Xometry

Sheet Metal Fabrication Design Guide

Table of Contents

Overview

Sheet metal fabrication is the process of forming parts from a metal sheet by punching, cutting, stamping, and bending. 3D CAD files are converted into machine code, which controls a machine to precisely cut a flat pattern, which can be formed into the final part using press brakes and die tools. Parts can be finished with inserts, welding, surface treatments, and part marking.

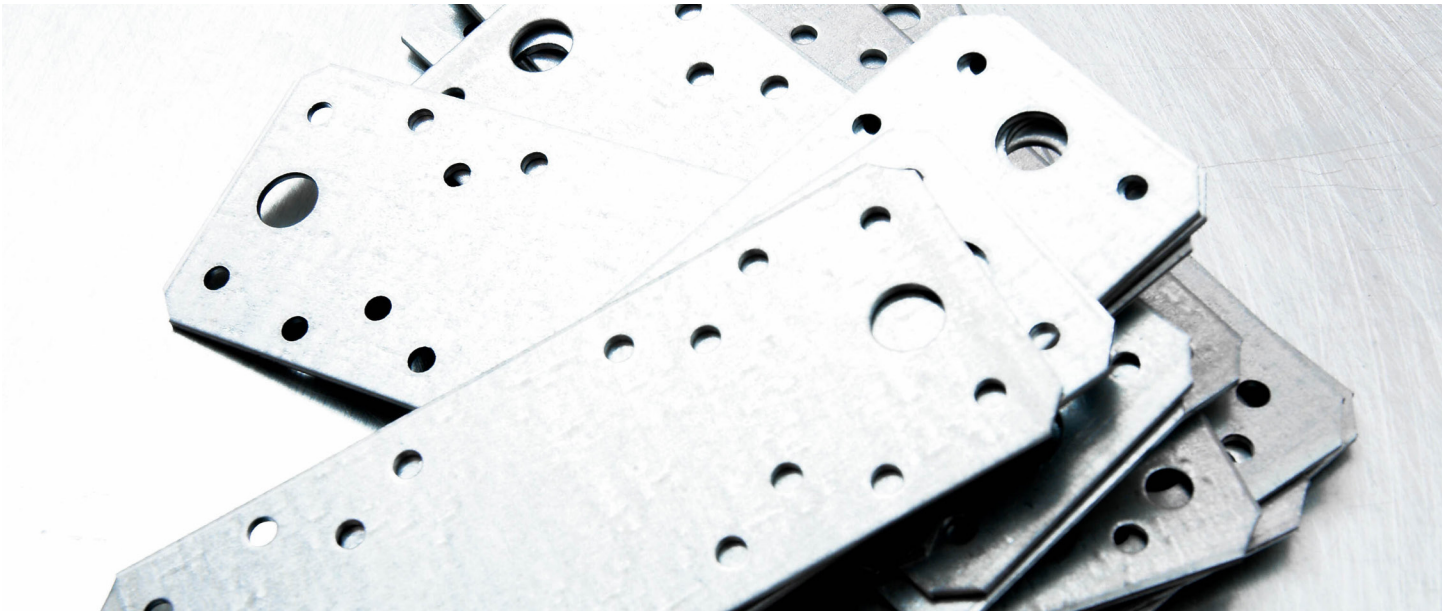
Sheet metal parts are known for their durability, which makes them great for end-use applications (e.g. chassis). Parts can be very inexpensive if they only require a 2D cut with a laser, plasma, or waterjet cutting tool. Because standardized tooling can make most sheet metal bends and features, this process can scale significantly and have a low price per part with high throughput.

Because parts are formed from a single sheet of metal, designs must maintain a uniform thickness. Factors like bend radius and feature-to-feature cut distances should be considered. Be sure to follow design requirements and tolerances in this guide to ensure parts fall closer to design intent.

Design Checkpoint:
Why Use Sheet Metal?

- ✓ *Often cheaper than milling*
- ✓ *Scales in volume production*
- ✓ *High strength-to-weight ratio*
- ✓ *Effective for large parts*

Tolerances



Xometry's Sheet Metal Tolerance Offerings

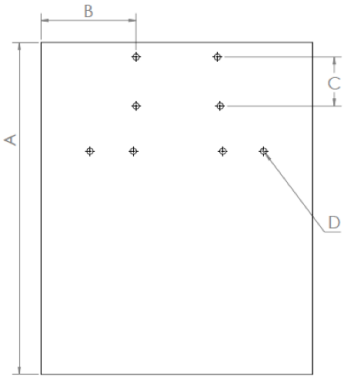
If a drawing or specification sheet has not been provided by the customer, Xometry will manufacture the product from the model to the specifications listed below:

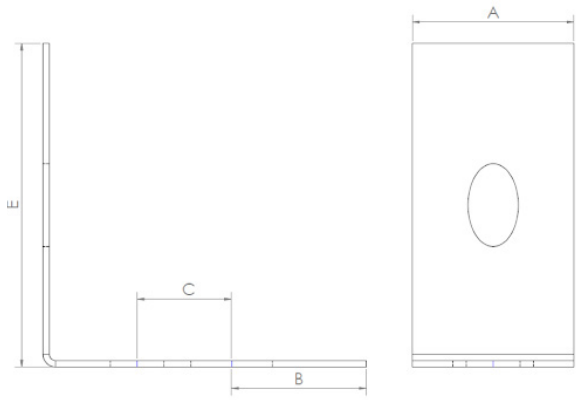
- Forming and bending: +/- 0.020"
- Bend to hole or feature: +/- 0.010"
- Linear dimensions excluding locations to bends: +/- 0.005"
- Diameters with inserts: +0.003" / -0.000"
- Angularity: +/- 2°
- Surface roughness (blank material): Ra 125 µin maximum
- Surface roughness (timesaver): Ra 100 µin maximum
- Sharp edges will be broken and deburred by default. Critical edges that must be left sharp should be noted and specified on a print.

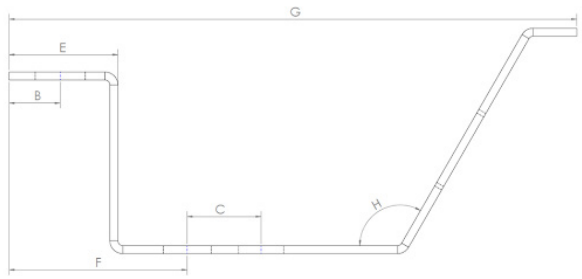
Tip: *Certain sheet metal designs like hems, curled flanges, and rolled sheets, stamped parts, and welded assemblies require custom tooling and will be flagged for a manual quote. If your part includes these features, please allow the Xometry manual quote team to review and provide you with an accurate cost and lead time.*

Xometry can provide 3 tiers of tolerance offerings based on the design of the desired part.

Single Surface		
Edge to Edge	+/-0.005	A
Edge to Hole	+/-0.005	B
Hole to Hole	+/-0.005	C
Hole to Diameters	+/-0.005	D
Bend to Edge/Hole	+/-0.010	E
Multiple Surface		
Hole to Hole/Edge: Edge to Edge	+/-0.030	F
Over Formed Part	+/-0.030	G
Bend Angle	+/-1 deg	H







Flat Parts

Flat sheet metal parts that utilize stock material gauge thicknesses and require no bending or milled features. All flat sheet metal parts can meet a +/-0.005” (+/- 0.127mm) tolerance.

Simple Bend / One Surface

When designing parts with some type of bend with a standard bend radius, sheet metal tolerances are required to open.

For simple bent parts, Xometry can offer a +/- 0.010” (+/- 0.254mm) tolerance for dimensions across a single bend. The bend radius itself is constrained to a +/- 1.0° tolerance.

Multiple Surface Bends

For dimensions measured over multiple bends, Xometry can offer a +/- 0.030” (0.762mm) tolerance.

Typical Stock Size and Gauge

Common Sheet Metal Materials:

- Aluminum alloy
- Steel
- Stainless Steel
- Brass or Bronze
- Copper

Raw stock material comes in various shapes and sizes. Most cutting platforms, such as waterjet tables or mechanical shears, can accommodate 6’ to 10’ lengths of materials. The table to the right shows some standard gauges, or stock material thicknesses, from different materials.

	Steel		Galvanized		Stainless		Aluminum	
Gauge	in.	mm	in.	mm	in.	mm	in.	mm
3	0.2391	6.07						
4	0.2242	5.69						
5	0.2092	5.31						
6	0.1943	4.94					0.1620	4.11
7	0.1793	4.55			0.1875	4.76	0.1443	3.67
8	0.1644	4.18	0.1681	4.27	0.1719	4.37	0.1285	3.26
9	0.1495	3.80	0.1532	3.89	0.1563	3.97	0.1144	2.91
10	0.1345	3.42	0.1382	3.51	0.1406	3.57	0.1019	2.59
11	0.1196	3.04	0.1233	3.13	0.1250	3.18	0.0907	2.30
12	0.1046	2.66	0.1084	2.75	0.1094	2.78	0.0808	2.05
13	0.0897	2.28	0.0934	2.37	0.0940	2.39	0.0720	1.83
14	0.0747	1.90	0.0785	1.99	0.0781	1.98	0.0641	1.63
15	0.0673	1.71	0.0710	1.80	0.0700	1.78	0.0570	1.45
16	0.0598	1.52	0.0635	1.61	0.0625	1.59	0.0508	1.29
17	0.0538	1.37	0.0575	1.46	0.0560	1.42	0.0450	1.14
18	0.0478	1.21	0.0516	1.31	0.0500	1.27	0.0403	1.02
19	0.0418	1.06	0.0456	1.16	0.0440	1.12	0.0360	0.91
20	0.0359	0.91	0.0396	1.01	0.0375	0.95	0.0320	0.81
21	0.0329	0.84	0.0366	0.93	0.0340	0.86	0.0280	0.71
22	0.0299	0.76	0.0336	0.85	0.0310	0.79	0.0250	0.64
23	0.0269	0.68	0.0306	0.78	0.0280	0.71	0.0230	0.58
24	0.0239	0.61	0.0276	0.70	0.0250	0.64	0.0200	0.51
25	0.0209	0.53	0.0247	0.63	0.0220	0.56	0.0180	0.46
26	0.0179	0.45	0.0217	0.55	0.0190	0.48	0.0170	0.43
28	0.0149	0.38	0.0187	0.47	0.0160	0.41	0.0126	0.32

Standard Gauges and Stock Material Thicknesses

Stock Material Thickness Tolerances

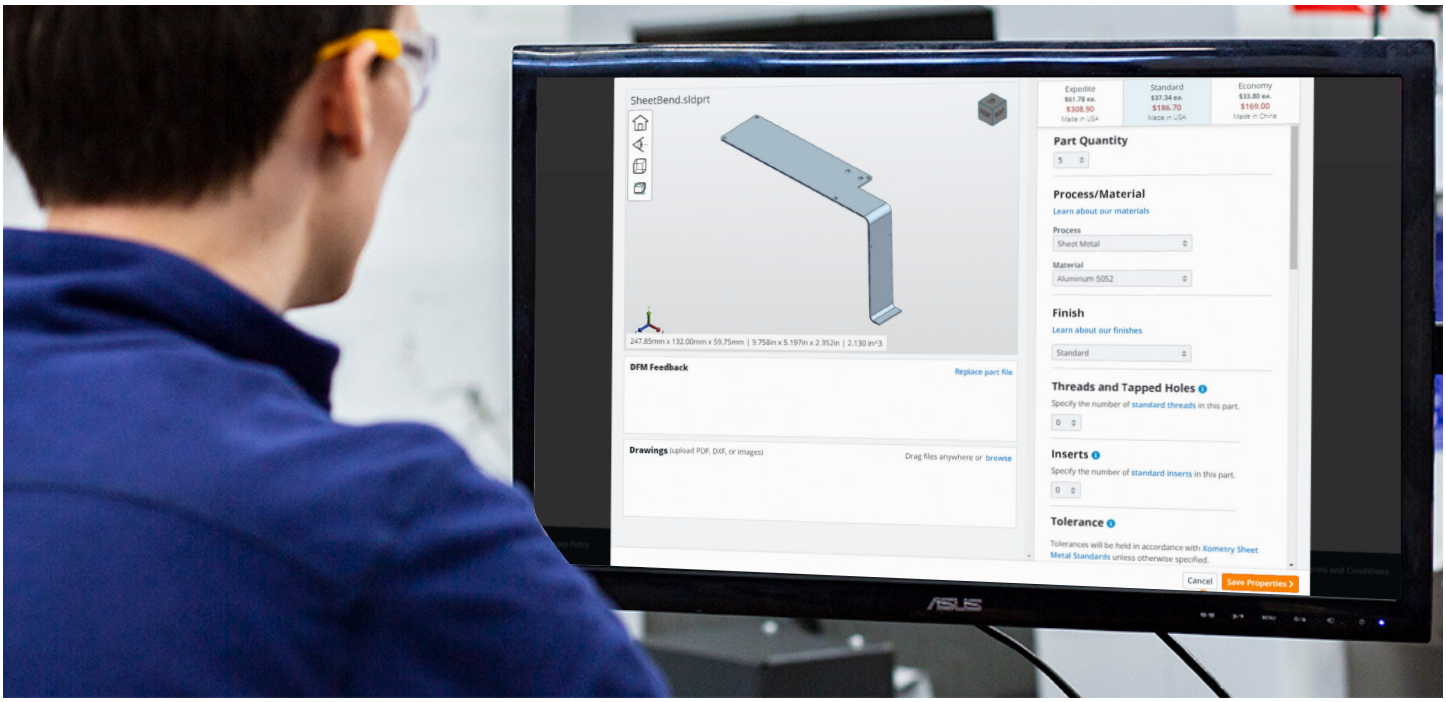
Sheet metal material stock has general tolerances for thickness. The following table is a quick reference for standard aluminum 5052 material thickness tolerances.

5052-H32 Aluminum Alloy Sheet Thickness Tolerances

Thickness (in.)	Tolerance range (+/- in.)
0.032″	-0.003″ to 0.003″
0.040″	-0.004″ to 0.004″
0.050″	-0.004″ to 0.004″
0.062″	-0.004″ to 0.004″
0.080″	-0.005″ to 0.005″
0.090″	-0.005″ to 0.005″
0.100″	-0.005″ to 0.005″
0.125″ (1/8″)	-0.006″ to 0.006″
0.190″	-0.011″ to 0.011″
0.250″ (1/4″)	-0.015″ to 0.015″
0.375″ (3/8″)	-0.020″ to 0.020″
0.500″ (1/2″)	-0.027″ to 0.027″

Standard stock material tolerances may vary between alloys and treatments. Sheet metal with tighter tolerances sheet can be specified if required.

Quoting Formats: 3D CAD and DXF Files



You can upload sheet metal design files to online quoting sites like the Xometry Instant Quoting Engine℠

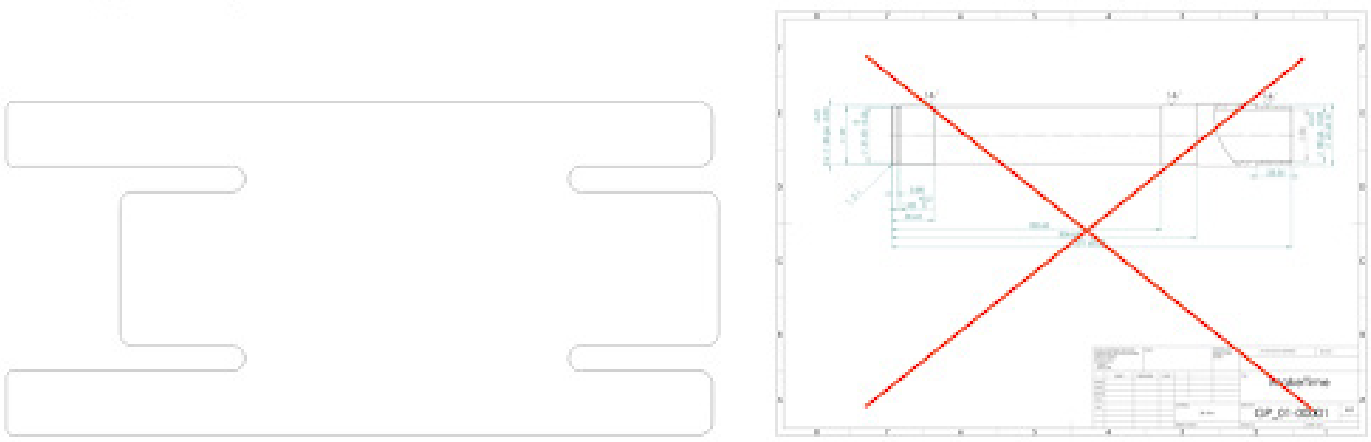
3D CAD files, such as STEP (.step, .stp), SOLIDWORKS (.sldprt), Parasolid (.x_t, .x_b), Autodesk Inventor (.ipt), Dassault Systems (.3dxml, .catpart), PTC, Siemens (.prt), and ACIS (.sat) can be used to quote and produce sheet metal parts. For bent components, it is important to provide the CAD with the bends designed in the model. Sheet metal fabricators can "unbend" the model on their end to produce the part to specifications. In some cases, a DXF file can be used for both quoting and production.

DXF (.dxf) is a popular vectored line format used for flat profile cut parts. DXF files may also be used as a flat-pattern (blanking) reference when building a bent sheet component.

You can get an instant quote using DXF files from Xometry by uploading and specifying the units and required thickness in an intuitive user interface. After you specify a thickness, you can specify material, finish, features, and inspection requirements as needed. In order to successfully upload a DXF, there are a few requirements that should be considered:

DXF Requirements

A DXF (Drawing Interchange Format or Drawing Exchange Format) is a CAD data file format for enabling data exchange between AutoCAD and other programs. DXF files are typically used for sheet metal fabrication.

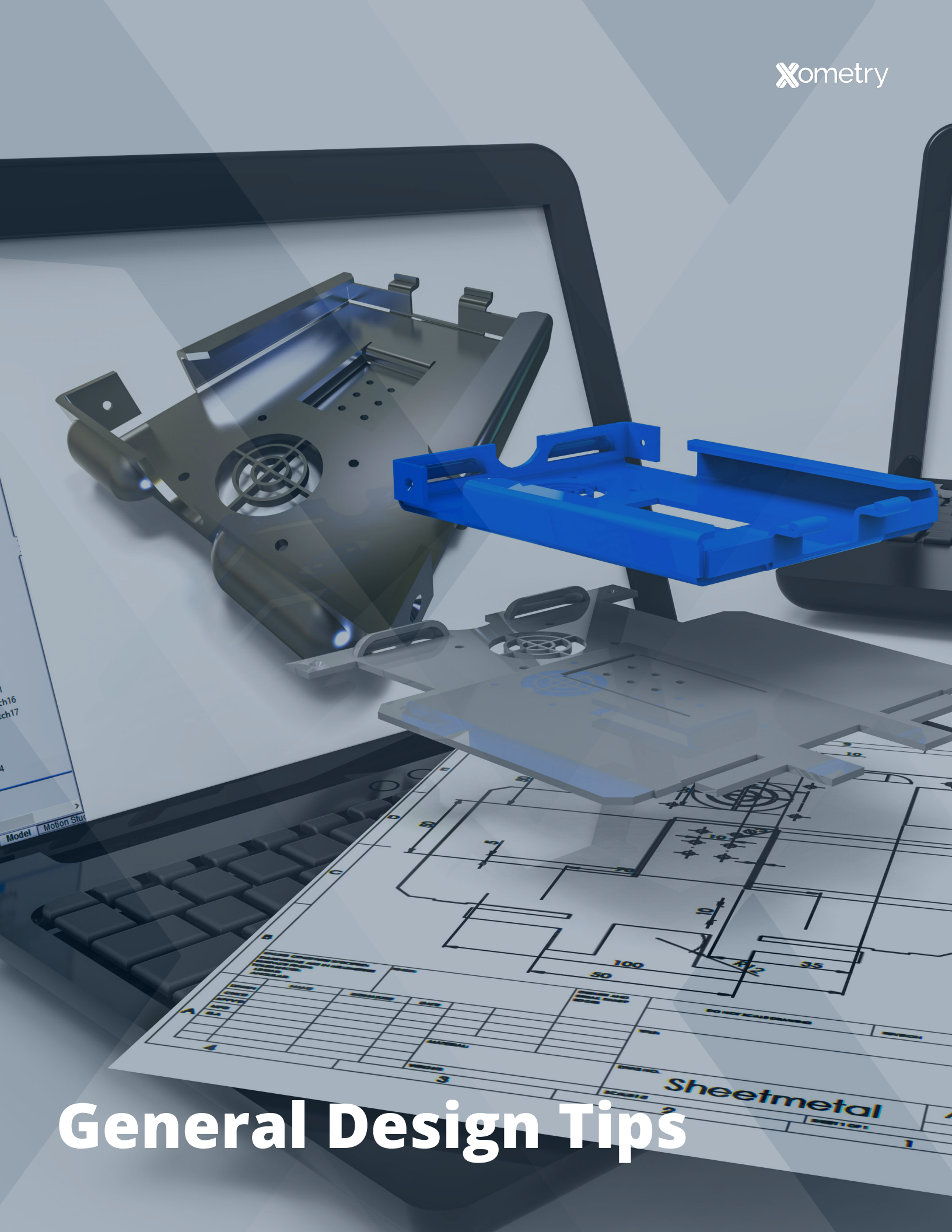


The DXF cut file is a continuous pathway perimeter or internal paths for holes and slots to guide a machine cutting operation.

The best way to get an instant quote with a DXF is to upload a cut file (and not the full drawing) to get.xometry.com/quote. Using a DXF will create a quote for a “flat cut” sheet metal part without bends. If there are bends in the part, the best way to quote is by using a 3D CAD model. A .pdf drawing can be attached to the uploaded model if further specifications are required.

To read more about Xometry DXF upload capabilities and requirements visit Xometry’s FAQ page.

Xometry

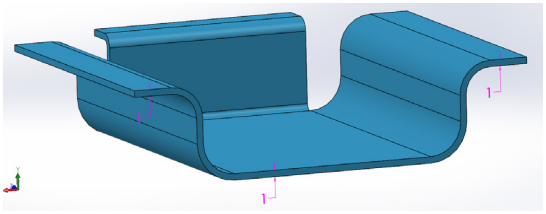


General Design Tips

Wall Thickness

Sheet metal parts are unique because they use common flat stock materials. The following design guide tips will often describe distances in material thickness, or **MT**, for relative measurements.

Material Thickness = MT



Model indicating uniform wall thickness

Wall Thickness

Parts must maintain a uniform wall thickness throughout their entirety. Generally, Xometry is capable of manufacturing bent sheet metal parts up to ¼" (6.35mm) in thickness, but this tolerance mainly depends on the geometry of the part.

All sheet metal parts start from flat metal stock and are cut and processed to specifications

> Flat cut sheet parts, such as flat brackets, can be cut thicker (above ¼") using plasma or waterjet but are typically not recommended for bending or forming.

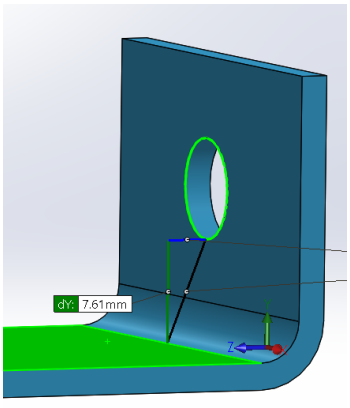
Holes and Slots

Hole and Slot Dimensions: Keep hole and slot diameters at least as large as 1X MT. Higher-strength materials may require larger diameters.

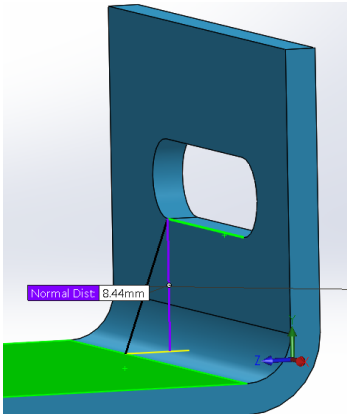
Hole and Slot Clearances: Holes and slots may become deformed when placed near a bend. Hole and slots should be at least 6X MT from each other. Holes can be at least 2.5X MT from bend edges while slots should have more clearance at 4X MT. Be sure to place both holes and slots at least 2X MT from part edges to avoid a “bulging” effect. The image below depicts the standard feature to feature minimum clearances.

Sheet Metal Feature Clearances (**Material Thickness**)					
	Countersink	Hole	Slot	Edge	Bend Edge
Countersink	8	8	8	8	8
Hole	8	6	6	2	2.5
Slot	8	6	6	2	4
Edge	8	2	2	NA	NA
Bend Edge	8	2.5	4	NA	NA

Sheet metal feature clearances cheat sheet using factors of the material thickness (MT)

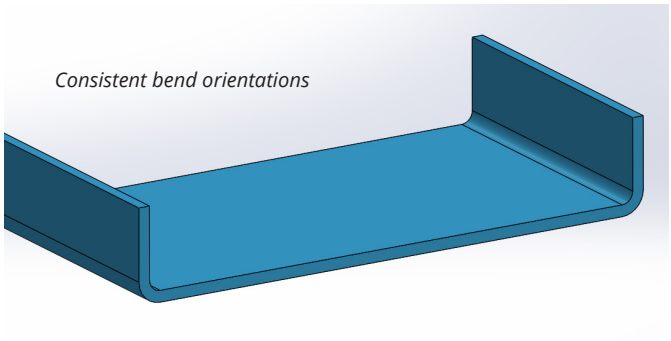


Hole-to-bend distance

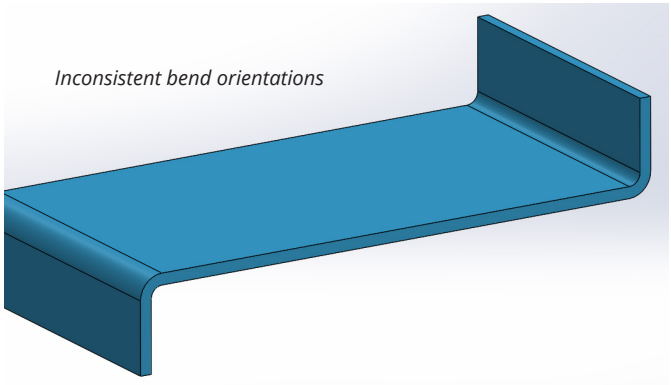


Slot-to-bend distance

Bends



Consistent bend orientations



Inconsistent bend orientations

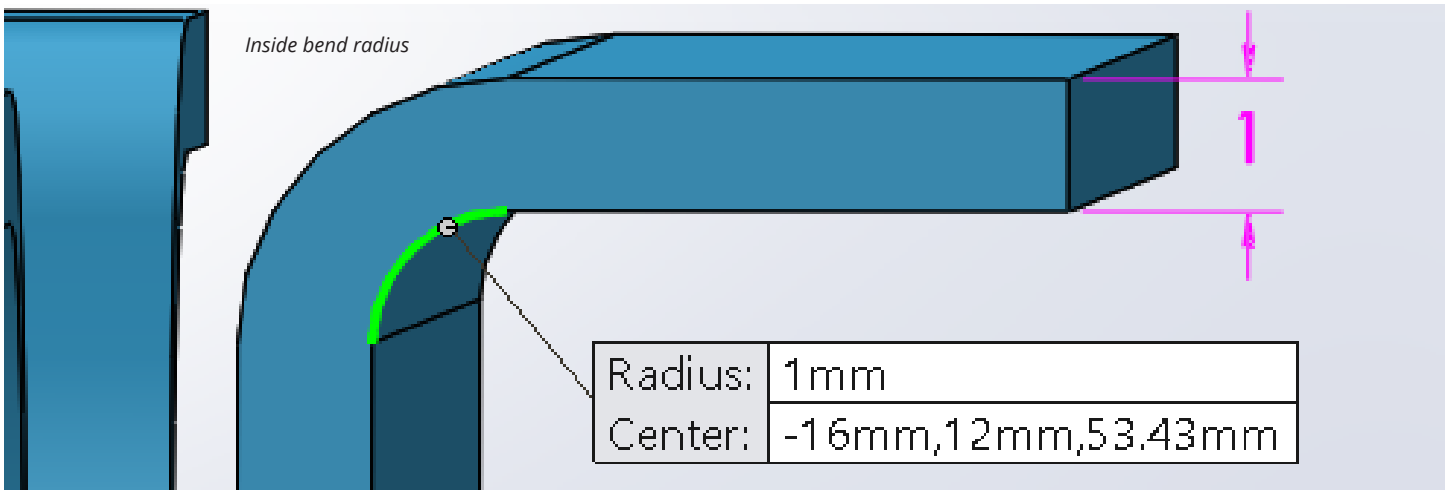
Bends

Designing Bends: Keep bend radius at least 1X material thickness to prevent parts from fracturing or having distortion.

Sheet metal brakes are used to bend the material into a part's desired geometry. Bends in the same plane should be designed in the same direction to avoid part reorientation, which will save both money and time.

Keeping a consistent bend radius will also make parts more cost-effective. Small bends to large, thick parts tend to become inaccurate, so they should be avoided if possible.

> ***Tip:*** *Be mindful of material alloys when designing sheet metal parts. Aluminum 5052 is the most common sheet material because of its ductility. Aluminum 6061, popular in CNC machining, can be brittle and crack if bent.*

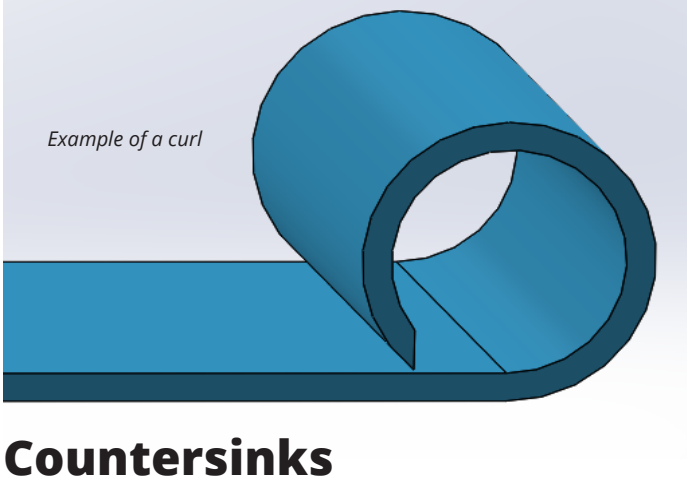


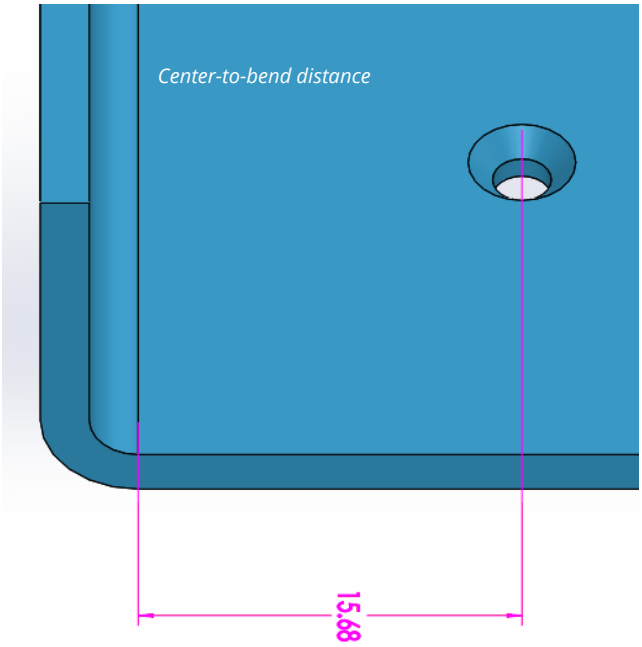
Curls and Countersinks

Curls

Designing Curls: The outside radius must be at least 2X material thickness.

Curl Clearances: Holes should be placed away from the curl at least a distance of the radius of the curl plus the material's thickness. Bends should be at least 6X material thickness plus the radius of the curl away.



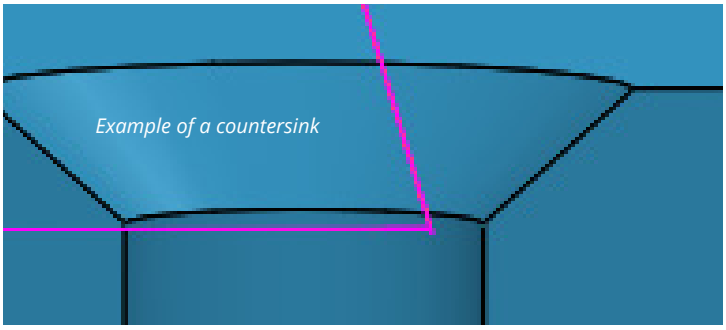


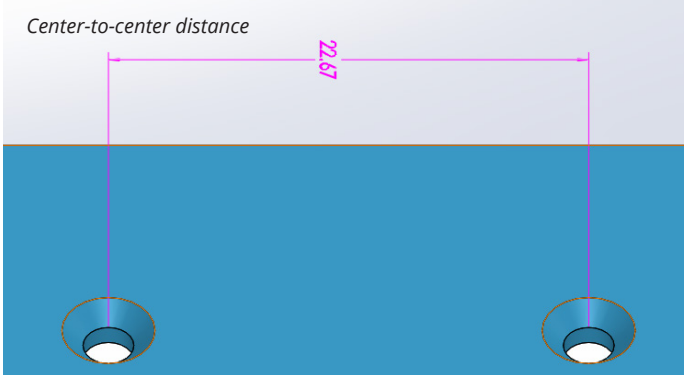
Countersinks

Countersinks are suitable for sheet metal designs and are typically produced with hand tools. Be sure to not extend the countersink more than 0.6X MT deep.

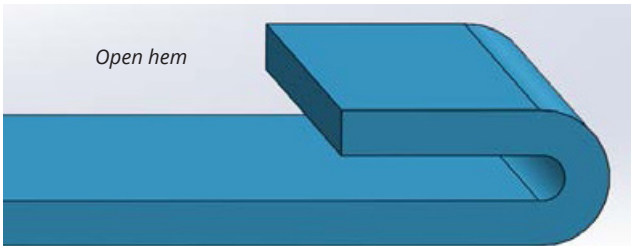
Countersink Clearance: Countersinks must be at least 8X MT from each other, 4X MT from an edge, and 3X MT from a bend.

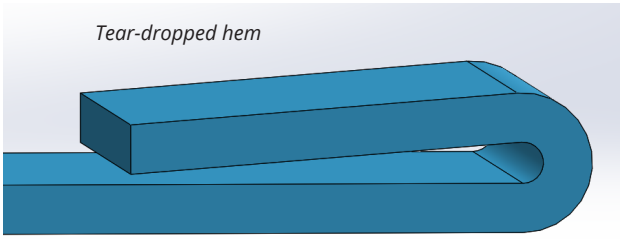
> ***Warning:*** *Counterbores in sheet metal should be avoided because they require machining operations. In some cases, they can be formed but require specialized punch tooling.*

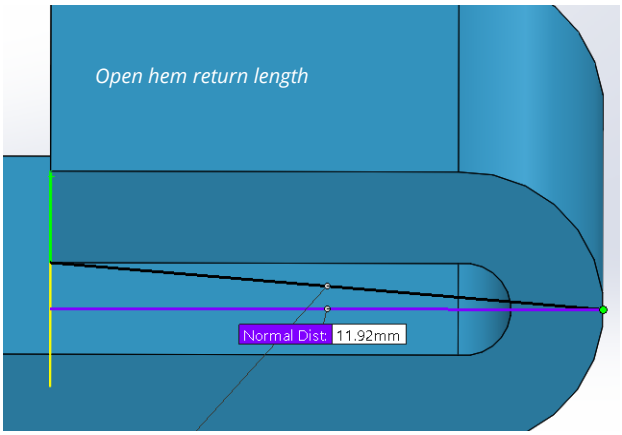




Hems







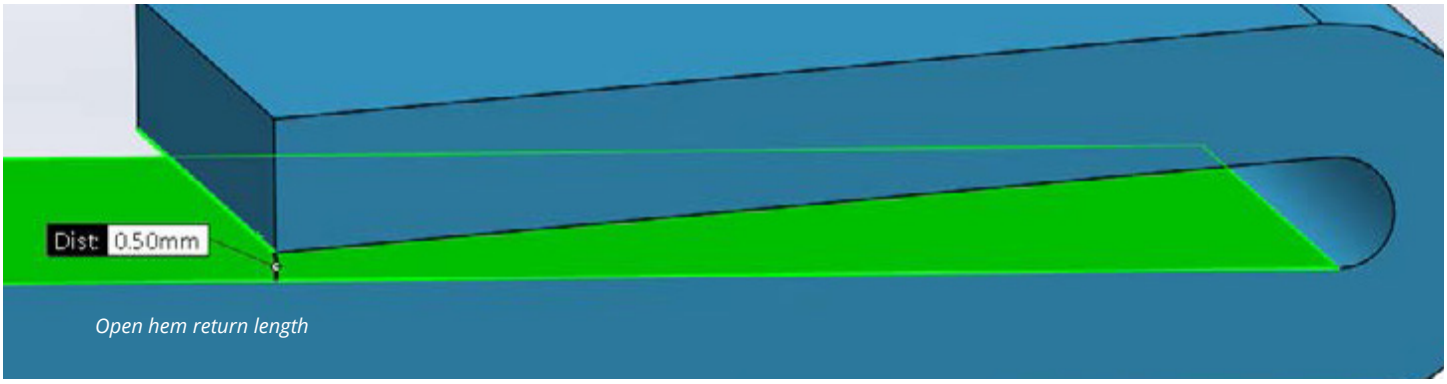
Hems are folds to the edge of a part to create a rounded, safe edge. Hems can also increase the stiffness of an otherwise flat edge.

Designing Hems: Hems may be open, flat, or tear-dropped, and tolerances depend on the hem's radius, material thickness, and features near the hems.

Open hems: Minimum inside diameter must be at least 1X MT. Larger insidediameter may lose circularity. The return length must be at least 4X MT after the radius.

Tear-drop hems: Minimum inside diameter must be at least 1X MT. The opening should be at least ¼X MT, and the return length is at least 4X MT after the radius.

Flat hems: Flat hems are pressed shut and require at least 4X MT return length. Please note that flat hems risk fracturing the material at the bend, and should be avoided for most designs.

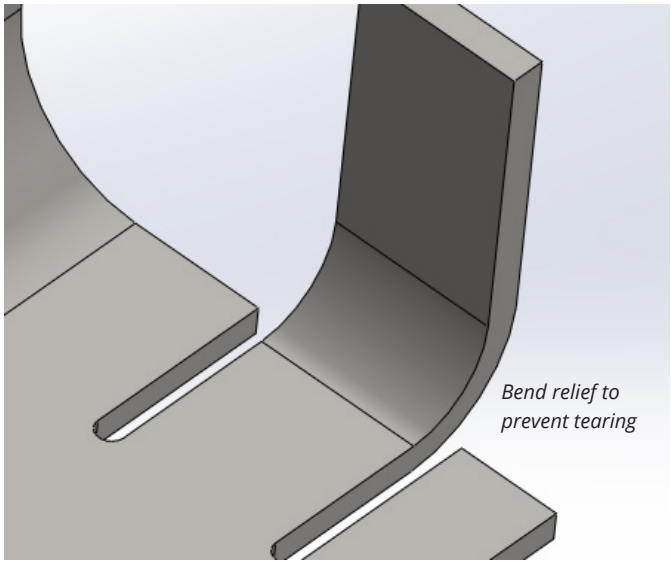


Relief Cuts and Corner Fillets

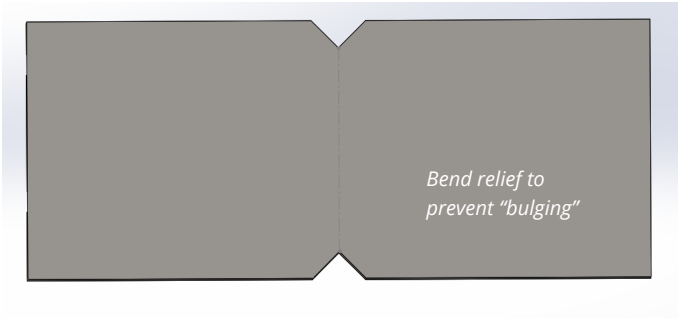
Relief Cuts

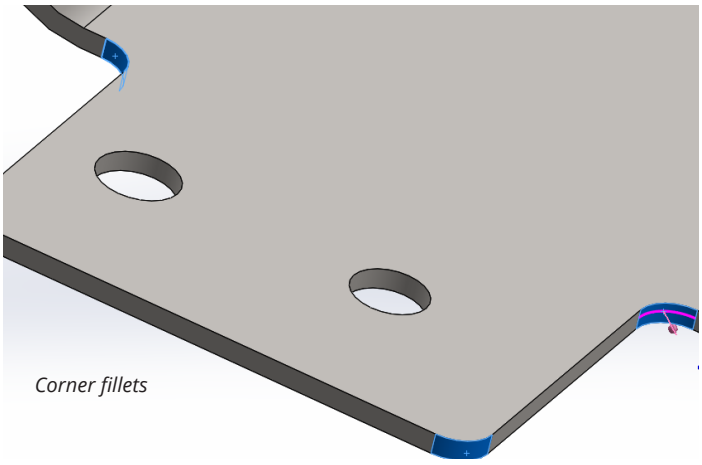
Relief cuts help parts fall closer to design intent to avoid bulging and tearing at bends. Overhangs become more prominent for thicker parts with a smaller bend radius, and may even be as large as ½ the material's thickness. Tearing may occur when bends are made close to an edge.

> ***Tip:*** *Relief cuts are powerful features to protect holes near bends and can maintain square edges for welding or riveting.*



Designing Relief Cuts: Relief cuts must be at least 1MT thickness in width and must be longer than the bend radius.





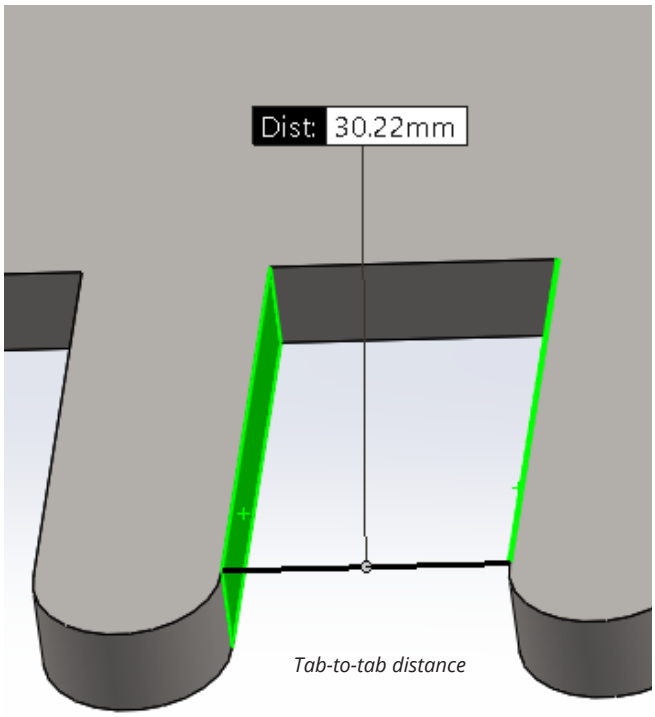
Corner Fillets

Sheet metal parts may have sharp corners, but designing a fillet of ½ MT will make parts more cost-effective and producible.

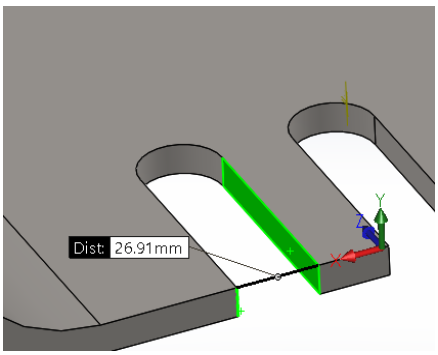
Notches and Tabs

Designing Notches and Tabs: The minimum thickness of a notch is at least 0.040” (~1mm) or 1X MT, whichever is greater. Tabs must be at least 0.126” (3.2mm) thick, or 2X MT, whichever is greater. Both tabs and notches should be no larger than 5X their width.

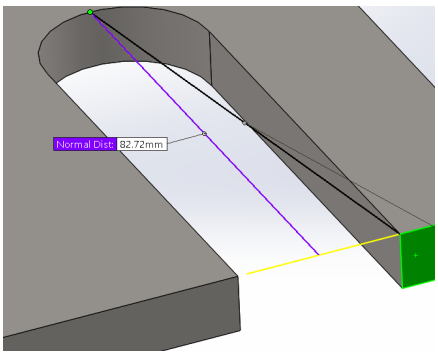
Notch and Tab Clearances: Notches must be at least ⅛” (3.175mm) away from each other. For bends, notches must be at least 3 times the material’s thickness plus the bend radius. Tabs must have a minimum distance from each other of 0.04” (1mm) or the material’s thickness, whichever is greater.



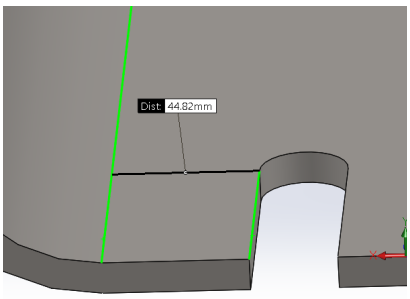
Tab-to-tab distance



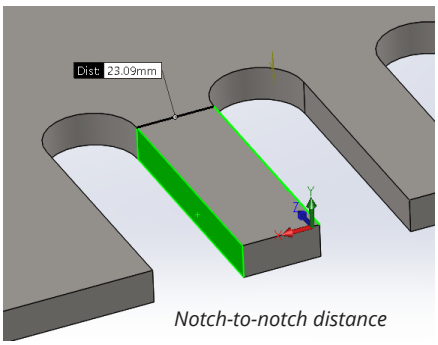
Notch thickness



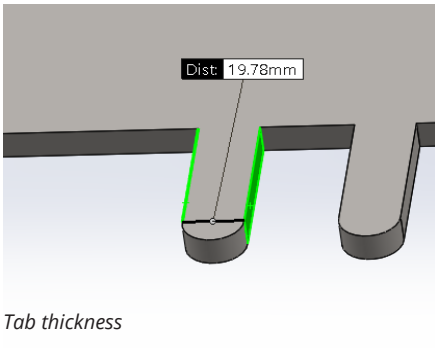
Notch length



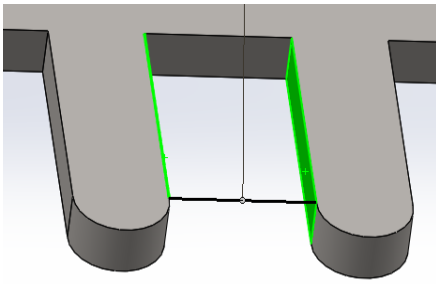
Notch-to-bend distance

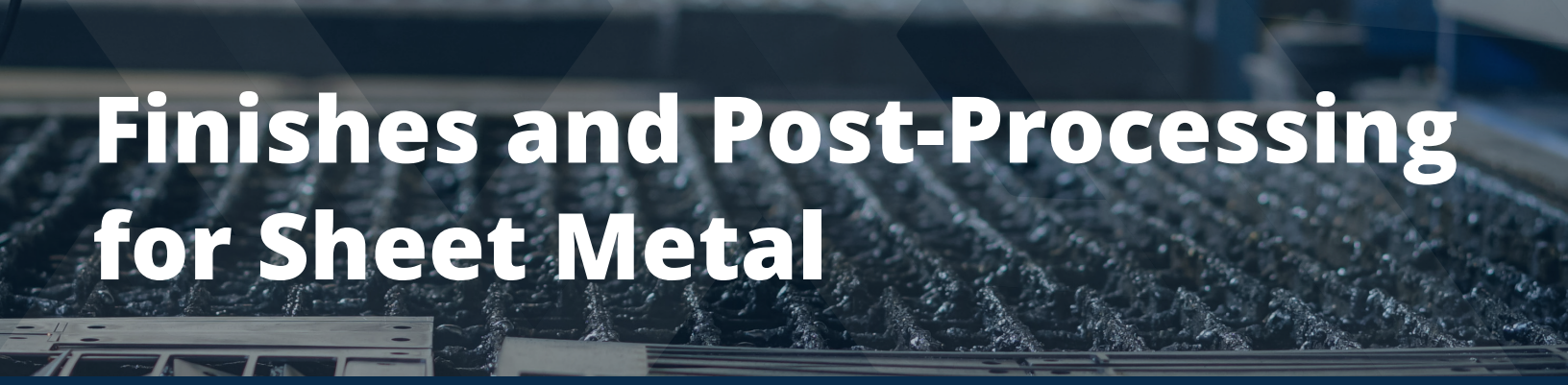


Notch-to-notch distance



Tab thickness


Tab-to-tab distance



Finishes and Post-Processing for Sheet Metal

Sheet metal parts have a variety of post-finishing options. Below are common finishing services for metal parts.

Finishing Options

Anodizing
- Type II Anodize (Black, Clear, or Color)
- Type III Hard coat
- Type III w/ PTFE

Metal Plating
- Copper Plating
- Electroless Nickel
- Electrolytic Nickel
- Nickel Plating
- Nickel Sulfamate
- Nickel Watts
- Tin
- Zinc

Adhesives and Coatings
- Black Oxide
- Dry Film Lubricants
- Powder Coat
- Wet Paint

Conversion and Pretreatments
- Chem Film
- Chromate Conversion Coating
- Heat Treat
- Passivation
- Pickle and Oil

Surface Treatment
- Bead Blasting
- Electropolishing
- Etching

Part Marking
- Ink Stamping
- Laser Marking
- Silk Screening

Heat Treatment
- Annealing
- Carburizing
- Cryogenic Treatment
- Flame Hardening
- Hydrogen Embrittlement Relief
- Induction Hardening
- Nitriding
- Stress Relieving

Precious Metal Plating
- Gold
- Palladium
- Palladium-Nickel
- Platinum
- Rhodium
- Silver

Installing Inserts and Fasteners

Features like boss standoffs, threads, and nuts are typically installed via press fit inserts. The most popular are PEM® Fasteners. Using inserts guarantees and standardizes features like:

- Nuts
- Pins and studs
- Standoffs
- Cable-tie mounts
- Captive hardware

Designing for Inserts: Follow the guidelines specified by the instructions included with the off-the-shelf inserts. Make sure to note the part ID and install direction in an accompanying print for reference.

Installed threaded inserts on a sheet metal part

Source: pemnet.com

Welding, Riveting, and Assemblies

This weld joint uses a folded tab and relief cut to maintain a square corner.

Sheet metal parts can be welded. Welded joints should be called out within a print on the type of weld desired. Be mindful of weld locations as there will be a rounded build-up of material that could cause fit interference if not considered in the design.

Rivets are useful for larger components and for serialized production as they use standardized tooling to create permanent bonds.

> ***Tip:*** *Services like Xometry's custom manufacturing can also perform light assemblies and weldments upon request.*

Resources at Xometry

Online Instant Quoting

Web: Upload your CAD file at get.xometry.com/quote
Accepted file types: STEP (.step, .stp), SOLIDWORKS (.sldprt), Mesh (.stl), Parasolid (.x_t, .x_b), DXF (.dxf), Autodesk Inventor (.ipt), Dassault Systems (.3dxml, .catpart), PTC, Siemens (.prt), ACIS (.sat)
Capabilities: CNC machining, sheet metal fabrication, 3D printing, urethane casting, injection molding

By Request

Post-manufacture assembly, post-processing finishes, and other manufacturing services

Live Engineering Support

Hours: M-F 8:00 AM - 9:00 PM EST
Email: support@xometry.com
Phone: (240) 252-1138

Xometry

xometry.com