Xometry

Design Guidelines and Best Practices for Sheet Metal Fabrication

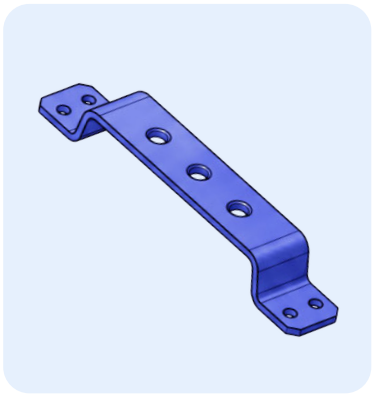


Table of Contents

Overview 03

What is Sheet Metal Fabrication?

Types of Sheet Metal Forming

Tolerances & Standards 08

General Manufacturing Standards

Geometry-Dependent Tolerances

File Formats for Sheet Metal 12

3D CAD Formats

2D DXF Requirements

Sheet Metal Design Guidelines 15

Wall Thickness

Holes and Slots

Bends

Curls and Countersinks

Hems

Relief Cuts

Notches and Tabs

Corner Treatments

Finishes & Post-processing 26

Mechanical & Chemical Surface Treatments

Anodizing & Conversion Coatings

Metal Plating

Paints and Coatings

Part Marking

Hardware & Assembly 33

Welding

Brazing and Soldering

Mechanical Fastening

Additional Resources 42

Stock Sizes and Gauge Lookup Chart

Stock Material Thickness Tolerances

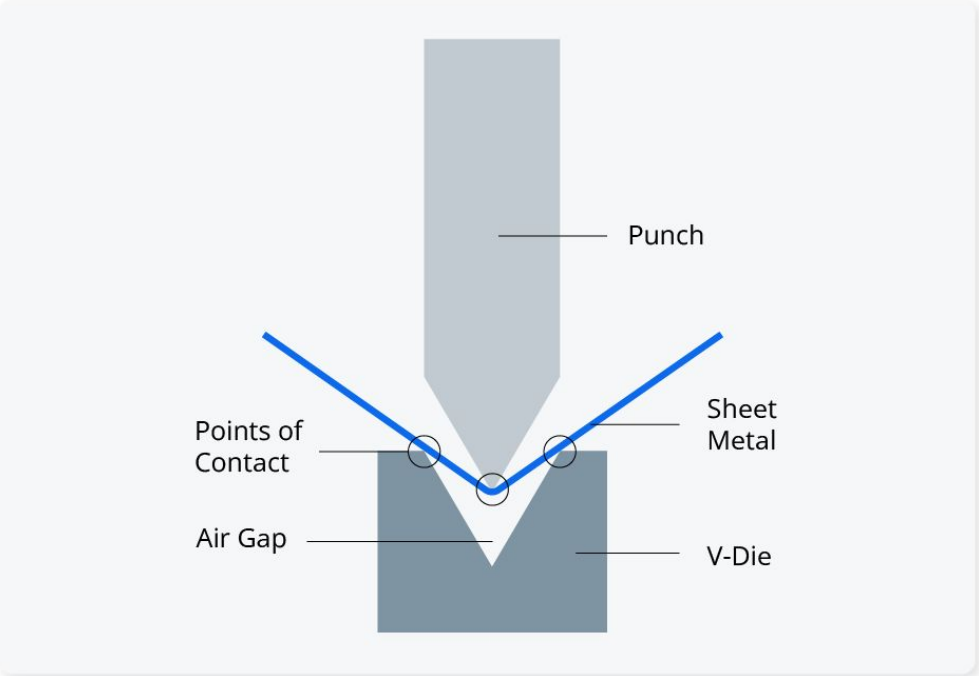
Additional Xometry Resources

SECTION 1

Sheet Metal Fabrication Overview

What is Sheet Metal Fabrication?

Sheet metal fabrication is the process of forming parts from flat metal sheets through methods such as punching, cutting, stamping, and bending. 3D CAD files are “unfolded” into a 2D flat pattern to prepare them for manufacturing. Computer-controlled machines precisely cut this pattern, which is then formed into the final 3D geometry using press brakes and die tools. The final parts can be welded, surface-treated, or receive inserts and part markings.



A diagram of sheet metal forming using the air bending method.

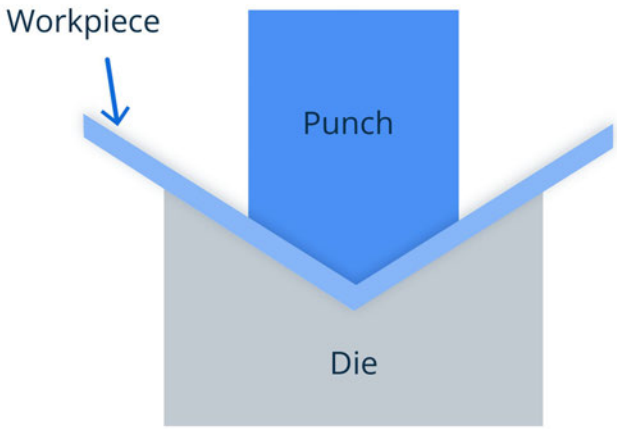
Because parts are formed from a single sheet, designs must maintain a uniform wall thickness throughout. Variations in thickness cannot be achieved without additional steps, such as post-machining. Additionally, designers must account for factors like bend radius and feature-to-feature distances. Following the best practices and standards outlined in this guide will ensure your parts closely align with your design intent.

Types of Sheet Metal Forming

Once a flat sheet metal blank is cut, forming processes shape the material by applying a force that exceeds its yield strength, creating a permanent new geometry. While there are several methods available to manufacturers, this section provides a high-level overview of the most common techniques.

Bending (Press Brake)

Press brake bending is the most common sheet metal fabrication process used to form material, capable of producing everything from simple 90-degree flanges to complex, multi-bend geometries. The machine uses a punch (the upper tool) to force a flat sheet into a V-shaped die (the lower tool).



Bending (Press Brake)

A critical challenge in press brake bending is springback. This is the material's elastic recovery—its tendency to partially return to its original flat shape after the forming force is removed. It occurs because the compressed inner layers of the bend push against the stretched outer layers once the tool retracts. Manufacturers typically compensate for this by overbending the part slightly (e.g., to 88°) so it naturally springs back to the precise target angle (e.g., 90°).

Because it is the most widely used forming method, many of the design recommendations later in this guide will focus on press brake bending techniques.

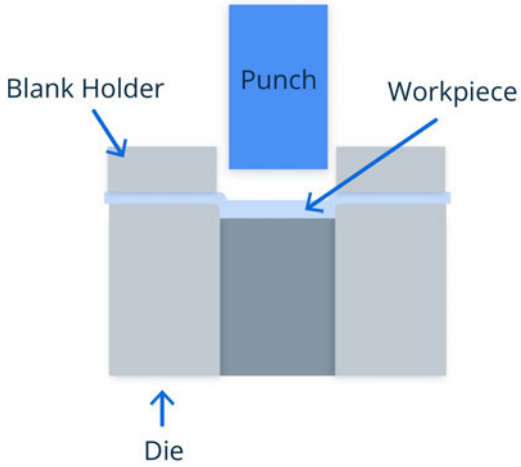
Deep Drawing

Deep drawing is a specialized process that pushes a blank through a die opening, stretching and compressing it to seamlessly take the shape of the punch. It is primarily used to create continuous, watertight parts with significant depth, such as enclosures, sinks, aluminum cans, and automotive oil pans.

High Setup Costs: The custom punches, dies, and blank holders required are complex and expensive to manufacture.

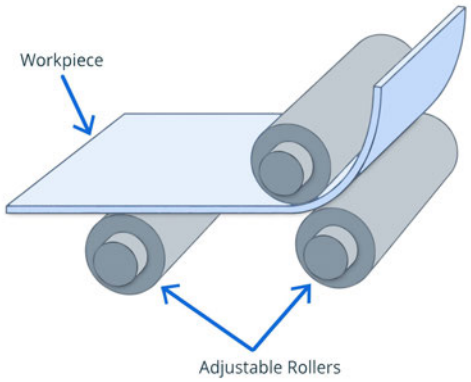
Mass Production: Once setup is complete, the process is exceptionally fast and highly repeatable, making it ideal for high-volume orders.

Material Requirements: It is limited to highly ductile materials (like low-carbon steel or 3003 aluminum) and requires perfectly uniform, burr-free blanks to prevent tearing during the stretch.



Rolling

Rolling is a forming method that uses a series of motorized, rotating rolls to continuously curve metal sheets into cylindrical or conical shapes. It is the go-to process for creating consistent, large-radius curves that are impractical to achieve with a standard press brake.



No Length Limitations: Because the material continuously feeds through the machine, it is ideal for long tank sections and pipes.

Heavy-Duty Capabilities: Rolling is highly efficient for thick plates and structural shapes.

Gentle Forming: The gradual forming process significantly reduces the risk of material cracking, with typical minimum radii ranging from 3 to 5 times the material thickness.

Sheet Metal Benefits

Sheet metal fabrication is a foundational manufacturing process renowned for producing highly durable, end-use components. Because it relies on shaping thin-gauge material rather than carving away solid blocks of metal, it offers a unique combination of efficiency, economy, and performance. Whether you are designing a simple bracket or a complex, multi-part electronic enclosure, sheet metal provides several distinct advantages:

- **Unmatched Cost Efficiency:** For parts with a uniform wall thickness, sheet metal is almost always more cost-effective than CNC milling. 2D cutting methods (like laser, plasma, or waterjet) process material at blazing speeds, and bending requires significantly less machine time than 3D contouring. Furthermore, because you are starting with a sheet rather than a thick block, material waste is kept to an absolute minimum.
- **Seamless Scalability:** One of sheet metal's greatest strengths is its ability to scale directly alongside your product's lifecycle. You can quickly and cheaply prototype a dozen units using laser cutting and standard press brake tooling. As demand grows, the same fundamental design can be transitioned to automated progressive **die stamping** for production runs of 100,000+ units, drastically reducing per-part cost.
- **Exceptional Strength-to-Weight Ratio:** By using strategic forming geometries such as flanges, hems, and embossed ribs, thin-gauge sheet metal can achieve remarkable structural rigidity. This allows engineers to design robust, load-bearing enclosures and chassis that are a fraction of the weight of fully machined or cast alternatives.
- **Versatility in Size and Finishing:** Unlike CNC machining, which is often constrained by the physical limits of the machine's work envelope, sheet metal processes can easily accommodate massive parts, such as full-size server racks, architectural panels, or automotive components. Additionally, the flat, smooth nature of sheet metal makes it highly suitable for hardware inserts, protective platings, or cosmetic coatings, which we will cover in greater detail later in this guide.

SECTION 2

Tolerances & Standards

General Manufacturing Standards

Xometry adheres to manufacturing standards that are closely aligned with established industry protocols. Typical tolerances are influenced by a part's specific geometry, particularly whether the design involves basic flat cutting or multiple bending operations. The details below provide a comprehensive overview of our standard tolerances for sheet cutting and sheet metal bending. Please note that Xometry always prioritizes the specific requirements and tolerances defined in your technical drawings and notes, so be sure to provide those if your expectations differ from the standards listed below.

Manufacturing Standards for Flat Sheet Cut Parts

- Holes of .100" or smaller in diameter may be slightly larger on the top face than standard tolerances due to the cutter piercing the material near the hole's cut line.
- A small bump or divot in the material, or a different edge condition, may be present at the lead-in and lead-out of a cutting profile.
- Pre-finished or textured sheets, such as brushed or polished stock, may have only one side with the cosmetic finish.
- Protective film may be shipped on cut products to prevent damage to cosmetic finishes.
- The edge condition of sheet-cut materials will result in vertical striations rather than smooth edges. This affects the transparency of edges on clear plastics cut with a waterjet.
- Some cut materials may show a small halo discoloration from backsplash (waterjet) or overspray near the cut edges (laser).

Standard Tolerances for Flat Sheet Cut Parts

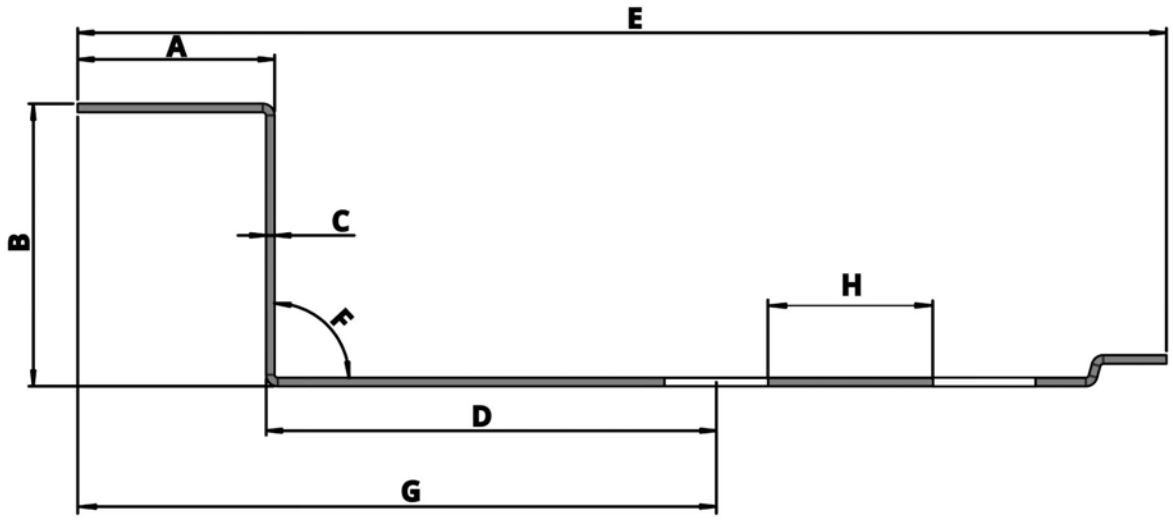
- ➔ Thickness tolerances are independent of cutting tolerances, as they rely on the raw stock material tolerance range.
- ➔ Edge-to-edge tolerances are +/- .010" nominal on the top face of the plate/sheet.
- ➔ Thicker materials may exhibit a tolerance deviation on the bottom face due to the tapers inherent in laser and waterjet cutting.
- ➔ Xometry cannot guarantee flatness callouts for sheet-cut materials, except for ground stock or tooling plate (e.g., MIC 6).

Manufacturing Standards for Bent Sheet Metal Parts

In addition to sheet cutting standards, the following also apply to bent sheet metal components:

- ➔ Sharp edges will be broken and deburred by default.
    - Critical edges that must be left sharp should be noted and specified on a drawing.
- ➔ Witness marks produced by tooling may be visible close to bend locations.
- ➔ Specific sheet metal designs require custom tooling and will require a manual review:
    - Hems
    - Curled flanges
    - Rolled sheets
    - Stamped parts
    - Welded assemblies

Standard Tolerances for Bent Sheet Metal Parts



A - ± .015" Edge-to-bend

B - ± .030" Across bend

C - Dependent on raw stock material and thickness tolerance

D - ± .015" Bend to hole or internal cut feature

E - ± .030" Overall edge-to-edge on formed part

F - ± 1 Degree Bend angle

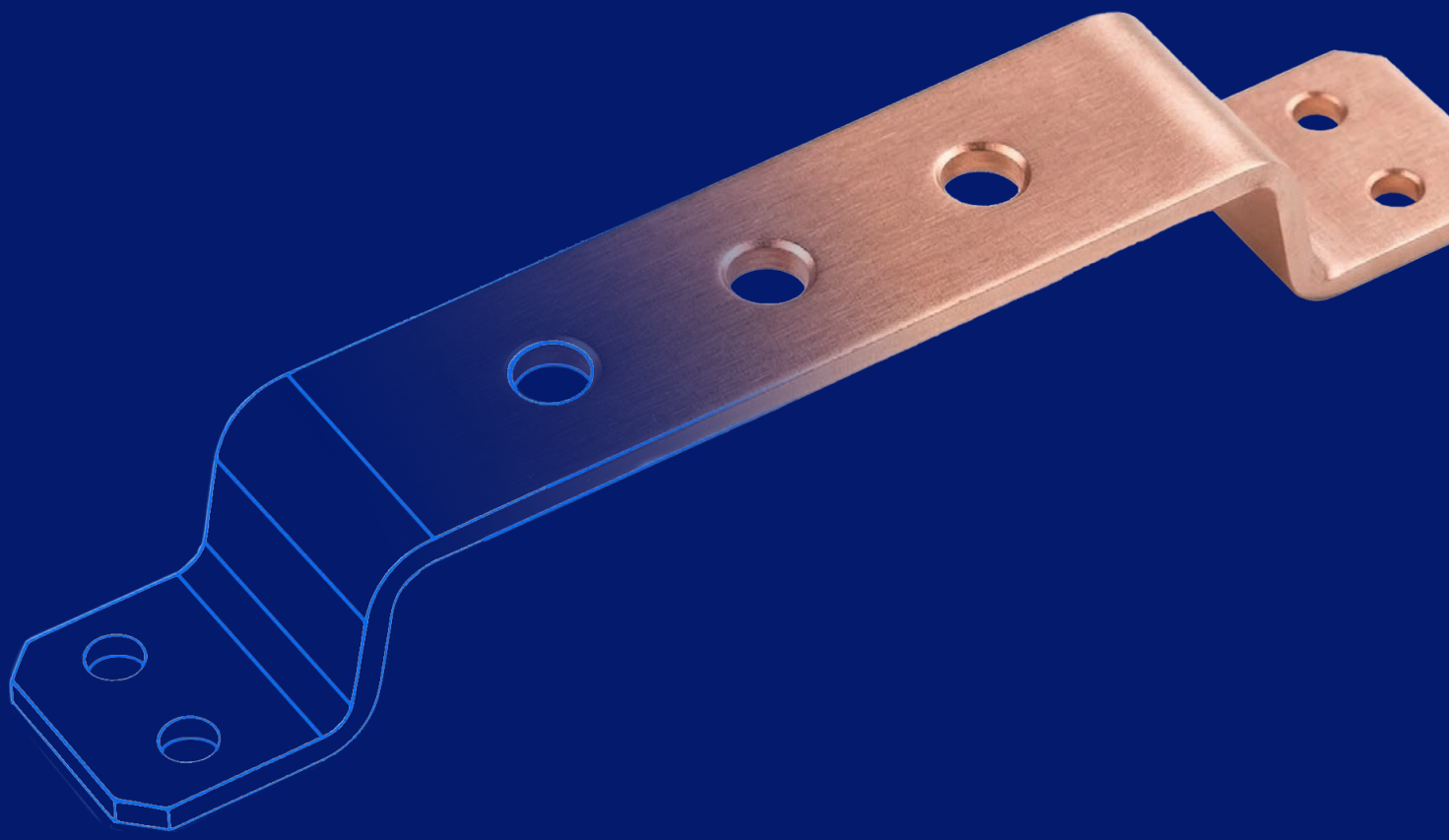
G - ± .030" Edge-to-feature over multiple formed surfaces

H - ± .010" Feature edge-to-edge over the same surface over the top of the cut face

See Sheet Cutting standards for edge-to-edge and other cutting tolerances over flat surfaces.

SECTION 3

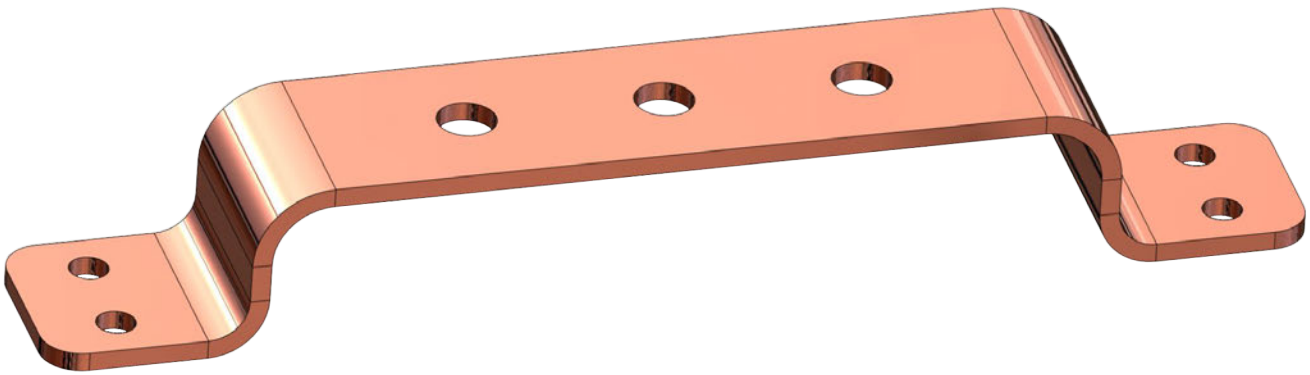
File Formats for Sheet Metal



3D CAD Files

3D CAD files are the industry standard for quoting and producing high-quality sheet metal parts. By providing a comprehensive 3D model rather than relying solely on traditional 2D drawings, you unlock several advanced manufacturing advantages:

- **Automated Quoting:** 3D CAD allows AI-powered platforms like Xometry's to provide you with instant pricing and lead times.
- **Design for Manufacturability (DFM):** 3D geometry allows us to run quick DFM checks, instantly flagging potential issues that could slow down production.
- **Elimination of Translation Errors:** Relying purely on 3D geometry reduces the risk of human error associated with misinterpreting 2D projections or manually calculating bend allowances. 2D drawings remain valuable for communicating specific tolerances, finish requirements, and threaded hole callouts.

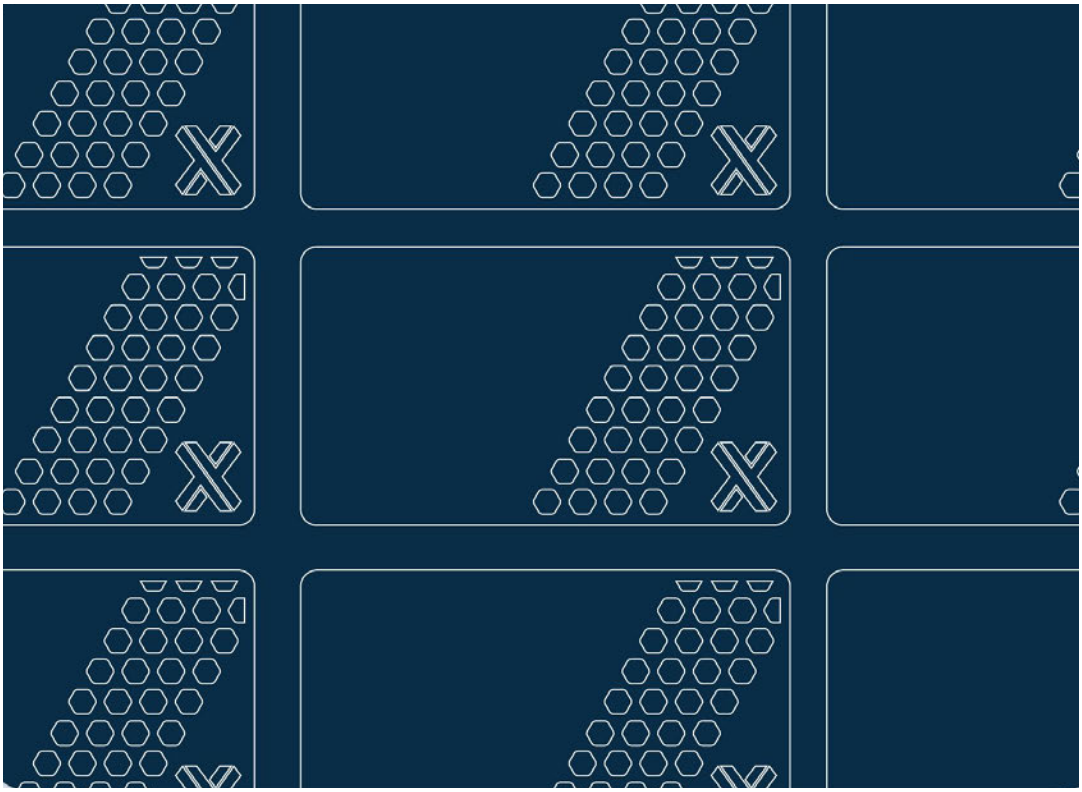


For bent components, it is crucial to provide the CAD file with the bends already designed into the model (the formed state). You do not need to provide a flat pattern; Xometry's sheet metal fabricators use specialized software to "unbend" your 3D model on their end.

Pro Tip: Xometry supports a variety of native 3D CAD formats for sheet metal quoting, including **.SLDPRT, .STEP, .IPT, .PRT, .SAT,** and more.

2D DXF Files

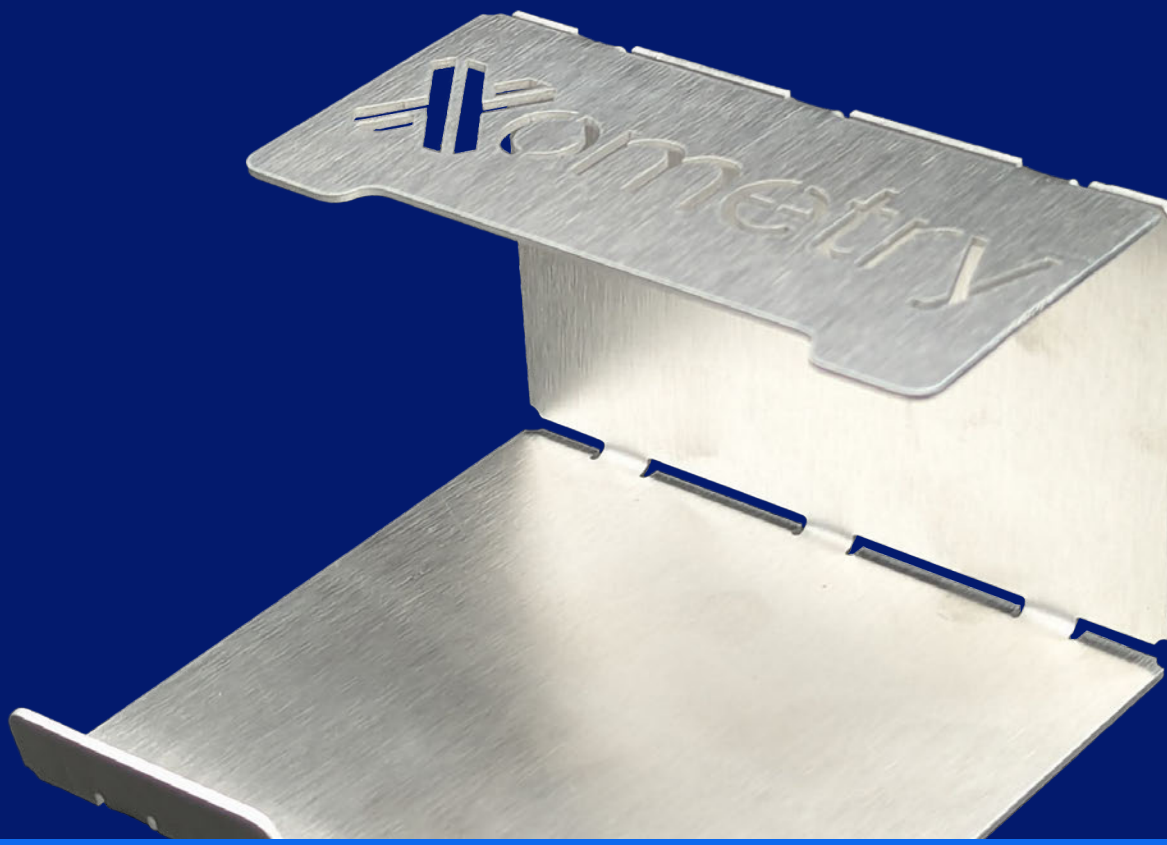
A DXF (Drawing Interchange Format or Drawing Exchange Format) is a CAD data file format for enabling data exchange between AutoCAD and other programs. 2D DXF files are typically used for sheet cutting. The DXF cut file is a continuous pathway perimeter or internal paths for holes and slots to guide a machine cutting operation.



The best way to get an instant quote with a DXF is to upload a cut file (not the full drawing). Xometry can quote flat sheet cut parts using a 2D DXF file, but bent sheet metal parts require a 3D CAD file format such as **.STEP, .SLDPRT, or .IPT** for quoting. A PDF technical drawing can be attached to the uploaded model to define specifications and callouts for features and finishes.

SECTION 4

Sheet Metal Design Guidelines



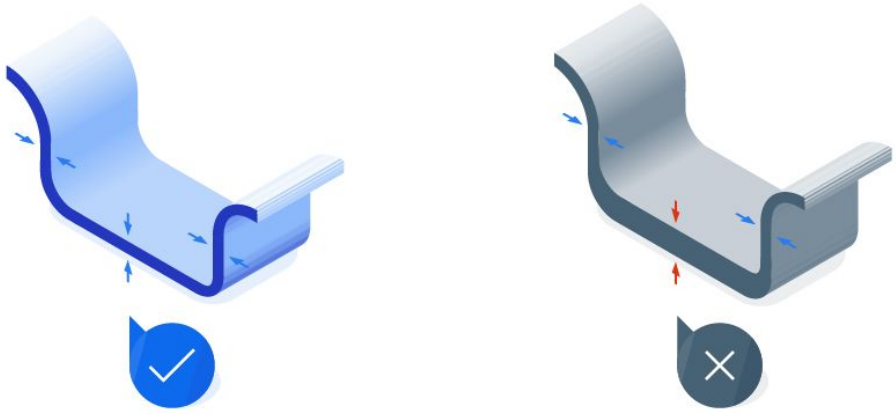
General Design Tips

Designing for sheet metal requires a distinct approach compared to processes like machining or 3D printing. Because the final part is cut and formed from a single flat piece of material, every feature must account for the physical realities of tooling clearances, bending forces, and material stretching. This section outlines critical best practices for your designs—covering aspects like wall thickness, optimal hole and slot placement, bend radii, curls, countersinks, notches, tabs, essential corner relief cuts, and more. Adhering to these guidelines will ensure your design is highly manufacturable, cost-effective, and free of common fabrication defects such as tearing or deformation.

Note: The tips in this section will often describe distances in material thickness, or MT, for relative measurements.

Wall Thickness

Since all sheet metal parts start from flat metal stock, which is cut and processed to specification, bent parts must maintain a uniform wall thickness throughout. Generally, Xometry is capable of bending sheet metal parts up to 1/4" (6.35mm) in thickness, but this capability also depends on the geometry of the part.



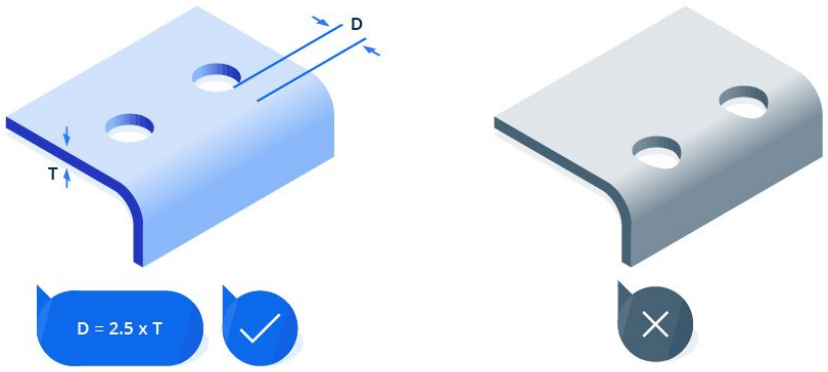
Note: Flat cut sheet parts, such as flat brackets, can be cut thicker (above 1/4") using plasma or waterjet, but are typically not recommended for bending or forming.

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.

- Holes and slots should be at least **6X MT** from each other.
- Holes should 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.



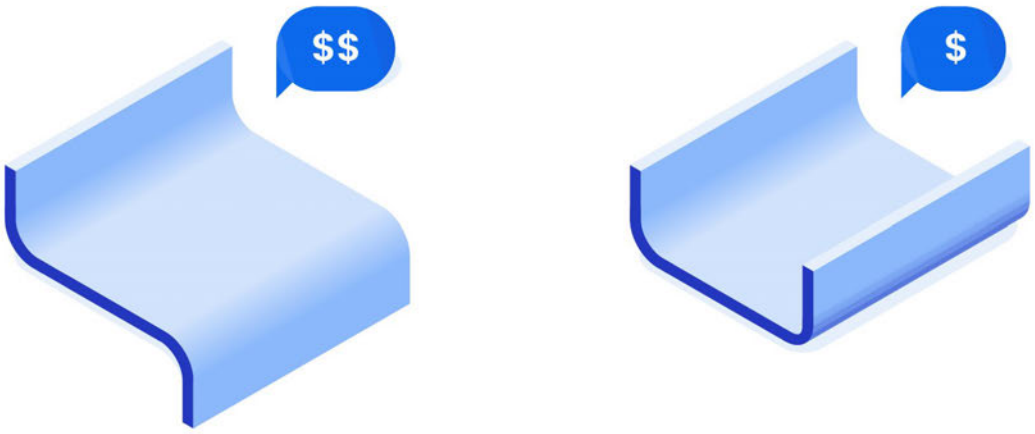
Sheet Metal Feature Clearances (Multipliers of Material Thickness)					
Feature	**Countersink**	**Hole**	**Slot**	**Edge**	**Bend Edge**
Countersink	8	8	8	8	8
Hole	8	6	6	2	2.5
Slot	8	6	6	2	4
Edge	8	2	2		
Bend Edge	8	2.5	4		

Bends

Designing Bends: Keep the bend radius at least **1X the material thickness** to prevent parts from fracturing or distorting.

Sheet metal brakes are typically used to bend sheet metal into the desired geometry of a part. Bends in the same plane should be designed in the same direction to avoid part reorientation, thereby saving time and money.

Keeping a consistent bend radius will also make parts more cost-effective. Additionally, avoid designing very short flanges or tight bend radii on thick materials, as these features are difficult to form accurately and often lead to material fracturing or distortion.

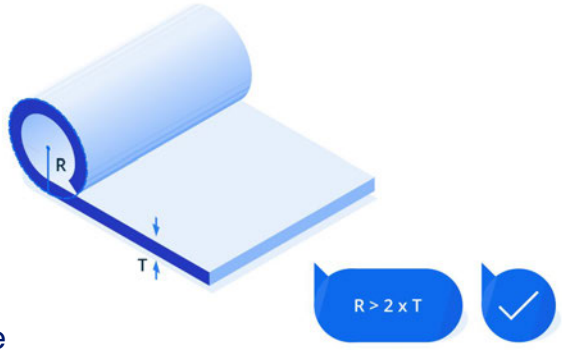


Pro Tip: Be mindful of material alloys when designing sheet metal parts. Aluminum 5052 is the most common sheet metal grade due to its ductility. Aluminum 6061, popular for CNC machining, can be brittle and prone to cracking when bent.

Curls and Countersinks

Curls

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

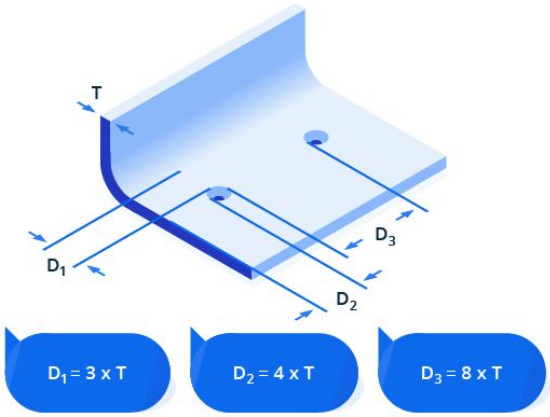


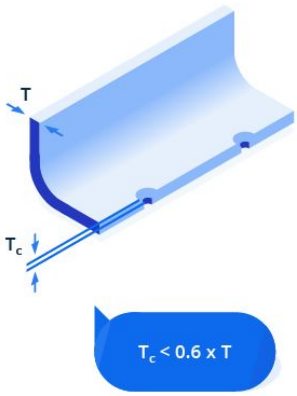
Curl Clearances: Holes should be placed away from the curl at least a distance of the radius of the curl plus the material's thickness. Bends should be at least **6X material thickness** plus the radius of the curl away.

Countersinks

Countersinks are suitable for sheet metal designs and are typically produced with hand tools. Be sure not to 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.





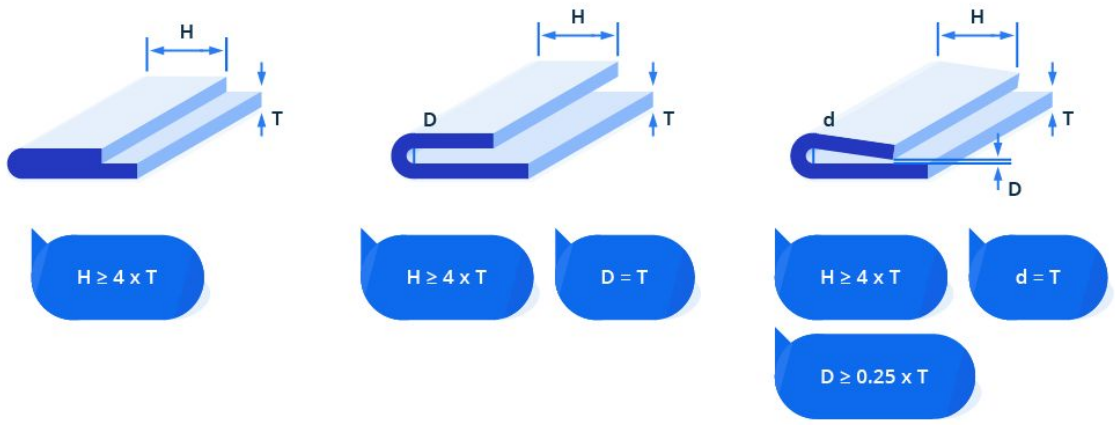
Warning: Counterbores in sheet metal should be avoided, as they require additional machining. In some cases, they can be formed but require specialized punch tooling.

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.

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

- **Flat hems:** Flat hems are pressed shut and require at least **4X MT** return length. Please note that flat hems can cause the material to fracture at the bend and should be avoided for most designs.
- **Open hems:** Minimum inside diameter must be at least **1X MT**. A larger inside diameter 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 **1/4X MT**, and the return length is at least **4X MT** after the radius.



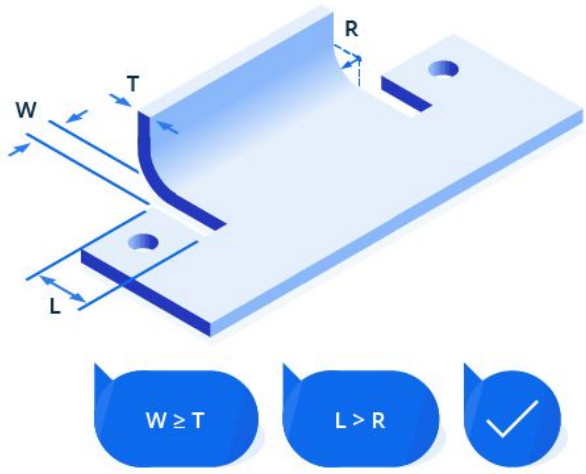
Relief Cuts

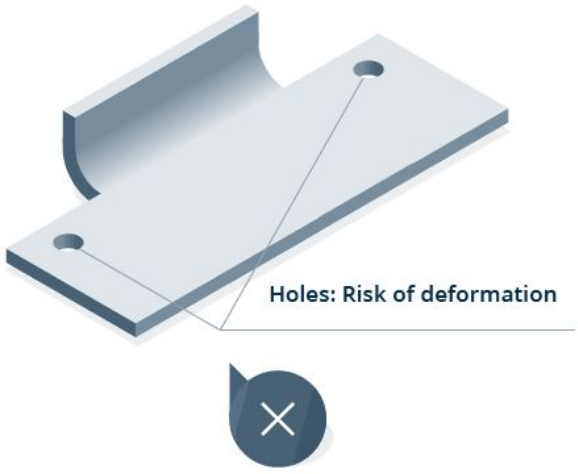
Relief Cuts

Relief cuts help parts fall closer to their design intent, reducing bulging and tearing at bends. Overhangs become more prominent in thicker parts with a smaller bend radius and may even reach half the material's thickness. Tearing may occur when bends are made close to an edge.

Pro 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 **1 MT** wide and longer than the bend radius.





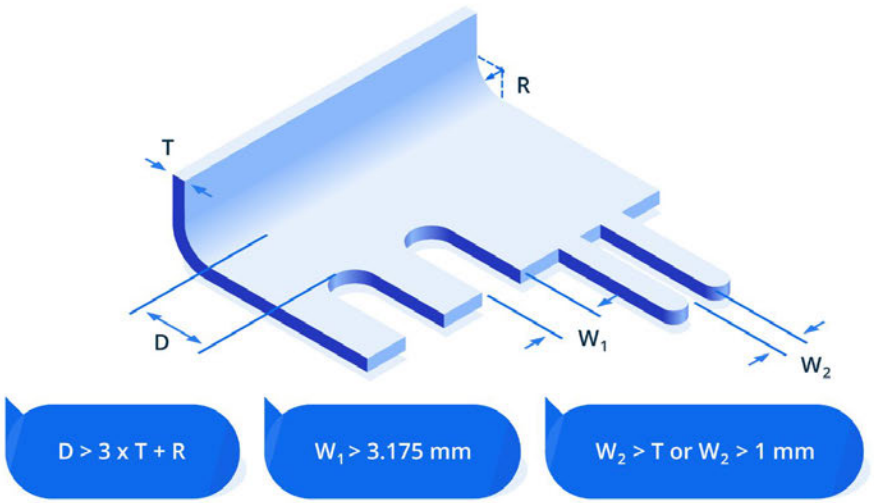
Notches and Tabs

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 **1/8" (3.175mm)** away from each other.
- For bends, notches must be at least **3X MT** plus the bend radius.
- Tabs must have a minimum distance from each other of **0.040" (~1mm)** or the material's thickness, whichever is greater.



Corner Treatments

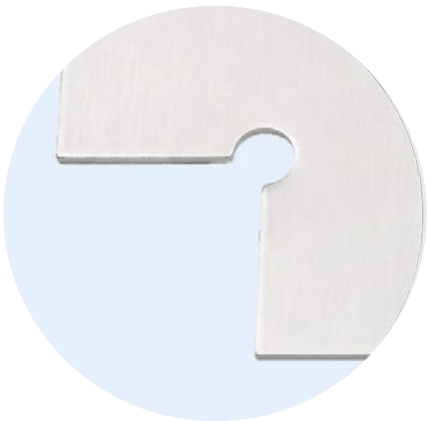
Corner treatments are vital, yet often overlooked, sheet metal features. Because 3D geometries are folded from flat sheets, the junctions where flanges meet require specific design attention. Proper corner treatments prevent material tearing, ensure safe handling, and facilitate accurate downstream assembly or welding.

Corner Reliefs

Parts designed with intersecting bends, such as box or pan corners, require adequate clearance to be formed accurately. Corner reliefs serve a function similar to bend reliefs by preventing material interference and potential tearing throughout the fabrication process. These essential cutouts are positioned at the intersection of the bends and must be sized to properly clear the bend radius.

- **When to use them:** They are critical when working with low-ductility materials, such as certain aluminum alloys (e.g., AL6061) or any thicker-gauge metals (e.g., >0.075" thick), where material displacement is higher.
- **Sizing Guideline:** As a general rule, circular corner reliefs should have a diameter equal to or slightly larger than the sum of your material thickness and your inside bend radius.

Flat Pattern

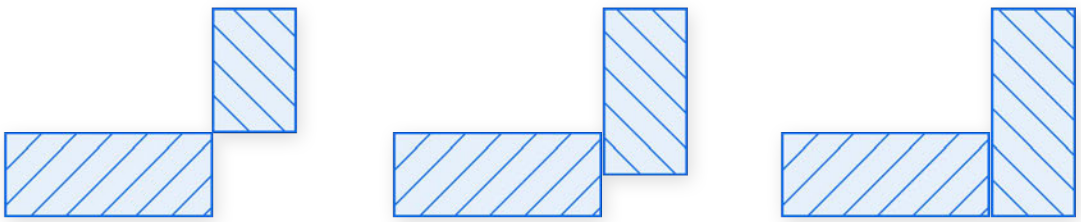


Bent Part

Gaps and Overlaps

When designing how your corner flanges meet, relying on your CAD software's default settings can often lead to unintended manufacturing issues. You must actively choose between a gap or an overlap based on your final assembly method:

- **Open Corners (Gaps):** Intentionally leaving a gap between flanges is a cost-effective choice because it requires slightly less bending precision. However, gaps should only be used for standalone parts, parts that will be mechanically fastened, or applications where the cosmetic appearance of an open gap is acceptable. Do not use gaps if the corner must be welded.
- **Closed Corners (Overlaps):** If the part requires welding or a seamless look, gaps must be closed using an overlap. There are three primary styles to consider:
  - **Corner-to-Corner:** Flanges meet perfectly at the apex, leaving a minimal, uniform seam. This enables full-penetration welding, resulting in an exceptionally strong structural joint.
  - **Half Overlap:** With one flange partially overlapping the other's thickness, this style increases welding efficiency for non-structural corners needing minimal filler, despite a slightly weaker joint.
  - **Full Overlap:** One flange entirely covers the other, providing a clean look without welding. For welded applications, this style is ideal for laser or autogenous (fusion) processes because no filler material is needed.

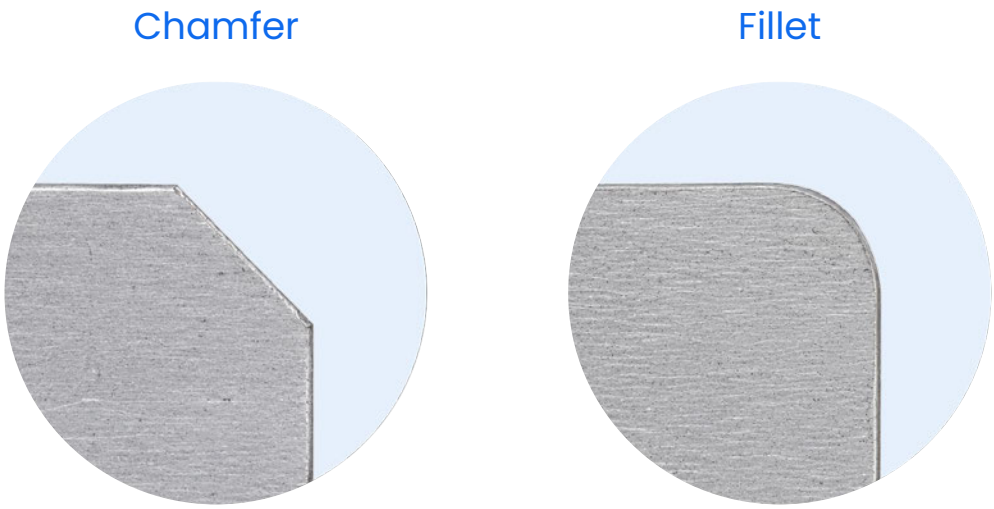


Corner-to-Corner | Half Overlap | Full Overlap

Corner Chamfers and Fillets

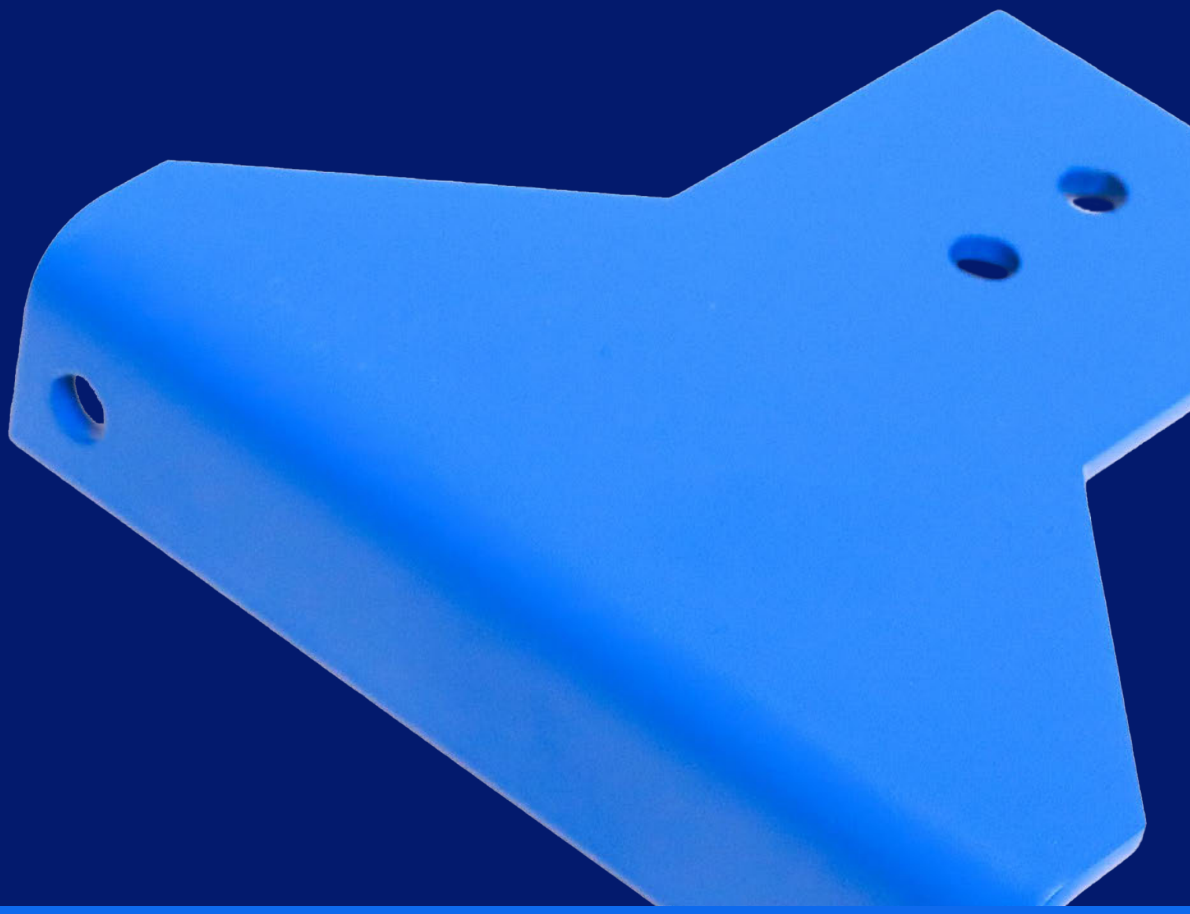
Applying a simple chamfer (an angled cut) or a fillet/radius (a rounded edge) to the sharp outer corners of your flat pattern requires minimal CAD effort but offers three distinct downstream benefits:

- **Eliminate Safety Hazards:** Dulling sharp corners prevents handling injuries for both the fabricators building the part and the end-users interacting with your product.
- **Improve Aesthetics:** Laser cutters can glide continuously around a radius or chamfer, producing a significantly cleaner, smoother cut than they can when pausing to change direction on a sharp 90-degree outer corner.
- **Aids Assembly:** Chamfers and radii simplify assembly by creating clearance between laser-cut corners and the bends of adjoining parts. This ensures fabricators have adequate space to precisely position parts before final fastening or welding.



SECTION 5

Finishes & Post-processing



Finishes & Post-processes for Sheet Metal

Xometry offers a wide range of finishing and post-processing options for sheet metal parts. The right finish will ultimately depend on your specific application and design goals. For example, will the part be highly visible and require a cosmetic appeal? Or will it prioritize function, requiring enhanced corrosion resistance to withstand the elements?

To make the selection process easier, we have organized our finishes into the following categories:

- **Mechanical & Chemical Surface Treatments**
- **Anodizing & Conversion Coatings**
- **Metal Plating**
- **Paints and Coatings**
- **Heat Treatments**
- **Part Marking**

The sections that follow provide detailed information on the specific options available within each category, helping you choose the perfect finish for your project.

Pro Tip: Keep in mind that some finishes can be combined (e.g., Anodizing after Bead Blasting, or Chem-Film before Powder Coat), but adding multiple steps will increase both cost and lead time.

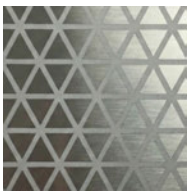
Mechanical & Chemical Surface Treatments

These processes alter the surface texture or cleanliness of the part, often by removing material or chemically cleaning it. They are useful for creating aesthetic matte finishes, deburring, or microscopic cleaning. Since these finishes are only treating the surface of the material, they generally do not affect thickness.

Bead Blasting: A pressurized stream of abrasive media (glass beads) is shot at the part to smooth out tool marks and create a uniform, matte texture.

Media Tumbling: Parts are rotated in a vibrating barrel with abrasive stones to deburr sharp edges and smooth the overall surface finish.

Electropolishing: An electrochemical process that removes a microscopic layer of material to leave a bright, clean, and highly corrosion-resistant surface.



Etching: A chemical bath is used to selectively dissolve material, often used for part marking or to create a specific surface profile for bonding.

Passivation: A chemical treatment, primarily for stainless steel, that removes free iron from the surface to enhance the metal's natural corrosion resistance.

Anodizing & Conversion Coatings

These methods use chemical or electrochemical reactions to convert the surface of the metal into a durable, protective layer. Unlike paints that sit entirely on top of the material, these coatings are integrated into the surface structure itself. They are primarily used to add corrosion resistance, electrical insulation, or a low-friction surface, typically with much less dimensional build-up than paints or powder coats.

Anodizing Type II: A standard electrochemical treatment that creates a corrosion-resistant oxide layer, which can be dyed various colors for cosmetic purposes.

Anodizing Type III (Hardcoat): A thicker, denser, and more durable oxide coating designed for applications requiring extreme wear and abrasion resistance.

Anodizing Type III w/ PTFE: Hardcoat anodizing infused with Teflon (PTFE) to create a self-lubricating surface with a very low coefficient of friction.

Chem Film (Chromate Conversion): A chemical coating that provides corrosion protection and maintains electrical conductivity, often serving as an ideal primer for paint.

Black Oxide: A chemical conversion coating that creates a matte black finish and offers mild corrosion resistance without adding significant thickness to the part.

Pro Tip: Type III (Hardcoat) anodizing adds measurable thickness. A standard hardcoat typically adds roughly **0.001" of build-up per surface**. Be sure to account for this dimensional change in your tolerances.

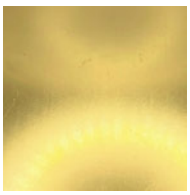
Metal Plating

Metal plating involves depositing a thin layer of another metal onto the substrate. It is an additive process, meaning it creates a physical build-up on the part surface that must be accounted for in the design. Platings are generally chosen to enhance specific mechanical or electrical properties, such as conductivity, solderability, corrosion resistance, or wear resistance.

Nickel (Electroless, Electrolytic, Sulfamate): A versatile plating used for corrosion resistance and hardness. Electroless Nickel is most popular for its uniform thickness, while electrolytic and sulfamate variants offer brighter finishes or greater ductility, respectively.

Tin: A soft, malleable white metal widely used to ensure excellent solderability on contacts and terminals.

Zinc: A cost-effective plating that acts as a sacrificial coating to protect steel parts from rusting.



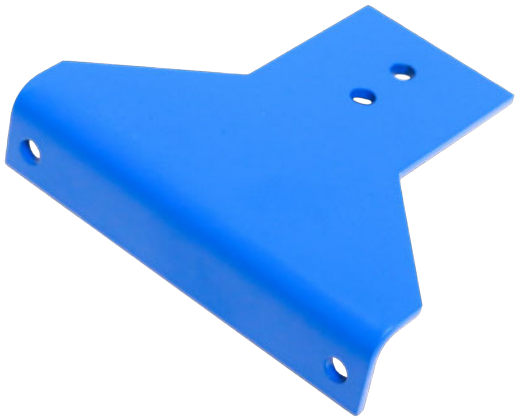
Gold: Offers superior electrical conductivity and oxidation resistance, making it the industry standard for high-reliability contacts and connectors.

Silver: Provides the highest electrical and thermal conductivity of any metal, though it is susceptible to tarnishing over time.

Paints and Coatings

This section covers finishing processes that apply a polymer or liquid layer over the sheet metal substrate. Like metal plating, these are additive processes that create physical build-up on the surface. However, paints and coatings typically add significantly more thickness than platings or conversion coatings, which must be carefully accounted for in your design. These finishes are primarily chosen to provide exceptional cosmetic appearances, custom color-matching, branding, and robust environmental protection. The two most common methods are Powder Coating and Wet Painting.

Powder Coating: A dry powder is applied electrostatically and then cured under heat to form a thick, hard, and uniform skin. It is highly resistant to scratching, chipping, and fading, making it the industry standard for durable, high-quality cosmetic finishes. Xometry offers various colors and sheens, and can accommodate custom requests if a specific powder is desired.



Wet Painting: Liquid paint is sprayed onto the part and allowed to dry or cure. While typically less durable than powder coating, it offers tighter control over thickness build-up and provides a virtually endless range of custom colors, gloss levels, and specialized textures.

Pro Tip: Thick coatings like powder coat typically add **.002"–.005" of material build-up per surface**. To ensure proper fit for fasteners and hardware, design holes and slots with extra clearance or specify masking requirements for critical features.

Part Marking

Part marking is often the final step required to add text, graphics, or branding to your sheet metal components. Markings are also used for enhancing part identification and traceability by applying part numbers or barcodes directly to parts or their packaging. Depending on the chosen method, a marking can be performed using an additive process, such as applying ink, or a subtractive/altering process, like burning or indenting the metal. Choosing the right method depends heavily on the part's final finish, required durability, and desired cosmetic appearance.

Silkscreening: Uses a woven mesh and stencil to apply crisp, durable ink directly onto the surface. It is ideal for multi-color logos, front panels, and detailed instructional graphics.

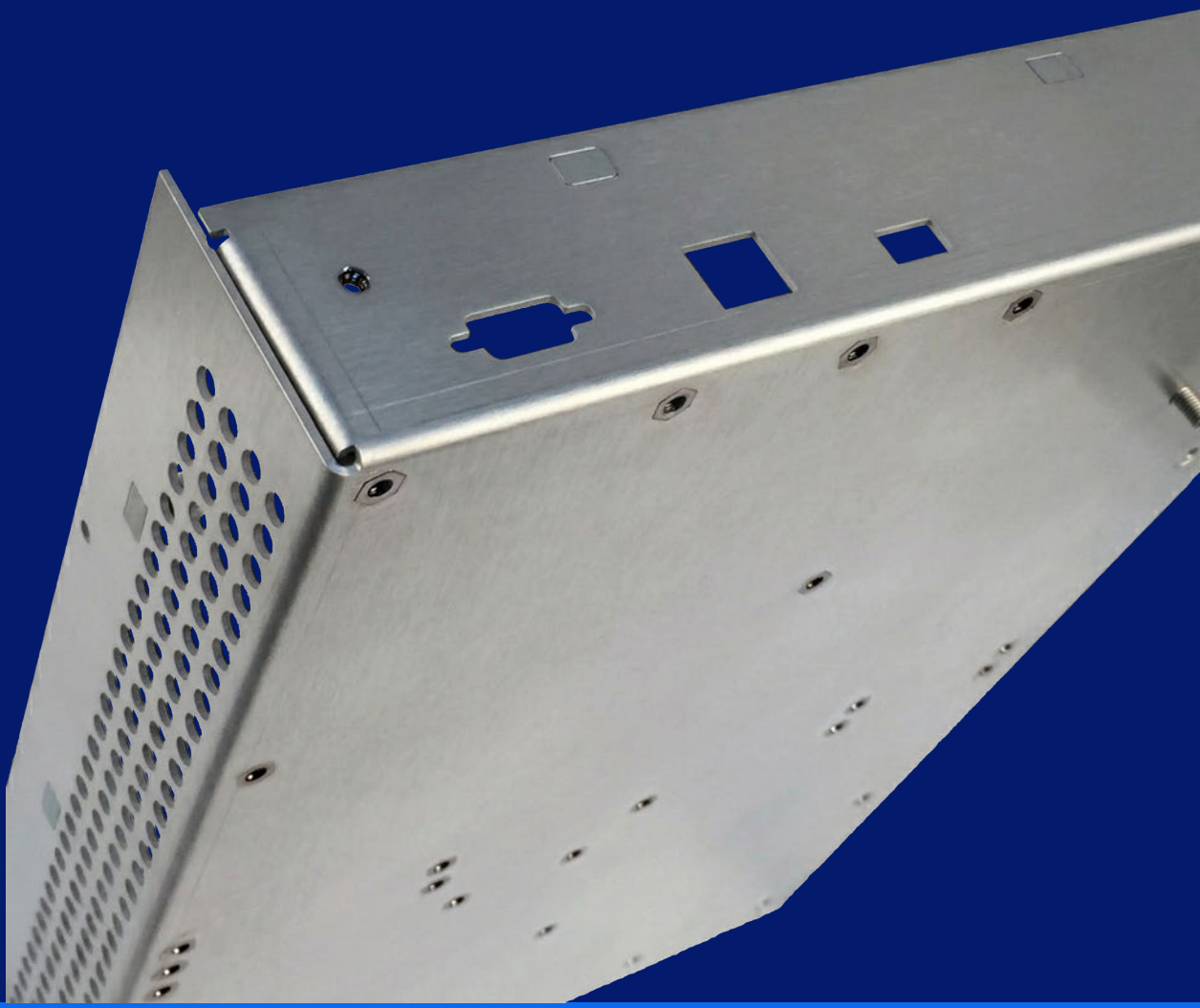
Ink Stamping: A fast, highly cost-effective method for applying basic text, date codes, or part numbers. It is best suited for internal components where cosmetic perfection is a secondary concern.

Laser Marking: Uses a highly focused laser beam to ablate or permanently discolor the surface material. This provides an extremely precise, permanent mark that is excellent for barcodes, serialized numbers, and branding on bare or anodized metals.

Bag and Tagging: Instead of marking the physical part, the component is packaged in a bag that receives a printed label. This is the perfect identification solution for parts that are too small to mark, or where a physical mark would compromise the part's cosmetic finish or functional integrity.

SECTION 6

Hardware & Assembly



Assembly Methods

Once parts are cut and formed, they are often joined into sub-assemblies or complete products. The choice of joining method affects the assembly's strength, appearance, cost, and serviceability. Common approaches include welding for permanent, high-strength bonds; brazing and soldering for joining delicate or dissimilar metals with less heat; and the integration of mechanical fastening to join components and provide robust threading in thin materials. Each method offers unique advantages and requires distinct design considerations based on the project's requirements.

Welding

Welding creates a permanent, high-strength joint by melting and fusing base materials together, often with the use of a filler material. It produces the strongest possible joint, and the processes range from precise, manual arc welding to high-speed, automated resistance welding.

Welding Processes

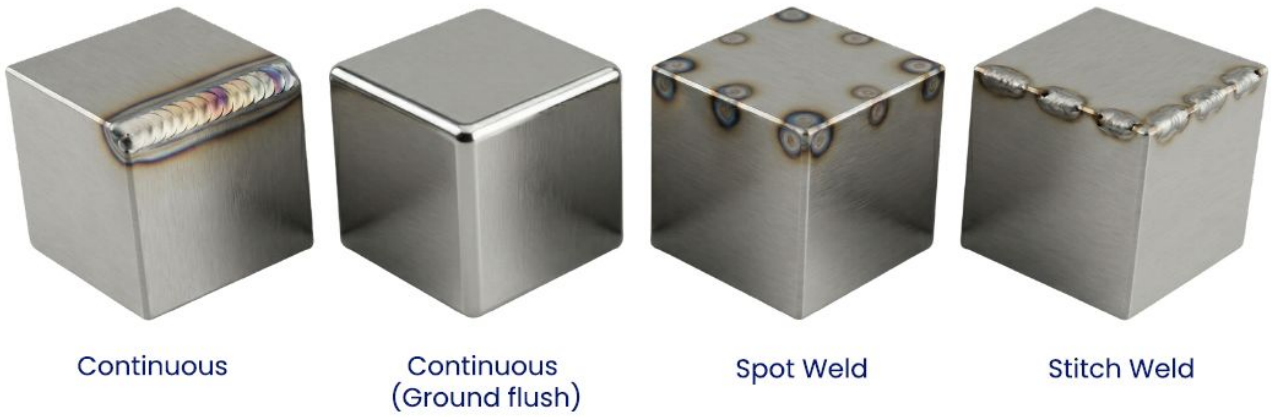
Arc welding uses an electric arc to melt base metals and filler materials, typically with a shielding gas to protect the weld. Common methods include TIG welding, which provides exceptional precision and clean finishes for thin materials; MIG welding, a faster, highly versatile option using a continuously fed wire; and Stick welding, a portable, self-shielding method best suited for outdoor applications where cosmetic perfection is less critical.

In contrast, **resistance welding** joins overlapping sheet metal using applied pressure and electrical current without the need for filler material. This category includes Spot welding, a high-speed process that creates localized fused points ideal for the rapid assembly of flat parts, and Seam welding, which uses rotating wheel electrodes to form a continuous, leak-tight seal for components like tanks and ducts.

Selecting the Optimal Weld Type

Selecting a weld requires balancing airtight/watertight seals, aesthetics, and manufacturing costs. Use the chart below to quickly compare the most common weld types and determine the best fit for your project.

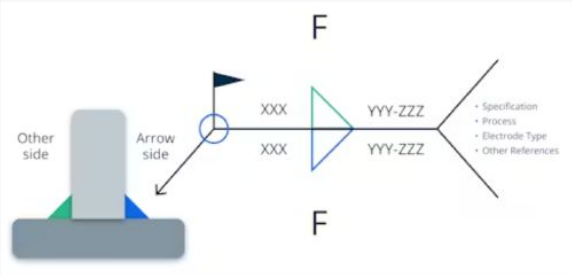
Weld Type	Result/Appearance	Common Use Case	Cost Impact
Continuous	Guaranteed seal, uniform bead	Tanks, reservoirs, pressure piping	Moderate ($$)
Continuous (Ground flush)	Flush-ground finish	Designer furniture, aesthetic housings, food industry	High ($$$)
Spot Weld	Discreet (small discs), very low distortion	Thin sheet metal assembly, ideal for structural bonding	Low ($)
Stitch Weld	Good mechanical strength, but not watertight	Structural frames, internal reinforcements, thin enclosures	Low ($)



Calling Out Welds

To ensure your manufacturing requirements are met without misinterpretation, it is essential to provide clear weld specifications. The following graphics detail the components of a comprehensive weld callout and the industry-standard symbols utilized by manufacturers to execute your precise design intent.

Basic Welding Symbols



XXX	Size of the weld
ZZZ	Pitch (center to center distance)
YYY	Length of the weld
F	Root angle, Root opening
	Field weld
	Weld all around

Supplementary Welding Symbols

TYP	Tail, for notes
	Weld all around
	Flush contour
	Backing strip or bar
	Flush surface finish
	Convex surface finish
	Concave surface finish
	Backing weld run

The checklist below can be used to ensure your weld callouts are well defined and ready for the manufacturing floor.

- ☐ **Identify weld placement –** Arrow side, other side, or both sides
- ☐ **Specify weld sizing –** Specify throat thickness (a) or required leg length (z)
- ☐ **Specify weld pattern –** Indicate length and pitch for intermittent/stitch welds
- ☐ **Define weld class/stringency –** Define the requirement level (Standard or Critical / ISO 5817)
- ☐ **Define post-weld treatment –** Specify as-welded, brushed, or ground flush

Material Applications and Considerations

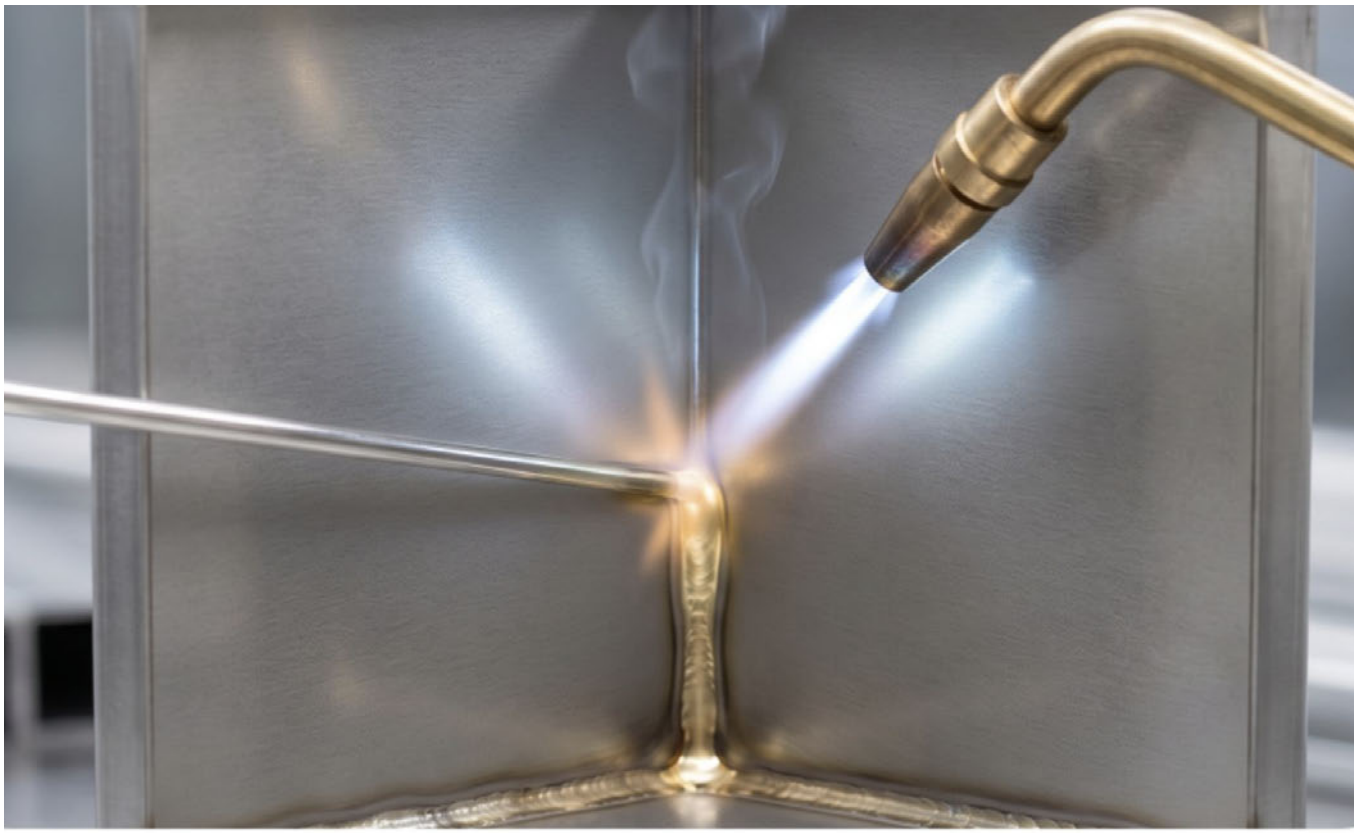
Most common metals can be welded, but the technique and filler must be chosen carefully.

- **Carbon steel** offers excellent weldability.
- **Stainless steel** requires careful heat control to prevent carbide precipitation, which reduces corrosion resistance.
- **Aluminum** is more difficult due to its rapid oxide formation and high thermal conductivity, requiring a clean surface and proper gas shielding (typically with TIG or MIG).

A major limitation of welding is the need to remove coatings (such as paint or powder-coat) at the weld joint, which traditionally adds complexity by forcing the part through multiple specialized suppliers for fabrication, welding, and final coating. When you order complete assemblies including post-processing through Xometry, we account for this complex workflow to ensure efficient production and fast lead times. As your single source for cutting, bending, welding, and assembly, Xometry simplifies the entire process.

Brazing and Soldering

Brazing and soldering are joining processes that utilize capillary action to draw a molten filler metal into the tight gap between closely fitted parts, creating a strong bond without melting the base metals. This lower-temperature approach avoids many of the thermal distortion issues associated with welding. The primary difference between the two methods lies in the filler material's melting point: **brazing** uses filler metals—such as silver alloys or copper-phosphorus alloys—that melt above **450°C**, while **soldering** relies on lower-temperature alloys, such as tin-lead or lead-free variants, that melt below **450°C**.



Advantages and Limitations of Brazing and Soldering

Advantages	Limitations
Low Heat Input: Reduces the risk of distortion (warping) and does not create a large HAZ, preserving base metal properties.	**Lower Strength:** The joint strength is limited to the strength of the filler metal, not the base metal.
Joins Dissimilar Metals: An excellent method for joining dissimilar metals (e.g., copper to steel) that cannot be easily welded.	**Precise Temperature Control:** The process requires careful temperature control to melt the filler but not the base.
Clean Appearance: Can produce very clean, neat joints with minimal post-processing.	**Flux Removal:** The flux used to clean the surface must be thoroughly removed after joining to prevent corrosion.

Material Applications and Considerations

Brazing and soldering are common for copper, brass, and many steels. Aluminum and stainless steel can be joined, but they require specific fluxes and filler alloys to overcome their protective oxide layers.

Soldering is generally used for thinner sheets (up to 3 mm for copper, less for steel), while brazing can be used on parts up to 6 mm thick.

Joint design is critical for both brazing and soldering. To allow for proper capillary action, the gap between the mating parts must be extremely small and consistent, typically in the **0.05 mm to 0.2 mm range**.

Mechanical Fastening and Inserts

Mechanical fastening is an essential joining method that provides distinct advantages over thermal processes like welding. Because it relies on physical hardware rather than intense heat, it produces no heat-affected zone (HAZ), easily joins dissimilar or pre-coated materials, and can be performed after final surface finishing. Importantly, mechanical fastening often creates removable joints, allowing for product disassembly, routine maintenance, and repair.

To facilitate these robust connections in thin sheet metal, features like internal threads and boss standoffs are typically added using press-fit inserts, with PEM® Fasteners being the industry standard. Common installable hardware Xometry supports includes:

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

Pro Tip: Always design your hole sizes and edge distances according to the manufacturer's specific guidelines for your chosen off-the-shelf insert. Be sure to clearly call out the manufacturer's exact part number and the required installation direction on your accompanying 2D print to ensure accurate assembly.

Sheet Metal Hardware Comparison Chart

Here is a simple, quick-reference chart comparing the most common types of press-fit hardware and inserts used in sheet metal design:

Insert Type	Primary Function	Key Advantage	Common Application
Press-Fit Nuts	Provides a strong, permanent female thread in thin sheet metal.	Remains completely flush on the reverse side of the sheet.	Securing brackets or mounting plates to an enclosure.
Press-Fit Studs	Provides a permanent protruding male thread (or unthreaded pin).	Eliminates the need to hold a loose bolt behind the panel during assembly.	Creating grounding terminals or mounting heavy components.
Standoffs	Acts as a threaded spacer to separate or stack multiple components.	Ensures perfectly consistent spacing between layers without loose washers.	Stacking printed circuit boards (PCBs) inside a chassis.
Captive Screws	A threaded screw that remains permanently attached to the panel, even when fully unthreaded.	Prevents dropped or lost hardware from falling into critical assemblies.	Removable access doors, server racks, and maintenance panels.
Cable-Tie Mounts	Provides a secure, low-profile loop for zip-ties to pass through.	Eliminates the need for adhesive mounts that can degrade and peel off over time.	Routing and securing wire harnesses inside electronic enclosures.

SECTION 6

Additional Resources

Sheet Stock Sizes and Gauge Lookup Chart

Gauge	Steel (in.)	Steel (mm)	Galvanized (in.)	Galvanized (mm)	Stainless (in.)	Stainless (mm)	Aluminum (in.)	Aluminum (mm)
3	0.2391	6.07						
4	0.2391	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

Sheet Stock Sizes and Gauge Lookup Chart (Cont.)

Gauge	Steel (in.)	Steel (mm)	Galvanized (in.)	Galvanized (mm)	Stainless (in.)	Stainless (mm)	Aluminum (in.)	Aluminum (mm)
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

Stock Material Thickness Tolerances

Sheet metal material stock has general tolerances for thickness. The following table provides a quick reference to thickness tolerances for standard aluminum 5052-H32, one of the most popular sheet metal materials.

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"
0.032"	-0.003" to 0.003"

Standard stock material tolerances may vary between alloys and treatments. Higher tolerance sheet materials are available, but may incur increased cost.

Additional Resources at Xometry

Upload your 3D CAD files to the **Xometry Instant Quoting Engine®** to get an instant quote for custom sheet metal components and more!

Online Resources:

- Sheet Metal Services Page
- Sheet Cutting Services Page
- Standard Sheet Thicknesses
- Metal Finishes Gallery
- Sheet Cutting Design Guide

Tools & Calculators:

- Sheet Metal Bend Radius Calculator
- ISO 286 Limits and Fits Calculator
- Find more on Xometry Pro

CNC MACHINING

- 5-axis capabilities
- Wire and RAM EDM
- Exotic materials

ADDITIVE

- 8+ printing processes
- Aerospace-grade polymers
- Tool-free metal parts

SHEET & TUBE FABRICATION

- Advanced forming capabilities
- Braking, punching, stamping, welding
- MIL-spec finishing options

INJECTION MOLDING & CASTING

- Instant pricing & lead times
- Low to high volume solutions
- All commercial materials & finishes

METAL TOOLING

- High-volume production
- Extrusion, die casting & more
- Support from industry experts

ASSEMBLY

Value-Added Service

- Sub-assemblies
- Electromechanical
- Precision grinding & welding

LIVE SUPPORT

Hours: Mon - Fri 8:00 AM - 8:00 PM EST | Sat - Sun 9:00 AM - 5:00 PM EST

1-800-983-1959 support@xometry.com