Understanding CNC MILLING: Key Terms and Concepts

Milling Cutter

Milling Cutter: A rotary cutting tool with one or more teeth used in milling operations. The cutter rotates about its axis and removes material from the workpiece.

Types of Cutters:

End Mills: Used for cutting slots, profiling, and plunge cutting.
Ball Nose Mills: Ideal for 3D contour work and complex surfaces.
Face Mills: Designed for cutting flat surfaces (faces) of the workpiece.
Slot Drills: Two-fluted end mills for plunge cutting and slotting.
Fly Cutters: Single-point cutters for fine surface finishes.

Feed Rate

Feed Rate: The distance the cutter advances into the workpiece per revolution or per minute, measured in millimeters per minute (mm/min) or inches per minute (IPM).
Importance

Surface Finish: Higher feed rates can lead to rougher finishes.
Tool Life: Incorrect feed rates can cause tool wear or breakage.
Machining Time: Optimizing feed rate improves productivity.

Depth of Cut

Depth of Cut: The thickness of the material removed in one pass of the cutter, measured perpendicular to the surface being machined.
Types

Axial Depth of Cut: Depth along the axis of the cutter.
Radial Depth of Cut: Width of the cut, or how much the cutter engages with the side of the workpiece.
Considerations

Tool Strength: Deeper cuts require stronger tools.
Machine Rigidity: Deeper cuts can cause vibrations if the machine isn't rigid enough.
Material Removal Rate: Increasing depth of cut increases material removal but may affect quality.

Step Over

Step Over: The lateral distance the cutter moves over between passes when machining flat surfaces or cavities.
Impact on Surface Finish

Smaller Step Over: Produces finer surface finish but increases machining time.
Larger Step Over: Reduces machining time but may leave visible tool marks.
Optimization

Balancing Quality and Efficiency: Choose a step over that meets surface finish requirements without unnecessary machining time.

Fixture and Workholding

Fixture: A device used to securely hold the workpiece in place during machining.

Fixture and Workholding

Types

Vises: Clamp the workpiece between jaws.
Clamps: Hold the workpiece directly against the machine table.
Custom Fixtures: Designed for specific parts or shapes.
Considerations

Stability: Prevents movement and vibration.
Accessibility: Allows the cutter to reach all necessary areas.
Repeatability: Facilitates consistent placement for multiple parts.

Surface Finish

Surface Finish: The texture of the machined surface, characterized by its roughness, waviness, and lay.
Measurement

Roughness Average (Ra): The average deviation from the nominal surface over a specified length.
Factors Affecting Surface Finish

Cutting Parameters: Feed rate, spindle speed, depth of cut.
Tool Condition: Sharpness and wear of the cutting tool.
Material Properties: Hardness and ductility of the workpiece.
Improvement Techniques

Finishing Passes: Light cuts at higher speeds to refine the surface.
Tool Selection: Using tools designed for fine finishes.
Vibration Control: Ensuring machine rigidity and proper fixturing.

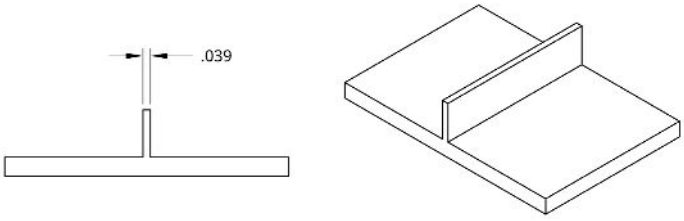
Tolerance

Tolerance: The permissible limit of variation in a physical dimension, ensuring parts fit and function as intended.
Types

Dimensional Tolerance: Variation in size (e.g., ±0.1 mm).
Geometric Tolerance: Variation in shape, orientation, or position.
Importance

Interchangeability: Ensures parts can be replaced without custom fitting.
Cost vs. Precision: Tighter tolerances increase manufacturing costs.

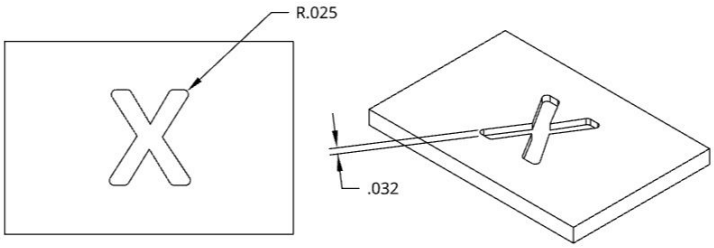
Understanding CNC MILLING: Key Terms and Concepts



MINIMUM THICKNESS AND THIN WALLS

The minimum recommended thickness for any part should exceed 1mm. It's important to exercise caution with features smaller than 1mm, as they might not withstand the machining process.

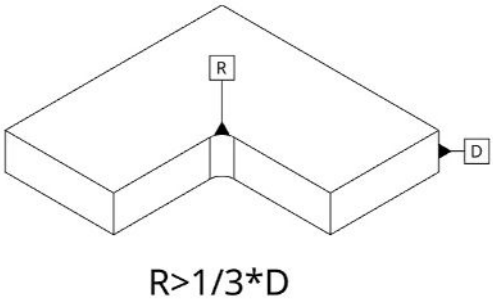
Thin walls cause Manufacting difficulty in regards to Surface Finish and Tolerance. This also can have an effect on the cost of the part to manufacter as Fixture,Workholding and Cutting Strategies have to accommodate for the lack of support



FONTS AND LOGOS

For incorporating fonts and logos, adhere to the following guidelines:

Ensure a minimum width of 1mm and a maximum depth of 1mm.
internal radius of R.025 or higher
engraved fonts are much less expensive than embossed fonts.



GORGES, POCKETS AND RADII

If the required radii are missing from the submitted file, the order will be paused, and a technician will reach out to request the necessary adjustments.

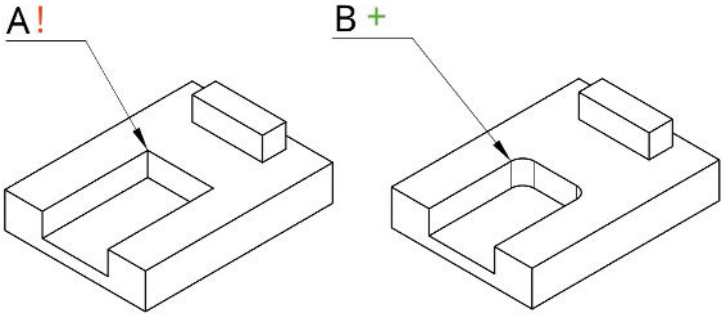
In some cases, our team of specialists may apply these radii automatically if the modification is minor.

GORGES, POCKETS AND RADII

We recommend r= 1mm minimum and 3 times less than the height (d)

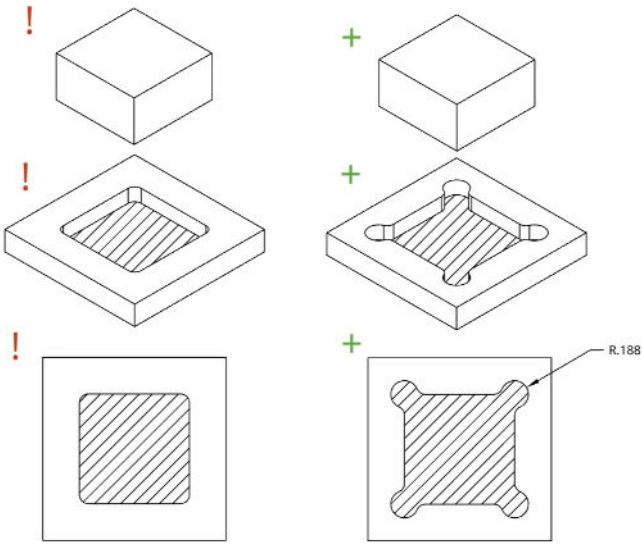
Money saver: 1mm radius is about 7 times more expensive than a 4mm radius!

Understanding CNC MILLING: Key Terms and Concepts



Internal Corner RADII

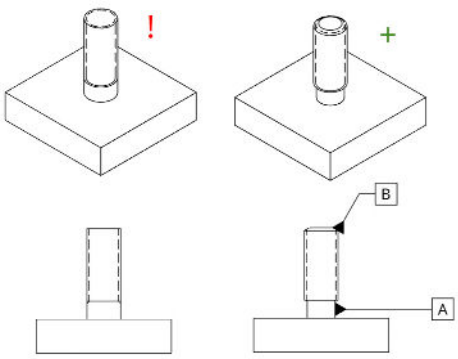
In CNC milling, internal corners typically have a radius to accommodate the round shape of the milling tool, making the process more efficient and the parts stronger. Sharp internal corners, like those in part A, are harder to machine and can create weak points in the part, whereas rounded corners in part B are easier to produce and result in a more durable component.



Internal Corner RADII

In CNC machining, it is not possible to create perfectly sharp internal corners due to the round shape of the cutting tool. A tool with a cylindrical profile will always leave a rounded corner with a radius that corresponds to the radius of the tool itself. This is why the cavity on the right shows rounded corners labeled "R.188", which indicates that the corner radius is 0.188 inches. This radius must be added to the design to allow the tool to properly clear the material and create a cavity that can be machined realistically.
The exclamation marks on the left side of the image indicate an issue, which is that sharp internal corners as depicted are not feasible with standard CNC machining practices.

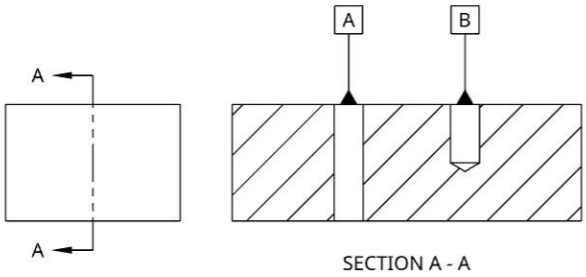
In essence, rounding the internal corners of the recess to match the radius of the external corners of the square part is a practical method to ensure a reliable and forgiving fit when assembling parts made by CNC machining. This approach takes into account the limitations and capabilities of the machining process, the reality of production tolerances, and the mechanical benefits of avoiding sharp corners.



EXTERNAL THREADS

External threads can be executed up to a maximum of 2 mm from the lower end of the cylinder.
We suggest a minimum height of 2 mm for the discharge groove to allow for the correct fit of a counterpart.

Understanding CNC MILLING: Key Terms and Concepts

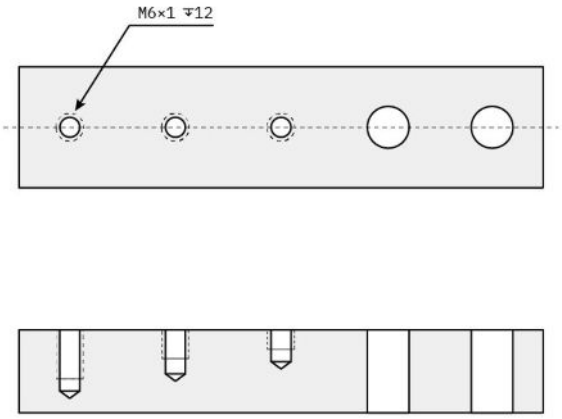


Holes and Drills

A: Blind hole threads are generally made up to a maximum of 3/4 of the hole depth.

If through-hole threads are too deep, they will be made half each side, except where indicated otherwise in the notes field or by agreement with an expert.

The maximum depth for H7 and h7 tolerance machining is generally 3 times the diameter.



Threads and H7/h7 Holes

Because there is no standard method for defining threads across various CAD software, it's not required to meticulously draw threads in your design. Instead, it's adequate to represent the intended threads by creating holes with the correct ISO diameter. For example, a simple cylindrical shape with a diameter of 4.2mm is sufficient for an M5 thread.

Understanding CNC MILLING: Key Terms and Concepts

MINIMUM AND MAXIMUM DIMENSIONS for 3-axis CNC		
Size	**IMPERIAL**	**METRIC**
Minimum dimensions:	(0.23x0.23 in)	6x6x6mm
Maximum dimensions: but may vary depending on the material	(18x18x15.7 in)	460X460X400mm
Min. feature size	Ø 0.019 in	Ø 0.50 mm

Smallest allowable dimensions: 6x6x6 mm (approximately 0.23x0.23x0.23 inches). Largest possible dimensions: 460x460x400 mm (about 18x18x15.7 inches), though this maximum size may differ based on the material selected. For material-specific size restrictions, please refer to the relevant guidelines.

Design guidelines for CNC Mill		
Feature	**Minimum size**	**Recommended size**
Min. feature size	Ø 0.50 mm	Ø 2.5 mm
Internal edgesl	R 0.25 mm	R 8 mm
Minimum wall thickness	0.8 mm	0.5 mm
Holes	Diameter: standard drill bit sizes Depth: 4 x diameter	Diameter: Ø 0.5 Depth: 10 x diameter
Threads	Size: M6 or larger Length: 3 x diameter	Size: M2

CNC Milling Nominal Tolerances		
Limits for nominal size	**Metals (ISO 2768- f)**	**Plastics (ISO 2768- m)**
0.5mm* to 3mm	±0.05mm	Soon
Over 3mm to 6mm	±0.05mm	Soon
Over 6mm to 30mm	±0.1mm	Soon
Over 30mm to 120mm	±0.15mm	Soon
Over 120mm to 400mm	±0.2mm	Soon
Over 400mm to 1000mm	±0.3mm	Soon

CNC milling nominal tolerances refer to the standard allowable deviations from specified dimensions in machined parts. Typically, CNC milling can achieve tolerances of ±0.005 inches (±0.13 mm) for metals and ±0.010 inches (±0.25 mm) for plastics. These tolerances ensure parts meet design requirements while accounting for minor variations inherent in the machining process.