

Sheet Metal Fabrication Tolerances

Although the machinery and tooling will repeat within .004", it is a mistake to simply engineer all mating parts expecting +/- .005" accuracy. Such overkill forces additional labor in sorting and inspection. The result of tolerances that are too tight is simply higher cost and lower productivity. Correctly toleranced parts still have excellent fit and function, with the added benefit of manufacturing efficiency. Below is an image for VIP's recommended default sheet metal tolerances.

[Hole Size](#) [Hole to Hole](#) [Hole to Edge](#) [Hole to Bend](#) [Bend to Bend](#)

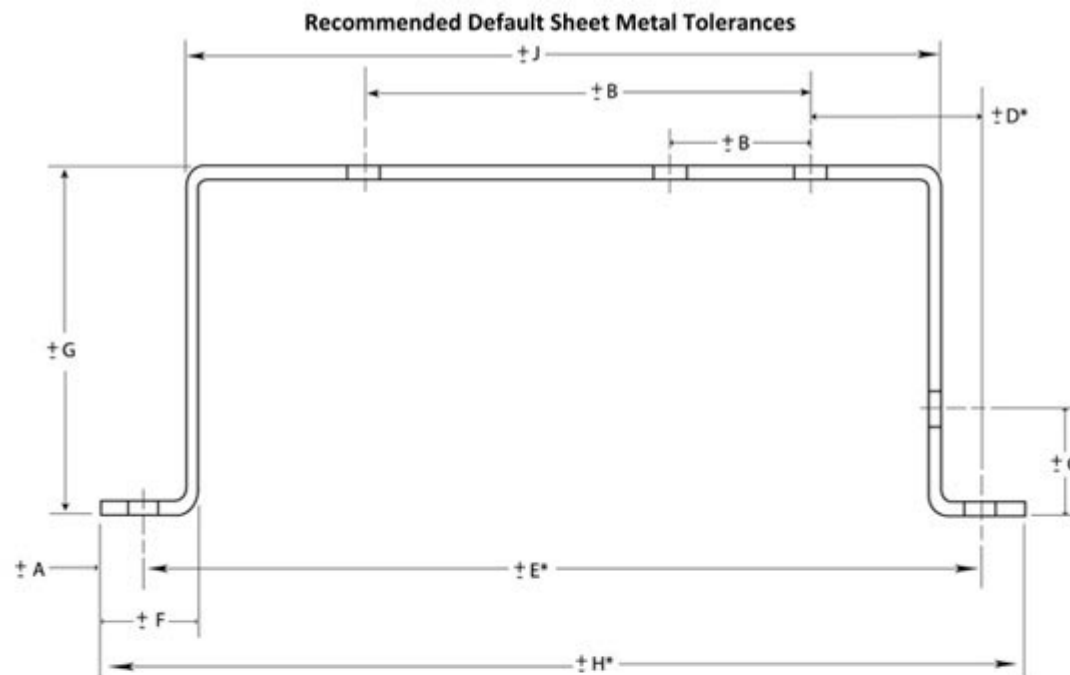
Click on Above Image to Open VIP's Recommended Tolerance Document

These tolerances are recommended and are best practices. We can obtain tighter tolerances, so feel free to [contact us](#) for more information or go to our [capabilities](#) page to see the services VIP has to offer.

Holes are produced by forcing a punch tool through the [sheet metal](#) in order to rip out a slug. When the punch retracts the slug remains stuck in the die tool and a hole is left in the [sheet metal](#). 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. Vista Industrial Products, Inc. has an extensive library of tooling, but we do not stock all possible tool sizes. This is why we look to the designers, engineers and draftsmen to give us a tolerance range that allows us some flexibility in using tooling that we have in house. When that is not possible, a capital investment in new tooling is required, or parts can be manufactured on our lasers. Generally speaking, +/- .003" is a reasonable and functional tolerance.

[Back to Top](#)

Accuracy of the distance from one hole to another is dependent primarily upon the machinery used to process the sheet. Vista Industrial Products, Inc. has equipment that 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, which can cause unwanted variation between holes or features. If this condition exists, a greater tolerance should be applied to certain areas surrounding this characteristic.

**Default Tolerances (Millimeters)**

DIM	Tolerance	Description
A	0.12	Sheared edge to hole
B	0.12	2 Holes on one surface
C	0.25	Formed edge to hole
D*	0.75	Holes across 2 bends
E*	0.75	Holes across 4 bends
F	0.25	Sheared edge to bend
G	0.25	Across 2 bends
H*	0.75	Formed part
J	0.25	Across 2 bends

Default Tolerances (Inches)

DIM	Tolerance	Description
A	0.005	Sheared edge to hole
B	0.005	2 Holes on one surface
C	0.010	Formed edge to hole
D*	0.030	Holes across 2 bends
E*	0.030	Holes across 4 bends
F	0.010	Sheared edge to bend
G	0.010	Across 2 bends
H*	0.030	Formed part
J	0.010	Across 2 bends

Noted dimensions are to be taken while the part is in the restrained condition! Noted dimensions are for parts within a 12" envelope.

* Dimensions D, E & H are not a recommended form of dimensioning!

[Back to Top](#)

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 ± 0.010 " hole-to-edge. Tolerances of ± 0.005 " should be used only when it is absolutely necessary.

[Back to Top](#)

Several factors have been introduced leading up to this stage in the fabrication process.

The deburring process may remove $.003$ " when cosmetic appearance is a priority.

Precision press brakes will position and repeat within the ± 0.002 " 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.

[Back to Top](#)

Considering the variables that effect “hole to bend” tolerance, now multiple material surfaces and thickness are introduced. Whenever possible, engineers should allow +/- .020” bend-to-bend. Resort to +/- .010” only when absolutely necessary.

Pages most viewed from here...

[Back to Top](#)

[Home](#) | **[Capabilities](#)** | **[Blog](#)** | **[Privacy Statement](#)** | **[Jobs](#)** | **[Site Map](#)** | **[FAQs](#)** | **[Contact Us](#)** | 760-599-5050

Vista Industrial Products, Inc. Est. 1956 | 3210 Executive Ridge (North County San Diego) Vista, California 92081
Updated January 2023 © Copyright