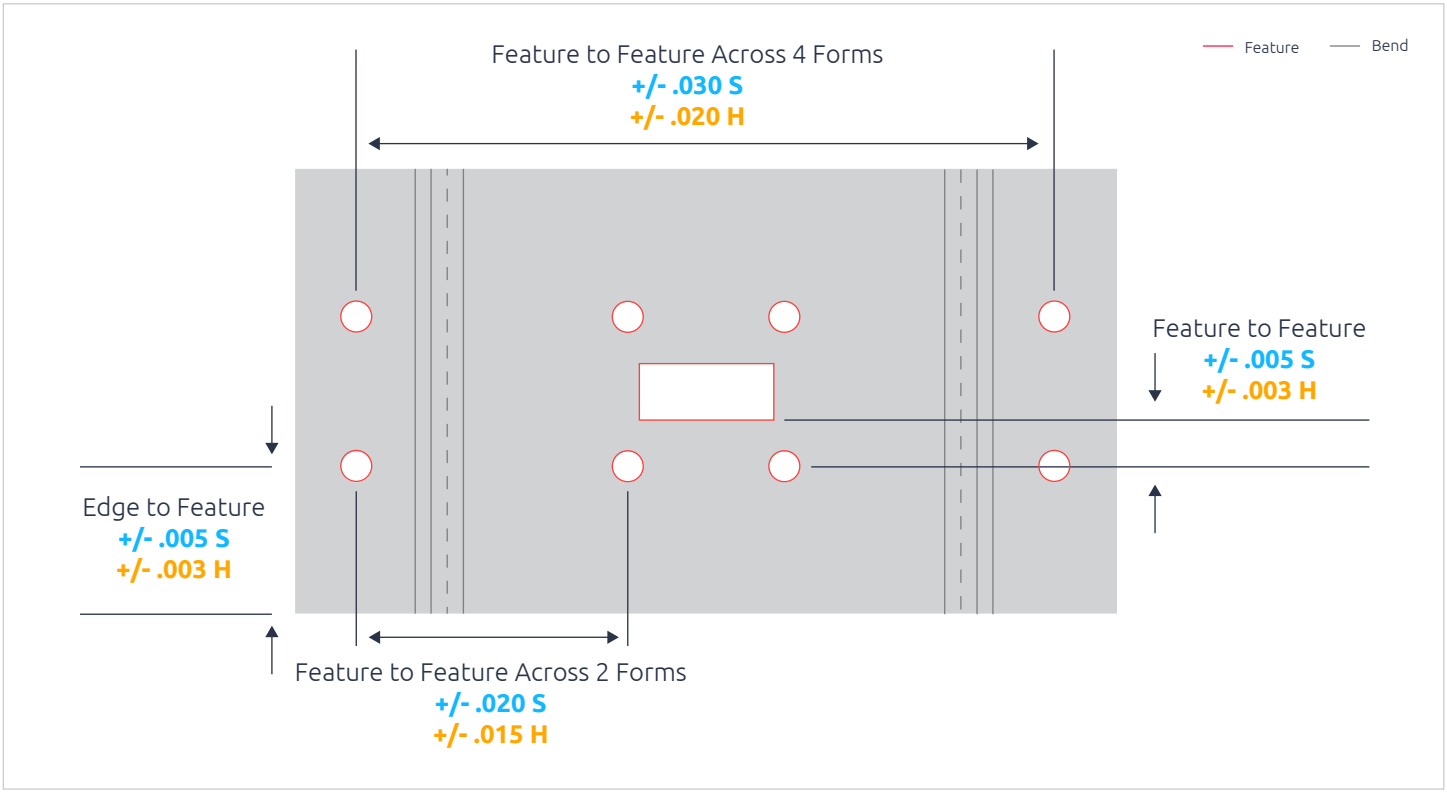
CRITICAL DIMENSIONS

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

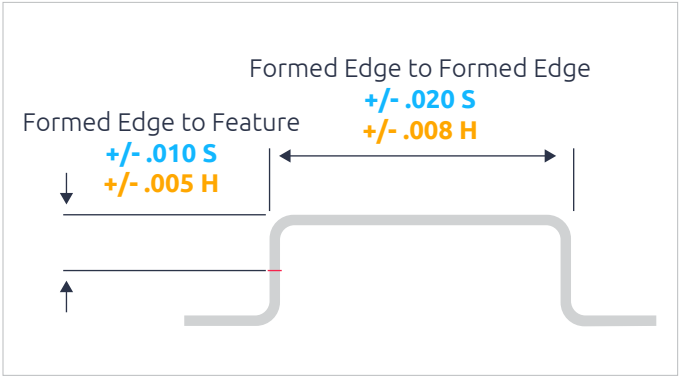
However, all holes and features punched through the sheet can introduce stress into the sheet metal. If the part has a closely spaced perforated pattern or formed features such as dimples or counter sinks, the result can cause the sheet to warp and distort. This can cause unwanted variation between holes 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 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@CADREX.COM