

ENGINEERING DRAWINGS FOR CNC MACHINING

IDEAS
TO
PRECISION
PARTS

Dimensioning, Tolerances, Surface Finish & GD&T

A practical 6CNC technical guide for preparing clearer manufacturing drawings, reducing quotation ambiguity, and improving machining and inspection communication.



DIMENSIONING

Clear & measurable



TOLERANCES
Critical where needed



SURFACE FINISH

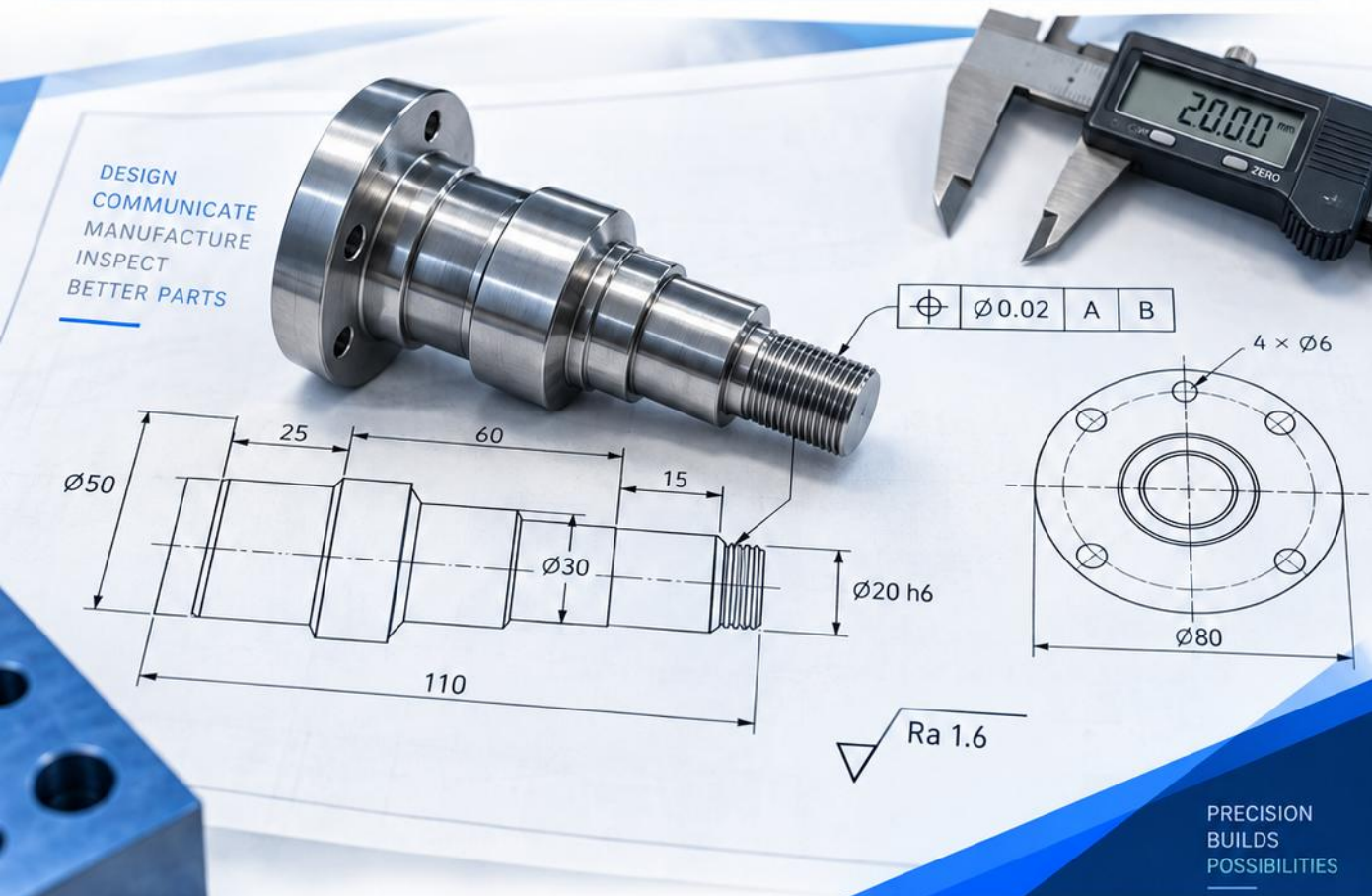
Functional surfaces



GD&T
Geometric control

WHY THIS GUIDE MATTERS

A good CNC drawing translates design intent into manufacturable, measurable requirements. It should tell the machinist what matters, tell inspection what to verify, and avoid unnecessary ambiguity.



Contents

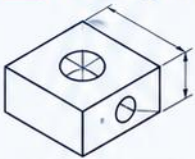
Section	Topic
01	What a manufacturing drawing must communicate
02	Dimensioning fundamentals
03	Dimensioning for manufacturability and inspection
04	Datums and dimensioning schemes
05	Dimensional tolerances and fits
06	Surface roughness
07	GD&T fundamentals
08	Common drawing problems
09	6CNC drawing-submission checklist

YOUR GUIDE AT A GLANCE

Nine essential topics to help you create clearer, more complete manufacturing drawings, reduce quotation ambiguity, and achieve better machining and inspection results.

CLEAR DRAWINGS
BETTER PARTS
FASTER RESULTS

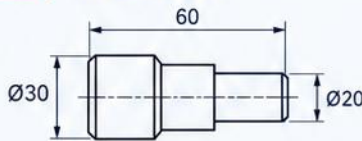
01 What a manufacturing drawing must communicate



DESIGN
MANUFACTURE
INSPECT
BETTER PARTS

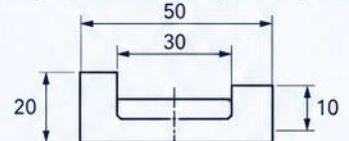
Ensure the drawing clearly communicates intent, requirements and critical information.

02 Dimensioning fundamentals



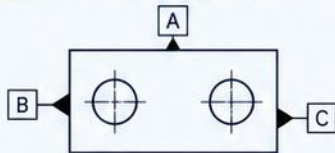
Use clear, consistent and measurable dimensioning practices.

03 Dimensioning for manufacturability and inspection



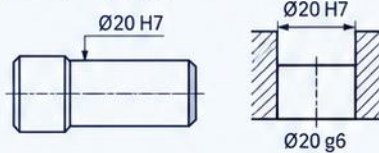
Apply dimensioning that supports cost-effective machining and reliable inspection.

04 Datums and dimensioning schemes



Define datums correctly and use appropriate dimensioning schemes (e.g. baseline, chain).

05 Dimensional tolerances and fits



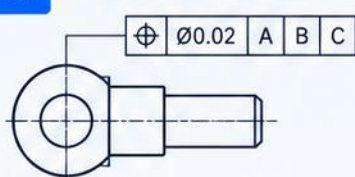
Specify appropriate tolerances and understand fits for functional requirements.

06 Surface roughness



Specify surface finish requirements to meet function and performance.

07 GD&T fundamentals



Use GD&T to control form, orientation and location where needed.

08 Common drawing problems



- Missing information
- Ambiguous dimensions
- Conflicting tolerances
- Unclear datums
- Non-manufacturable detail

Recognize and avoid common issues that cause delays, higher costs or rejects.

09 6CNC drawing-submission checklist



- Complete information
- Clear dimensions
- Tolerances and GD&T
- Material and finish
- Ready for quotation

Use our checklist to submit high-quality drawings and get accurate, faster quotes.

SECTION 01

What a Manufacturing Drawing Must Communicate

A complete part drawing should communicate geometry, manufacturing and inspection dimensions, technical requirements, and identifying information.

Requirement	Meaning
Correct	Use consistent symbols, dimensions and technical callouts.
Complete	Provide all information required to manufacture and inspect the part.
Clear	Avoid ambiguity, clutter and unnecessary duplication.
Reasonable	Dimension with design function, manufacturing method and measurement access in mind.

A complete drawing normally contains

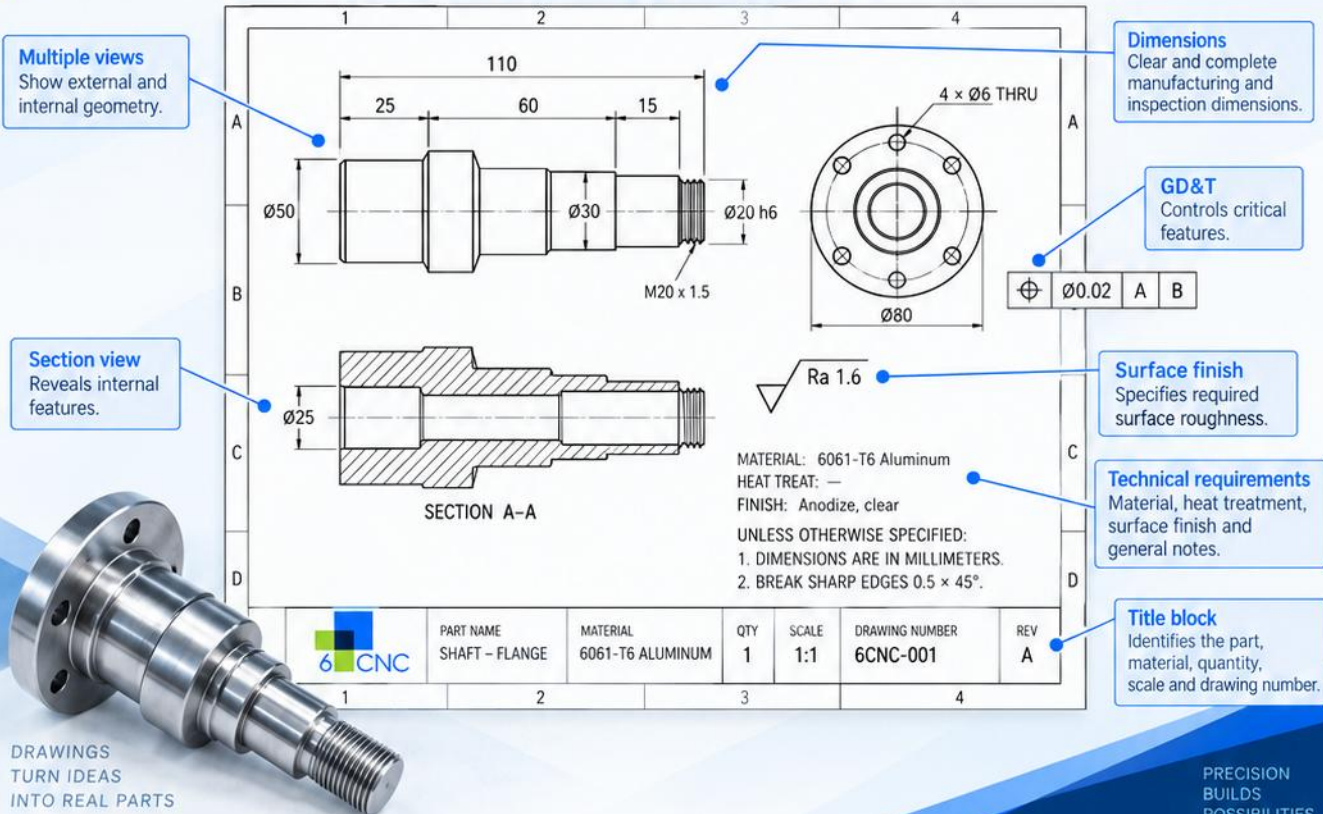
- Views that clearly describe external and internal geometry.
- Complete dimensions needed for manufacturing, inspection and assembly.
- Technical requirements such as dimensional tolerance, geometric tolerance, surface roughness, heat treatment or surface treatment where applicable.
- A title block identifying the part, material, quantity, scale and drawing number.

6CNC PRACTICE

A 3D CAD model is valuable for quotation and programming, but a 2D drawing remains important whenever tolerances, GD&T, surface finish, threads or special requirements must be controlled.

EXAMPLE: A COMPLETE MANUFACTURING DRAWING

Key elements that communicate design intent, manufacturing requirements and inspection criteria.



SECTION 02

Dimensioning Fundamentals

IDEAS
TO
PRECISION
PARTS

The manufactured part is controlled by the stated dimension values, not by the apparent size or scale of the drawing. Dimensions should normally represent the finished condition unless another condition is explicitly identified.

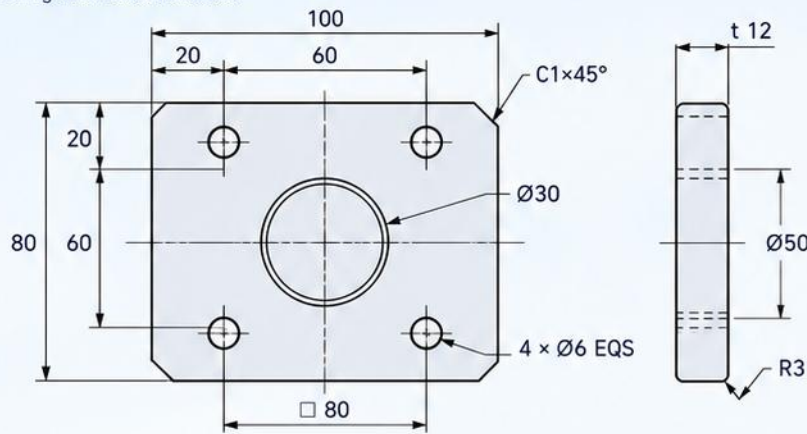
- A dimension should normally be stated only once.
- Place the dimension on the view that shows the feature most clearly.
- Dimension lines and extension lines should remain distinct from object outlines.
- Dimension text should not be crossed by other drawing lines.
- Use standard feature notation for diameter, radius, chamfer, depth and countersink.

Feature	Common callout
Diameter	Ø
Radius	R
45° chamfer	C
Thickness	t
Square	□
Equal spacing	EQS

DIMENSIONING EXAMPLE

A machined plate showing common dimensioning features and callouts.

CLEAR DIMENSIONS | STANDARD CALLOUTS | MANUFACTURABLE DRAWINGS



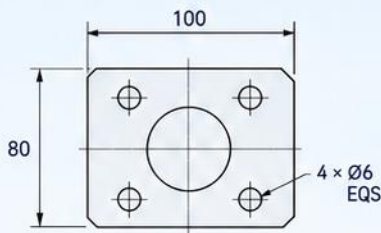
FEATURE CALLOUTS IN THIS EXAMPLE

- 1 Ø30 - Diameter
- 2 R3 - Radius
- 3 C1 x 45° - 45° chamfer
- 4 t 12 - Thickness
- 5 □ 80 - Square
- 6 4 x Ø6 EQS - Equal spacing



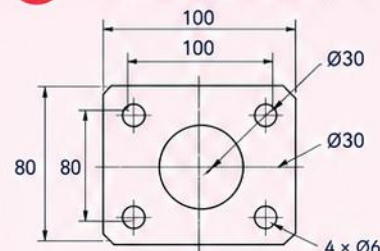
GOOD PRACTICE

State each dimension once and place it clearly.



POOR PRACTICE

Dimensions repeated, unclear placement and crossed lines.



- Dimension stated more than once.
- Crossing lines through dimension text.
- Unclear placement.
- Harder to read and can cause errors.

DESIGN
COMMUNICATE
MANUFACTURE
INSPECT
BETTER PARTS

SECTION 03

Dimensioning for Manufacturability & Inspection

A drawing can be geometrically correct and still be difficult to machine or inspect. The source repeatedly emphasizes that dimensions should be arranged for practical manufacturing and measurement.

Important dimensions should be direct

Critical functional dimensions should be stated directly instead of being derived from a chain of other dimensions. This reduces accumulated variation and makes inspection requirements clearer.

Avoid closed dimension chains

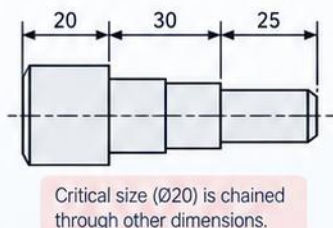
A fully closed dimension chain can over-constrain a drawing. Leave a non-critical dimension as the open link so important dimensions remain controlled without conflicting requirements.

Poor practice	Better practice
Critical size derived from several chained dimensions.	State the critical size directly.
Depth defined only through a difficult internal reference.	Dimension from an accessible end face or inspection datum.
Mixed references without clear datum logic.	Group dimensions around functional and process references.

1 Critical dimension should be direct

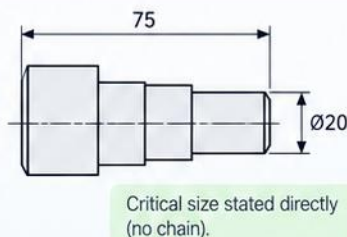
✗ POOR PRACTICE

Critical size derived from several chained dimensions. Variation accumulates and inspection is less clear.



✓ BETTER PRACTICE

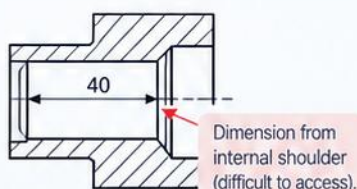
State the critical size directly. Other dimensions can still be given, but the key functional requirement is independent.



2 Use accessible references for inspection

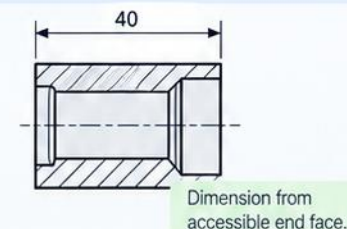
✗ POOR PRACTICE

Depth defined only through a difficult internal reference. Hard to measure and may require special gages.



✓ BETTER PRACTICE

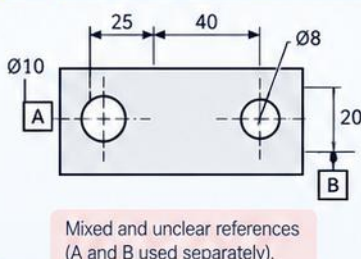
Dimension from an accessible end face or inspection datum. Easier to measure and explains inspection clearly.



3 Use consistent, grouped references

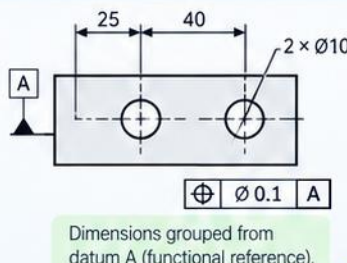
✗ POOR PRACTICE

Mixed references without clear datum logic. May cause confusion in manufacturing and inspection.



✓ BETTER PRACTICE

Group dimensions around functional and process references. Use a consistent datum scheme.



SECTION 04

Datums & Dimensioning Schemes

A datum is the point, line or surface from which dimensions are established. The source distinguishes design datums from process datums and primary references from auxiliary references.

Datum type	Purpose
Design datum	Defines functional position and geometric relationship in the design.
Process datum	Supports locating, machining, measurement or inspection.
Primary datum	Controls the main dimensional direction or functional relationship.
Auxiliary datum	Provides additional reference where needed.

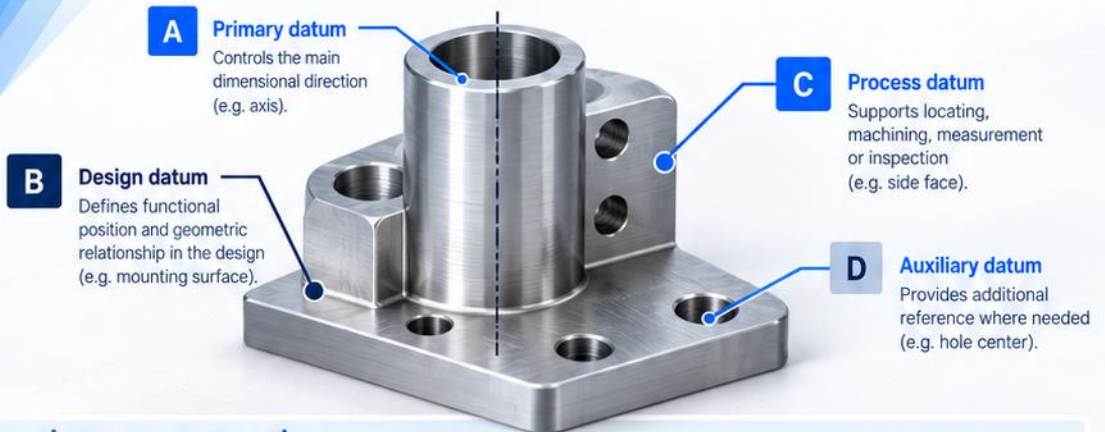
Three dimensioning schemes

Scheme	Behavior	Effect
Baseline / coordinate	Dimensions originate from one datum.	Reduces propagation of dimensional error.
Chain	Each dimension starts from the previous feature.	Variation can accumulate along the chain.
Combined	Uses baseline and chain dimensions together.	Useful when functional priorities are clear.

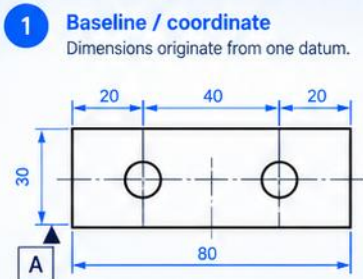
DATUM COINCIDENCE

The source recommends making the principal dimensioning datum coincide with design and process references where practical.

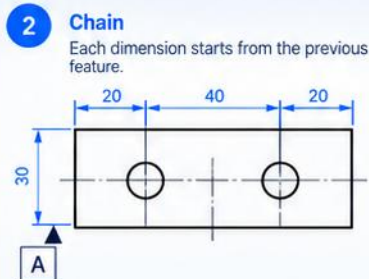
CLEAR
REFERENCES
BETTER
PARTS



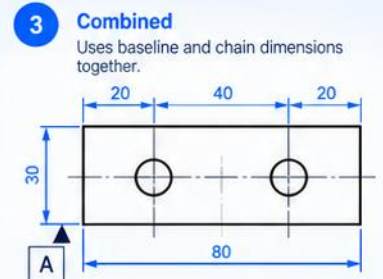
Dimensioning schemes — comparison



Reduces propagation of dimensional error.



Variation can accumulate along the chain.



Useful when functional priorities are clear.

SECTION 05

Dimensional Tolerances & Fits

IDEAS
TO
PRECISION
PARTS

Tolerance is the permitted variation in a dimension. The source distinguishes basic size, actual size, limit sizes, upper and lower deviations, and the tolerance zone.

Term	Meaning
Basic size	Nominal design dimension.
Actual size	Measured size of the manufactured feature.
Limit sizes	Maximum and minimum permitted sizes.
Upper / lower deviation	Difference between a limit size and the basic size.
Tolerance	Maximum limit size minus minimum limit size.
Tolerance zone	Region bounded by upper and lower deviations.

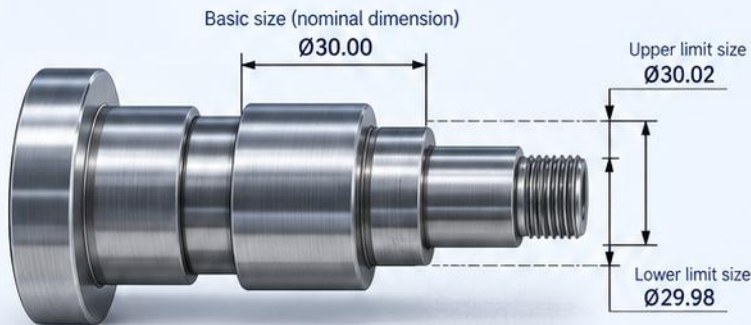
Fits

Fit type	Relationship
Clearance fit	The mating condition provides clearance.
Interference fit	The mating condition requires interference.
Transition fit	Actual sizes may produce small clearance or small interference.

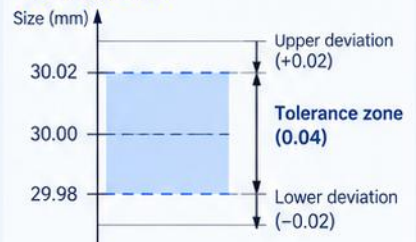
6CNC NOTE

Do not apply tight tolerances to every dimension by default. Critical tolerances should correspond to real functional or assembly requirements.

DIMENSIONAL TOLERANCE TERMS



Tolerance zone

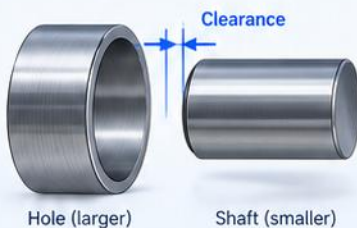


$$\text{Tolerance} = \text{Upper limit size} - \text{Lower limit size} = 0.04 \text{ mm}$$

TYPES OF FITS (HOLE AND SHAFT)

CLEARANCE FIT

The mating condition provides clearance.



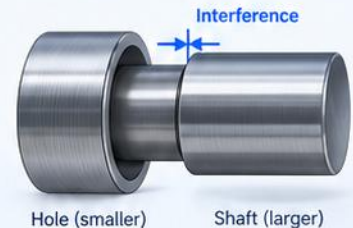
TRANSITION FIT

Actual sizes may produce small clearance or small interference.



INTERFERENCE FIT

The mating condition requires interference.



SECTION 06

Surface Roughness

Surface roughness describes the small-scale peaks and valleys on a manufactured surface. The source notes that roughness influences wear, corrosion behavior and fit quality.

Ra range from source	Source examples
12.5 - 6.3 μm	Rough turning, milling, planing, drilling and other rough-machined surfaces.
3.2 - 1.6 μm	Finish turning, finish milling, reaming and general mating/contact surfaces.
0.8 - 0.2 μm	Fine reaming, grinding and polishing for higher-demand surfaces.
0.1 - 0.025 μm	Lapping / superfinishing for very high precision or sealing surfaces.

- Apply roughness to the specific functional surface.
- When most surfaces share one requirement, a general note may be used with exceptions identified separately.
- Repeated identical features generally need the same roughness callout only once when the scope is clear.

IMPORTANT

These process examples are source-derived teaching ranges, not a guaranteed 6CNC capability table. Actual finish depends on material, geometry, process, machine, tooling and inspection method.

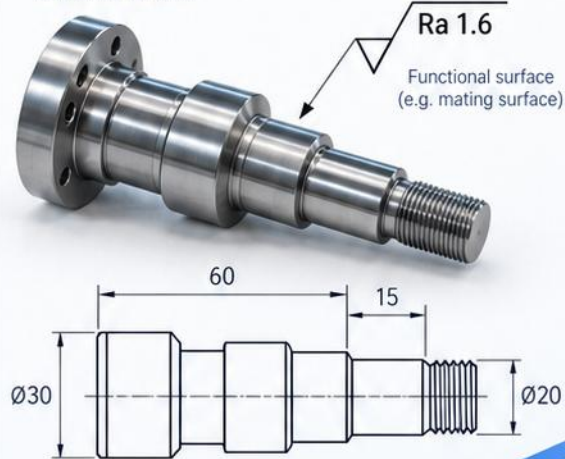
SURFACE ROUGHNESS COMPARISON

Lower Ra values mean a smoother surface with smaller peaks and valleys.

	Ra (μm)	Surface profile (illustrative)	Typical process / application
ROUGHER SURFACE ↑	12.5		Rough turning, milling, planing, drilling and other rough-machined surfaces.
	6.3		
	3.2		Finish turning, finish milling, reaming and general mating/contact surfaces.
	1.6		
↓ SMOOTHER SURFACE	0.8		Fine reaming, grinding and polishing for higher-demand surfaces.
	0.2		Lapping / superfinishing for very high precision or sealing surfaces.

EXAMPLE: ROUGHNESS CALLOUT ON A PART DRAWING

Apply the surface finish symbol to the specific functional surface.



SMOOTHER SURFACES
BETTER PERFORMANCE

GOOD SURFACE FINISH
BUILDS BETTER PARTS

PRECISION
BUILDS
POSSIBILITIES

SECTION 07

GD&T Fundamentals

Geometric tolerances control permitted variation in form and position. The source groups them into form tolerances and position-related tolerances, including orientation, location and runout.

Category	Examples in the source
Form	Straightness, flatness, circularity, cylindricity, line profile, surface profile
Orientation	Parallelism, perpendicularity, angularity
Location	Position, coaxiality/concentricity, symmetry
Runout	Circular runout, total runout

Feature-control frames and datums

A geometric tolerance callout combines the geometric characteristic, tolerance value, leader and datum references where required.

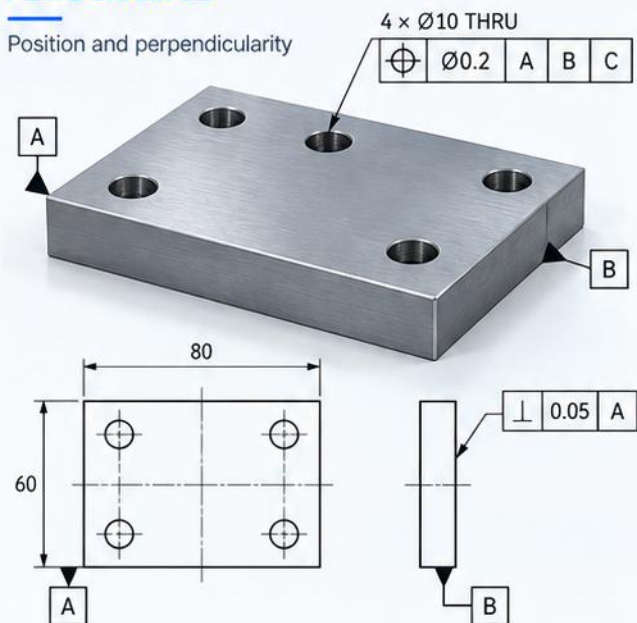
- For a controlled surface or outline, the leader points to that surface or its extension.
- For an axis or center plane, the leader aligns with the corresponding size dimension.
- Datum symbols distinguish a datum surface from a datum axis or center plane.

6CNC PRACTICE

Use GD&T when the functional requirement is geometric - for example alignment, perpendicularity, position or runout - rather than trying to control geometry indirectly with multiple $\pm$ dimensions.

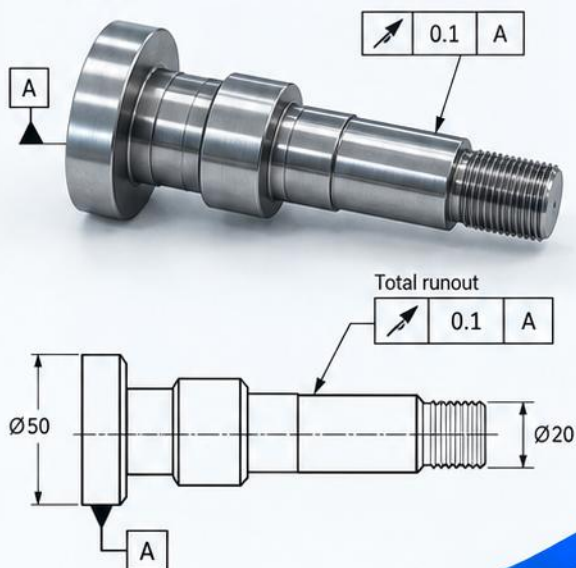
PLATE EXAMPLE

Position and perpendicularity



SHAFT EXAMPLE

Total runout



Common Drawing Problems in CNC RFQs

Problem	Why it creates risk	Better approach
Missing dimensions	Manufacturer must infer geometry or request clarification.	Provide all manufacturing and inspection dimensions.
Duplicate / conflicting dimensions	Two callouts may define one feature differently.	State each dimension once in the clearest view.
No tolerance on a critical size	Acceptable variation is undefined.	Specify the functional tolerance directly.
Closed dimension chain	Requirements may conflict after tolerance accumulation.	Leave a non-critical open link.
Poor datum logic	Machining and inspection references become unclear.	Choose functional datums and use them consistently.
Hard-to-measure dimensions	Inspection requires indirect calculation or special setup.	Dimension from accessible references.
Incomplete technical requirements	Finish, heat treatment or surface treatment may be missed.	State all applicable requirements.
Unclear title block	Material, revision or drawing identity may be ambiguous.	Complete material, part name, drawing number and revision.

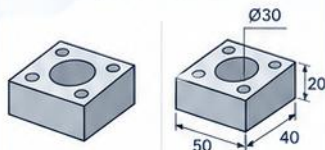
VISUAL SUMMARY

Common issues are easy to fix. These examples show typical problems and better approaches to help you create clearer, more manufacturable drawings and get accurate quotes.

DESIGN
COMMUNICATE
MANUFACTURE
INSPECT
BETTER PARTS

1 Missing dimensions

Manufacturer must infer geometry or request clarification.

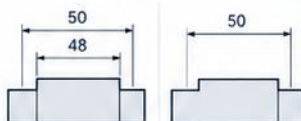


✗ Missing dimensions

✓ All key dimensions provided

2 Duplicate / conflicting dimensions

Two callouts may define one feature differently.

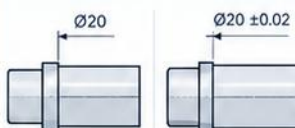


✗ Conflicting dimensions

✓ One clear dimension

3 No tolerance on a critical size

Acceptable variation is undefined.

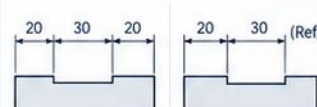


✗ No tolerance on critical size

✓ Specify functional tolerance

4 Closed dimension chain

Requirements may conflict after tolerance accumulation.

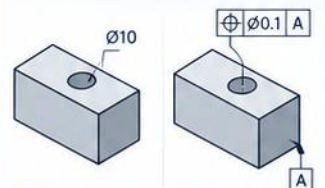


✗ Closed dimension chain

✓ Leave a non-critical open link

5 Poor datum logic

Machining and inspection references become unclear.



✗ No datums or inconsistent datums

✓ Use functional datums consistently

6 Hard-to-measure dimensions

Inspection requires indirect calculation or special setup.



✗ Hard to measure (deep pocket)

✓ Dimension from accessible references

7 Incomplete technical requirements

Finish, heat treatment or surface treatment may be missed.



✗ No finish or process requirements

✓ State all applicable requirements

8 Unclear title block

Material, revision or drawing identity may be ambiguous.

PART NAME	Shaft
MATERIAL	4140 Steel
DWG NO.	6CNC-001
REV.	A

✗ Incomplete title block

✓ Complete material, part name, drawing number and revision



SECTION 09

6CNC Drawing Submission Checklist

Use this checklist before submitting a CNC machining RFQ.

Check	Requirement
☐	3D CAD model attached where available
☐	2D drawing attached for controlled requirements
☐	Material grade specified
☐	Critical dimensions identified
☐	Dimensional tolerances specified
☐	Datum scheme and GD&T defined where required
☐	Surface roughness specified on functional surfaces
☐	Threads fully defined, including depth where applicable
☐	Heat treatment specified if required
☐	Surface treatment / coating specified if required
☐	Inspection or reporting requirements specified
☐	Drawing number and revision are clear

Recommended RFQ package

- 3D CAD model, preferably a neutral manufacturing format such as STEP/STP.
- PDF 2D drawing for dimensions, tolerances, GD&T, surface finish and notes.
- Quantity and required delivery timing.
- Any inspection-document requirements such as FAI, CMM report or certificates.

FINAL REVIEW

A clear drawing does not need to be crowded. It needs to make design intent, manufacturing requirements and inspection criteria unambiguous.



EXPANDED ILLUSTRATED SECTION

Correct vs. Incorrect Drawing Examples

The following English engineering diagrams are redrawn from the teaching logic of the supplied source PPT. They are not screenshots; they are clean 6CNC-style technical illustrations intended for customer and engineering use.

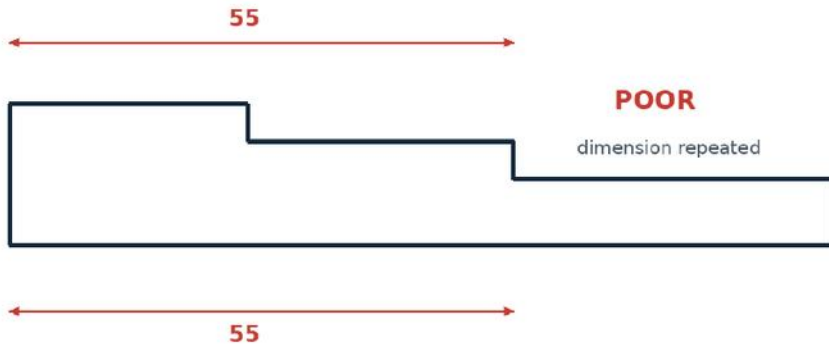
Each example explains why a drawing practice creates manufacturing or inspection risk and how to improve it.

EXAMPLE 01

Missing & Duplicate Dimensions

The source identifies incomplete, missing and repeated dimensions as a common drawing problem.

Missing & Duplicate Dimensions



Recreated English engineering example based on the corresponding source-PPT teaching case.

Recommended Practice

State each required size once, on the clearest view. Critical dimensions should be explicit rather than inferred.

6CNC Review Point

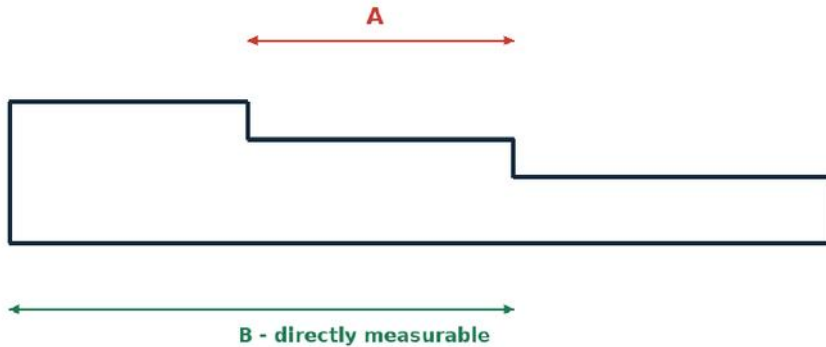
During RFQ review, confirm that the drawing communicates the functional requirement directly and can be interpreted consistently by machining, quality and the customer.

EXAMPLE 02

Dimensions Should Be Easy to Measure

The source compares a difficult internal dimension with a dimension referenced from an accessible face.

Dimensions Should Be Easy to Measure



Recreated English engineering example based on the corresponding source-PPT teaching case.

Recommended Practice

Choose dimensions that can be verified directly with practical inspection methods.

6CNC Review Point

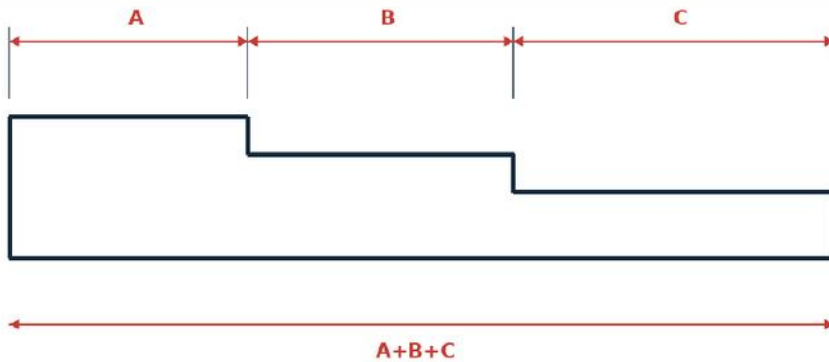
During RFQ review, confirm that the drawing communicates the functional requirement directly and can be interpreted consistently by machining, quality and the customer.

EXAMPLE 03

Avoid Closed Dimension Chains

The source explicitly warns against closed dimension chains.

Avoid Closed Dimension Chains



Recreated English engineering example based on the corresponding source-PPT teaching case.

Recommended Practice

Leave a non-critical dimension as the open link so tolerances do not create conflicting requirements.

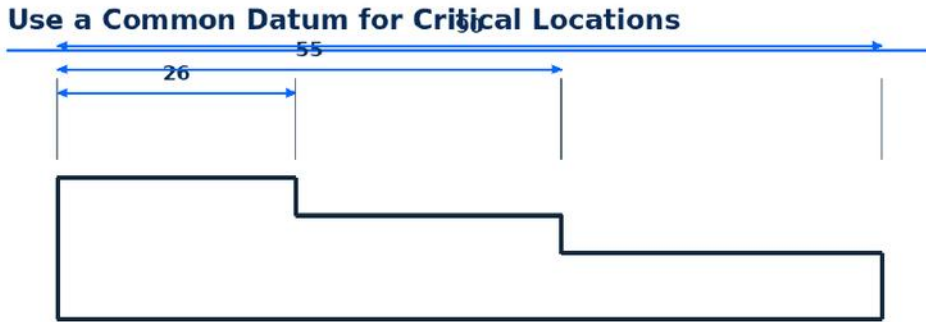
6CNC Review Point

During RFQ review, confirm that the drawing communicates the functional requirement directly and can be interpreted consistently by machining, quality and the customer.

EXAMPLE 04

Use a Common Datum for Critical Locations

The source explains coordinate/baseline dimensioning from one reference.



GOOD - key locations reference one datum.

Recreated English engineering example based on the corresponding source-PPT teaching case.

Recommended Practice

A common datum prevents one feature error from automatically propagating into the next feature location.

6CNC Review Point

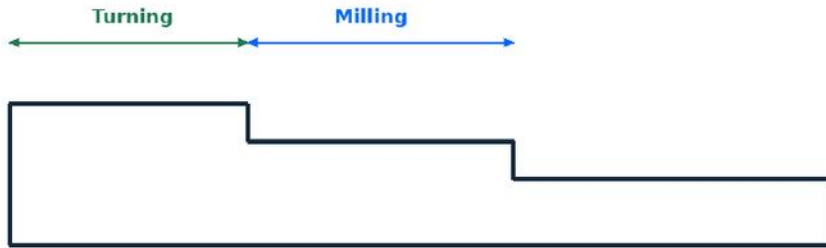
During RFQ review, confirm that the drawing communicates the functional requirement directly and can be interpreted consistently by machining, quality and the customer.

EXAMPLE 05

Group Dimensions for Manufacturing

The source recommends dimensioning for machining and grouping dimensions made by the same process.

Group Dimensions for Manufacturing



Recreated English engineering example based on the corresponding source-PPT teaching case.

Recommended Practice

Organize turning, milling and other feature dimensions around the references used to create them.

6CNC Review Point

During RFQ review, confirm that the drawing communicates the functional requirement directly and can be interpreted consistently by machining, quality and the customer.

EXAMPLE 06

45° vs. Non-45° Chamfers

The source states that a 45° chamfer can use C notation, while non-45° chamfers should state length and angle.

45° vs. Non-45° Chamfers



Recreated English engineering example based on the corresponding source-PPT teaching case.

Recommended Practice

Use an unambiguous chamfer callout that directly describes the geometry.

6CNC Review Point

During RFQ review, confirm that the drawing communicates the functional requirement directly and can be interpreted consistently by machining, quality and the customer.

EXAMPLE 07

Common Hole Callouts

The source contains a dedicated set of common hole-dimensioning examples.

Common Hole Callouts



Recreated English engineering example based on the corresponding source-PPT teaching case.

Recommended Practice

Define through/blind condition, depth, counterbore or countersink geometry as applicable.

6CNC Review Point

During RFQ review, confirm that the drawing communicates the functional requirement directly and can be interpreted consistently by machining, quality and the customer.



EXAMPLE 08

Machined vs. Non-Machined Surfaces

The source shows a cast-part example where too many links between machined and non-machined surfaces are unreasonable.

Machined vs. Non-Machined Surfaces



Keep machined-to-cast links to the minimum needed.

Recreated English engineering example based on the corresponding source-PPT teaching case.

Recommended Practice

Keep only the dimensional relationships necessary to locate the machined surfaces relative to the blank.

6CNC Review Point

During RFQ review, confirm that the drawing communicates the functional requirement directly and can be interpreted consistently by machining, quality and the customer.

EXAMPLE 09

Tolerance Zones & Fits

The source defines basic size, deviations, tolerance zones and clearance/interference/transition fits.

Tolerance Zones & Fits



Recreated English engineering example based on the corresponding source-PPT teaching case.

Recommended Practice

Use tolerances to express acceptable variation and fits to express the intended hole-shaft relationship.

6CNC Review Point

During RFQ review, confirm that the drawing communicates the functional requirement directly and can be interpreted consistently by machining, quality and the customer.

EXAMPLE 10

Surface Roughness Selection

The source maps Ra ranges to common machining processes and surface applications.

Surface Roughness Selection



Recreated English engineering example based on the corresponding source-PPT teaching case.

Recommended Practice

Specify roughness only where function requires it; finer finish usually means additional process control.

6CNC Review Point

During RFQ review, confirm that the drawing communicates the functional requirement directly and can be interpreted consistently by machining, quality and the customer.

EXAMPLE 11

GD&T Feature-Control Frames

The source explains form, orientation, location and runout tolerances plus datum references.

GD&T Feature-Control Frames



Recreated English engineering example based on the corresponding source-PPT teaching case.

Recommended Practice

Use GD&T to control geometric function such as position, perpendicularity or runout.

6CNC Review Point

During RFQ review, confirm that the drawing communicates the functional requirement directly and can be interpreted consistently by machining, quality and the customer.



APPLICATION NOTE 01

Typical Part Classes: Shafts and Disks

The source analyzes shaft/disc dimensioning with the rotational axis as the radial datum and an end face as an axial datum.

6CNC Application

For turned parts, organize diameters around the spindle axis and axial locations around a functional end face. Avoid unnecessary axial chains when a shoulder location is critical.

Buyer / Engineer Check

Could a machinist manufacture the feature and could an inspector verify it without asking what the designer intended? If not, revise the drawing before release.



APPLICATION NOTE 02

Design Datum vs. Process Datum

The source distinguishes design datums from process datums used for setup, machining and inspection.

6CNC Application

Where practical, align the main drawing datum with both function and manufacturing. When they differ, keep the design requirement clear and plan the process to reproduce it.

Buyer / Engineer Check

Could a machinist manufacture the feature and could an inspector verify it without asking what the designer intended? If not, revise the drawing before release.



APPLICATION NOTE 03

Coordinate, Chain & Combined Dimensioning

The source compares coordinate, chain and combined schemes and notes that chain dimensions accumulate error.

6CNC Application

Use coordinate dimensions for critical feature locations, chain dimensions where sequential relationships are intentional, and combined schemes when both needs exist.

Buyer / Engineer Check

Could a machinist manufacture the feature and could an inspector verify it without asking what the designer intended? If not, revise the drawing before release.



APPLICATION NOTE 04

Tolerance Callout Reading

The source explains upper/lower deviations, symmetric $\pm$ tolerance and fit notation.

6CNC Application

Keep decimal precision consistent, align deviations clearly, and distinguish dimensional tolerance from geometric tolerance.

Buyer / Engineer Check

Could a machinist manufacture the feature and could an inspector verify it without asking what the designer intended? If not, revise the drawing before release.



APPLICATION NOTE 05

Drawing Release Checklist

The source closes with drawing-review requirements: drawing rules, technical requirements, consistency between assembly and part drawings, title-block completion and final review.

6CNC Application

Before release, verify geometry, dimensions, tolerances, GD&T, surface finish, material, treatment, revision and inspection requirements.

Buyer / Engineer Check

Could a machinist manufacture the feature and could an inspector verify it without asking what the designer intended? If not, revise the drawing before release.