Understanding Sheet Metal Bending: Key Terms and Concepts

Bend Allowance

The bend allowance is the length of the material that undergoes deformation during the bending process. When a piece of metal is bent, the material around the bend stretches and compresses, affecting the final dimensions of the part. Accurately calculating the bend allowance is essential for determining the correct size of the flat sheet metal before bending so that the finished product meets the desired specifications. It's important to avoid placing holes, slots, or other cutouts within the bend allowance area unless they are precisely centered along the bend line. Placing such features off-center can cause the bend to shift toward these weaker areas, leading to inaccuracies in the final part size and potential structural weaknesses.

Bend Angle

The bend angle is the measure of how much the metal is bent, defined by the angle between the two flanges (sides) of the bent part. This angle dictates the shape and functionality of the final piece. Angles less than 90 degrees are considered "more closed" or acute, resulting in sharper bends. Angles between 90 and 180 degrees are "more open," producing gentler bends. Understanding and specifying the correct bend angle is crucial for ensuring that the part fits correctly in its intended application.

Bend Deduction

Bend deduction is the amount subtracted from the total length of the flat sheet to achieve the desired dimensions after bending. It represents the difference between the sum of the outer dimensions of the finished part and the original flat length before bending. Calculating the bend deduction allows engineers and fabricators to determine the precise size of the flat metal needed. For example, if you're creating an L-shaped bracket where each leg needs to be exactly 6 inches long, and the bend deduction is 0.5 inch, you would subtract the bend deduction from the total length of the two legs. Therefore, the flat piece should be
6 in + 6 in − 0.5 in = 11.5 inches
6in+6in−0.5in =11.5 inches long to achieve the correct final dimensions.

Bend Line

The bend line is the exact location on the flat sheet where the bending operation will occur. It indicates where the punch or bending tool will make contact with the material to form the bend. In technical drawings and fabrication plans, the bend line guides the operator in positioning the metal accurately within the bending machine. Precise placement of the bend line ensures that the bends are made in the correct locations, which is essential for the part to meet design specifications.

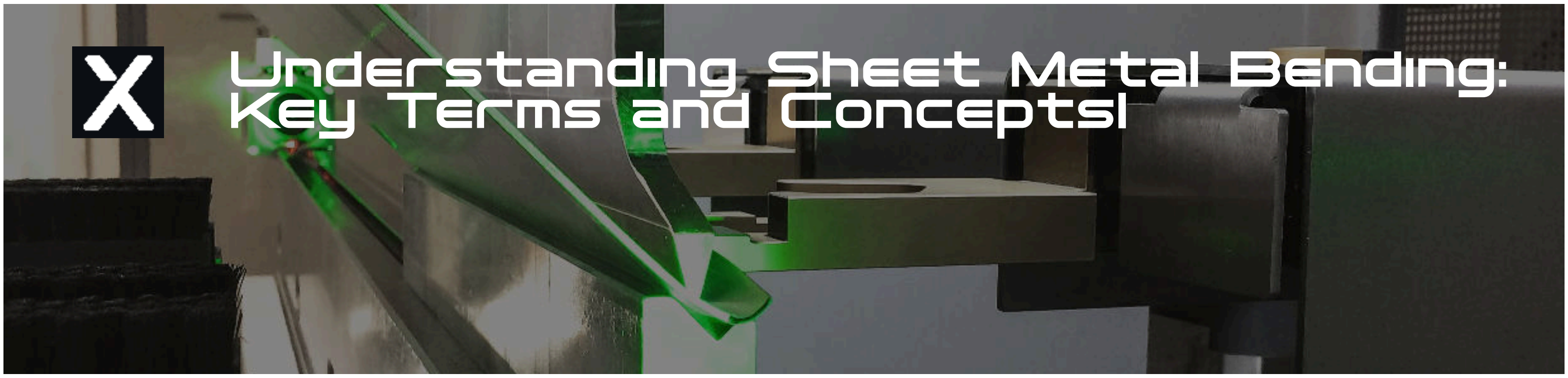
Inner Radius

The inner radius is the radius of the inside curve of the bend. While bends in sheet metal are not perfectly circular due to material properties and the bending process, the inner radius serves as a critical design parameter. It influences the bend allowance calculation and affects the material's flow during bending. A larger inner radius can reduce the risk of cracking or damaging the material, especially in thicker or harder metals. Designers specify the inner radius to control the bend's characteristics, although the actual radius may vary slightly in practice due to material springback and tooling limitations.

K-Factor

The K-factor is a ratio that describes the location of the neutral axis relative to the material's thickness during bending. The neutral axis is the theoretical line within the material that experiences no compression or tension. The K-factor is calculated by dividing the distance from the inner surface of the bend to the neutral axis by the total thickness of the material. This coefficient is crucial for accurately calculating the bend allowance and bend deduction. It reflects the material's tendency to stretch or compress during bending. Softer metals like aluminum typically have a K-factor around 0.4, indicating that the neutral axis is closer to the inside surface due to more significant stretching on the outer surface. Harder metals like steel and stainless steel have a K-factor around 0.45, showing a slightly different neutral axis position due to their material properties.

Understanding Sheet Metal Bending: Key Terms and Concepts!



Minimum Flange Support

Minimum flange support refers to the minimum required distance between the bend line and the edge of the material (the flange). This distance ensures that there is enough material to support the bending operation without causing deformation or inaccuracies. Material within the minimum flange support area will contact the die during bending, providing stability. Larger bend radii require larger dies, which in turn need more flange support. To achieve precise bends and maintain the structural integrity of the part, it's recommended to minimize holes, notches, or other cutouts in the flange support region. Such features can weaken the material and lead to defects or deviations in the bend.

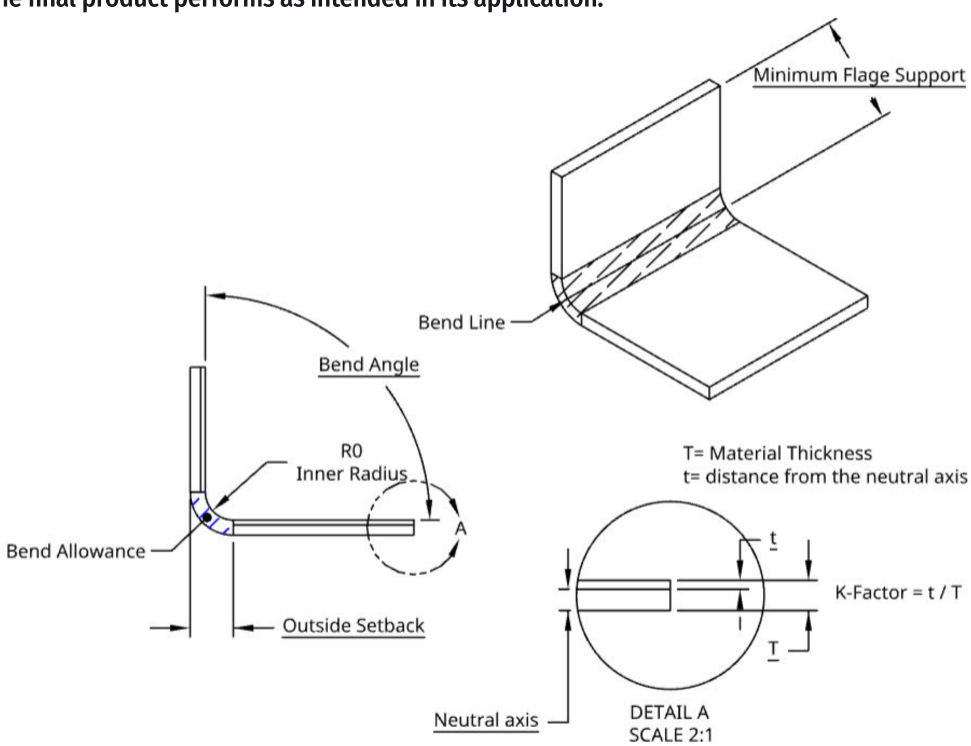
Neutral Axis

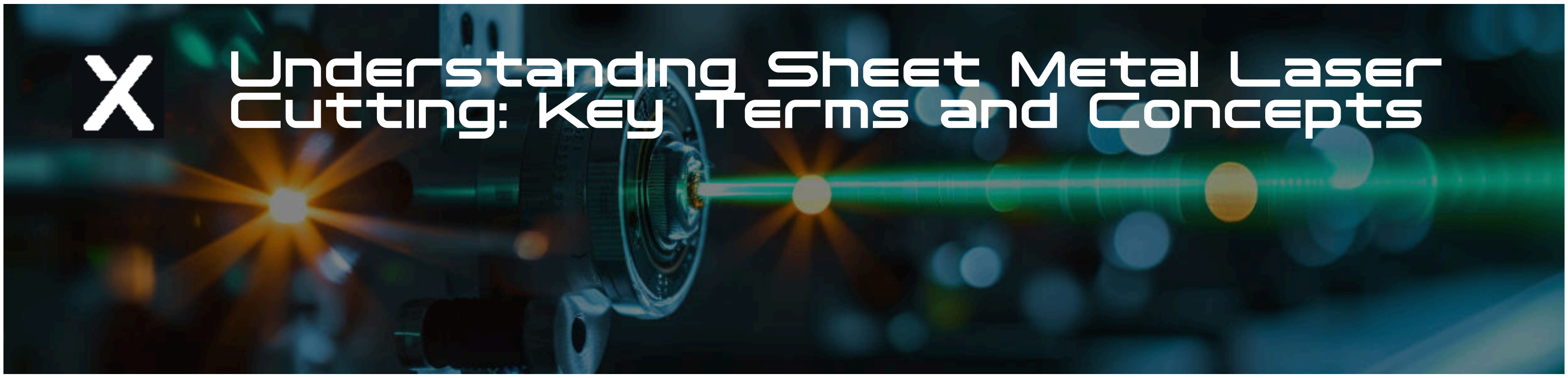
The neutral axis is an imaginary line within the material being bent where the fibers neither compress nor stretch—they remain neutral. During bending, the material on the inside of the bend compresses, while the material on the outside stretches. The neutral axis shifts toward the inside of the bend based on the material properties and thickness. Understanding the position of the neutral axis is essential for calculating accurate bend allowances and for predicting how the material will behave during bending. It helps in minimizing errors and ensuring that the final part dimensions are as intended.

Outside Setback

Outside setback is the distance from the beginning of the bend allowance area to the outer surface of the flange on the finished part. It accounts for the material that "wraps around" the bend and is essential for determining the correct flat pattern dimensions. The outside setback is used in combination with the bend allowance to calculate where to start and end the bend in the flat layout. Accurately calculating the outside setback ensures that, after bending, the part's outer dimensions align with the design requirements.

By thoroughly understanding these terms and their implications in the bending process, designers and fabricators can create precise and functional sheet metal parts. Accurate calculations and careful planning help in reducing material waste, avoiding costly errors, and ensuring that the final product performs as intended in its application.





CNC Laser Cutting

CNC (Computer Numerical Control) Laser Cutting is a precise manufacturing process that employs a high-powered laser beam directed by computer-controlled machinery to cut or engrave materials into specific designs and shapes. The laser beam melts, burns, or vaporizes the material, resulting in a clean and accurate cut edge. This technology is widely used across various industries, including aerospace, automotive, electronics, and metal fabrication, due to its ability to produce complex shapes with high precision and repeatability.

Laser Types

CO_2 Lasers: Utilize a gas mixture (primarily carbon dioxide) to produce the laser beam. Ideal for cutting non-metal materials like wood, acrylic, plastics, and also capable of cutting some metals with appropriate power levels.

Fiber Lasers: Use optical fibers doped with rare-earth elements to generate the laser. Highly efficient and suitable for cutting metals, offering faster cutting speeds and lower maintenance compared to CO_2 lasers. This is the laser type we uses at Xeon NC.

Nd Lasers: Solid-state lasers that provide high-intensity beams for applications requiring deep penetration, such as cutting thicker metals or for precision engraving.

Kerf

Kerf refers to the width of the material that is removed during the cutting process. In laser cutting, the kerf is typically very narrow, ranging from 0.1 mm to 0.5 mm, depending on the material and laser settings. Understanding the kerf is crucial for designing parts that require precise fits, as it affects the final dimensions of the cut pieces.

Heat-Affected Zone (HAZ)

The Heat-Affected Zone is the area of the material that experiences thermal alteration due to the laser's heat during cutting. This zone may exhibit changes in microstructure, hardness, or mechanical properties, which can affect the performance of the final part. Minimizing the HAZ is important to preserve material integrity, which can be achieved by optimizing laser parameters like power, speed, and focus.

Assist Gas

Assist gas is used in laser cutting to aid in the cutting process and improve cut quality. Common assist gases include:

:Oxygen: Enhances the cutting of mild steel by exothermic reaction, increasing cutting speed but may cause oxidized edges.

:Nitrogen: Used to prevent oxidation, producing clean, oxide-free edges, especially important for stainless steel and aluminum.

:Compressed Air: A cost-effective option for cutting thin materials where edge quality is less critical.

The choice of assist gas affects the cutting speed, edge quality, and the amount of dross (residual material) produced.

Piercing

Piercing is the initial step where the laser creates a hole in the material before starting the cut path. Effective piercing is crucial for thicker materials:

Piercing Time: Needs to be controlled to prevent excessive melting or spatter.

Piercing Techniques: Techniques like high-speed piercing or pulse piercing can reduce piercing time and improve quality.

Understanding Sheet Metal Laser Cutting: Key Terms and Concepts

Cutting Speed

Cutting speed is the rate at which the laser head moves across the material. It is a critical parameter that influences:

Cut Quality: Too fast can result in incomplete cuts; too slow can cause excessive melting.

Production Efficiency: Optimizing speed enhances throughput without compromising quality.

Cutting speed must be balanced with laser power and material properties to achieve desired results.

Material Thickness

Material thickness affects laser cutting parameters:

Thin Materials: Allow for higher cutting speeds and require less laser power.

Thick Materials: Require higher laser power and slower cutting speeds to ensure full penetration.

Each laser machine has a maximum effective cutting thickness, depending on its power and the material type.

Dross and Slag

Dross (also known as slag) is the residue or roughness left on the underside of the material after cutting:

Causes: Incomplete ejection of molten material due to incorrect cutting parameters.

Prevention: Adjusting assist gas pressure, cutting speed, and laser power can minimize dross.

Post-Processing: Dross may require secondary operations like grinding or sanding to achieve desired edge quality.

Nesting

Nesting is the arrangement of multiple parts on a material sheet to maximize material utilization:

Manual Nesting: Operator arranges parts, which may not be optimal.

Automatic Nesting Software: Uses algorithms to optimize part placement, reducing waste and cutting time.

Efficient nesting lowers material costs and increases production efficiency.

Lead-In and Lead-Out

Lead-in and lead-out are small paths that the laser follows before and after cutting the main geometry:

Purpose: Prevents defects at the start and end points of cuts.

Placement: Should be positioned in non-critical areas of the part.

Types: Straight or arc-shaped paths depending on the material and cut quality requirements.

Proper use of lead-ins and lead-outs enhances edge quality and part accuracy.

Tabbing and Micro-Joints

Tabs or micro-joints are intentional connections left between the part and the surrounding material:

Function: Prevents parts from tipping or shifting during cutting.

Size: Small enough to be easily broken or removed post-cutting.

Application: Useful for small parts or intricate designs where movement could affect cut quality.

Including tabs in designs helps maintain part integrity throughout the cutting process.

Minimum Hole Sizes

Importance of Minimum Hole Sizes

Laser Beam Diameter: The laser beam has a finite width, known as the kerf, which typically ranges from 0.1 mm to 0.5 mm. Designing holes smaller than the kerf width is impractical because the laser cannot create features smaller than its own diameter.
Material Thickness: The minimum hole size is also influenced by the thickness of the material. Thicker materials require larger minimum hole sizes to ensure structural integrity and prevent deformation during cutting.
General Guidelines

Minimum Hole Diameter: As a rule of thumb, the minimum hole diameter should be at least equal to the material thickness. For example, if you're cutting a 3 mm thick sheet, the minimum hole diameter should be 3 mm.
Avoiding Heat Accumulation: Small holes in thick materials can cause excessive heat buildup, leading to poor edge quality or warping. Ensuring adequate hole size minimizes these risks.

Scaling Features: If your design includes intricate patterns or small holes, consider scaling up the entire design or adjusting the size of these features to meet minimum size requirements.
Alternative Methods: For features smaller than the laser can handle, post-process machining or drilling might be necessary.

Nesting Parts on Standard Sheet Sizes

Standard Sheet Sizes

Efficient material utilization is achieved by nesting parts on standard sheet sizes. Common sheet sizes include:

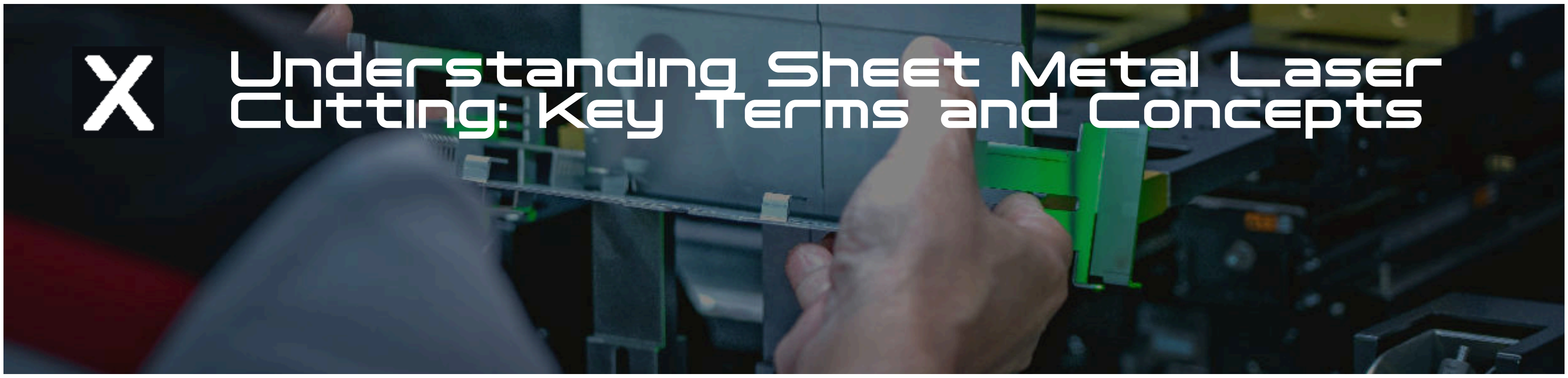
72" x 144" (1829 mm x 3658 mm)

72" x 120" (1829 mm x 3048 mm)

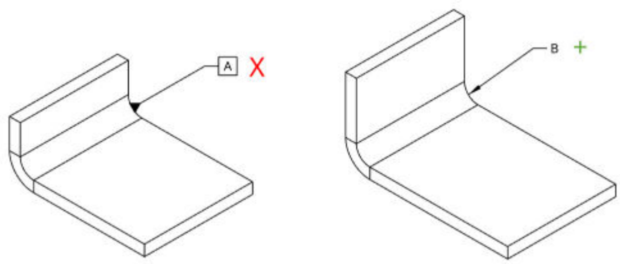
60" x 120" (1524 mm x 3048 mm)

48" x 120" (1219 mm x 3048 mm)

36" x 120" (914 mm x 3048 mm)



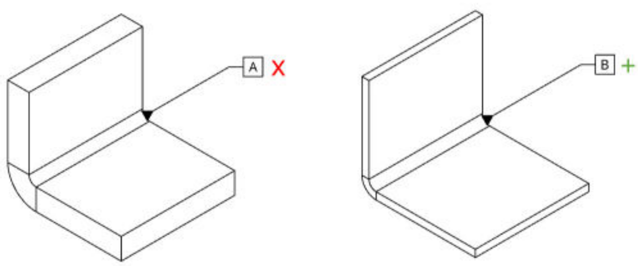
Understanding Sheet Metal Laser Cutting: Key Terms and Concepts



Minimum Flange Length Requirement

Having a flange that is too short can indeed make it impossible to manufacture a CNC-bent part using a CNC bending press brake. This limitation is primarily due to the physical constraints of the bending equipment and the tooling used.

Material Stability: During bending, the material must be supported adequately to prevent slipping or shifting. A flange that is too short doesn't provide enough surface area for the press brake's back gauge to hold the material securely.
Risk of Deformation: Without sufficient support, the metal can buckle or warp during the bending process, resulting in a part that doesn't meet specifications.



bend radius that is tighter smaller than the material thickness poses several challenges:

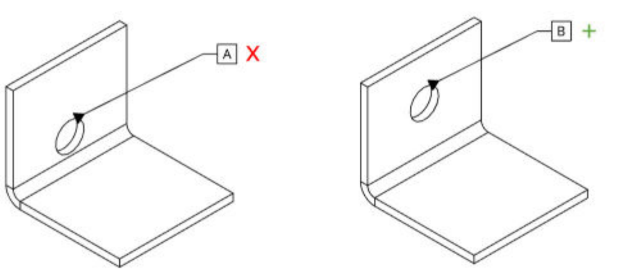
Stress Concentration: Bending a material beyond its minimum bend radius concentrates stress on the outer fibers of the bend. When the bend radius is smaller than the material thickness, the outer surface is stretched excessively.
Material Limitations: Different materials have varying levels of ductility. Less ductile materials, such as high-carbon steels or certain aluminum alloys, are more prone to cracking when bent sharply.
Cracks and Fractures: Excessive stress can lead to micro-cracks or complete fractures at the bend, compromising the structural integrity of the part.

Force Requirements: Tighter bends require significantly more force to achieve. The press brake may not have sufficient tonnage to perform the bend without exceeding its operational limits.

Distortion of Holes During Bending

Material Deformation: When sheet metal is bent, the material on the outside of the bend radius stretches (tension), while the material on the inside compresses. If a hole is too close to the bend, this stretching and compressing can distort the shape of the hole.

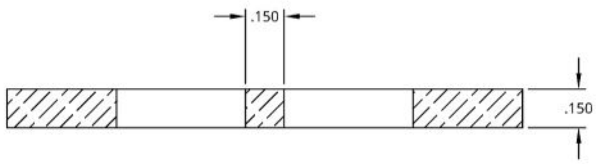
Elongation and Warping: The hole may become elongated or warped, changing from a perfect circle to an oval or irregular shape. This distortion can affect the fit and function of components that interact with the hole, such as fasteners or alignment pins.



Industry Standards: There are general guidelines for the minimum distance between a hole and a bend, often expressed as a multiple of the material thickness (e.g., at least 2 to 2.5 times the material thickness).

Minimum Distance = 2 × Material Thickness + Bend Radius

Understanding Sheet Metal Laser Cutting: Key Terms and Concepts

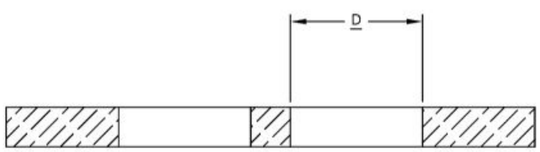


Minimum Hole Distance Guidelines.

General Rule of Thumb: The minimum recommended distance between two holes, or between a hole and an edge, is typically at least the material thickness. For example, if the material is 1 mm thick, the minimum distance should be 1 mm.
A more conservative approach is to use 1.5 to 2 times the material thickness, especially for thicker materials or critical applications.

Formula: Minimum Distance = Material Thickness × K

Where K is a factor ranging from 1.5 to 2, depending on the material and cutting process.



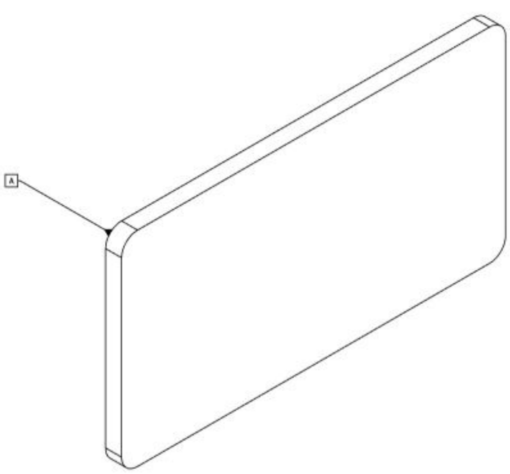
Minimum Laser Cut Hole Diameter

Thin Materials: For thin sheet metal (less than 1 mm), the laser can cut very small holes, sometimes as small as 0.2 mm in diameter.

Thick Materials: As the material thickness increases, the minimum achievable hole size increases. A common guideline is that the minimum hole diameter should be at least equal to the material thickness.

Consider Alternative Methods
Punching or Drilling: For very small holes, mechanical methods like punching or drilling might be more effective.

Secondary Operations: Small holes can be laser marked and then drilled in a separate process.



Sharp Corners

Injury Potential: Sharp corners and edges can pose a hazard to workers during handling, assembly, or installation.

Cut Hazards: There's a risk of cuts or punctures when interacting with parts that have sharp points.

Smooth Cutting Path: Adding a radius allows the laser to maintain a more constant speed and smoother motion.

Enhanced Precision: Results in cleaner cuts with more accurate dimensions.

Reduced Heat Affected Zone (HAZ): Minimizes thermal distortion and improves edge quality.