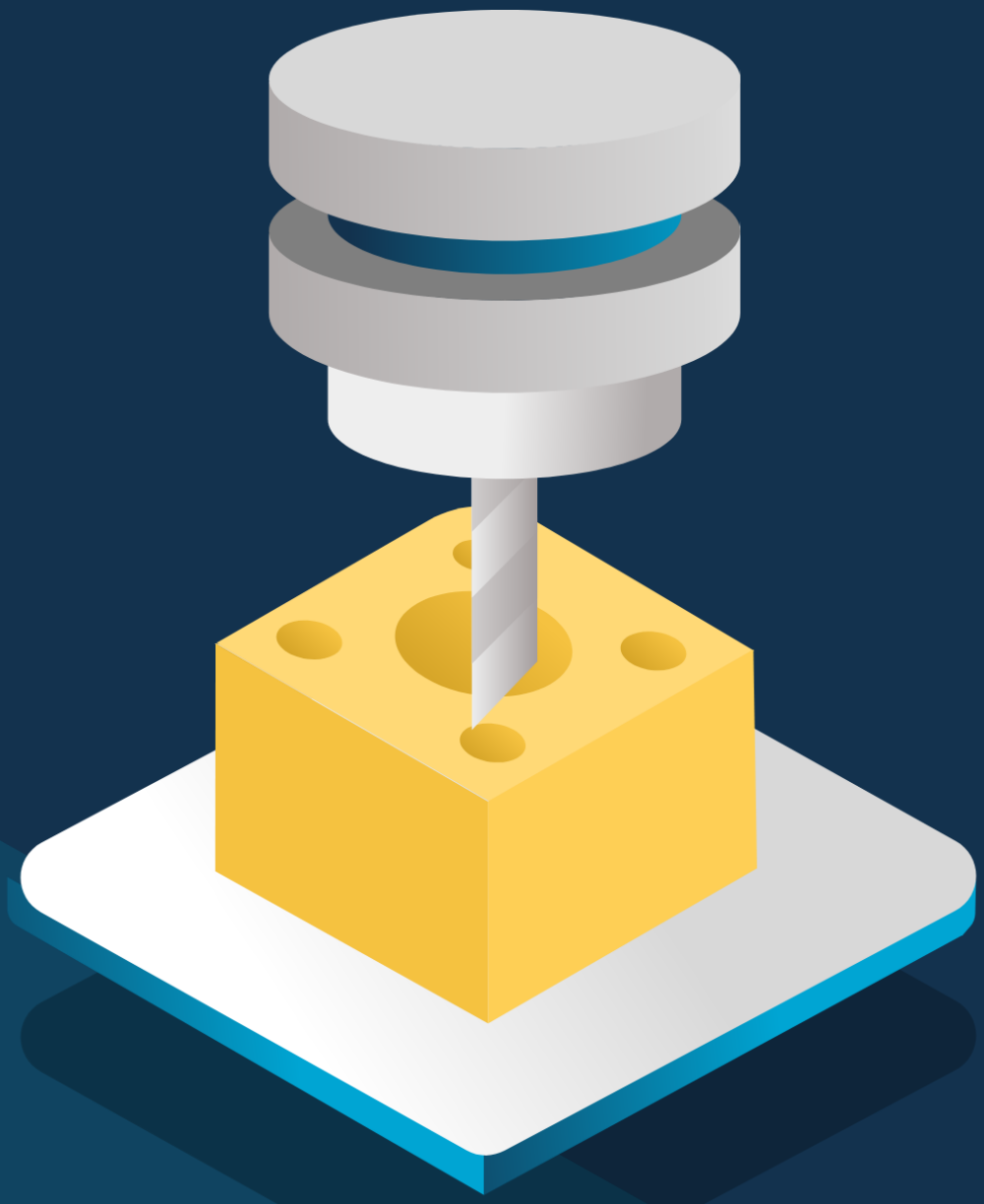
WIZ-PARTS



CNC MACHINING DESIGN GUIDE

v1.0 September 2024

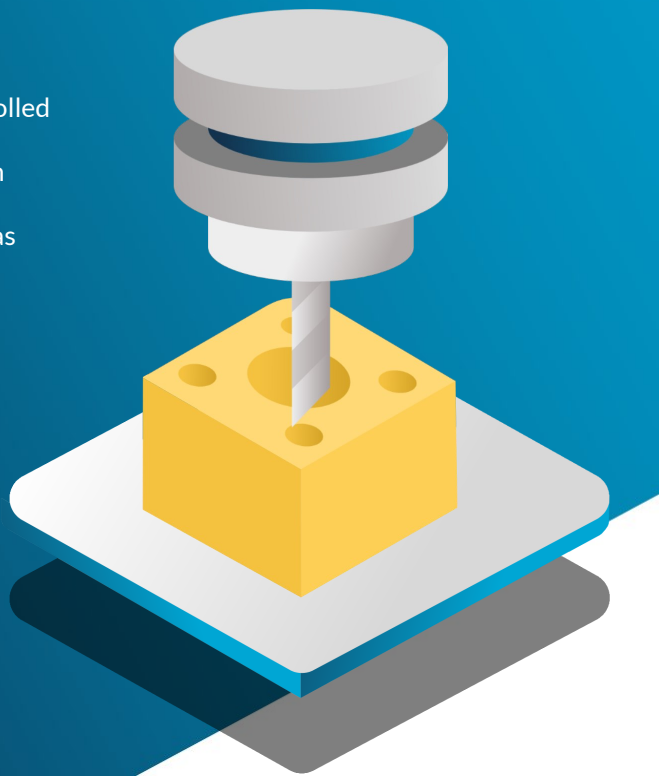
What Is CNC Machining?

CNC machining is a manufacturing process that utilizes computerized machines to precisely control and automate the removal of material from a part. CNC machines interpret CAD data to execute precise movements and cuts on the material, such as milling, drilling and turning. This technology allows for high accuracy, repeatability and efficiency whilst reducing manual labor and minimizing errors.

Contents

CNC Milling

CNC milling is a machining process where computer-controlled machines remove material from a workpiece using rotary cutters. It involves precise movements of the cutting tool in multiple axes to create complex shapes and features. CNC milling machines can perform a variety of operations such as drilling, slotting, and threading.



Use cases of CNC Milling

Prototyping: CNC milling is widely used in product development and prototyping due to its ability to quickly and accurately produce test parts and components.

Production of Complex Parts: Manufacture parts with intricate geometries and tight tolerances that are difficult to achieve with conventional machining methods.

Tool and Die Making: Used to fabricate molds, dies, and tooling used in injection molding, stamping, and other manufacturing processes.

CNC Turning

CNC turning is a machining process where computer-controlled lathes remove material from a rotating workpiece to create cylindrical parts with symmetrical features. The workpiece is held in a chuck and rotated while a cutting tool moves along the length of the part to remove material. CNC turning machines can perform operations such as facing, drilling and grooving.

Use cases of CNC Turning

Production of Rotational Parts: CNC turning is used extensively to manufacture cylindrical components such as shafts, pins, bushings, and bolts with precise dimensions.

Prototyping and Small Batch Production: CNC turning is ideal for rapid prototyping and small batch production due to its ability to quickly set up and produce parts with minimal material waste.

What Materials Can Be Machined?

CNC machining can efficiently process a variety of materials, including metals such as aluminum, mild steel, and stainless steel, as well as plastics like nylon. Each material offers unique properties that cater to specific needs, so selecting the appropriate one depends on your project's functional, aesthetic and environmental requirements.

Comparison of CNC Machining Material Properties

	Aluminium	Mild Steel	Stainless Steel	Nylon
Strength	Average	Very High	Very High	Low
Hardness	Average	High	Very High	Low
Machinability	Excellent	Very Good	Good	Excellent
Weight	Good	Poor	Poor	Very Good
Cost	£	££	£££	£

Aluminium

Ideal for machining parts where weight or corrosion resistance is paramount such as in the automotive and aerospace industries.

Mild Steel

Ideal for machining high strength components where corrosion resistance is not important or a surface finish is planned. Ideal for shafts and bosses.

Stainless Steel

Ideal for machining high strength but corrosion resistant parts such as those used in the medical, aerospace and robotics industries.

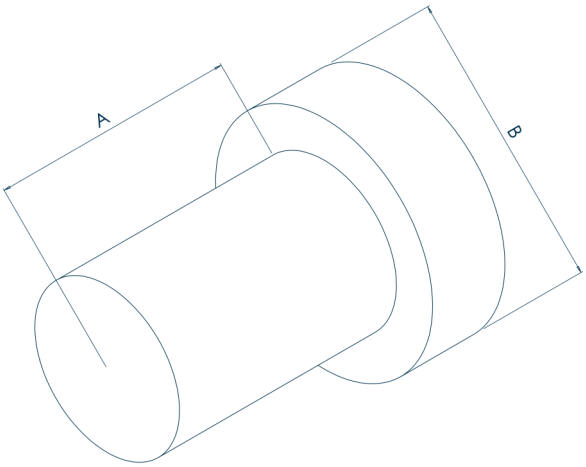
Nylon

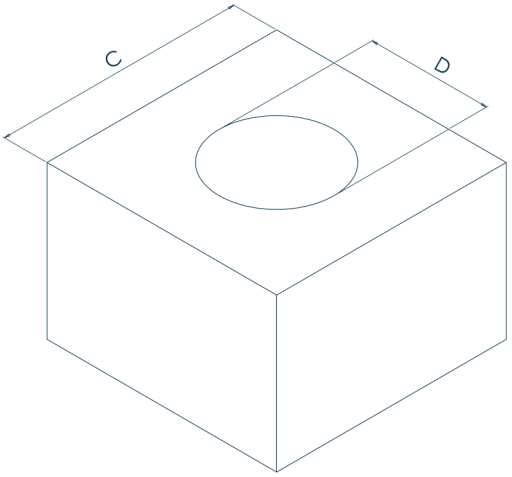
Ideal for machining low strength parts or parts which require low resistance, low wearing, contact faces such as bushings, bearings and gears.

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, CNC machined parts will be manufactured to the tolerances in the table below.





Feature	Dimension Range	Tolerance
Lathe Linear Machining (A)	0 - 120mm	+/- 0.25mm
Lathe Linear Machining (A)	120mm - 400mm	+/- 0.50mm
Lathe Radial Machining (B)	0 - 120mm	+/- 0.25mm
Lathe Radial Machining (B)	120mm - 400mm	+/- 0.50mm
Milling Linear Machining (C)	0 - 120mm	+/- 0.25mm
Milling Linear Machining (C)	120mm - 400mm	+/- 0.50mm
Drilling Diameter (D)	0 - 30mm	+/- 0.25mm
Surface Roughness	-	125RA

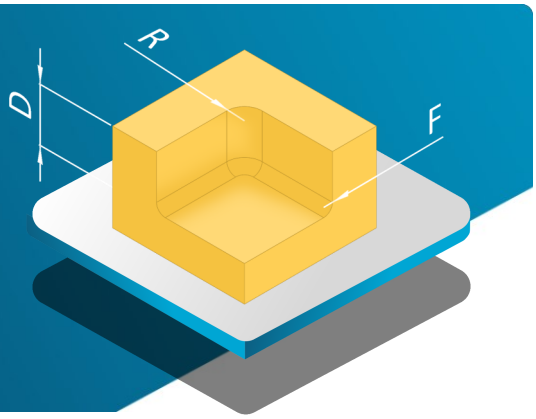
Consider Internal Radii

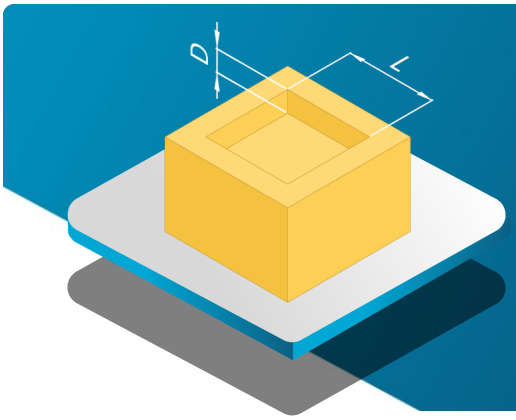
For vertical corner radii it is recommended to make the radius (R) equal to greater than one third of the cavity depth (D).

It is recommended to eliminate floor radii (F) where possible, however a radius of 0.5mm or 1mm is acceptable.

R ≥ 0.33 x D

F ≤ 1mm





Limit Cavity Depth

It is recommended to limit the depth of any cavity as much as possible. To reduce flex on the cutting tool, it is recommended that the cavity depth (D) should be less than or equal to one quarter of its length (L).

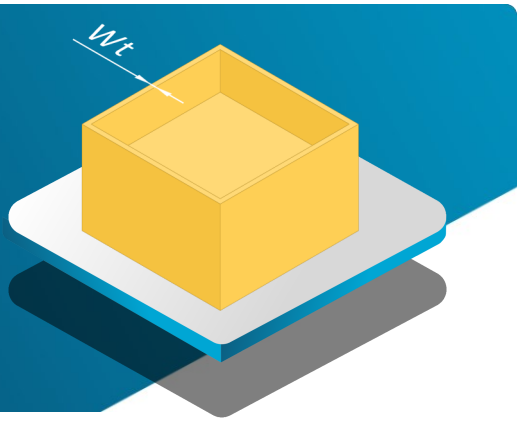
D ≤ 0.25 x L

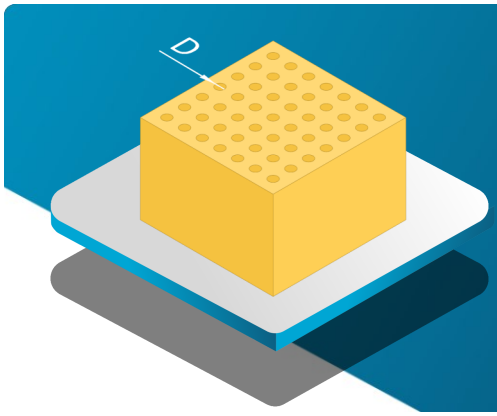
Minimum Wall Thickness

It is recommended to ensure that all machined parts have a minimum wall thickness (Wt) greater than 0.8mm for metal parts and 1.5mm for plastic parts.

Wt ≥ 0.8mm

Wt ≥ 1.5mm





Eliminate Small Features

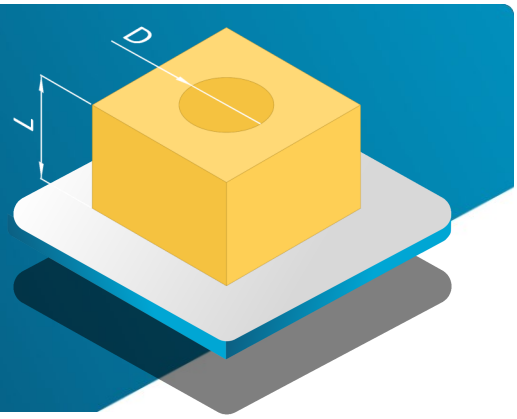
When designing CNC machined parts, it is recommended to keep machined holes and cavities equal to or greater than 2.5mm in diameter (D).

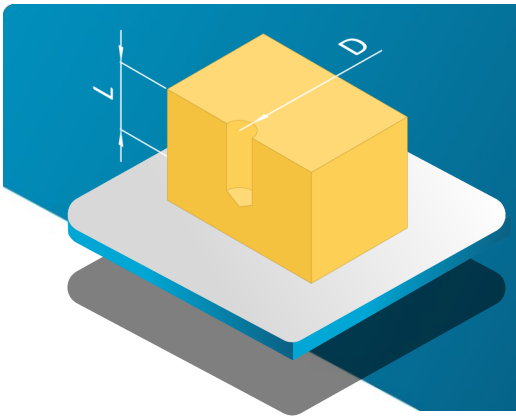
Wt ≥ 2.5mm

Use Standard Hole Sizes

As holes are machined using a drill bit or end mill tool, it is recommended when creating a hole less than 20mm to use a standard diameter. It is recommended that holes depth (L) are equal to or less than ten times its diameter (D).

L ≤ 10 x D





Blind Thread Depth

It is recommended that the depth (L) of any blind threads are between 1.5 and three times its nominal diameter (D). It is not recommended to create threaded holes smaller than M6 in size.

1.5 x D ≤ L ≤ 3 x D

Avoid Edge Holes

It is recommended to avoid edge or partial holes wherever possible. If partial holes cannot be avoided it is recommended to ensure the distance (L) is greater than or equal to two-thirds its diameter (D).

L ≥ 0.66 x D

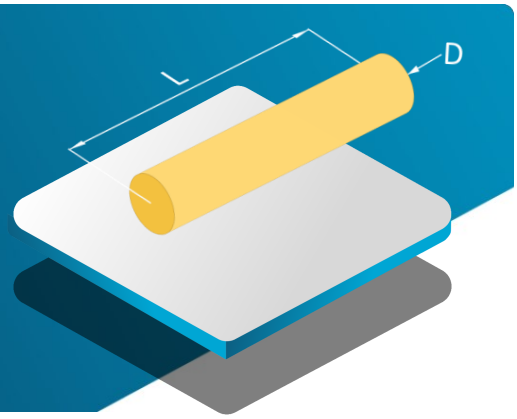
Avoid Undercuts

Wherever possible, undercuts should be avoided when designing CNC machined parts. Where it is not possible to eliminate, undercuts should be limited to T-Slot or dovetail cutting profiles.

Turned Part Dimensions

It is important to avoid long, thin turned parts as they may become unstable when machining. It is recommended to limit the part length (L) to equal to or less than eight times its diameter (D).

L ≤ 8 x D



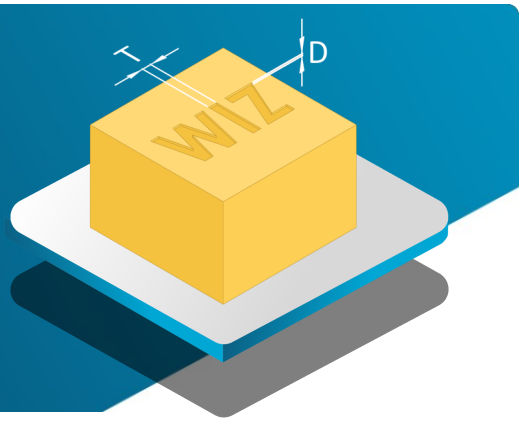
Avoid Unnecessary Features

Unlike when creating sheet metal or 3d printed parts, it is recommended to avoid adding unnecessary edge fillets or chamfers to CNC machined parts as these will increase machining time and cost.

Adding Text & Lettering

Where it is required to add text to a part, it is recommended to avoid embossed text and instead engrave text to a depth (D) between 0.3 and 0.8mm. All text should be in a sans serif font and a minimum of stroke thickness of 0.8mm.

0.3mm ≤ D ≤ 0.8mm **T ≥ 0.8mm**



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

2D CAD Files

.DXF

.DWG

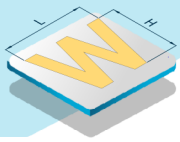
3D CAD Files

.STEP .IGES

.X_T .SLDPRT

2D Drawings

.PDF



WIZ-PARTS

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