Sheet Metal Fabrication

The complete design guide

UPTIVE
ADVANCED MANUFACTURING

Table of contents

3 Overview

How sheet metal fabrication works
Sheet metal fabrication
Benefits of sheet metal fabrication
Limitations of sheet metal fabrication

8 Forming Basics

Press brake forming
Bending
Springback
Bend relief and height
Forming near holes

14 Cutting

Laser cutting
Plasma cutting

17 Design Guidelines

Curls
Hems
Holes and slots
Notches and tabs

22 Materials

24 Additional Resources

Overview

UPTIVE
ADVANCED MANUFACTURING

How sheet metal fabrication works

Sheet metal fabrication is a subtractive manufacturing process that create a part from thin metal sheets into cuts, bends, and formed parts or assemblies directly from your CAD file.

Materials include aluminum, stainless steel, cold-rolled steel, and beryllium copper. Each metal of alloy is selected based on mechanical properties, aesthetics, and performance characteristics based on part or component specifications.

How sheet metal fabrication can help you:

- Choose materials for desired properties: aesthetics, corrosion resistance, higher strength-to-weight ratio, etc...
- Remove the need for tooling with more automated prototype-to-production sheet metal fabrication solutions.
- Save production costs by eliminating potential design errors that can negatively impact part maneuverability.
- Increase accuracy for rapid prototypes in quantities as low as one, or for higher-volume production.



Sheet metal fabrication



Cutting

Laser, plasma, waterjet, shearing, and sawing...we combine traditional and advanced cutting methods for just about any kind of metal or alloy with a wide variety of sheet stock sizes, tolerances, and thickness to ensure your design integrity.



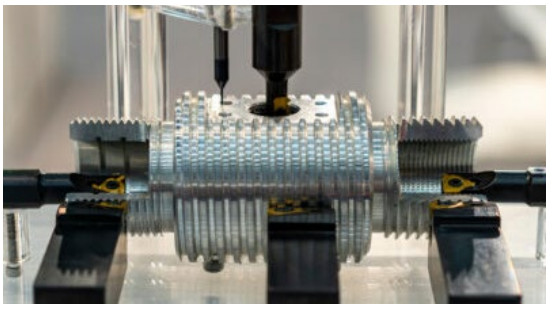
Stamping

From bending and punching to progressive die-stamping, we can ensure every one of your parts meets your performance expectations - even for complex designs for mass production.



Welding

MIG, TIG, robotic... we offer a wide range of welding processes, materials, and finshes based on your specific application, including any post-processing required for your technical specifications.



Value-add hardware

We have more hardware options to alleviate your supply chain headaches, including standos, threaded inserts, threaded nuts, captive screws, pins, and more based on your supply chain requirements.



Finishing

Powder coating, buff polishing, sand blasting, brushing plating, and a wide selection of finishing levels available for all processes to produce targets ranging from unfinished to photo-realistic results.

Benefits of sheet metal fabrication

Versatility

There are a wide variety of materials that can be used in sheet metal fabrication, like steel, aluminum, brass copper and more. Sheet metal fabrication is also suitable for a wide range of industries including automotive, aerospace, electronics, and more.

Cost-effective

There is less material waste in sheet metal fabrication compared to other methods. It is also an economical solution for both small and large-scale production runs.

Strength and durability

Many of the sheet metal fabrication metals offer significant strength while remaining lightweight. There is also a lot of corrosion-resistant materials, especially with treatments like galvnizing.

Precision and accuracy

The use of modern CNC machinery ensures high precision and repeatability, and also sheet metal fabrication is capable of achieving tight tolerances for complex parts.

Speed and efficiency

Sheet metal fabrication is able to meet quick turnaround times for prototypes and final products. It also has high production speeds for processes like stamping, laser cutting, and bending.

Customization and flexibility

Sheet metal fabrication allows for creating complex shapes and designs, and it is easier to make design modifications without significant cost increases.

Surface finishes

Sheet metal fabrication has options for various finishes such as painting, powder coating, anodizing, and more to improve aesthetics and functionality.

Limitations of sheet metal fabrication

Material thickness constraints

Sheet metal fabrication materials are not suitable for very thick or very thin materials beyond certain limits.

Initial setup cost

There is a high initial setup cost for the custom tooling and machinery for sheet metal fabrication.

Complexity and design restrictions

Some complex shapes may be difficult or impossible with sheet metal fabrications, and there are also limitations on shape bends and very tight radii.

Material properties

Certain metals may be brittle and prone to cracking under stress, and high thermal conductivity of metals ban poses challenges in welding and other thermal processes.

Precision limitations

Accumulation of tolerances can affect the final assembly if not managed properly.

Surface treatment

The need for additional surface treatments or coatings to prevent corrosion and improve appearance can add to costs and processing time.

Environmental concerns

Scrap metal and waste materials can be an environmental concern if not recycled properly.

Forming Basics

UPTIVE
ADVANCED MANUFACTURING

Press brake forming

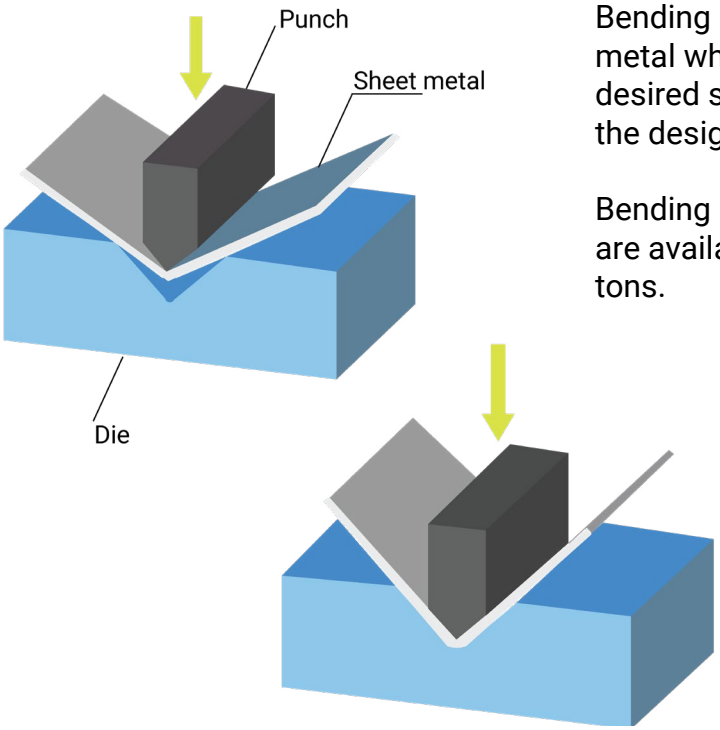
Press brake forming is the process where metal is bent or formed into shapes. The process works when a die and a punch are used to bend the sheet of metal.

This is most often used to form parts from ferrous alloy sheets and plates. Press brake forming is great for both prototype and production runs.

The maximum sheet thickness is ideally **less than 6.5mm**, but thickness **up to 25mm**.



Bending



Bending is a process where a force is applied to sheet metal which causes it to bend at an angle and form the desired shape. Bends can be short or long depending on the design requirements.

Bending is performed by the press brake machines, which are available in a variety of sizes and lengths, from 20-200 tons.

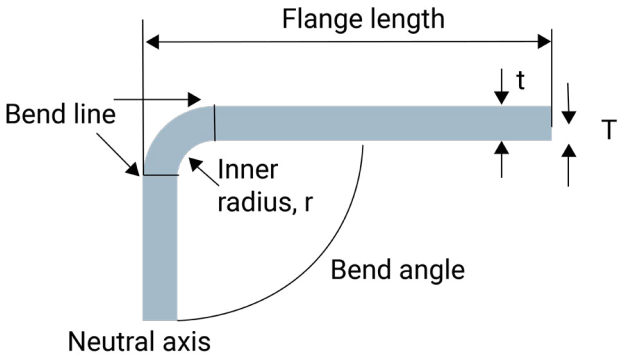
Parts to be bent are supplied as flat patterns with the bending information, including bend etches or notches.

Bending

Bend line - The straight line on the surface of the sheet, on either side of the bend, that defines the end of the level flange and the start of the bend.

Bend radius - The distance from the bend axis to the inside surface of the material, between the bend lines.

Bend angle - The angle of the bend, measured between the bent flange and its original position, or as the included angle between perpendicular lines drawn from the bend lines. Sometimes specified as the inside bend radius. **The outside bend radius is equal to the inside bend radius plus the sheet thickness.**



Neutral axis- The location in the sheet that is neither stretched nor compressed, and therefore remains at a constant length.

Bend allowance- The length of the neutral axis between the bend lines or the arc length of the bend. The bend allowance added to the flange lengths is equal to the total flat length.

K-factor - The location of the neutral axis in the material, calculated as the ratio of the distance of the neutral axis (T), to the material thickness (t). The K-factor is dependent on several factors (material, bending operation, bend angle, etc..) and it is **greater than 0.25, but not greater than 0.50**.

$$K = \frac{180^{*}(\text{Allowable Bend})}{\pi\,^{*}(\text{Bend Angle})\,^{*}T} - \frac{r}{T}$$

K-factor Chart

Radius	Soft/Aluminum	Hard/Stainless Steel
Air Bending		
0 - t	.33 - .38	.40
y - 3*t	.40 - .43	.45
3*t. - > 3*t.	.50 - .50	.50
Bottom Bending		
0 - t	.42 - .44	.46
y - 3*t	.46 - .47	.48
3*t. - > 3*t.	.50 - .50	.50

Springback

Springback is what occurs when a metal is bent. When the bending force is distributed across the surface is not enough to overcome the material's modulus of elasticity, elastic deformation occurs, not plastic deformation.

This elastic deformation impedes permanent deformation needed for a bend to remain. Interestingly, near the geometrical center of sheet metal, there is a zone with low stress that only experiences elastic deformation, even when under large bending forces. This means that this part is susceptible to springback.

As the stiffness of a material increases, the amount of springback you can expect increases. As a general rule of thumb, this is the maximum amount of springback that may occur (you should plan for this variation when designing sheet metal parts):

- Soft metals like copper and aluminum: 0.5°
- Steel: to 1°
- Stainless steel: 3°

Bend angle also plays a crucial role. The smaller the relative effect on the elastic area in the neutral zone, the smaller the springback. This is particularly true with small bend angles and small bend radii (a sharp tool).

In general, you can expect less springback for a sharper tool, smaller bend angles, and smaller bend radii.

The expected springback may be calculated as follows:

$$D = a \frac{R}{2.1*t}$$

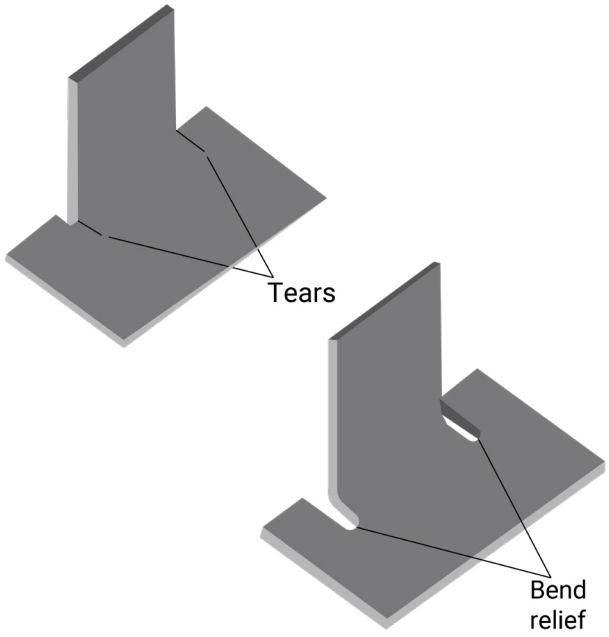
D= springback angle expected
R= radius of the bend angle
t= sheet metal thickness
a= correction factor

Bend relief and height

When a bend is made close to an edge the material may tear unless bend relief is given.

The image to the left shows a torn part, while the part on the right shows bend reliefs cut into the part.

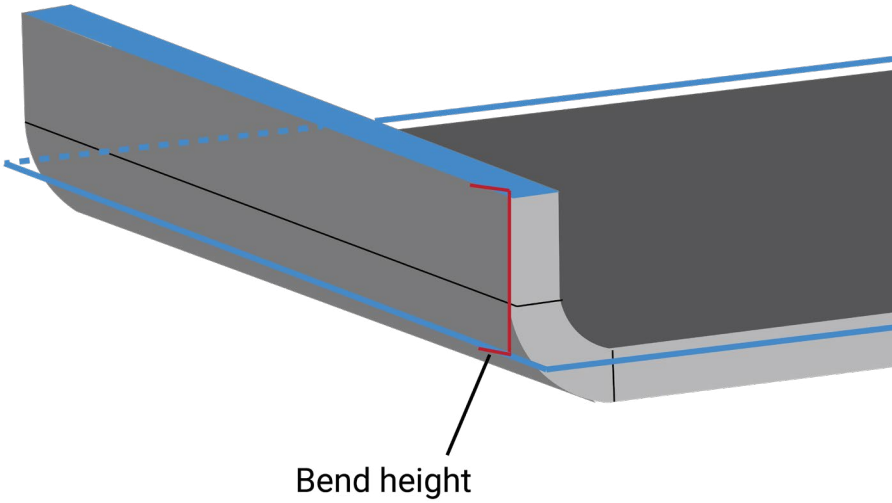
The depth of the relief should be greater than the radius of the bend. The width of the relief should be a **material thickness or greater, preferably a material thickness + 1/64".**



Bend height should be at least twice the thickness of the sheet metal plus the bend radius.

$$H = 2t + r$$

If the bend height is too small, it will result in deformation and low bending quality.



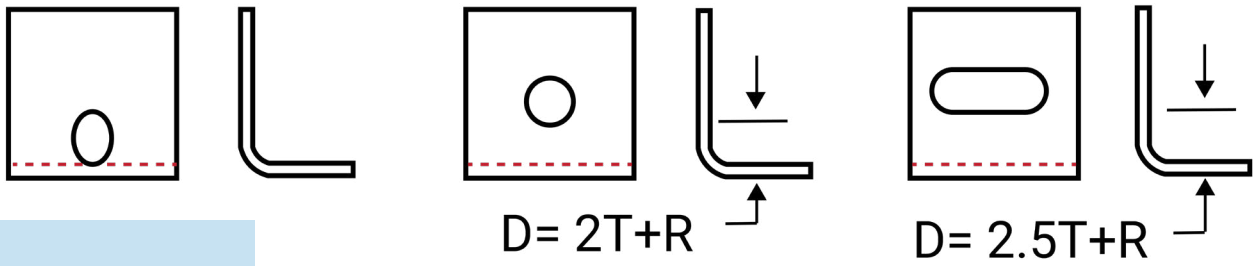
Forming near holes

When a bend is made too close to a hole, the hole may become deformed. To save the cost of punching or drilling in a secondary operation, the following formulas can be used to calculate the minimum distance required:

As a rule of thumb, the distance from the outside of the material to the bottom of the cutout should be equal to the minimum flange length as prescribed by the air bend force chart.

For a **hole <1" in diameter** the minimum distance "D" = 2T+R
For a **slot or hole >1" diameter** then the minimum distance "D" = 2.5T +R

When using a punch press, or laser cutting, holes should never be less than that of the material thickness.



D= Minimum distance
T= Material thickness
R= Bend radius

Feature Type	Minimum distance guidelines
Between hole and a bend	±2.5 times the material length + bend radius
Between slot and a bend	±4 times the material thickness + bend radius
Between a hole or slot and edge	±2 times the material thickness
Minimum distance between holes	±2-3 times the material thickness
Between an extruded hole and an edge	±3 times the material thickness
Between an extruded hole and bend	±3 times the material thickness + bend radius
Between semi-pierced hole and a bend	±3 times the material thickness + bend radius
Between semi-pierced holes	±8 times the material thickness

Cutting

UPTIVE
ADVANCED MANUFACTURING

Laser cutting



Laser cutting is a type of production that uses a laser to cut different metals. The laser has a high-energy beam that easily burns through the material.

Localized Hardening

Localized hardening takes place on the edges where the laser has cut. This hardening produces a durable and smooth edge without the need for finishing after using the laser cutter.

Distortion

The heat-affected zone (HAZ) is produced during laser cutting. In carbon steel, the higher the hardenability, the greater the HAZ. Distortion from laser processing is a result of the sudden rise in temperature of the material near the cutting zone.
Distortion is also created by the rapid solidification of the cutting zone.

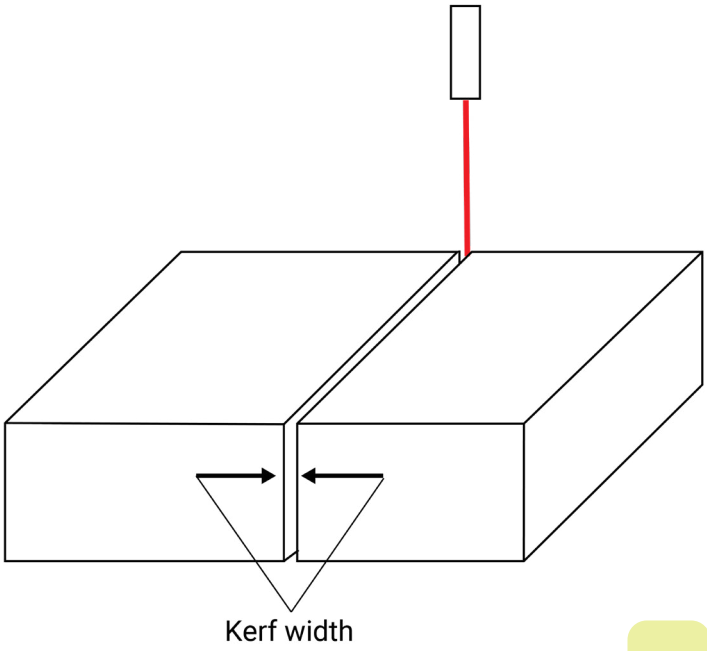
Hole diameter

The entry diameter is larger than the exit diameter, creating a slightly tapered hole. Minimum hole sizes are related to stock thicknesses and can be as low as 20% of the stock thickness.

Kerf

In laser cutting, “kerf” refers to the width of the material that is removed by the laser beam as it cuts through the sheet metal. It is a crucial consideration in sheet metal fabrication because it affects the precision and fit of the final product.

The kerf size is influenced by various factors, including the type of material, the thickness of the sheet, the laser power, and the cutting speed.



Plasma cutting



While handled plasma cutters are available, a CNC plasma cutter is the best option for cutting parts.

In plasma cutting, a gas is superheated with an electric arc and compressed to the 4th state of matter, plasma that flows through the sheet metal. Superheated means up to 40,000°F; the electric arc is about 5,000 VAC.

A precision CNC plasma cutter has a uniquely shaped shield cap and orfice diameter to reduce the plasma arc, resulting in a smaller kerf and low bevel angles, which means a much cleaner cut.

Plasma cutting is not ideal for thin, detailed parts that require tight tolerances or high precision. The kerf is much wider, and the cut edges are not as clean laser-cut edges.

Hole diameter

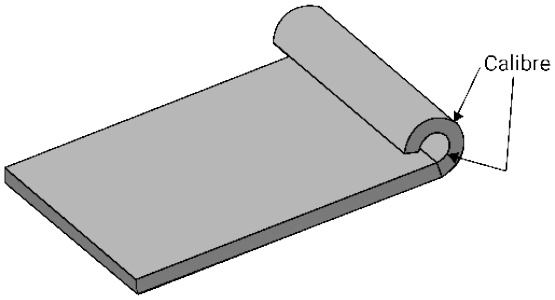
1. The hole diameter cut with a plasma cutter should be greater than or equal to the sheet metal thickness. A smaller hole will result in quality issues
2. Spacing parts creatively and trying to connect them will reduce the amount of time the machine has to start and stop resulting in a cost/piece reduction
3. Since the kerf is larger with plasma cutting, you need to account for it during design. Subtract half the kerf size from the part's frame and then add the other half to the inner section
4. Adding cutting nodes opposite each slot will reduce tension during cutting and lead to less distortion
5. Utilize part nesting software to reduce waste as much as possible
6. Parts must have a smooth radius on all sharp corners. the plasma kerf will not allow for shape corners

Design Guidelines

UPTIVE
ADVANCED MANUFACTURING

Curls

Curls in sheet metal fabrication are employed to impart strength to the edge of the sheet and mitigate the exposure of sharp edges, enhancing both safety and durability. Curls can be designed to be off-center or on-center, depending on the specific application and design requirements.



Key Design Guidelines for Curls

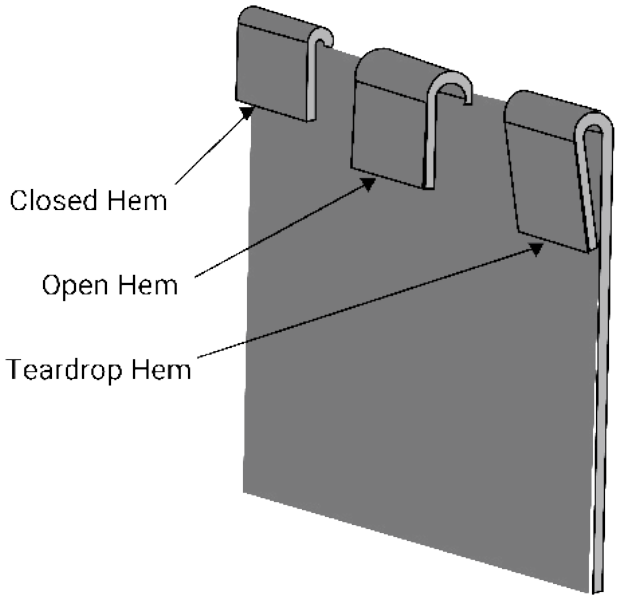
- **Outside Radius**: It should be at least double the thickness of the material to ensure structural integrity
- **Curl Opening Radius**: Utilizing a radius that is twice the material thickness results in a curl opening radius that is equal to the material thickness
- **Opening Size**: The opening of the curl should be at least equivalent to the thickness of the material to maintain consistency and strength

Feature Type	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

Hems

Hems are an essential element in sheet metal design, enhancing the strength of edges and eliminating sharpness, though they contribute additional weight to the part.

Flat hems can lead to material fractures, so the design should focus on open or teardrop hems to mitigate the risk.



Key Design Guidelines for Hems

Open Hems
- Larger diameters may lose their shape
- The return flange should be at least 4x the material thickness for stability
- The inside diameter should be at minimum equal to the material thickness

Teardrop Hems
- The inside diameter should be at minimum equal to the material thickness
- Openings should at minimum be 1/4 the material thickness
- The return flange should be 4x the material thickness

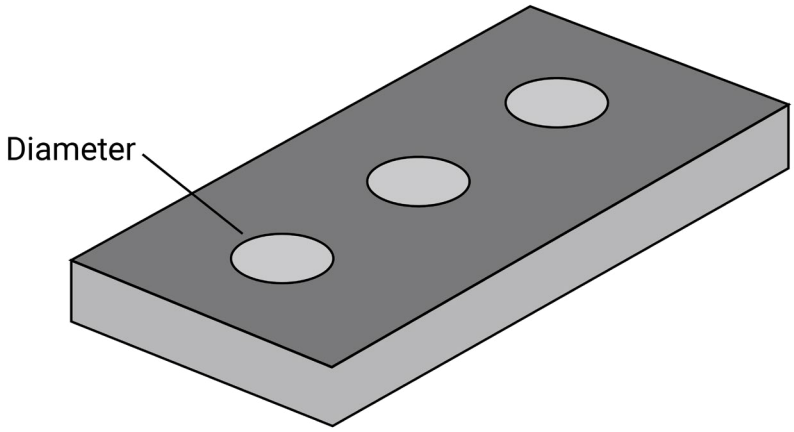
Feature Type	Minimum distance guidelines
Between hem and a hole	±2 x material thickness + radius of the hem
Between hem and an internal bend	±5 x material thickness
Between hem and an external bend	±8 x material thickness

Holes and slots

In sheet metal manufacturing, the design of holes is a critical aspect that requires precision and adherence to specific guidelines. The diameter of holes should be at least equivalent to the material's thickness to prevent excessive burring and ensure the longevity of the part. Proper spacing between holes is also essential to maintain their shape during subsequent processing steps.

Key Design Guidelines for Hole Design

- **Diameter Requirements:** Holes and slots should have a diameter at least equal to the
- material thickness or 1.00mm (0.04"), whichever is greater. In the case of alloy or stainless steel, the diameter should be at least twice the material thickness
- **Consideration for Material Strength:** For materials with higher strengths, the requirement for the diameter of holes and slots increases to ensure structural integrity and functionality

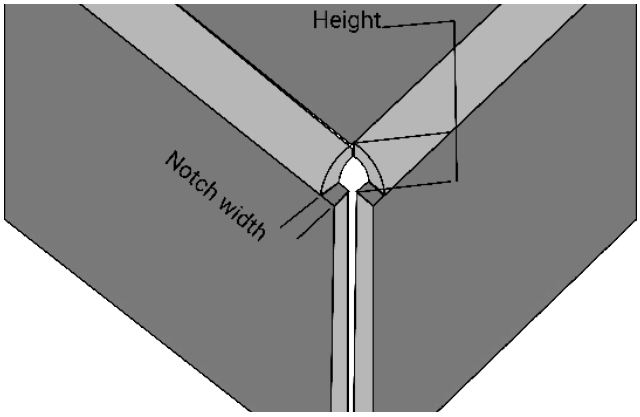


Notches and tabs

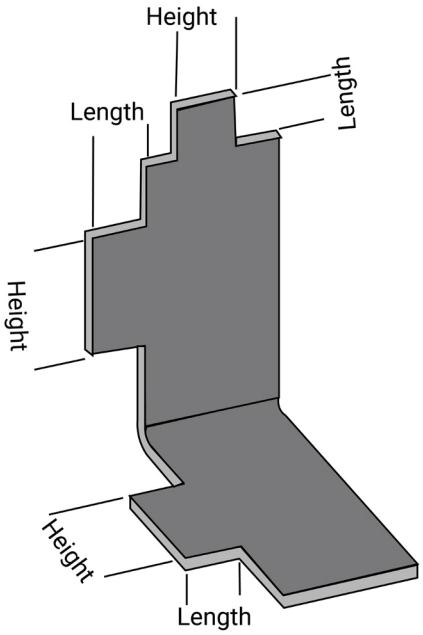
Notches and tabs are two approaches to get additional space in a sheet metal design to avoid interference between features and to allow for tool access during fabrication. Notches involve the removal of an unwanted section of sheet metal from the outer edge of the part, while tabs are protrusions extending from the edge, often used for bending or joining purposes.

Key Design Guidelines for Notches

- Thickness of at least 1mm or equivalent to the material thickness, depending which is greater
- The length of a straight or radius end notch does not exceed 5x the material thickness
- The length of V notches should be no more than 2x its width
- The corner radius of a notch should be 0.5 x the material thickness



Key Design Guidelines for Tabs



- Minimum width should be 3.2mm (0.126”) or two times the material thickness, whichever is greater
- The depth of a tab should not exceed 5 times its width

Feature Type	Minimum distance guidelines
Between notch and bend in a perpendicular plane	±3x material thickness + bend radius
Between notch and bend in parallel plane	±8x material thickness + bend radius
Between notch and hole	±1.2x material thickness
Between notches	±3.200mm (0.125”) or 2x material thickness (whichever is greater)
Between tabs	±1mm (0.04”) or material thickness (whichever is greater)

Materials

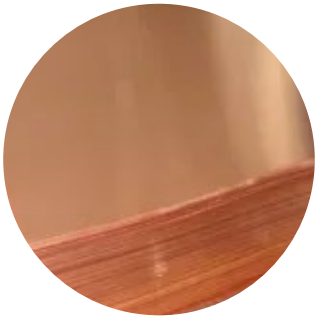
UPTIVE
ADVANCED MANUFACTURING

Materials



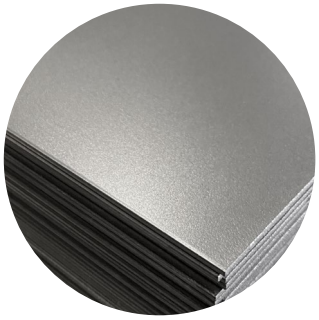
Aluminum

Lightweight and strong material that is known for its good corrosion resistance and machinability.



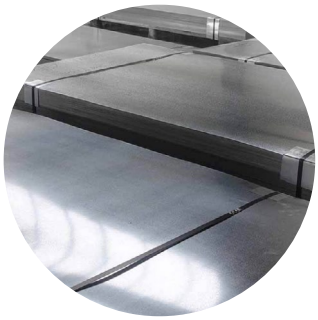
Beryllium Copper

High-strength, non-magnetic alloy that contains beryllium and copper.



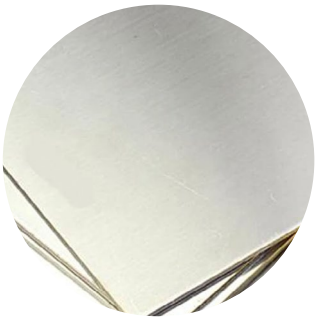
Cold-Rolled Steel

Steel that has been rolled at room temperature to increase its strength and smoothness.



Pre-Plated Cold-Rolled Steel

Cold-rolled steel that has been pre-coated with a layer of metal, such as zinc or tin, for corrosion resistance.



Stainless Steel

Corrosion-resistant steel that contains chromium.

Start your sheet metal project

Get a Quote

Head over to Get a Quote and upload your CNC ready design file with your information.
Accepted file formats: .dwg, .dxf, .igs, .pdf, .sat, .step, .x_t

Other UPTIVE Capabilities

- Additive Manufacturing
- Fused Deposition Modeling
- HP Multi Jet Fusion
- Selective Laser Sintering
- Direct Metal Laser Sintering
- Rapid Prototyping
- Injection Tooling & Molding
- CNC Machining
- Post-Processing & Finishing

Other Resources

- What is Sheet Metal Fabrication?
- Bending, Folding and Shaping: The Art of Sheet Metal Fabrication

Let's Connect

Email: hello@uptivemfg.com
Phone: (888) 467-7686
Website: Uptivemfg.com