Xometry

Design Guide: Sheet Metal Fabrication

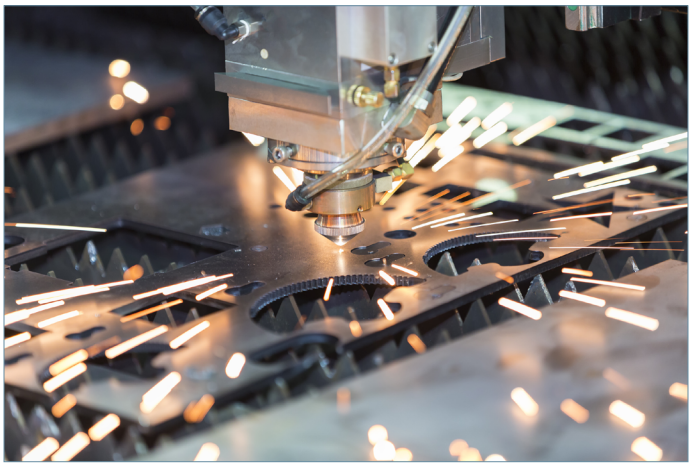
VERSION 2.1

Table of Contents

Overview

Sheet metal fabrication is the process of forming parts from a metal sheet by punching, cutting, stamping, and/or bending. 3D CAD files are converted into machine code, which controls a machine to precisely cut and form the sheets into the final part. Sheet metal parts are also known for their durability, which makes them great for end-use applications (e.g. chassis). Parts used for low volume prototypes, and high volume production runs are most cost-effective due to large initial setup and material costs.

Because parts are formed from a single sheet of metal, designs must maintain a uniform thickness. Be sure to follow design requirements and tolerances in this guide to ensure parts fall closer to design intent.

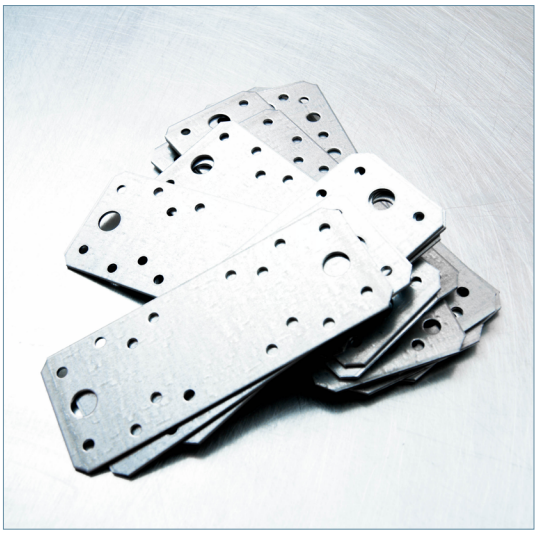


Tolerances

General Tolerances

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:

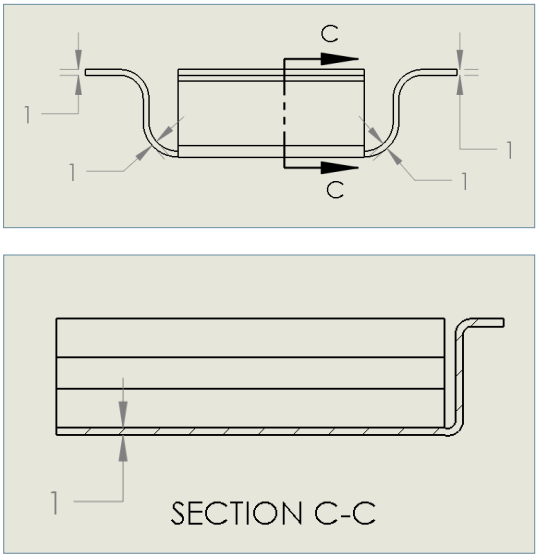
- Forming and bending: +/- 0.020”
- Bend to hole or feature: +/- 0.010”
- Linear dimensions excluding locations to bends: +/- 0.005”
- Diameters with inserts: +0.003/-0
- Angularity: +/- 2 degrees
- Surface roughness (blank material): Ra 125 uin max
- Surface roughness (timesave): Ra 100 uin max



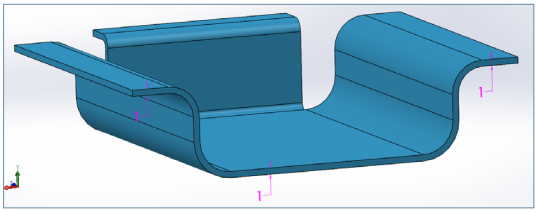
Wall Thickness

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

When considering sheet metal thickness, a single sheet with punches (holes) is a good rule of thumb. Some features, such as countersinks are doable, but counterbores and other machined features are difficult to produce as they require post-machining.



Drawing indicating uniform wall thickness



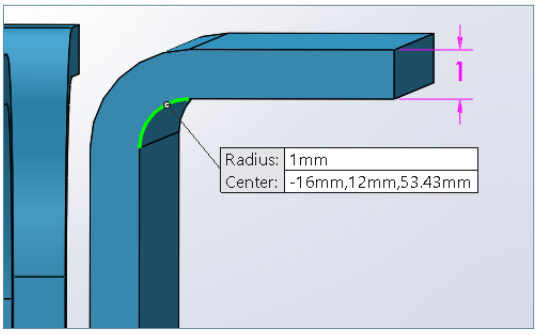
Model indicating uniform wall thickness

Bends

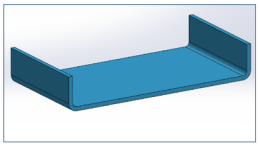
Sheet metal brakes are used to bend material into a part's desired geometry. Bends in the same plane should be designed in the same direction to avoid part reorientation, which will save both money and time. Keeping a consistent bend radius will also make parts more cost-effective. Small bends to large, thick parts tend to become inaccurate, so they should be avoided if possible.

DIMENSIONS

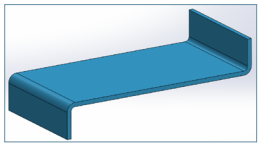
To prevent parts from fracturing or having distortions, make sure to keep inside bend radius at least one material's thickness.



Inside bend radius



Consistent bend orientations



Inconsistent bend orientations

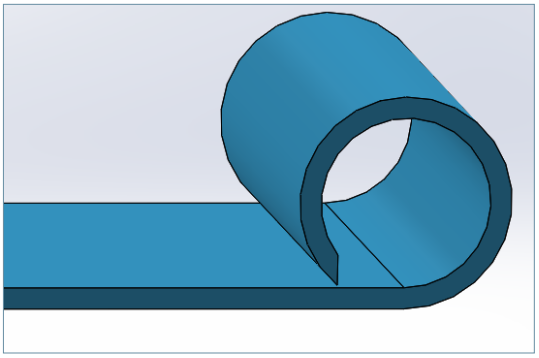
Curls

DIMENSIONS

Outside radius must be at least twice the material's thickness.

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 6 times the material's thickness plus the radius of the curl.



Example of a curl

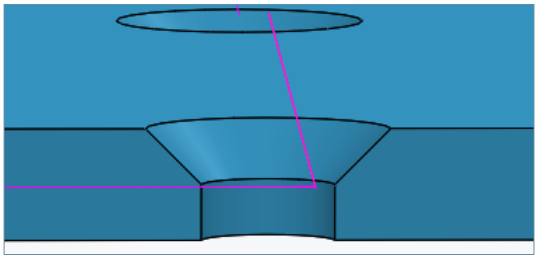
Countersinks

DIMENSIONS

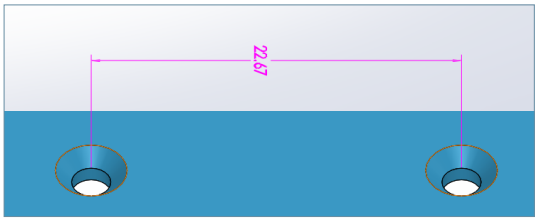
The maximum depth a countersink may have is 3.5 times the material's thickness.

CLEARANCES

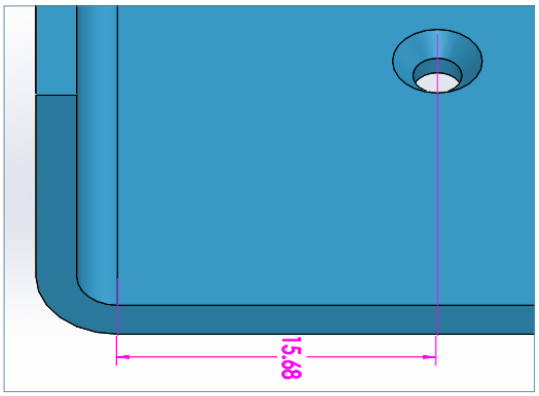
Countersinks must be at least 8 times the material thickness from each other, 4 times the material's thickness from an edge, and 3 times the material's thickness from a bend.



Example of a countersink



Center-to-center distance



Center-to-bend distance

Hems

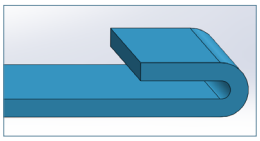
Hems are folds to the edge of a part to create a rounded, safe edge. Hems may be open, flat, or tear-dropped, and tolerances depend on the hem's radius, material thickness, and features near the hem.

NOTE:

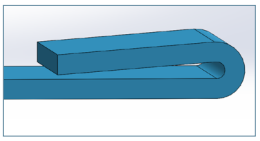
Flat hems risk fracturing the material at the bend, and should be avoided if possible.

DIMENSIONS

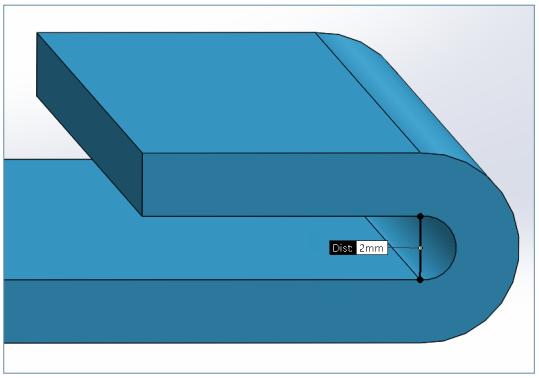
For open hems, minimum inside diameter is equal to the material thickness (larger diameters tend to lose circular shape), and the return length is at least 4 times the material's thickness. Tear-dropped hems must maintain an inside diameter of at least one material's thickness, an opening of at least ¼ the material's thickness, and return length is also 4 times the material's thickness.



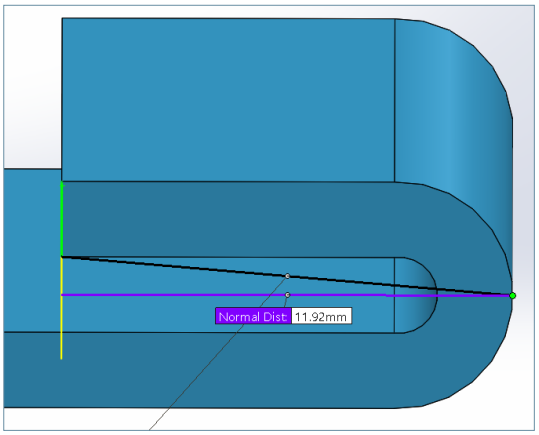
Open hem



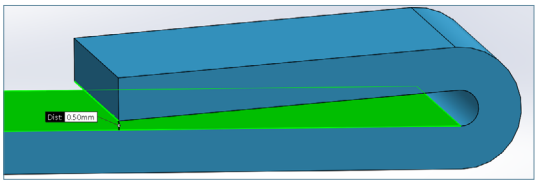
Tear-dropped hem



Open hem inside diameter



Open hem return length



Tear-dropped hem opening distance

Holes & Slots

DIMENSIONS

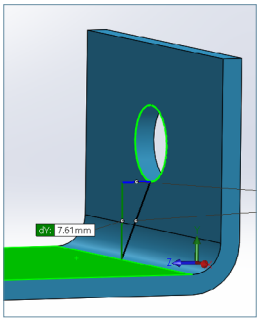
Keep hole and slot diameters at least as large as material thickness. Higher strength materials require larger diameters.

CLEARANCES

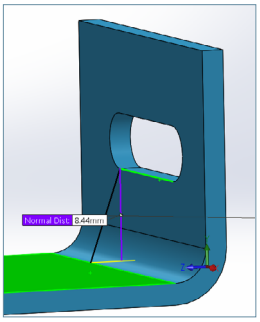
Holes and slots may become deformed when placed near a bend. The minimum distance they should be placed from a bend depends on the material thickness, the bend radius, and their diameter. Be sure to place holes away from bends at a distance of at least 2.5 times the material's thickness plus the bend radius. Slots should be placed 4 times the material's thickness plus the bend radius away from the bend.

Be sure to place holes and slots at least 2 times the material's thickness away from an edge to avoid a "bulging" effect.

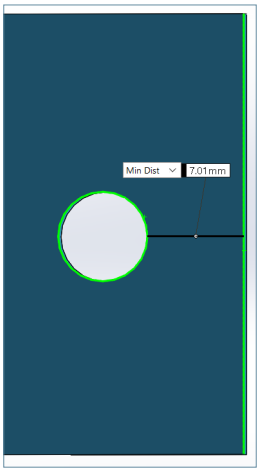
Holes should be placed at least 6 times the material's thickness apart.



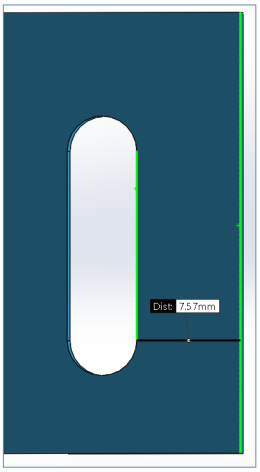
Hole-to-bend distance



Slot-to-bend distance



Hole-to-edge distance



Slot-to-edge distance

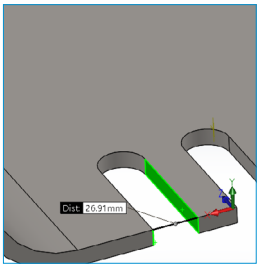
Notches & Tabs

DIMENSIONS

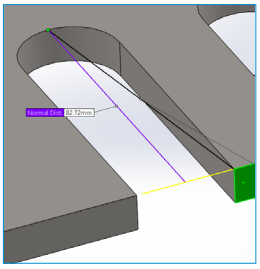
The minimum thickness a notch must maintain is at least 0.04" (1mm) or the material's thickness, whichever is greater; the length must be no larger than 5 times its width. Tabs must be at least 0.126" (3.2mm) thick, or two times the material's thickness, whichever is greater; the length must also be no larger than 5 times its width.

CLEARANCES

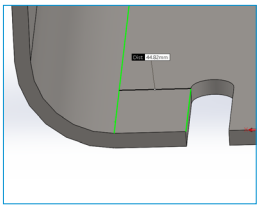
Notches must be at least ⅛" (3.175mm) away from each other. For bends, notches must be at least 3 times the material's thickness plus the bend radius. Tabs must have a minimum distance from each other of 0.04" (1mm) or the material's thickness, whichever is greater.



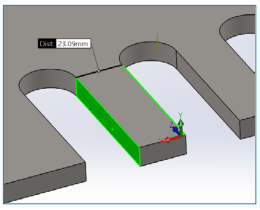
Notch thickness



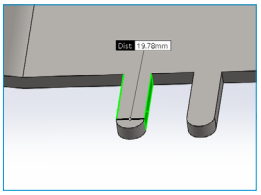
Notch length



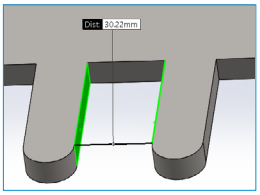
Notch-to-bend distance



Notch-to-notch distance



Tab thickness

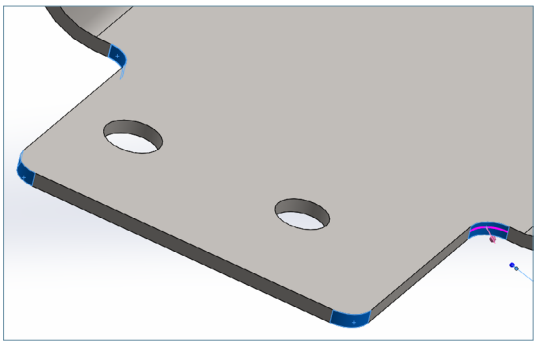


Tab-to-tab distance

Features

Corner Fillets

Sheet metal parts may have sharp corners, but designing a fillet of ½ the material's thickness will make parts more cost-effective.



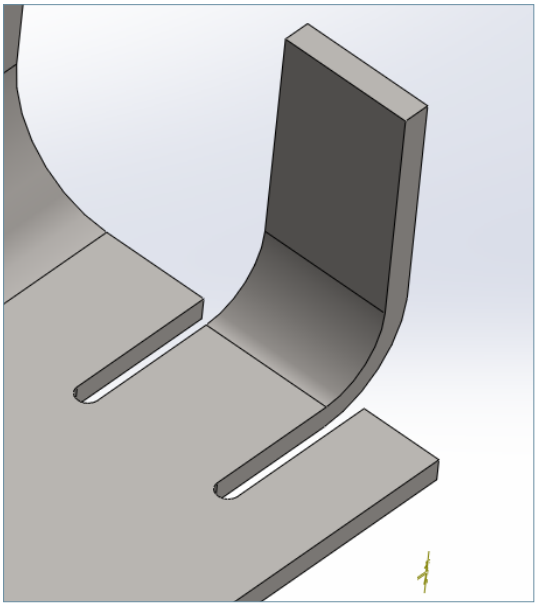
Corner fillets

Relief Cuts

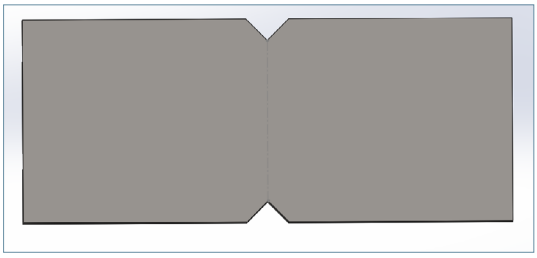
Relief cuts help parts fall closer to design intent to avoid "overhangs" and tearing at bends. Overhangs become more prominent for thicker parts with a smaller bend radius, and may even be as large as ½ the material's thickness. Tearing may occur when bends are made close to an edge.

DIMENSIONS

Relief cuts for bends must be at least one material's thickness in width, and must be longer than the bend radius.



Bend relief to prevent tearing



Bend relief to prevent "bulging"

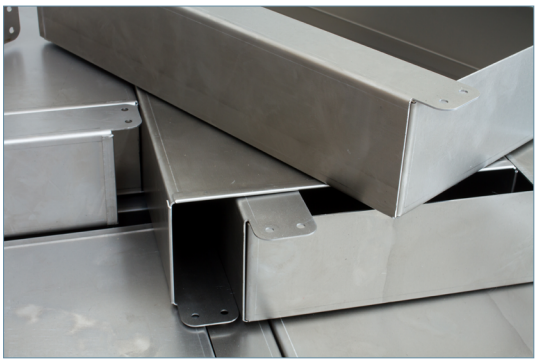
Finishes & Post-Processing

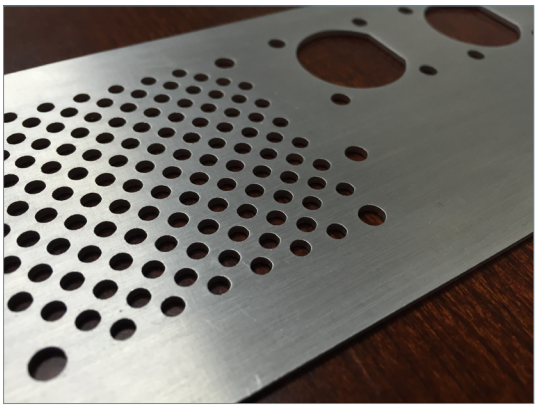
Xometry offers sheet metal parts in a wide array of materials including:

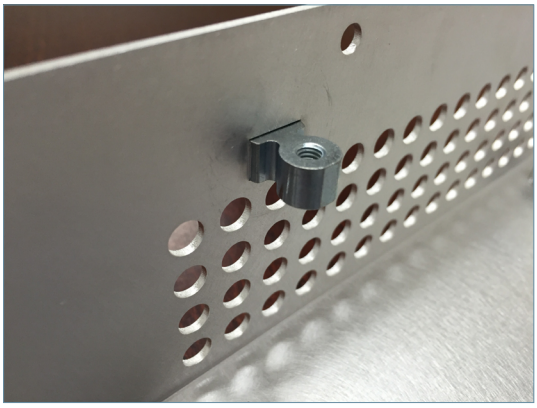
- Aluminum
- Stainless steel
- Bronze/Brass
- Copper
- Steel

To further customize parts, Xometry offers post-processing options to add to sheet metal parts such as:

- Bead Blasting
- Plating
- Welding
- Inserts
- Other custom finishes upon request







Resources at Xometry

Online Instant Quoting

Web: Upload your CAD file at get.xometry.com/quote

CAD: Download the free Xometry Add-In for SOLIDWORKS: xometry.com/solidworks

Accepted File Types: .stl, .step, .stp, .x_t, .x_b, .sldpart, .ipt, .prt, .sat, .catpart (max file size: 300MB)

Capabilities: CNC Machining, Sheet Metal Fabrication, 3D Printing, Urethane Casting, Injection Molding

Live Engineering Support

Hours: M-F 8:00 AM - 9:00 PM EST

Email: support@xometry.com

Phone: (240) 252-1138

Online: xometry.com/support offers live chat, FAQs, and other helpful articles.

ITAR Registered

All uploads are secure and confidential. Contact us for assistance with large orders and existing Xometry quotes.

Need to use a PO? We offer same day credit application processing. Contact us for details.