KOMACUT
LASER CUTTING PARTS ON DEMAND

Designing Sheet Metal Components Using Laser Cutting and CNC Sheet Bending

Product designs begin with CAD files and in order to get the parts or products you desire; those CAD files should follow specific design requirements. Placing a bend too close to a hole for example can result in fractures or deformed pieces during the sheet metal fabrication. Designing something that isn't manufacturable is not uncommon. In fact, some research suggests that ***30-50% of manufacturer's design time is spent correcting errors, 24% of which are related to manufacturability.*** So, using a Design for Manufacturing (DFM) strategy increases the chances of getting the design right from the start, saving you both time and money.

Tolerance

Most custom fabricators will tell you that tolerances are the most important factor when designing for precision sheet metal fabrication. Tolerance refers to the allowable or acceptable variation between the design and the finished product or part. The tolerance allows for a degree of variation that will not impact function, structure, or general acceptability of the product. Achieving the correct tolerance is vital and can make the difference between something that aligns properly for example, and something that doesn't.

While modern sheet metal fabrication techniques and equipment can achieve high tolerances, keep in mind that extremely tight tolerances can make a process more difficult or even impossible. Aiming for very tight tolerances can increase costs so a best practice is to design for the tightest tolerances required for the part or product instead of the tightest tolerance achievable. There is no point in paying for a very tight tolerance if it's not actually required for the part.

Different processes during fabrication have different tolerances. It's important to understand how they impact your design. Common tolerances are included in the table below but its important to check with your custom fabricator to confirm their standards.

CONFIRM TOLERANCE, CLEARANCES, AND DESIGN STANDARDS WITH YOUR FABRICATOR
All of the numbers we've included in this chapter are meant as guidelines only. Always check with your custom fabricator to get specific tolerances from them as it varies by material type, fabricator, equipment, tools, and other factors. Tolerances and clearances will be adjusted on a project by project basis to accommodate your needs.

Tolerances Guideline for Sheet Metal Fabrication *

	Tolerance Guidelines*
Forming or bending	+/- 0.508 mm (0.020")
Bend to hole or feature	+/- 0.254 mm (0.010")
Diameters with inserts	+/- 0.0762 mm (0.003")
Angularity	+/- 1˚
Holes	+/- 0.127 mm (0.005")
Edge to edge	±0.127 mm (0.005")
Edge to hole	±0.127 mm (0.005")
Hole to hole	±0.127 mm (0.005")
Hole to hardware	±0.254 mm (0.010")
Edge to hardware	±0.254 mm (0.010")
Hardware to hardware	±0.381 mm (0.015")
Bend to hole	±0.381 mm (0.015")
Bend to hardware	±0.381 mm (0.015")
Bend to edge	±0.254 mm (0.010")
Bend to bend	±0.381 mm (0.015")

*Tolerances Guideline for Sheet Metal Fabrication **

FACTORS THAT EFFECT TOLERANCES DURING FABRICATION

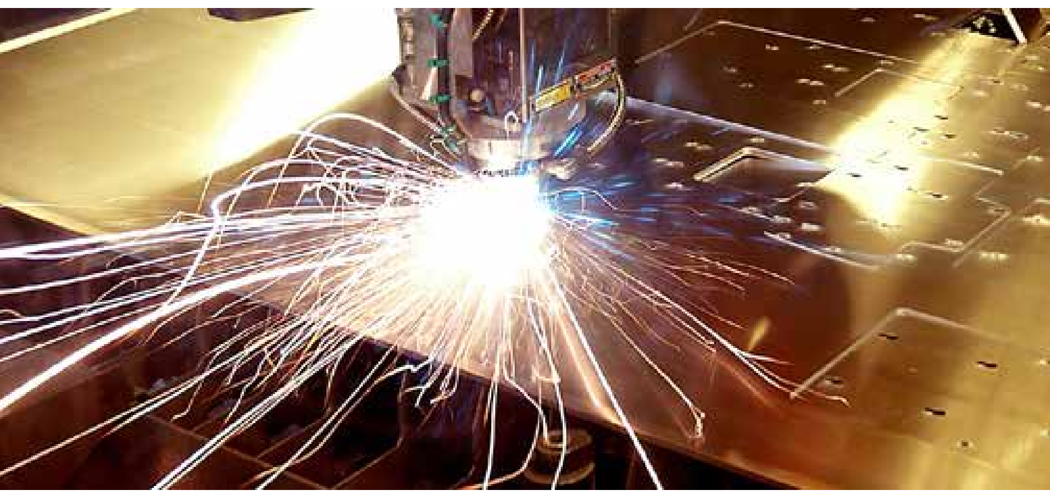
There are many factors which effect actual tolerances, which reinforces the need to confirm with your fabricator before beginning your design.

- The more processes a part undergoes, the more difficult it is to achieve tight tolerances
- The material you choose, including type and thickness
- Some equipment used for sheet metal fabrication can achieve tighter tolerances than others
- Custom fabrication and assembly companies have different capabilities and equipment to accommodate tolerance requirements
- CAD designs should be created using Design for Manufacturing standards, so tolerances are realistic

LASER CUTTING TOLERANCE

Laser cutting is often a best choice for parts requiring precision sheet metal fabrication. Lasers can operate with tolerances from +/- 0.127mm of precision and can accommodate material up to 20 mm (0.78") thick.

- Hole sizes are limited by the size of the laser beam
- Tapered cuts or other 3D features are not possible with laser cutting



CNC SHEET BENDING TOLERANCE

Bending sheet metal with press brakes to achieve the desired shape. Tooling is required although not all bending processes require custom dies. The process of bending does require manual operators and different types of equipment can achieve different bend radius tolerances.

- You can't typically achieve a true 90˚ corner. Corners will have radius
- Bend angles standard tolerance of ± 1˚
- Bend length tolerance are typically ±0.25 mm (0.010")

Design Standards

When designing for laser cutting and CNC bending processes for your sheet metal fabrication, it's important to keep common standards top of mind for different design elements.

Material thickness

The material thickness not only determines the strength and weight of your part, but also minimum bend radii, hole and slot sizes, flange length, and other important design features. Parts should keep a uniform thickness throughout.

Bends and Relief Radius

Designing bends with appropriate radii and flange length helps reduce any spring-back effect and minimizes the chance of torn metal. A standard die sets include 0.80 mm (0.030") tools for internal corners so you may want to design to this standard or ask your fabricator what tools they have available. It's best to design bends on the same plane in the same direction to avoid part reorientation during the process. Using a consistent bend radius can also help reduce costs. Also keep in mind that it is difficult to make accurate small bends to thick material and it's best not to bend very large parts with small flanges.

Offsets use a double bend to create a Z-like shape. It's common when creating tiers in sheet metal or for creating brackets or clamps. Offsets should follow the same design guidelines as other bends.

- The radius of an inside bend should be at least equivalent to the materials thickness to avoid fractures or distortion. For low carbon steel, the minimum radius can be ½ the material thickness or 0.80 mm (0.03") – whichever is greater.
- The flange length of a bend should be at least 3 times the material thickness
- For offset bends keep parallel planes at least 2 times material thickness from each other

CLEARANCES FOR BENDS

	Minimum Distance Guidelines*
Between a curl and an internal bend	±6 times the curl's radius plus the thickness of the material
Between a curl and an external bend	±9 times the curl's radius plus the thickness of the material
Between a hem and an external bend	±8 times the material thickness
Between a hem and an internal bend	±5 times the material thickness
Between a counterbore and a bend	±4 times the material thickness plus bend radius
Between a countersink and a bend	±3 times the material thickness
Between hole and a bend	±2.5 times the material length + radius of the bend
Between a slot and a bend	±4 times the material thickness plus the radius of the bend
Between an extruded hole and a bend	±3 times the material thickness plus the bend radius
Between semi-pierced hole and a bend	±3 times the material thickness plus the bend radius
Between a notch and a bend in a perpendicular plane	±3 times the material thickness plus the bend radius
Between a notch and a bend in a parallel plane	±8 times the material thickness plus the bend radius
Between a dimple and a bend	±2 times the material thickness plus inside radius of the dimple plus the radius of the bend
Between rib to a bend perpendicular to the rib	±2 times material thickness plus the radius of the rib plus the radius of the bend

Curls

Curls provide a circular roll to the edge of the sheet to provide strength and minimize exposure of the sharp edge. Unlike a hem, the edge of a curl turns in on itself. Curls can be off center or on center.

- The outside radius of a curl should be at least twice the thickness of the material being used
- Using a radius 2 times the material thickness will create a curl opening radius equal to the material thickness
- The opening of the curl should be at least equivalent to the material thickness

CLEARANCES FOR CURLS

	Minimum Distance Guidelines*
Between a curl and a hole	The curl's radius plus the thickness of the material
Between a curl and an internal bend	±6 times the curl's radius plus the thickness of the material
Between a curl and an external bend	±9 times the curl's radius plus the thickness of the material

Countersinks and counterbores

Countersinks and counterbores are used to create a flush surface where parts are fastened together. A countersink creates a cone shaped hole typically designed to accommodate a screw. Counterbores however are drilled straight down and have a flat bottom, often to accommodate bolts or nuts. Neither are practical for thin materials.

- Countersinks can be no more than 0.6 times the thickness of the material
- A countersink and a fastener should have at least 50% contact

CLEARANCES FOR COUNTERSINKS AND COUNTERBORES

	Minimum Distance Guidelines*
Between countersink and edge	±4 times the material thickness
Between a countersink and a bend	±3 times the material thickness
Between countersinks	±8 times the material thickness
Between counterbore and edge	±4 times the material thickness
Between a counterbore and a bend	±4 times the material thickness plus bend radius
Between counterbores	±8 times the material thickness

Hems

Hems provide strength to the hemmed edge and eliminate sharp edges but also add weight to the part. Trying to achieve a flat hem can cause fractures to the material so design for open or teardrop hems instead.

- Open hems should have an inside diameter at least equal to the material thickness. Large diameters can lose their shape.
- The return flange of an open hem should be at least 4 times the material thickness
- Teardrop hems should have an inside diameter at least equal to the material thickness
- Openings for a teardrop hem should be at least ¼ of the material thickness
- The return flange of a teardrop hem should be at least 4 times the material thickness

CLEARANCES FOR HEMS

	Minimum Distance Guidelines*
Between a hem and a hole	±2 times the material thickness plus the radius of the hem
Between a hem and an internal bend	±5 times the material thickness
Between a hem and an external bend	±8 times the material thickness

Holes and Slots

Holes should always be designed with a diameter at least equivalent to the thickness of the material. Smaller holes can cause excessive burring and impact the life of the part. Spacing holes appropriately also helps them hold their shape during further processing steps.

- Holes and slots should have a diameter at least as large as the thickness of the material or 1.00 mm (0.04") – whichever is greater. For alloy or stainless steel, it should be at least 2 times the material thickness.
- Materials with higher strengths require that holes and slots have a larger diameter

CLEARANCES FOR HOLES, AND SLOTS

	Minimum Distance Guidelines*
Between hole and a bend	±2.5 times the material length + radius of the bend
Between a slot and a bend	±4 times the material thickness plus the radius of the bend
Between a hole or slot and edge	±2 times the material thickness
Between holes	±2-3times the material thickness
Between an extruded hole and an edge	±3 times the material thickness
Between an extruded hole and a bend	±3 times the material thickness plus the bend radius
Between semi-pierced hole and a bend	±3 times the material thickness plus the bend radius
Between semi-pierced holes	±8 times the material thickness

Notches and Tabs

Notches or tabs can sometimes be part of the piece itself or be intended for further bending or for joining. Notches are created by removing a section of the sheet metal from the outside edge. Tabs on the other hand are protrusions from the sheet metal edge.

- Notches must have a thickness of at least 1mm (0.04") or equivalent to the material thickness (whichever is greater)
- The length of a straight or radius end notch should be no more than 5 times material thickness
- The length of a V notch should be no more than 2 times its width
- Tabs must have a width of at least 3.2mm (0.126") or two times the material thickness (whichever is greater)
- Tab depth should be no more than 5 times the width of the tab
- Notch corner radius should be 0.5 times material thickness

CLEARANCES FOR NOTCHES AND TABS

	Minimum Distance Guidelines*
Between a notch and a bend in a perpendicular plane	±3 times the material thickness plus the bend radius
Between a notch and a bend in a parallel plane	±8 times the material thickness plus the bend radius
Between a notch and a hole	±1.2 times the material thickness
Between notches	±3.200 mm (0.125") or two times the material thickness (whichever is greater)
Between tabs	±1mm (0.04") or the material thickness (whichever is greater)

Corner Fillets

Corner fillets create rounded edges to eliminate sharp corners.

- Corner fillets should be ½ the material thickness

Relief Cuts

Relief cuts can be used to prevent overhangs which are more common on thick parts with a small bend radius. They can also help prevent tearing from bends placed close to an edge

- A relief cut should be at least equal to the material thickness in width
- Length of a relief cut should be longer than the bend radius

Welding

Welding of some materials requires preparation, including grinding, which should be considered when designing. When designing welds, using very tight tolerances minimizes the need for the welder to use wire.

- Hand welding should be for gauges greater than 20 gauge

CLEARANCES FOR WELDING

	Minimum Distance Guidelines*
Between a weld and the edge	±2 times the diameter of the spot weld
Between welds	±10 times the material thickness

Dimples

- Dimples should have a maximum diameter of 6 times material thickness
- Dimple depth should be no more than ½ the inside diameter of the dimple

CLEARANCES FOR DIMPLES

	Minimum Distance Guidelines*
Between a dimple and the edge	±4 times the material thickness plus inside radius of the dimple
Between a dimple and a bend	±2 times the material thickness plus inside radius of the dimple plus the radius of the bend
Between a dimple and a hole	±3 times the material thickness
Between dimples	±4 times the material thickness plus inside radius of the dimple

Gussets

Gussets can add strength to a flange without any welding. They generally require custom tooling to produce.

CLEARANCES FOR GUSSETS

	Minimum Distance Guidelines*
Between a gusset and the edge	±8 times the material thickness plus the radius of the gusset
Between a gusset and hole	±8 times the material thickness

Lances

Lances involves cutting and bending on a piece without removing any material. It changes the shape and is often used to allow air flow through a part (vents and louvers). Special tooling is often required.

- Open lances should have a minimum width of 3.00mm (0.125") or 2 times the material width (whichever is greater)
- The maximum width of an open lance is 5 times the width
- Closed lances should have a minimum width of 1.60mm (0.06") or 2 times the material width (whichever is greater)
- The maximum height of a closed lance is 5 times the material thickness at a 45˚ angle

CLEARANCES FOR LANCES

	Minimum Distance Guidelines*
Between a lance and a hole	±3 times the material thickness

Ribs and Embossments

- The inside radius of a rib is no more than 3 times the material thickness
- Maximum depth for a round embossment or rib is equal to its inside radius
- Maximum depth for a flat embossment is equal to its inside radius plus the outside
- Maximum depth for a v embossment is equal to 3 times the material thickness

CLEARANCES FOR RIBS

	Minimum Distance Guidelines*
Between rib or embossment to a hole	±3 times material thickness plus the radius of the rib
Between rib to an edge in a perpendicular plane	±4 times material thickness plus the radius of the rib
Between rib to an edge in a parallel plane	±8 times material thickness plus the radius of the rib
Between rib to a bend perpendicular to the rib	±2 times material thickness plus the radius of the rib plus the radius of the bend
Between parallel ribs	±10 times material thickness plus the radii of the ribs

**Please use these numbers as guidelines only and always check with your fabricator for their recommendations before completing your design.*