



Sheet Metal Fabrication

Design Guide

Unlock Precision in Sheet Metal Design: Your
Essential Guide to Making Flawless Creations.

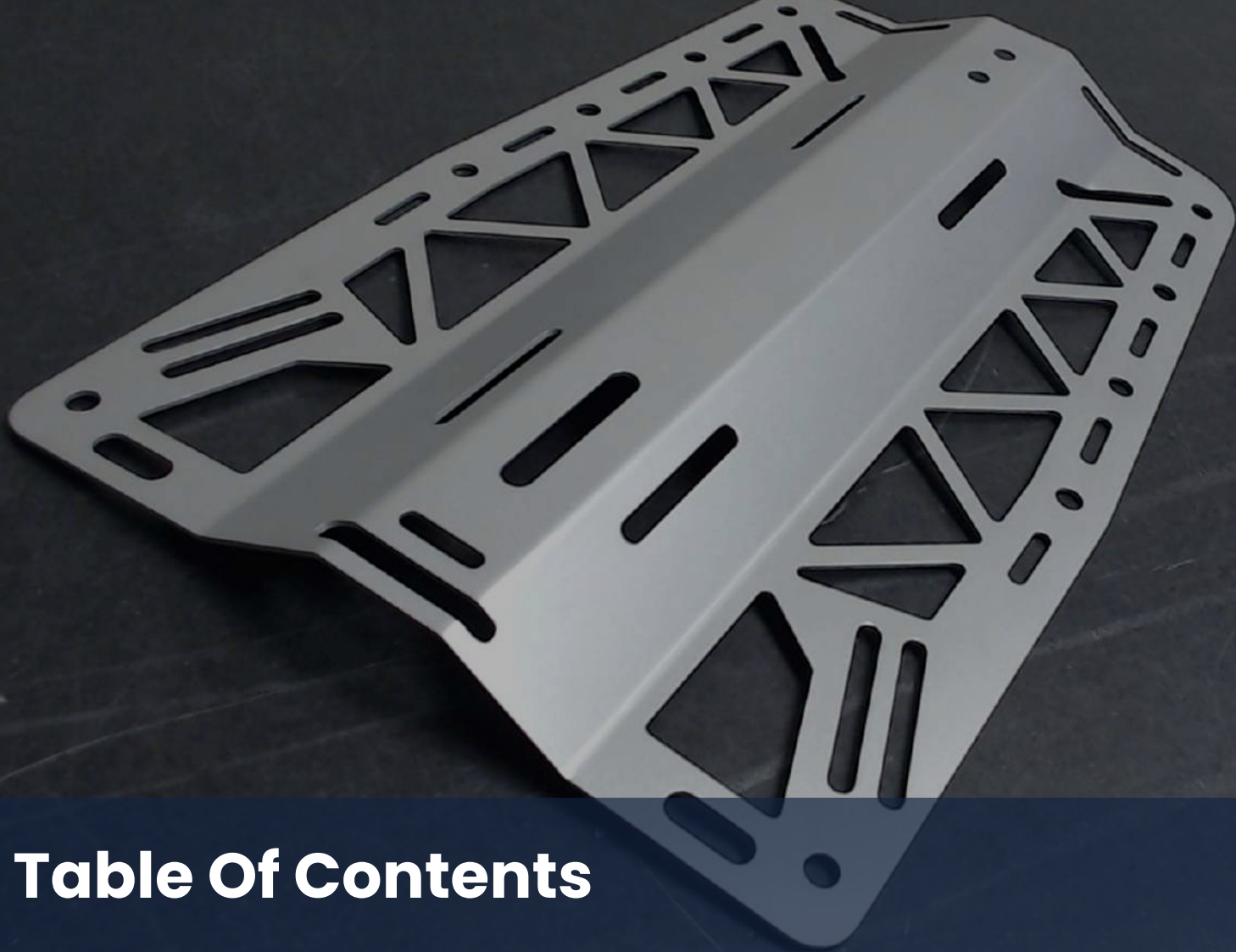


Table Of Contents

Chapter One

Guidelines for Sheet Metal Fabrication Design

Chapter Two

Common Sheet Metal Design Mistakes to Avoid

A close-up photograph of several sheet metal parts, likely aluminum or steel, showing various bends, flanges, and holes. The parts are arranged on a dark, textured surface. The lighting is soft, highlighting the metallic sheen and the precision of the fabrication.

Chapter One

Guidelines for Sheet Metal Fabrication Design

General Tolerance

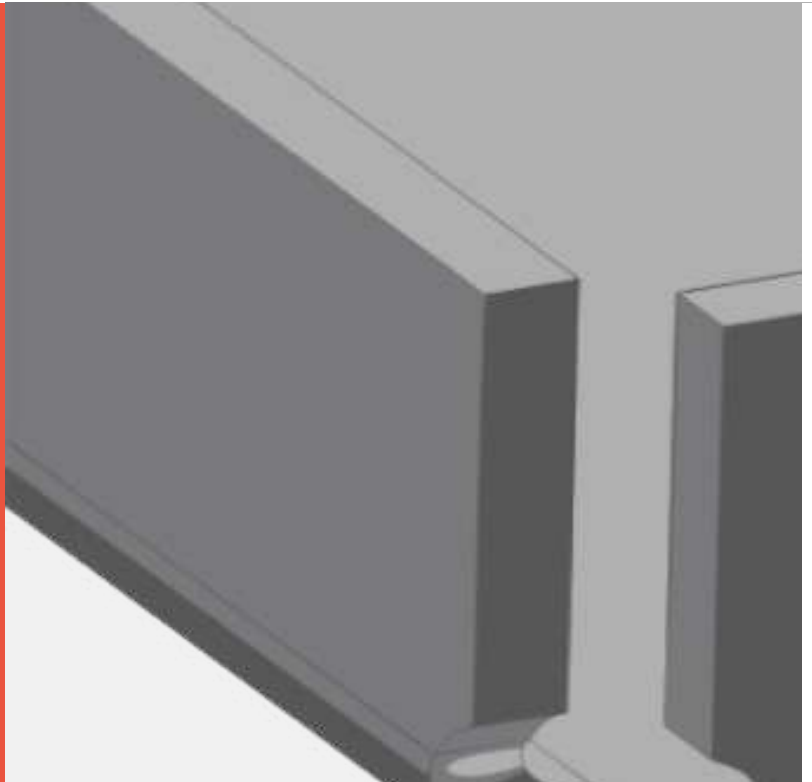
Sheet metal fabrication follows general industry tolerances aligned with international standards. Exceptions exist in precision-critical areas like aerospace and automobiles.

Feature	Prevalent Tolerance Range	Additional Notes
Wall thickness	0.9mm to 20mm	/
Offsets	0.3mm to 0.7mm	/
Curls	>2x material thickness	Anything less than the recommended curl dimension would make the sheet brittle.
Bends	0.9mm – 1.2mm 1.8mm – 2.4mm 3.8mm – 5.0mm 7.5mm – 10mm 15mm – 20mm	A +/- degree deviation is expected on all bends. Moreover, any other specifications will add to your costs.
Hems	Inside dia = material thickness with return length to be 4x the thickness	/
Countersinks	Major dia = +/- 0,254 mm Minor dia > 2/3 thickness	/
Holes and slots	Dia > material thickness	A diameter less than the material thickness would cause cracks in the sheet.
Notches and tabs	Notch width > 1.5x thickness Length > 5x thickness	/

Common Features In Sheet Metal Parts

Corner Fillets

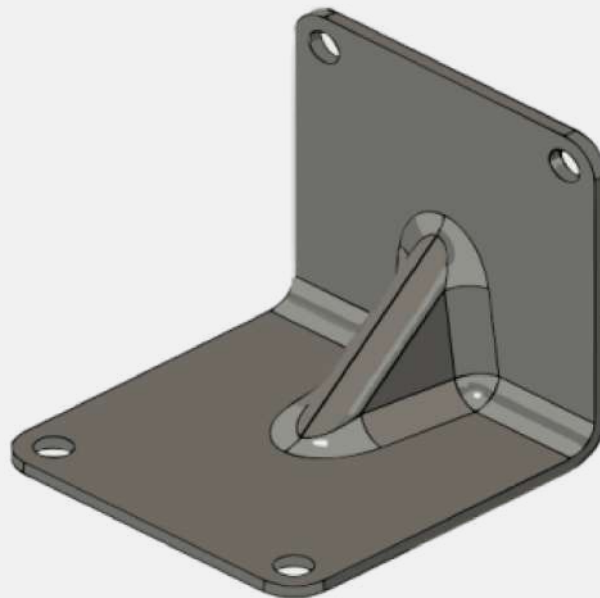
Corner fillets are rounded edges on sheet metal, preventing sharpness and reducing stress concentration to avoid failure.



- For sheet metal, use a fillet size equal to or greater than its thickness; e.g., 2mm metal needs a 2mm fillet.
- Fillets on a part should be symmetrical; opposing corners must have equal sizes.
- Fillets on a part should be uniform; all corners must be the same size.
- Place fillets in stress-prone areas, like bends or shape changes in sheet metal.
- Maximize fillet radius for even stress distribution and reduced stress concentrations.
- Design the part for easy fillet addition without compromising its integrity.

Ribs

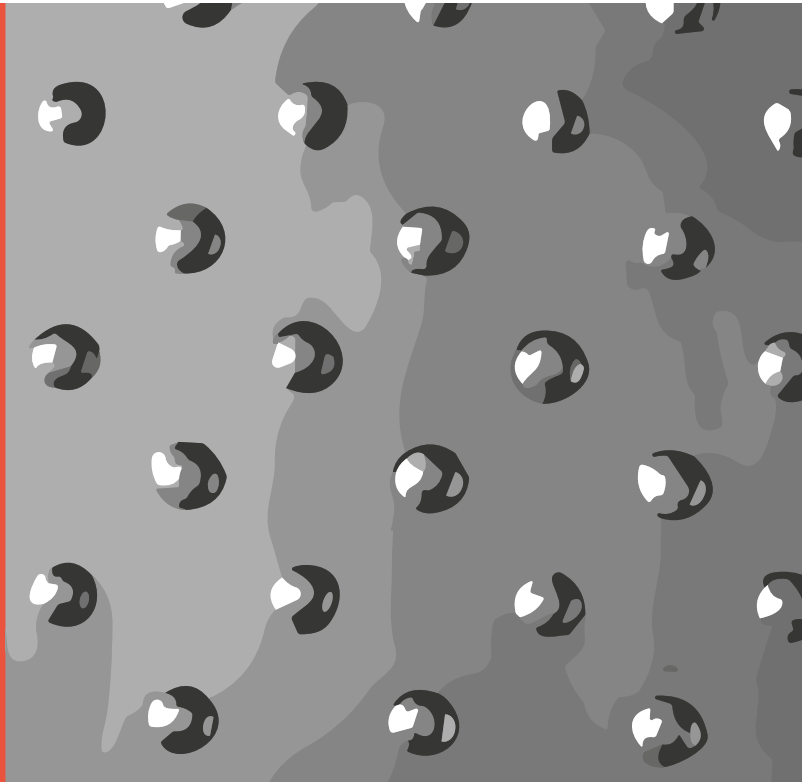
Raised features on sheet metal enhance strength and stiffness without significant weight addition.



- Keep the rib thickness to no more than 60% of the sheet metal thickness to avoid creating stress concentrations.
- Use fillets to smooth the transition between the rib and the surrounding material, which will help to distribute stresses more evenly.
- Avoid placing ribs too close together or too close to bends, as this can create weak spots in the material.
- Consider using tapered or variable-height ribs to distribute stresses more evenly.

Embossment

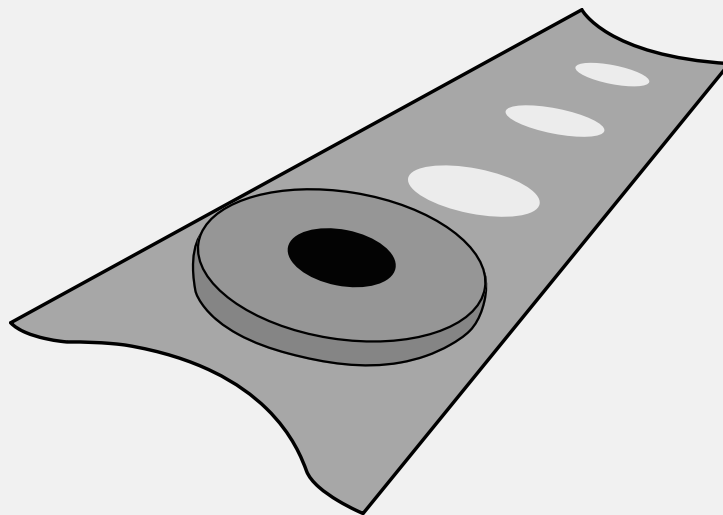
Recessed features on sheet metal add depth, texture, or create space for component fitting.



- Keep the depth of the embossment to no more than 50% of the sheet metal thickness to avoid creating stress concentrations.
- Use fillets to smooth the transition between the embossment and the surrounding material, which will help to distribute stresses more evenly.
- Avoid placing embossments too close together or too close to bends, as this can create weak spots in the material.
- Consider the impact of the embossment on the overall appearance of the part, and make sure it is aligned with any branding or design requirements.

Dimple Feature

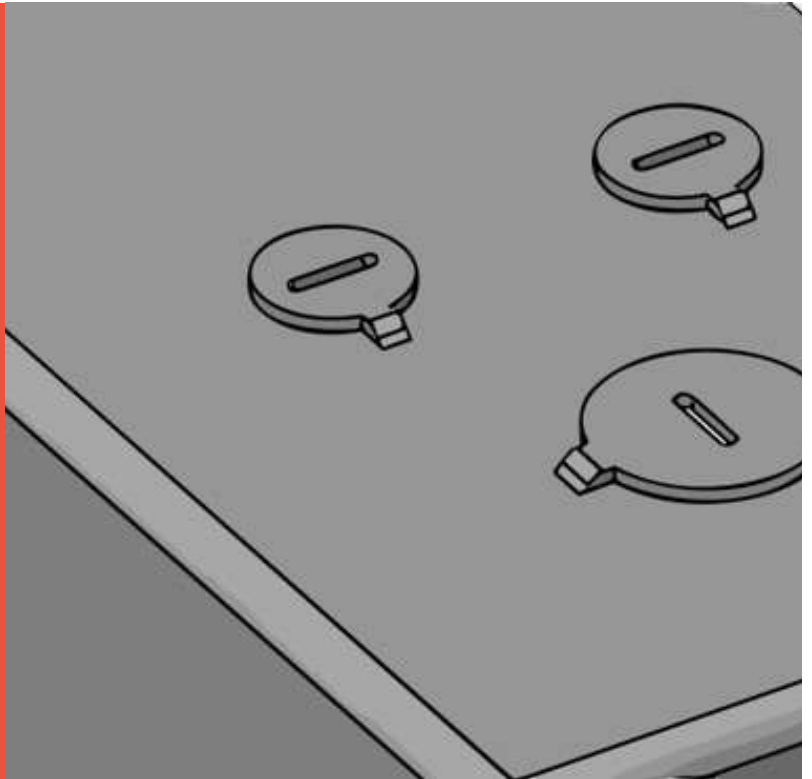
Dimples in sheet metal enhance stiffness, offer a flush surface for fasteners, and provide clearance for components.



- Choose dimple size and location strategically for maximum benefit, ensuring appropriateness for the application.
- Excessively large or deep dimples can weaken the material, while undersized ones might not offer enough reinforcement.
- Select the appropriate tool for dimpling, considering dimple size, shape, and material type. Options include punches, dies, and form tools.
- Consider sheet metal thickness and material type. Different metals might need varied techniques or tools for dimpling; thicker sheets demand more force or bigger tools.
- Consider design limitations; dimples aren't always suitable. Ensure designs address potential challenges from their inclusion.

Round Knockout

Round knockouts create holes in sheet metal, varying by punch and die size, commonly used in electrical boxes, HVAC systems, and enclosures.



- Ensure correct punch and die size; wrong sizes lead to improper holes.
- Use knockout tools suitable for the specific sheet metal type.
- Maintain tool sharpness to prevent poor-quality holes.
- Round knockouts favor thinner materials; thicker ones might need alternative methods.

Louver Feature

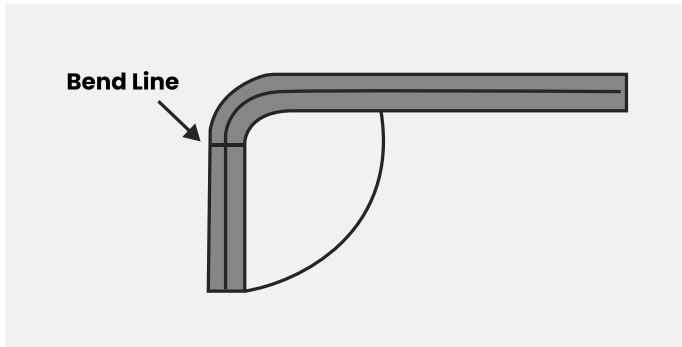
Louvers enhance airflow and ventilation in enclosures. They can direct air, reduce noise, and shield against dust, dirt, or moisture.



- Choose louver size based on required airflow and installation space. Too small limits ventilation; too large risks structural integrity.
- Choose louver orientation considering airflow direction and potential obstructions to ensure effective ventilation.
- Louver shape affects ventilation efficiency; streamlined, aerodynamic designs improve airflow and minimize turbulence.
- Choose louver material based on application and environmental exposure; stainless steel or aluminum is ideal for outdoor settings.
- Select louver manufacturing method based on precision, consistency, and cost-effectiveness of the fabrication.

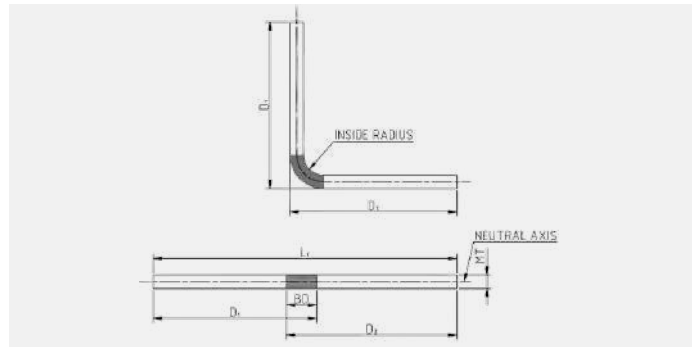
For Sheet Metal Bending

Here are the 6 most important parameters for any sheet metal bending operation.



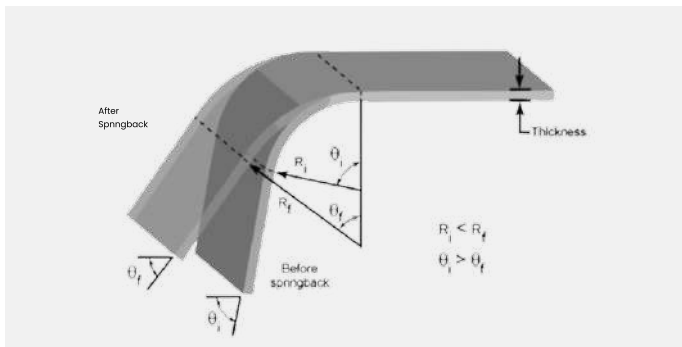
Bend Line

Bend line marks the start and end of a sheet bend, with industry standards maintaining a 5x sheet thickness distance between its edges.



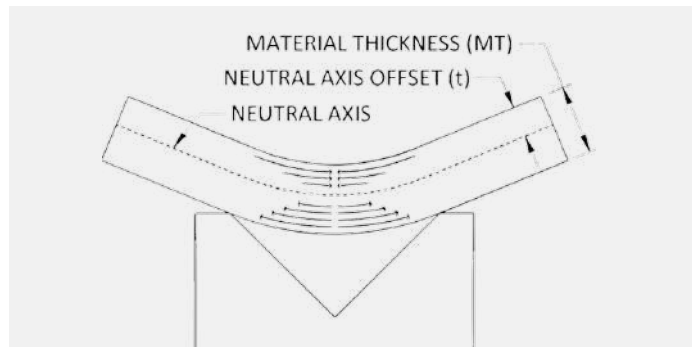
Bend Radius

Bend radius should be at least equal to material thickness to ensure the part's load-bearing capacity.



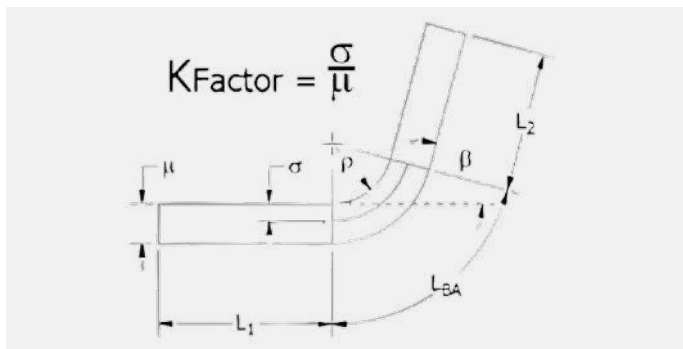
Bend Angle

Bend angle relates to its perpendicular axis. Standard: flange length is 4x thickness, maintaining uniform bend angles.



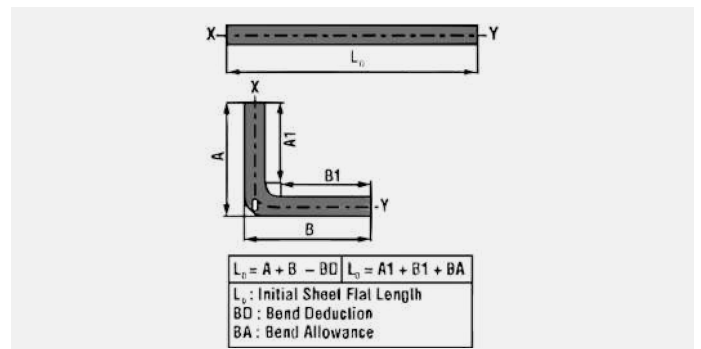
Neutral Axis

Neutral axis stays constant in a sheet. Its accuracy, combined with bend radius and angle, impacts product quality.



The K-Factor

K-factor measures material's location, influenced by material type, bending process, and angle. Optimal range: 0.25–0.50. Calculated as $K = T/t$.

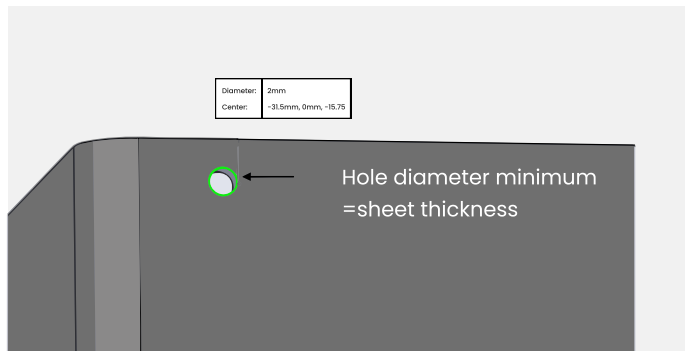


Bend Allowance

Ensure precise bending by measuring arc length, distance between neutral axis and bend lines, and selecting suitable bend allowances for material and process.

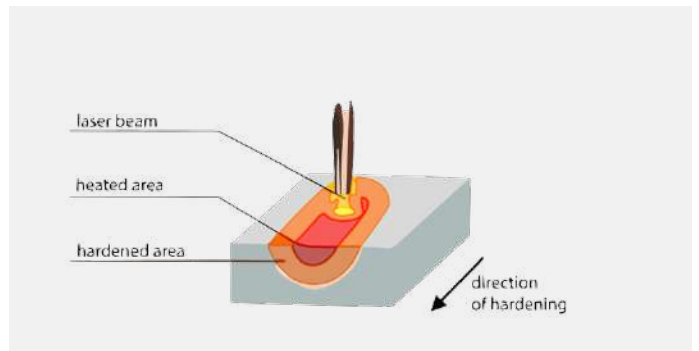
For Cutting

Another important process in sheet metal fabrication is cutting. In many cases, it's an easier alternative that delivers fast results with acceptable precision.



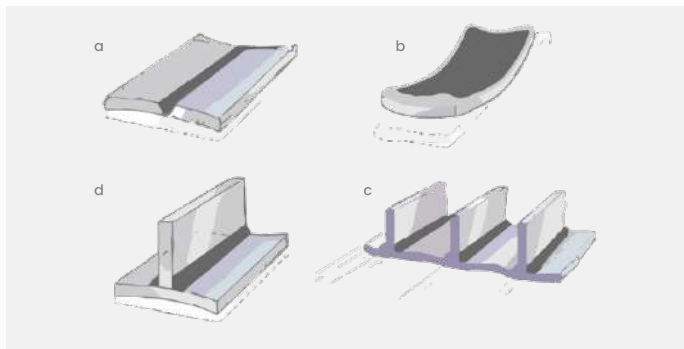
Hole Diameter

For drilling in sheets, ensure hole diameter equals or exceeds sheet thickness. Too small diameters risk cracks and affect product durability. Hole size is crucial for structural integrity.



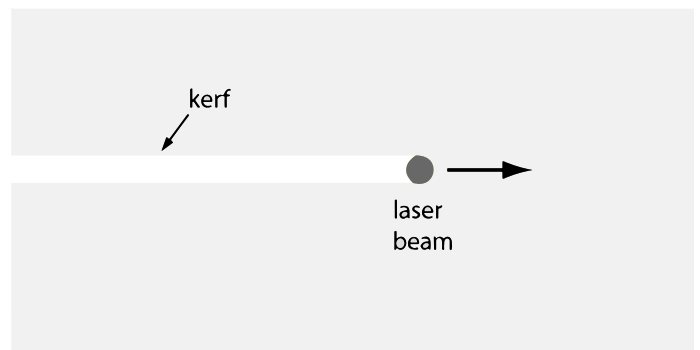
Localized Hardening

Cutting materials generates heat, potentially causing localized hardening. To avoid this, reduce cutting speed and use coolants to regulate temperature.



Distortion

Distortion in sheet metal fabrication involves warping or bending due to factors like temperature changes or stress, leading to inaccuracies and reduced product strength.



Kerf

Kerf denotes the material's width removed during cutting, influenced by the tool's width and material thickness. It's pivotal in determining material wastage in the cutting process.

A close-up, high-angle shot of a robotic welding arm in a factory setting. The arm, with its black cables and silver joints, is positioned over a series of rectangular sheet metal components arranged in rows on a dark, textured surface. The components have various slots and holes, and some are already welded. The lighting is industrial, with a cool blue tone.

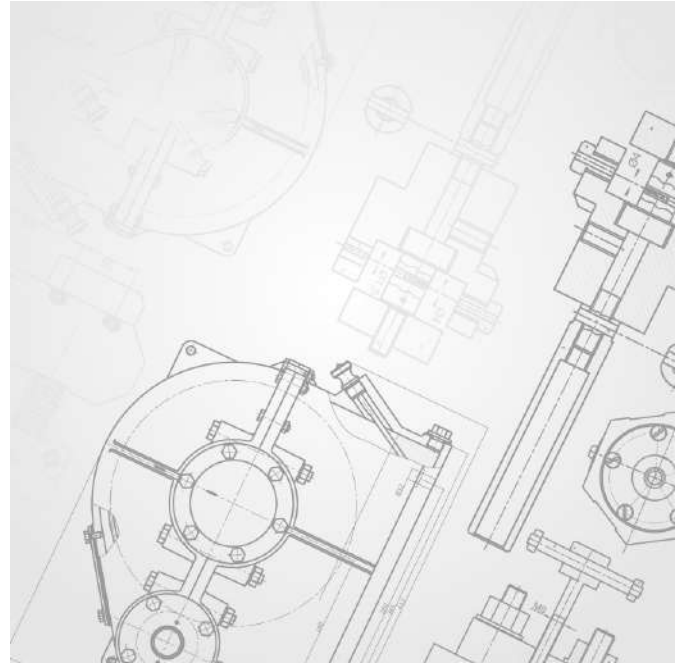
Chapter Two

Common Sheet Metal Design Mistakes to Avoid

Mistake 1:

A CAD File With No Bends

One common mistake to avoid is providing a CAD file with no bends. A sheet metal part without bends cannot be fabricated as a single piece and may require additional parts and labor to join multiple pieces together. It is important to include bends in the design and specify the bend angles and radii to ensure the part can be manufactured correctly.



Mistake 2:

Features Too Close To A Bend

Another similar mistake includes accidentally placing features such as holes, tabs, etc. too close to a bend. What happens if you keep the features too close? You'll end up with a deformed metal part that just wastes your money and time. To avoid making this mistake, you can just implement the 4T rule in all your CAD designs. The 4T rule dictates that all features should be 4x material thickness away from any bend line at least.

Mistake 3:

Perfectly Perpendicular Internal Bend Radius

In CAD design, while sharp corners seem ideal, bending sheet metal often results in a radius, preventing perfect corners. Trying for sharp bends can cause deformation and cracks. It's advised to set a minimum bend radius based on material and thickness for integrity. For consistency and cost-effectiveness, using the same radius for all bends is recommended.

Mistake 4:

Not Including Hardware Details In CAD File

For efficient fabrication and product accuracy, always provide comprehensive details in your CAD files, including hardware specifications, sizes, and locations. Overlooking specifics, like needing a CLS-440-2 clinching nut, can result in waiting for hardware arrangements. Such oversights invariably lead to increased assembly delays and costs.



Mistake 5:

Choose An Unsuitable Finish

Finishing is crucial in manufacturing, serving beyond mere aesthetics. While some finishes, like powder coating, enhance appearance and offer minimal protection, others like Silk Screening only add designs. Chemical Conversion Finishes provide a protective layer, extending product lifespan. Chromate Conversion finishing not only ensures electrical connectivity but also primes for painting. Choosing the correct finish is pivotal, depending on the metal part's intended application.

Mistake 6:

Select The Wrong Sheet Of Metal

When designing parts, always consider their end-use environment. Using unfinished steel in marine settings invites rust. Select the appropriate sheet metal by addressing key questions:

○ What is the daily expected wear?

○ How easily can sheet metal be manufactured?

○ Does your part require conductivity?

○ Is your metal part being used in an environment prone to corrosion and rust?

○ How important is the cosmetic appearance of your metal part?

○ What mechanical properties do your metal parts need to have?

Mistake 7:

Not Considering Material Strength For U Channels

U channels are an important part of any product design and their strength mostly depends on the overall strength of the material. Neglecting to take the material strength into account can result in U channels that are too weak, leading to bending or breaking under stress. To avoid this mistake, it's essential to select the appropriate material and thickness for the U channel. Based on the expected load and to factor in any additional stresses, such as vibrations or impacts, that the channel may experience in use.



Mistake 8:

Designing Unachievable Welding Requirements

No matter how simple a design is, there are high chances that it would require some welding or other mechanical joints. Some designers make the common mistake of overestimating the welding capabilities of the unit, which in turn increases the complexity and costs.

The best way forward to avoid such issues is to implement strict design for manufacturing (DFM) practices. This ensures that all features are according to the prevalent standards.



Get Started

Designing is just the start; impeccable fabrication brings visions to life. At RapidDirect, we merge decades of expertise with cutting-edge machinery, ensuring your sheet metal designs materialize flawlessly. Benefit from our swift turnarounds, cost-effective solutions, and professional support.

Ready to elevate your project?

Choose RapidDirect for unparalleled sheet metal fabrication today!




Quality Parts Made Easier, Faster

Get In Touch

 +86-0755-85276703

 info@rapiddirect.com

 www.rapiddirect.com

 Room 407, Building F, Tianyou Chuangke Industrial Park
Qiaotou Community, Fuhai Street
Baoan District Shenzhen, China, 518103

