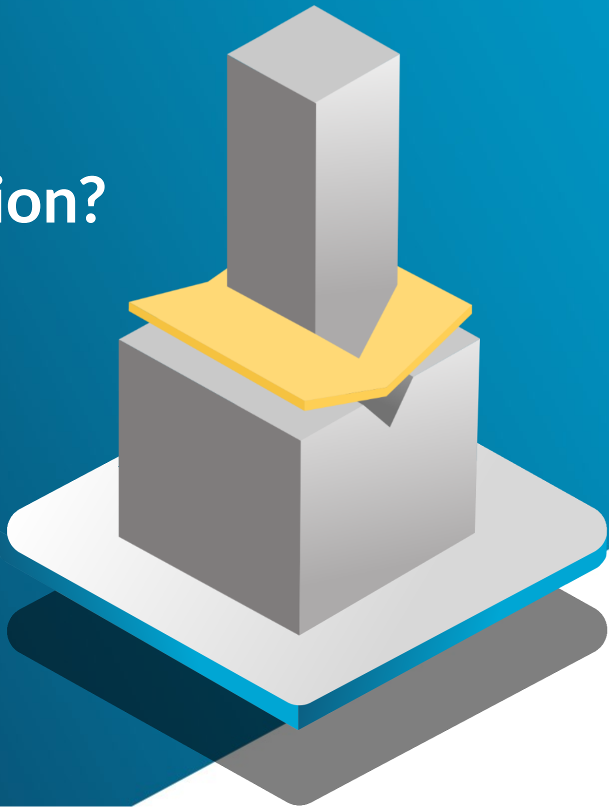
WIZ-PARTS

SHEET METAL FABRICATION DESIGN GUIDE

v1.0 September 2024

What Is
Sheet Metal Fabrication?



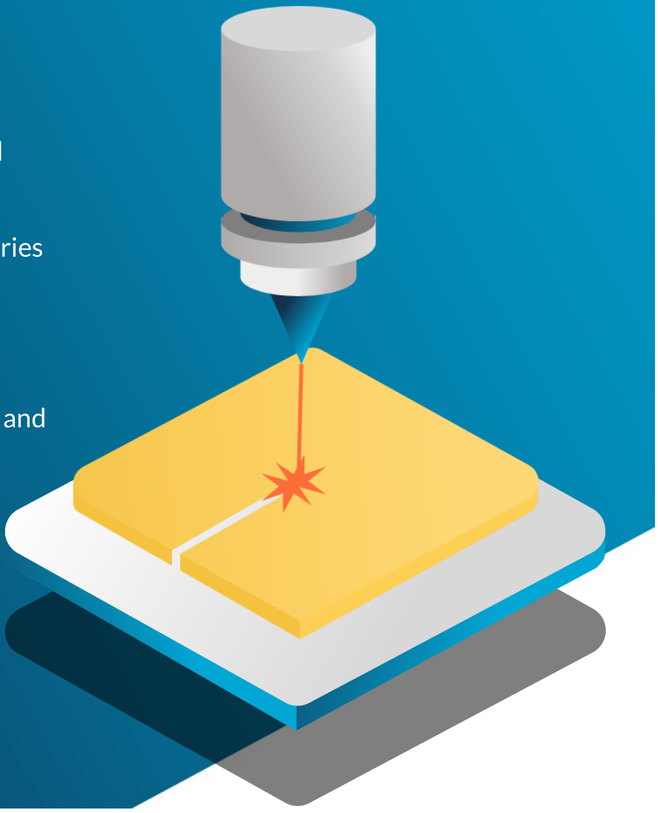
Sheet metal fabrication is a versatile manufacturing process that involves cutting, bending, and assembling thin metal sheets into various shapes and structures.

This process transforms raw metal into finished products through techniques such as Laser cutting, press brake bending and welding. Sheet metal fabrication is ideal for creating durable, precise, and custom metal components for a wide range of applications.

Contents

Laser Cutting

Laser cuttings employs a high-powered laser beam directed through optics to penetrate through the metal, resulting in clean and accurate cuts. Laser cutting is highly versatile, capable of producing intricate designs and complex geometries with minimal waste and high repeatability.



Benefits of Laser Cutting

Precision: Offers high accuracy, making it ideal for detailed and complex designs.

Efficiency: Fast cutting speeds and reduced setup times increase productivity.

Versatility: Suitable for various metals and thicknesses, including steel, stainless steel and aluminium.

Quality: Provides smooth edges and minimal distortion, reducing the need for additional finishing processes.

CNC Bending

CNC bending with a press brake is a precise and automated method used in sheet metal fabrication to bend metal sheets into specific angles and shapes. Using a CNC system to guide the press brake, ensures high accuracy and consistency in the bending process. This technology allows for complex bends and tight tolerances, making it ideal for parts with intricate designs and high repeatability.

Benefits of CNC Bending

Precision: CNC control ensures exact bending angles and dimensions.

Efficiency: Automated process reduces manual labor and increases production speed.

Consistency: High repeatability ensures uniformity across large production runs.

Selecting The Right Material

With a wide range of materials available for sheet metal fabrications, making the optimal material choice is crucial to its success. Selecting the right material involves considering both the type of metal and its physical properties. Each material has unique characteristics and benefits and therefore, the desired end product and its applications will guide the appropriate sheet metal material selection.

Comparison of Sheet Metal Material Properties

	Aluminium	Mild Steel	Stainless Steel
Strength	Low	Good	Very Good
Weight	Low	High	High
Corrosion Resistance	High	Low	Very High
Weldability	Good	Very Good	Very Good
Cost	££	£	£££

Aluminium

5251 N4 (BS 1470)

Available Sizes v

- 1mm
- 1.2mm
- 1.5mm
- 2mm
- 3mm

Mild Steel

DC01 /CR4 (EN 10130)

S275JR (EN 10025-2)

Available Sizes v

- 1mm
- 1.2mm
- 1.5mm
- 2mm
- 2.5mm
- 3mm
- 4mm
- 5mm
- 6mm

Stainless Steel

304 (EN 1.4301)

316 (EN 1.4401)

Available Sizes v

- 0.7mm
- 0.9mm
- 1mm
- 1.2mm
- 1.5mm
- 2mm
- 2.5mm
- 3mm

Calculating K-Factor

K-Factor is the ratio between the distance between the neutral axis and inner radius (D) and the material thickness (Mt).

K = D / Mt

The K-Factor is dependent on the material used, the material thickness, bend radius and the bend method utilised. It is important to select the correct K-Factor for your application to ensure accuracy and repeatability in production.

Recommended K-Factors For Sheet Metal Fabrication

Bend Radius	Aluminium	Mild Steel	Stainless Steel
Air Bending			
≤ Mt	0.33	0.38	0.40
Mt ≤ 3x Mt	0.40	0.43	0.45
≥ 3x Mt	0.50	0.50	0.50
Bottom Bending			
≤ Mt	0.42	0.44	0.46
Mt ≤ 3x Mt	0.46	0.47	0.48
≥ 3x Mt	0.50	0.50	0.50

Air Bending

Air bending is known as air bending because the sheet metal is not fully pressed into the die during bending leaving an air gap.

Benefits Of Air Bending v

- Faster Than Bottom Bending
- Less Pressure Required To Bend Material
- No Tooling Change Needed For Different Bend Angles
- Less Surface Contact Reducing Risk of Surface Damage

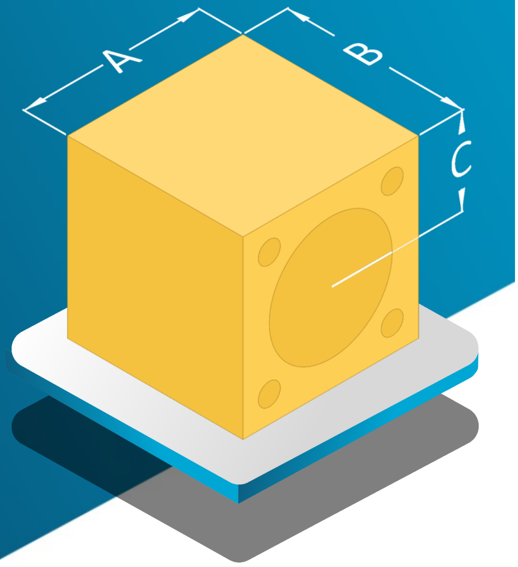
Bottom Bending

Bottom bending gets its name because the metal is forced to the bottom of the die making full contact with the punch and die.

Benefits Of Bottom Bending v

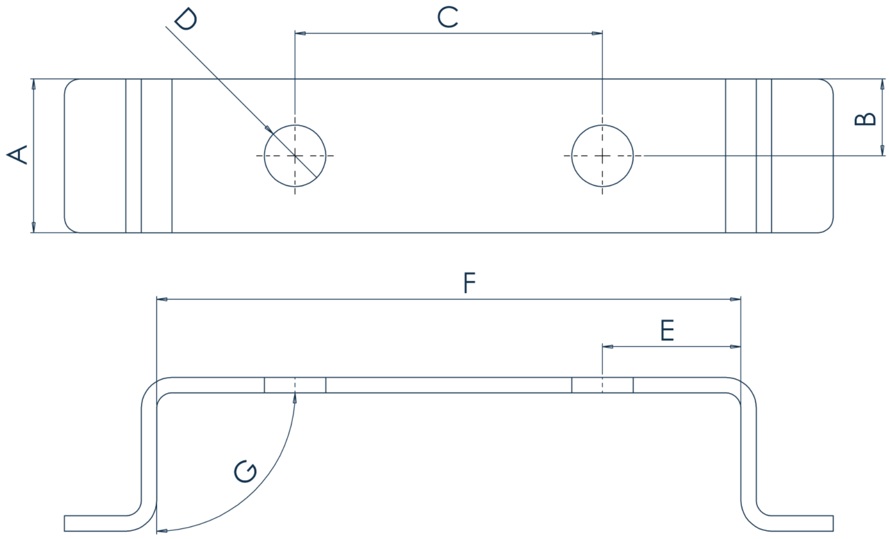
- Produces More Reliable and Accurate Bends
- Less Spring Back Than Air Bending
- More Accurate Angles Than Air Bending
- Ideal For Precise Parts and With Consistent Bend Angles

General Tolerances



Tolerances refer to the permissible limits of variation in a physical dimension of a manufactured part. These precise measurements ensure that parts fit and function correctly, maintaining quality and consistency throughout the production process. Tight tolerances are crucial for achieving high performance and reliability in the final product.

Unless otherwise stated on your drawings, sheet metal parts will be manufactured to the tolerances in the table below.

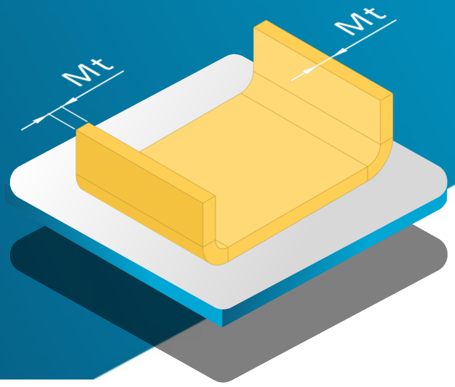


Feature	Tolerance
Edge to Edge (A)	+/- 0.25mm
Edge to Hole (B)	+/- 0.25mm
Hole to Hole (C)	+/- 0.25mm
Hole Diameter (D)	+/- 0.25mm
Bend to Hole (E)	+/- 0.50mm
Bend to Bend (F)	+/- 0.50mm
Bend Angle (G)	+/- 1 Degree

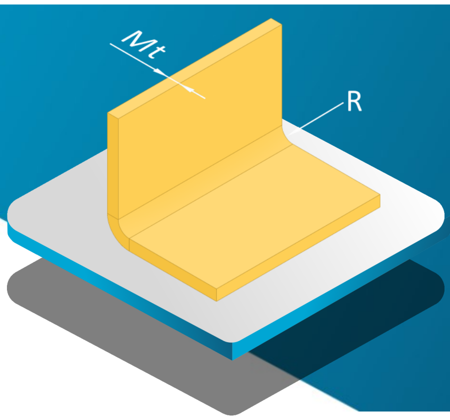
04 Sheet Metal Fabrication Design Tips

Uniform Material Thickness

Sheet metal components must have a uniform material thickness (Mt) throughout the part. This material thickness must be between 1 - 3mm for aluminium parts, 1 - 6mm for mild steel parts and 0.7 - 3mm for stainless steel parts.



Correct Bend Radii



It is recommended that bend radii (R) should be equal to or greater than the material thickness (Mt).

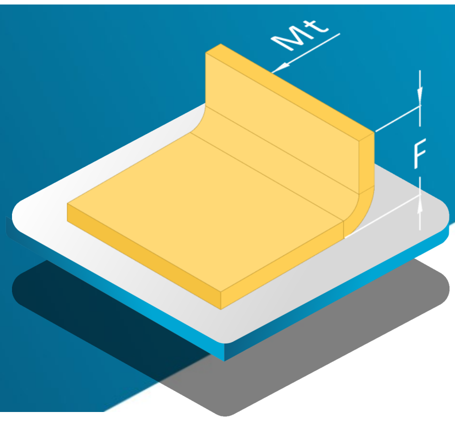
It is recommend to keep the bend radii consistent throughout the part if possible.

R ≥ Mt

Minimum Flange Lengths

It is recommended to make all sheet metal bends to have a flange length (F) equal to or greater than four times the material thickness (Mt).

F ≥ 4 x Mt



Distance Between Bends

It is recommended to avoid successive bends wherever possible.

To ensure adequate tool access bends need to be spaced such that flange length (F) is less than or equal to base length (L).

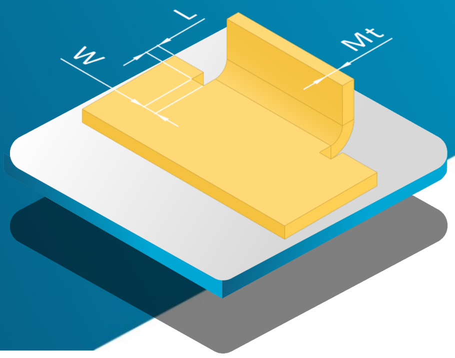
F ≤ L

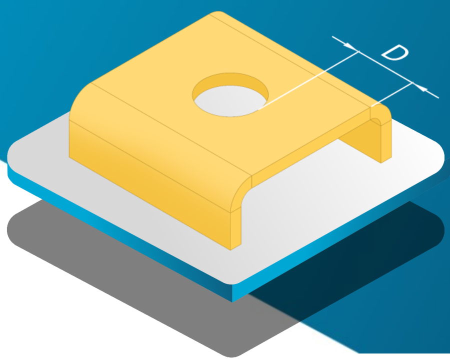
Creating Bend Reliefs

Where bends are made close to the material edge it is recommended to create relief cuts to reduce risk of tearing.

Reliefs should be made such that both width (W) and length (L) is equal to or greater than the material thickness (Mt).

W ≥ Mt **L ≥ Mt**





Hole Clearances to Edges

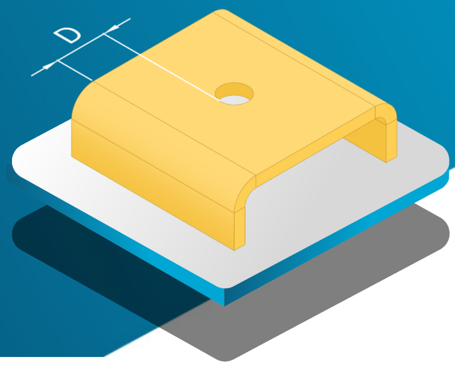
It is recommended that all holes are placed at a distance (D) equal to or greater than three times the material thickness (Mt) from the material edge.

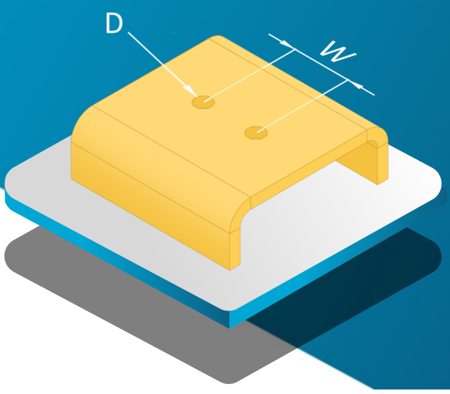
D ≥ 3 x Mt

Hole Clearances to Bends

It is recommended that all holes are placed at a distance (D) equal to or greater than four times the material thickness (Mt) from the bend edge.

D ≥ 4 x Mt





Correct Hole Sizing

Holes are to have a minimum diameter (D) equal to or greater than the material thickness (Mt).

It is recommended to space holes such that the minimum distance between centres (W) is equal to or greater than four times material thickness (Mt).

D ≥ Mt **W ≥ 4 x Mt**

04 Sheet Metal Fabrication Design Tips

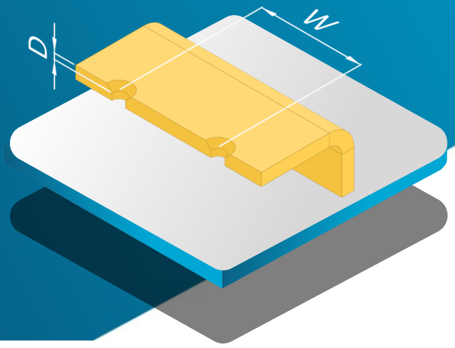
Adding Countersunk Holes

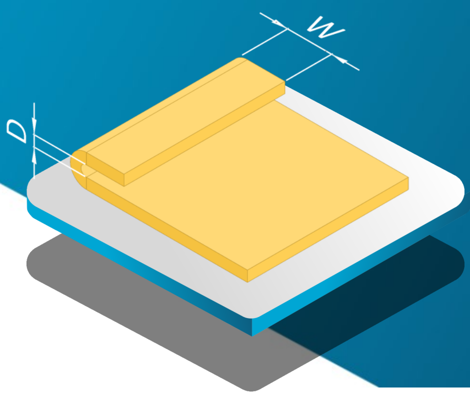
Countersinks should have a depth (D) less than or equal to 60% of the material thickness (Mt).

It is recommended to space countersunk holes such that the minimum distance between centres (W) is equal to or greater than eight times material thickness.

D ≤ 0.6 x Mt

W ≥ 8 x Mt





Adding Open Hem Edges

It is recommended that open hem edges have an inside diameter (D) equal or greater than the material thickness (Mt).

Hem Length (W) should be equal or greater than four times the material thickness (Mt).

D ≥ Mt

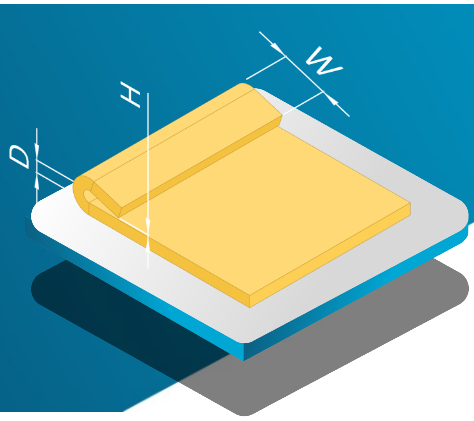
W ≥ 4 x Mt

Adding Tear Drop Hem Edges

Tear Drop hem edges are to follow the same rules as open hem edges for internal diameter and hem length dimensions.

It is recommended that the hem opening (H) is equal to or greater than 25% of the material thickness (Mt).

H ≥ 0.25 x Mt



Adding Curled Edges

It is recommended that curled edges have an internal radius (R) equal to or greater than the material thickness (Mt).

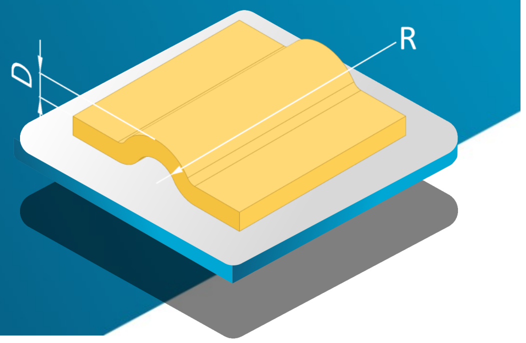
R ≥ Mt

Swaging Sheet Metal Parts

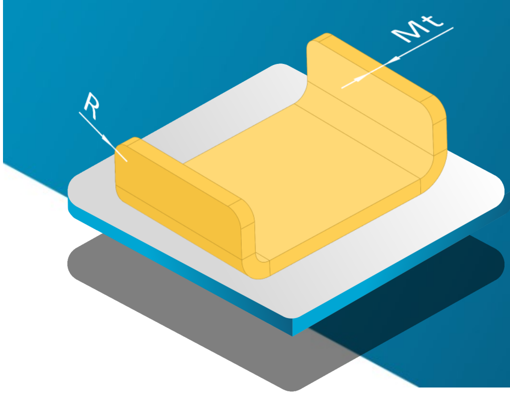
Swaging can be used to add strength to sheet metal parts with a material thickness (Mt) up to 3mm.

The depth of swage (D) must be less than or equal to three times the material thickness (Mt) and less than or equal to its internal radius (R).

$D \leq 3 \times Mt$ | $D \leq R$



Adding Corner Fillets



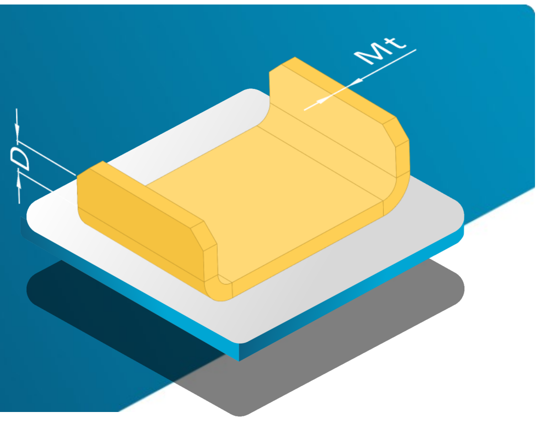
It is recommended that all exposed corners of a sheet metal part have a corner fillet applied with a radius (R) equal to or greater than 50% of the material thickness (Mt).

$R \geq 0.5 \times Mt$

Adding Corner Chamfers

It is recommended that any chamfered flanges leave a minimum distance between bend and chamfered edge (D) equal to or greater than two times material thickness (Mt).

$D \geq 2 \times Mt$



And Remember to Make Sure Your Files Are in the Correct Format!

2D CAD Files

.DXF

.DWG

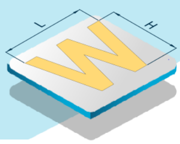
3D CAD Files

.STEP .IGES

.X_T .SLDPRT

2D Drawings

.PDF



Rapid Precision, Every Time

Unit D, Glebe Farm, Welsh Road
Cubbington, Leamington Spa
Warwickshire, CV32 7UB, UK

///decide.action.organs

01926 426 255

enquiries@wiz-parts.co.uk

www.wiz-parts.co.uk

v1.0 September 2024