Small-Batch
Custom Parts
Machining

Mastering Tolerances for Machined Parts

General machining tolerances and specific tolerances guidelines

6-cnc.com

TABLE OF CONTENTS

6CNC FOR ENGINEERING & SOURCING PROFESSIONALS

6CNC supports engineers, procurement teams, and product developers with precision custom parts from prototype through repeat small-batch production. Our team reviews CAD and drawings, confirms critical requirements, and builds a clear manufacturing plan around material, quantity, tolerance, finish, and inspection needs.

Precision machining for low-volume, high-mix custom parts

Engineering-led DFM review

Prototype to 1-600 pcs

Dedicated project & quality support

6CNC PRECISION MACHINING FACILITY

Based in Bao'an District, Shenzhen, 6CNC focuses on fast, cost-effective precision manufacturing for custom metal and plastic parts. Our in-house capacity supports engineering prototypes, pilot builds, and repeat low-volume production.

Quality control is integrated from pre-production through final inspection. Depending on project requirements, inspection can include CMM, image measurement, surface roughness testing, material verification, first-article inspection, and documented reports.

10+

Years of machining experience

140+

CNC machines

50,000+

Precision parts delivered

1-600

Typical project quantity

Small-batch custom parts machining with engineering support and documented quality control.

6CNC RFQ & ENGINEERING SUPPORT

Engineer-led review instead of a generic instant-quote workflow

Upload your 3D CAD file and controlled drawing with material, quantity, finish, critical tolerances, inspection requirements, target date, and delivery destination. Our engineering team reviews the project before quotation so important manufacturing and quality requirements are clear up front.

What you receive:

- Clear quotation based on the actual project requirements
- DFM feedback when geometry, tolerance, or process choices create risk
- Confirmed lead time and inspection scope
- Engineering support through production and delivery

6CNC MANUFACTURING CAPABILITIES

Capabilities for custom parts from prototype to small-batch production

CNC Machining

Precision milling for custom metal and plastic parts, from simple prismatic geometry to demanding multi-feature components.

5-Axis Machining

Multi-face and complex geometry with fewer setups, supporting tighter feature relationships and difficult tool access.

CNC Turning & Mill-Turn

Rotational parts plus off-axis features, drilling, slots, flats, threads, and live-tool operations.

Precision Grinding

Cylindrical, surface, and centerless grinding for critical dimensions, roundness, fit, and surface finish.

3D Printing & Rapid Prototyping

Fast concept and prototype support for design validation before or alongside CNC production.

Value-Added Manufacturing

Die casting, extrusion, wire EDM, vacuum casting, injection molding, and finishing support for broader project needs.

WHEN TO USE SPECIFIC TOLERANCES

There will be variation between parts

Tolerances help identify what range of variation is acceptable for the purpose.

Tolerances that have a lower range than **General Tolerances** should be called out in a drawing.

GENERAL MACHINING TOLERANCES

Bilateral Tolerances (inches)

For features of size (L, W, H, Ø) and location (position, concentricity, symmetry): ± 0.127 mm.

Sharp edges will be broken and deburred by default. Critical edges that must be left sharp should be noted or specified on a print.

For features of orientation (parallelism || and perpendicularity ⊥) and form (cylindrical, flatness ▱, circularity ○, and straightness —) apply tolerances as follows:

Distance Length (inches)	Bilateral Tolerances (inches)	Angularity degrees
0-12”	± 0.005”	± 0.5°
12-34”	± 0.010”	± 0.5°
24-36”	± 0.016” (1/64”)	± 1°
36-60”	± 0.031” (1/32”)	± 1°
Over 60”	± 0.063” (1/16”)	± 1°

GENERAL MACHINING TOLERANCES

Tolerance Standards Used in Europe

The most common tolerance standards that are used by engineers in Europe for subtractive manufacturing (e.g. CNC machining) are defined by ISO 2768 and ISO 286.

Tolerance Standard	General Linear Tolerances	General Tolerances for External Radius and Chamfer Heights	Angular Dimensions
ISO 2768 – Medium (Standard)	±0.1-2 mm depending on the nominal length from 0.5 to over 4000 mm	±0.2-1 mm depending on the nominal length from 0.5 to over 6 mm	±1°-0°5’ depending on the nominal length from up to 10 to over 400 mm
ISO 2768 – Fine	±0.05-0.5 mm depending on the nominal length from 0.5 to over 2000 mm	±0.2-1 mm depending on the nominal length from 0.5 to over 6 mm	±1°-0°5’ depending on the nominal length from up to 10 to over 400 mm
ISO 286 – Grade 8	Standardized tolerance value ranges from 0.014 to 0.33 mm depending on the nominal size from up to 3 to over 2500 mm		
ISO 286 – Grade 7	Standardized tolerance value ranges from 0.01 to 0.21 mm depending on the nominal size from up to 3 to over 2500 mm		
ISO 286 – Grade 6	Standardized tolerance value ranges from 0.006 to 0.135 mm depending on the nominal size from up to 3 to over 2500 mm		

GENERAL MACHINING TOLERANCES

ISO 2768 - Linear Dimensions

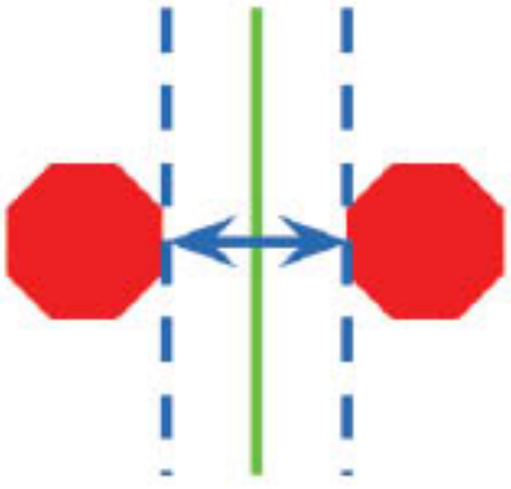
ISO 2768 and the derived geometrical tolerance standards are mainly employed for parts that are manufactured by machining or other material removing processes.

Nominal Length (mm)		Tolerance Class Values (mm)	
Above	**Up to and including**	**Fine**	**Medium**
0.5	3	±0.05	±0.1
3	6	±0.05	±0.1
6	30	±0.1	±0.2
30	120	±0.15	±0.3
120	400	±0.2	±0.5
400	1000	±0.3	±0.8
1000	2000	±0.5	±1.2
2000	4000	–	±2.0

BASIC TOLERANCES

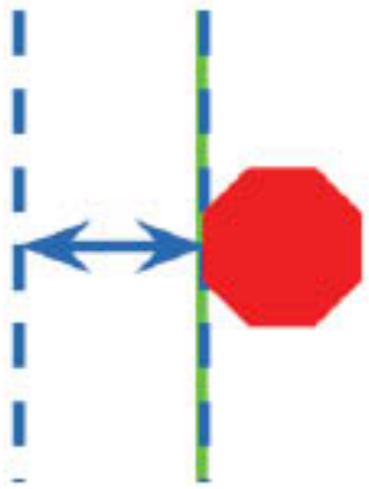
Bilateral, Unilateral, and Limit

What's the difference?
Bilateral is designer-focused Limit is inspector-focused.



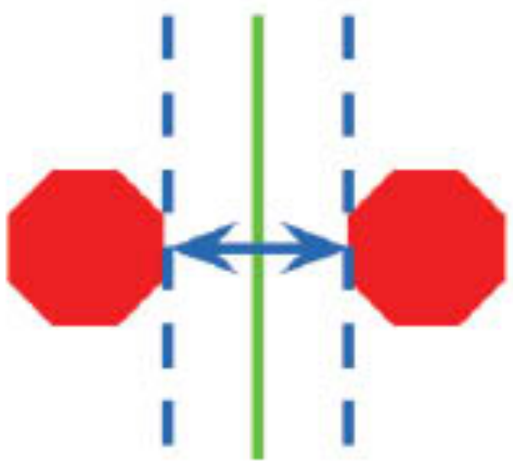
Bilateral

15.9 mm
±0.1 mm



Unilateral

16 mm
+0 mm
-0.2 mm

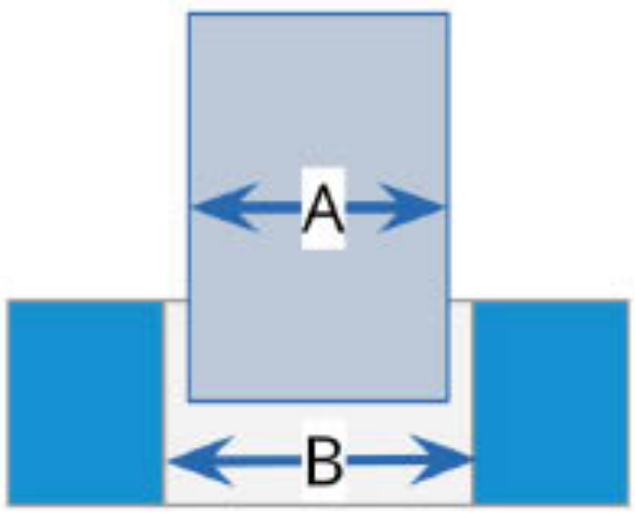


Limit

16 mm
15.8 mm

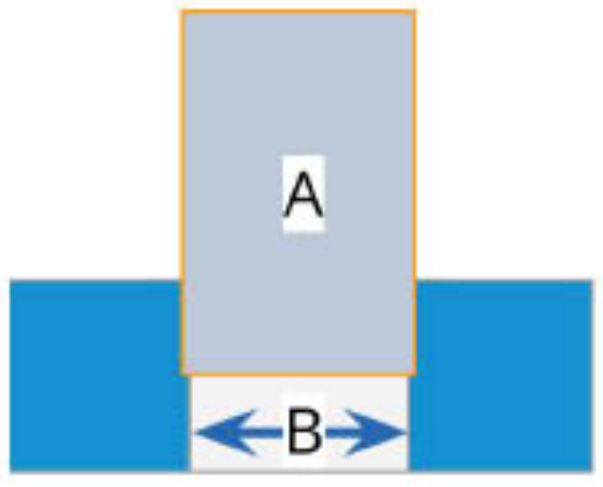
CLEARANCE AND INTERFERENCE FITS

Clearance



- Clearance fits need to be toleranced so: Any condition of A is less than any condition of B
- This should not require tools to install (loose to tight)

Interference

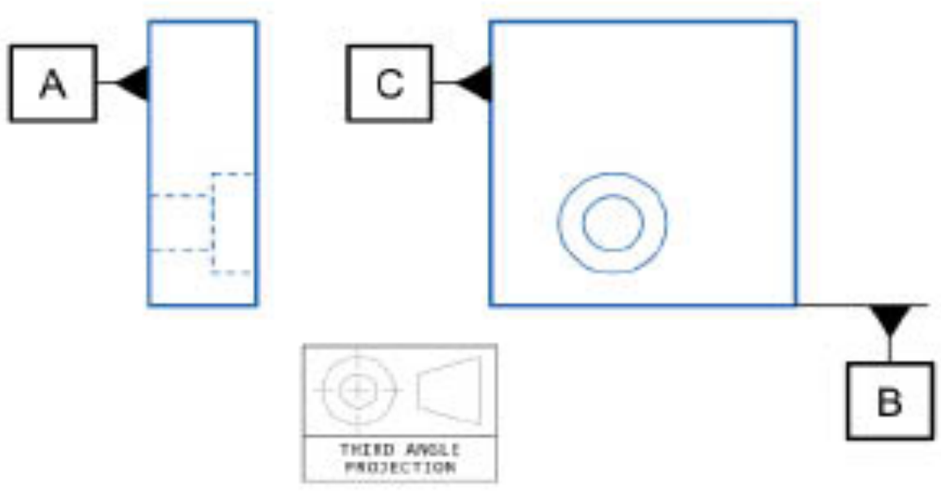


- Interference fits need to be toleranced so: Any condition of A is more than any condition of B
- This requires tools to install (press fit, shrink fits, etc.)

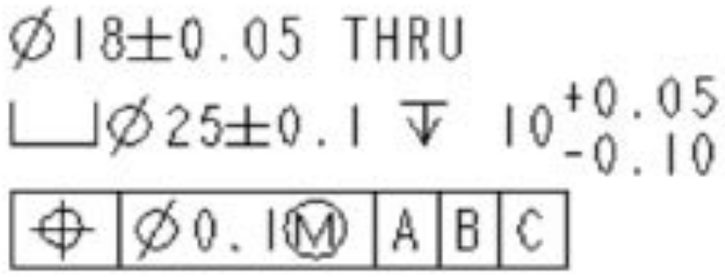
GEOMETRIC DIMENSIONING & TOLERANCE (GD&T) EXPLAINED

- GD&T is a symbol-based system of communicating feature-based tolerances

- GD&T uses set:
  - DATUMS as references for defininig feature tolerances
  - Datums are typically planes, axes, or cylinders Datums become a “universal reference” between designers, machinists, and quality assurance



GD&T can call out the counterbore dimensions, location, and tolerances by using the DATUM references.



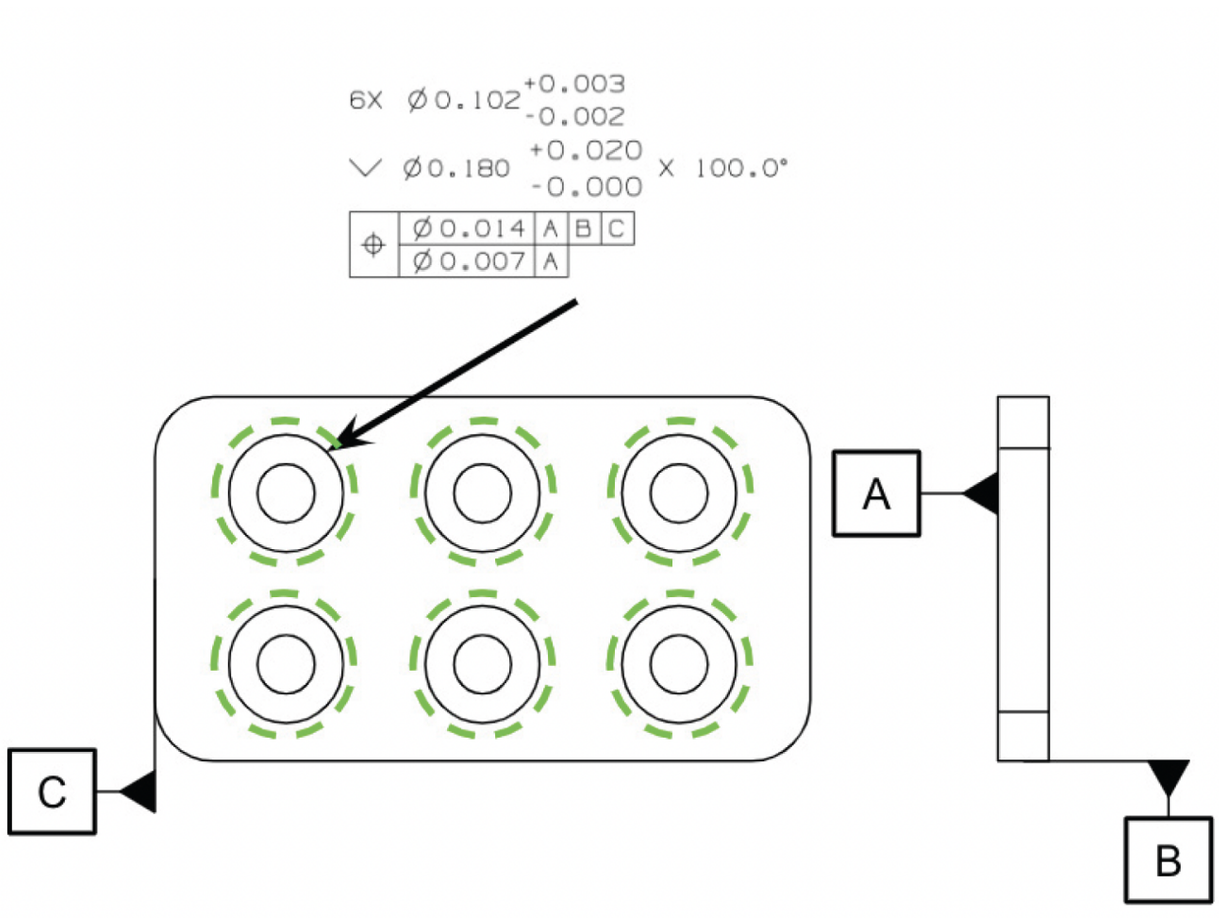
GD&T GLOSSARY

Symbol	Description
∠	Angularity
○	Circularity
◎	Concentricity
⌭	Cylindricity
▱	Flatness
||	Parallelism
⊥	Perpendicularity
⌖	Position
⌒	Profile (line)
⌓	Profile (surface)
⌰	Run-out (total)
↗	Run-out (circular)
—	Straightness
≒	Symmetry

GD&T: TRUE POSITION

True Position ⌖ is used typically for hole locations:

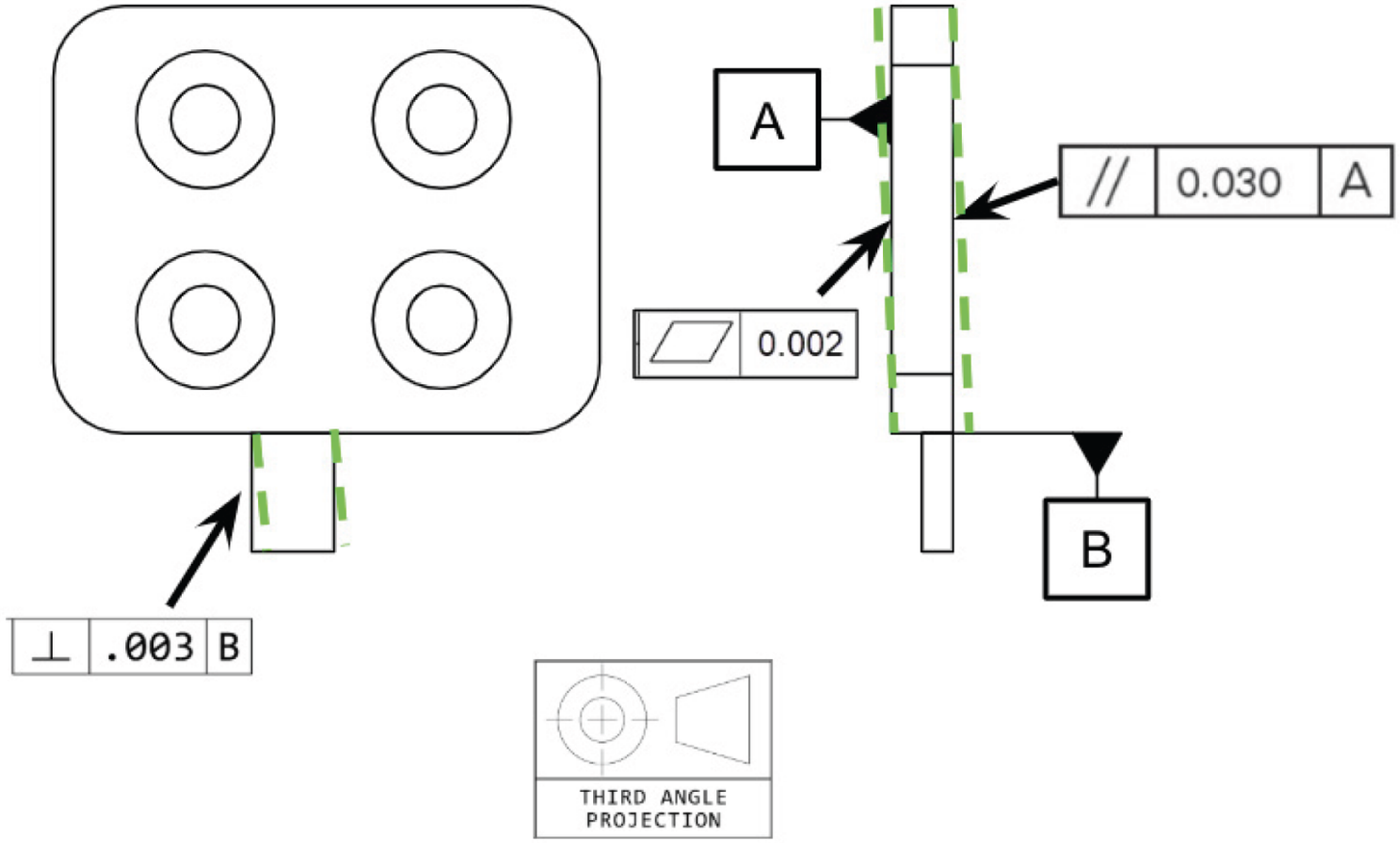
- It determines how much variance a feature can have from CAD before it is out-of-spec by referencing datums
- True position is usually used for sets of holes
- Features are machined in the same setup



These measurements typically require equipment like comparators or CMM.

GD&T: FLATNESS, PARALLELISM & PERPENDICULARITY

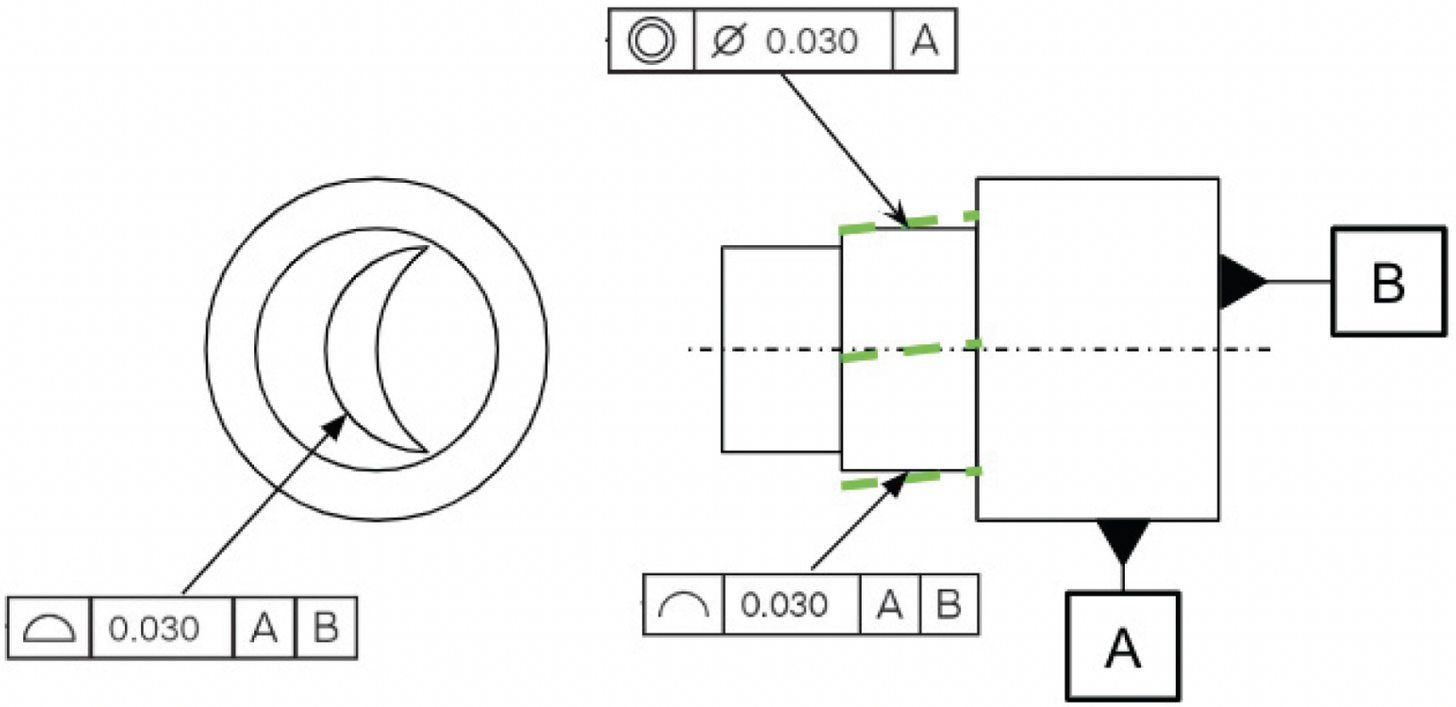
- **Flatness** ▱: non-datum call out for the acceptable variation on a plane
- **Parallelism** ||: call out to reference the acceptable variation of a surface from a parallel datum plane
- **Perpendicularity**⊥: call out to reference the acceptable variation of a surface from a perpendicular datum plane



These can typically be measured with hand tools and gauges.

GD&T: CONCENTRICITY & PROFILE

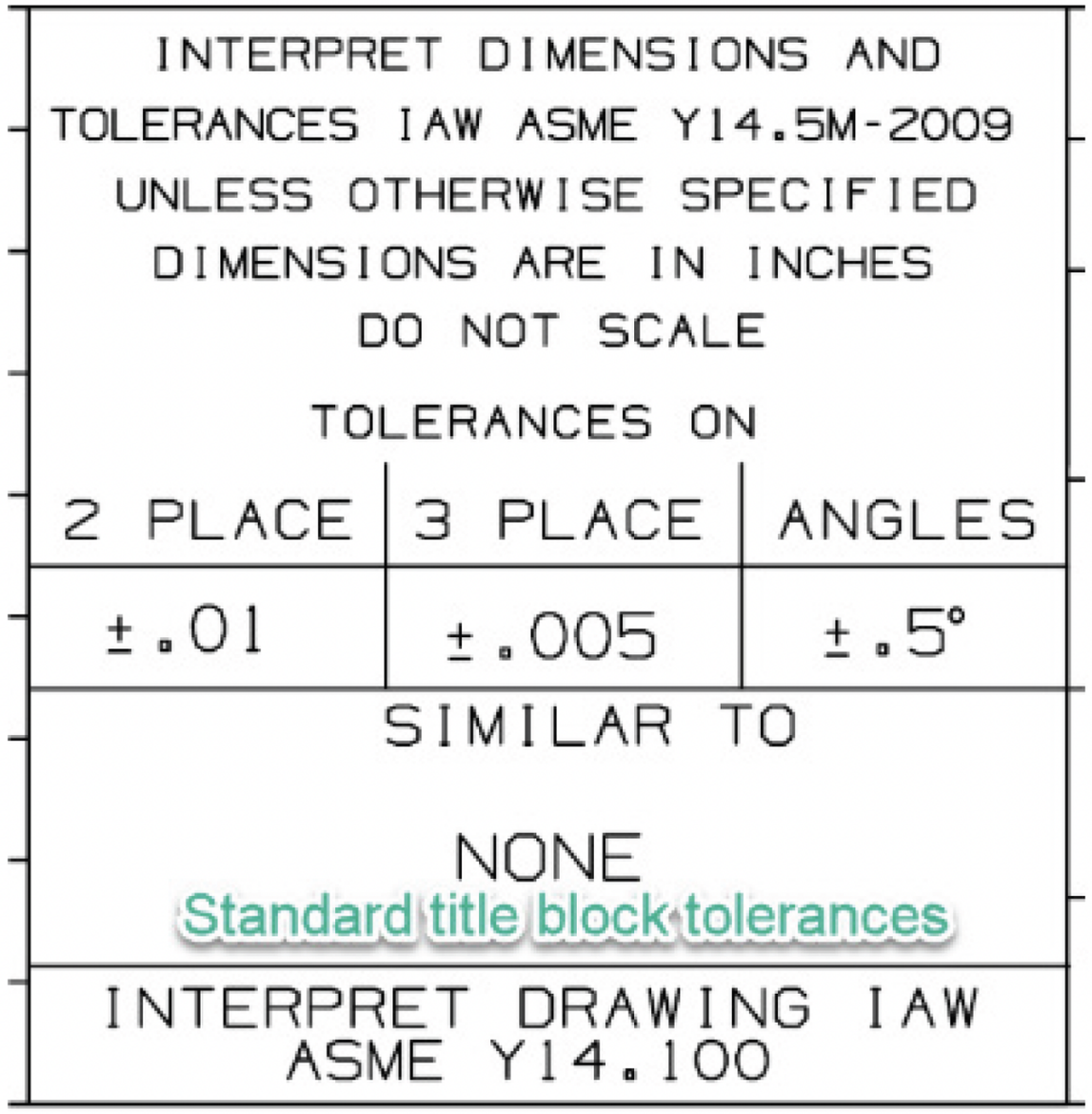
- **Concentricity** ◎ calls out the acceptable deviation from a datum axis (coaxiality)
- Profile ⌒ **(line)** or ⌓ **(surface)**, calls out the acceptable deviation of a line or surface feature, typically with curvature



These measurements typically require equipment like comparators or CMM.

WARNING: AVOID OVER-TOLERANCING

- Use your title block to cover general tolerances
- Call out tighter tolerances or GD&T on purpose-critical features only
- GD&T and very tight tolerances may require CMM, comparators, or non-manual tools



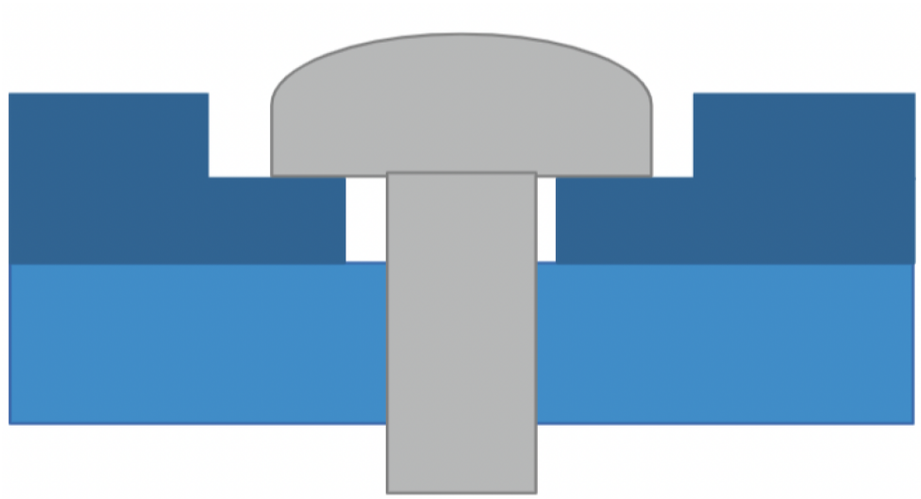
OVER-TOLERANCING ALTERNATIVES

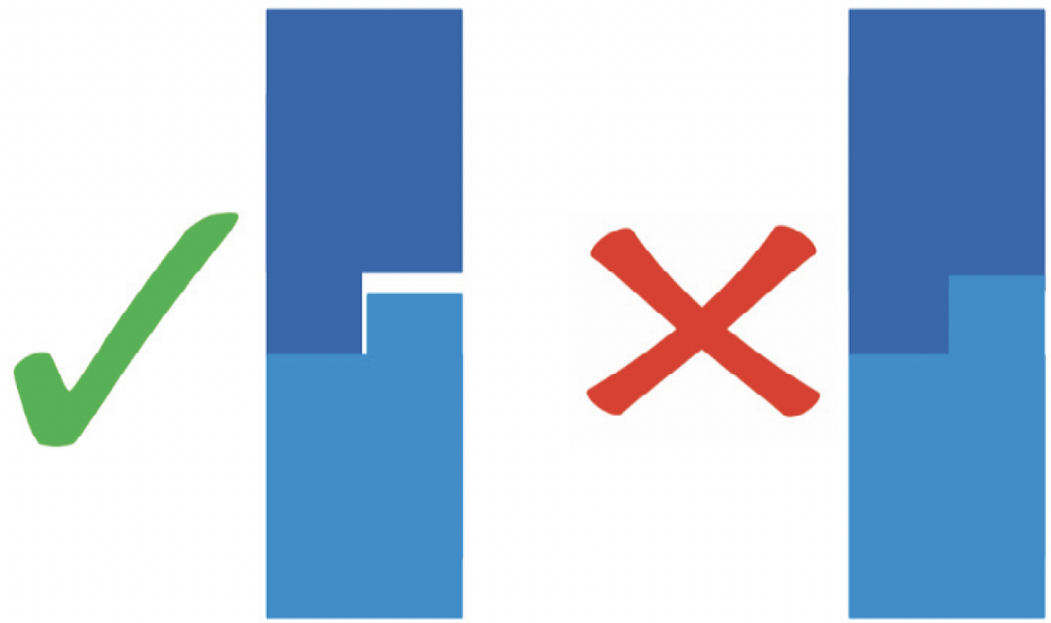
Alternatives to overtolerancing include the use of slots:

Give ample clearance fits to:

- Thru-features like bolts: counterbores and countersinks should not be the concern
- Lip mates should be designed to be flush on only one face

Dowels and indexed features mitigate the need for excessive tight tolerances.



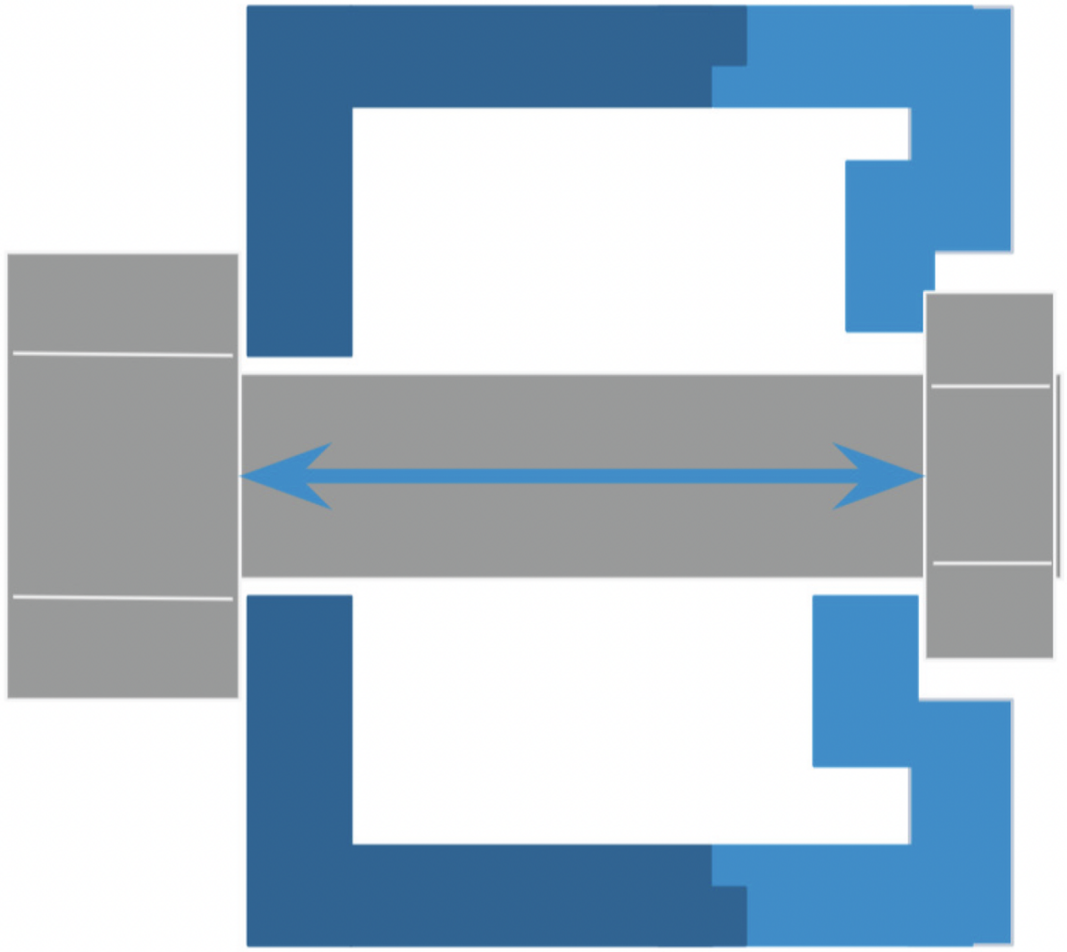


BE MINDFUL OF STACK-UP

Stack-up is when parts, typically in their maximum tolerance condition, do not assemble properly.

• When reviewing an assembly, look at where parts mate: lips, faces, edges, boss, etc.

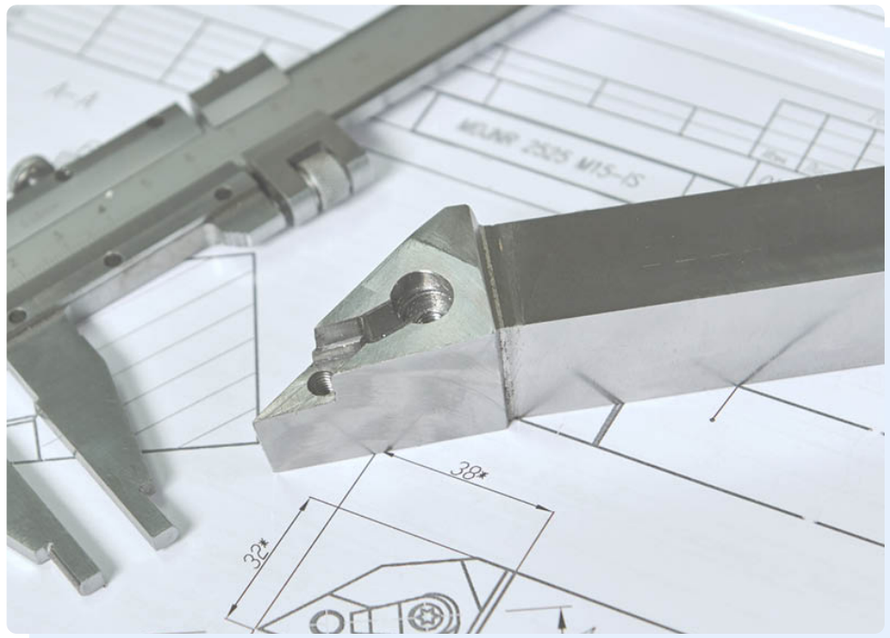
• Ensure these features are toleranced in a way where the part can function in maximum, minimum, and varied conditions



PREPPING YOUR CAD AND DRAWINGS

CAD Best Practices

- Model the part to the nominal (middle) of the tolerances. This mitigates machine operators mistakenly cutting out-of-tolerance on the first pass
- Use Hole wizards and other templates to reduce guesswork on nominals
- Model-based definitions may get lost in conversion: drawings are still necessary



Drawing Best Practices

- Use the Title Block tolerances to reduce clutter
- Only call out critical tolerances, tight tolerances, GD&T, and unilateral tolerance

Main takeaway: Make sure the drawing and CAD match.

SPECIFYING TIGHT TOLERANCES

What to send with a 6CNC RFQ

1 **Upload 3D CAD + controlled 2D drawing**

2 **Mark only function-critical tight tolerances**

3 **State manufacturing requirements**

4 **Define inspection and documentation**

6CNC Engineering Review

Before production, our team reviews:

- Datum strategy and setup sequence
- Process capability for critical features
- Tool access, wall thickness, and distortion risk
- Appropriate inspection method for each requirement
- Any tolerance or DFM risk that should be clarified before machining

Clear requirements reduce rework, inspection ambiguity, and unnecessary cost.

INSPECTION METHODS MATTER

Machining has a variety of selectable inspection options, from basic report to CMM to custom requests.

Some GD&T callouts may require CMM to complete:

- ⌖ True Position
- ◎ Concentricity
- ⌒ Line Profile
- ⌓ Surface Profile

FAQ & ENGINEERING SUPPORT

Contact 6CNC

Website: 6-cnc.com

Email: info@6-cnc.com

Phone: +86 177 4495 3189

Send your CAD files and drawing for an engineering review and project-specific quotation.

For faster engineering support

Include:

- 3D CAD model
- Controlled 2D drawing
- Material and quantity
- Surface finish / treatment
- Critical tolerances or GD&T
- Inspection and certification needs
- Required delivery date and destination

Our engineering team can help clarify manufacturability, tolerance strategy, inspection planning, and production requirements before your order is released.

Small-Batch Custom Parts Machining

6-cnc.com