



QUALITY STANDARD / CNC

CNC Machining Tolerance Standard

Default dimensional, geometric, thread, surface-finish and inspection requirements for CNC machined parts.

DOCUMENT STATUS	RELEASED
DOCUMENT ID	MQS-CNC-TOL-001 / REV A
EFFECTIVE DATE	31 JULY 2026
DEFAULT SYSTEM	Metric units / ISO 2768-mK unless a higher-priority requirement applies
CONTROL RULE	Approved drawing, purchase requirements and signed inspection plan take precedence

Controlled standard: this document defines Mockup default CNC machining tolerances and acceptance requirements when higher-priority project documents are silent.

mockuptech.com | app.mockuptech.com

01 / DOCUMENT CONTROL

Purpose, scope and requirement hierarchy.

This standard defines default acceptance criteria when project documents are silent. It does not replace a controlled engineering drawing, tolerance analysis, approved DFM record or inspection plan.

Priority	Requirement source	Application
1	Approved customer drawing and specifications	Controls dimensions, GD&T, datum scheme, fits, threads, surface finish and inspection requirements.
2	Purchase order, quotation and approved DFM record	Controls project-specific exceptions, capability risks and agreed deviations.
3	Approved inspection plan or signed sample	Controls inspection method, sampling and acceptance where traceable to the active revision.
4	This Mockup tolerance standard	Applies only to dimensions or characteristics not otherwise controlled.

IN SCOPE	REQUIRES PROJECT-SPECIFIC REVIEW
CNC milling and turning Metals and rigid engineering plastics Linear and angular dimensions General geometric tolerances Threads, holes, shafts and surface finish	Thin-wall or flexible parts Heat-treated or stress-relieved parts Parts longer than 1000 mm Optical, sealing and bearing interfaces Tolerances below the precision band Temperature-controlled metrology

Interpretation rule. Dimensions apply at 20 C unless otherwise specified. Units, decimal places and CAD model precision do not create an acceptance tolerance unless a controlled requirement says so. Basic and reference dimensions are not inspected as bilateral size tolerances.

02 / DEFAULT DIMENSIONAL TOLERANCES

Use one clear default when the drawing is silent.

ISO 2768-1 class m applies to general linear and angular dimensions, with ISO 2768-2 class K for general geometric tolerances. Dimensions below 0.5 mm require an explicit tolerance.

General linear dimensions - ISO 2768-m reference

Nominal length range (mm)	Default tolerance (mm)
0.5 to 3	+/- 0.10
over 3 to 6	+/- 0.10
over 6 to 30	+/- 0.20
over 30 to 120	+/- 0.30
over 120 to 400	+/- 0.50
over 400 to 1000	+/- 0.80
over 1000 to 2000	+/- 1.20
over 2000 to 4000	+/- 2.00

External radii and chamfer heights - ISO 2768-m reference

Nominal size range (mm)	Default tolerance (mm)
0.5 to 3	+/- 0.20
over 3 to 6	+/- 0.50
over 6	+/- 1.00

Angular dimensions - ISO 2768-m reference

Shorter side length (mm)	Default angular tolerance
up to 10	+/- 1 degree
over 10 to 50	+/- 0 degree 30 minutes
over 50 to 120	+/- 0 degree 20 minutes
over 120 to 400	+/- 0 degree 10 minutes
over 400	+/- 0 degree 5 minutes

03 / DRAWING-SPECIFIED CAPABILITY

Separate default, precision and special features.

Capability depends on feature size, material, geometry, setup, thermal condition, finishing and measurement access. The values below are feature-specific review bands, not blanket whole-part promises.

Capability band	Indicative feature tolerance	Control requirement
Standard	+/- 0.10 mm or ISO 2768-m, whichever is applicable	Routine machining and standard inspection.
Precision	+/- 0.025 mm	Requires controlled drawing callout, stable datum strategy and suitable inspection method.
High precision	+/- 0.010 mm	Engineering review, process capability plan and enhanced inspection required.
Selected critical feature	down to +/- 0.005 mm	Only after written feasibility approval; may require grinding, honing, lapping or climate-controlled metrology.

Feature-specific starting points for review

Feature type	Metals / PEEK / PEI	Other rigid plastics	Notes
Linear dimension	+/- 0.025 mm	+/- 0.05 mm	Subject to length, wall stiffness and datum accessibility.
Drilled hole diameter	+/- 0.025 mm	+/- 0.05 mm	Reaming or boring required for tighter size or form.
Reamed / bored hole	+/- 0.010 mm	project review	Requires gauge or suitable bore measurement.
Turned shaft diameter	+/- 0.010 mm	+/- 0.05 mm	Grinding may be required for smaller tolerance or improved roundness.
Hole position	diametral 0.05 mm	diametral 0.10 mm	Requires defined datum reference frame and accessible verification.

NO BLANKET CLAIM

A tight value on one selected feature does not imply the same tolerance across every feature, setup, axis or finished condition of the part.

04 / GENERAL GEOMETRIC TOLERANCES

Control form and orientation explicitly.

When the drawing invokes ISO 2768-K and does not provide an individual GD&T; callout, the following general geometric limits apply. Datum-dependent location tolerances should be individually specified.

Straightness and flatness - class K reference

Nominal length (mm)	Tolerance (mm)
up to 10	0.05
over 10 to 30	0.10
over 30 to 100	0.20
over 100 to 300	0.40
over 300 to 1000	0.60
over 1000 to 3000	0.80

Perpendicularity - class K reference

Shorter side length (mm)	Tolerance (mm)
up to 100	0.40
over 100 to 300	0.60
over 300 to 1000	0.80
over 1000 to 3000	1.00

Characteristic	Class K default	Application note
Symmetry	0.6 mm up to 300 mm; 0.8 mm over 300 to 1000 mm; 1.0 mm over 1000 to 3000 mm	Use individual position or profile control when assembly function depends on location.
Circular runout	0.2 mm	Applies relative to the appropriate functional datum feature.
Parallelism	Equal to the applicable size tolerance or the straightness/flatness tolerance, whichever is greater	Individually specify when a functional relationship is critical.

APPLICATION RULE

General parallelism follows the applicable ISO rule. Use an explicit drawing callout whenever the functional relationship is critical.

05 / HOLES, SHAFTS, FITS AND THREADS

Specify functional interfaces, not just nominal sizes.

Fit and thread requirements must be communicated on a controlled drawing. A CAD model alone does not define fit class, thread class, inspection gauge or allowance after finishing.

Feature	Minimum drawing information	Default / review rule
Clearance hole	Diameter, depth or through condition, quantity	General tolerance applies unless a fit or fastener standard is called out.
Precision hole	Size tolerance or ISO fit, depth, datum and position	Reaming, boring or honing may be required. Confirm gauge and inspection access.
Shaft / journal	Diameter tolerance, length, datum, runout and finish	Turning may be followed by grinding when size, form and finish are jointly tight.
Press / transition fit	Hole and shaft basis, fit designation and material condition	Evaluate assembly force, coating thickness, temperature and production capability.
Counterbore / countersink	Diameter, depth or included angle and controlled surface	Measure the functional seating condition, not an inferred CAD edge.

Thread requirements

Thread type	Required callout	Inspection expectation
Metric ISO	Nominal diameter x pitch, tolerance class, depth / through	Use appropriate GO / NO-GO gauges when required.
Unified inch	UNC / UNF / UNEF size, class, depth / through	State internal or external class and any plating allowance.
Pipe thread	Standard, taper / parallel form, class and sealing requirement	Confirm gauge type, engagement and leak-test requirement.
Thread insert	Insert specification, installed depth and locking requirement	Inspect thread before and after installation as applicable.

FINISH ALLOWANCE

Threads, precision holes, bearing seats and sealing features must state whether dimensions apply before or after anodizing, plating, coating, heat treatment or other secondary processing.

06 / SURFACE ROUGHNESS AND EDGE CONDITION

Define the surface that function requires.

Surface roughness is not interchangeable with cosmetic appearance. Specify Ra or another required parameter only on the surfaces where it controls sealing, friction, fatigue, wear or appearance.

Roughness level	Process category	Application note
Ra 3.2 um / 125 uin	Standard machined finish	General non-sealing machined surfaces.
Ra 1.6 um / 63 uin	Fine machined finish	Precision interfaces and selected sealing surfaces after review.
Ra 0.8 um / 32 uin	Precision finish	May require optimized cutting, grinding or additional operations.
Ra 0.4 um / 16 uin	Special finish	Typically requires grinding, honing, lapping or polishing and enhanced metrology.

Edge / feature	Default requirement	Exception
Unspecified external edge	Break sharp edges 0.1-0.3 mm	Thin walls, sealing edges, knife edges, optical or customer-visible geometry.
Hole and slot edge	Deburr; no raised material	Press fits, O-ring grooves, fluid paths and threaded leads.
Internal intersection	No loose burr or contamination	Inaccessible passages require an agreed verification method.
Internal corner radius	As defined by model; drawing controls minimum radius where functional	Very small radii may require EDM or special tooling.

MEASUREMENT METHOD

Specify cutoff, evaluation length, direction and measurement location when roughness is critical. Cosmetic comparators or photographs do not replace a numerical roughness requirement.

07 / MATERIAL, SIZE AND PROCESS EFFECTS

Tolerance capability changes with the part.

The same numerical tolerance does not create the same manufacturing risk across different materials, geometries, sizes, setups and secondary processes.

Condition	Common risk	Required review
Engineering plastics	Thermal expansion, moisture absorption, creep and clamping distortion	Conditioning, measurement temperature, support method and time after machining.
Thin walls / long slender features	Deflection during cutting and inspection, residual stress movement	Tool access, stock strategy, stress relief, fixture support and free-state inspection.
Large parts	Thermal gradient, machine travel, datum transfer and cumulative error	Machine capacity, setup count, inspection volume and transport handling.
Heat treatment	Distortion, scale and hardness change	Define machining allowance, intermediate inspection and post-treatment correction.
Anodizing / plating / coating	Dimensional buildup, masking transition and thread/fit change	Define pre- or post-finish dimensions and nominal coating thickness.
Multiple setups / 5-axis work	Datum transfer, reach, tool deflection and cross-face relationships	Functional datum scheme, setup strategy and CMM accessibility.

Temperature and free-state rules

Rule	Standard requirement
Reference temperature	20 C unless the drawing or inspection plan specifies otherwise.
Flexible parts	Inspect in the free state unless a defined restraint condition is specified.
Plastic parts	Allow stabilization after machining and define humidity conditioning when material behavior makes it necessary.
Finished parts	Final acceptance applies after all dimension-changing secondary processes unless the drawing explicitly controls an earlier stage.

PROJECT CONTROLS

Use climate-controlled inspection, stabilization time, restraint fixtures or temperature compensation whenever required by the drawing, inspection plan or approved engineering review.

08 / INSPECTION AND ACCEPTANCE

Match the measurement method to the requirement.

A tolerance is valid only when the feature can be manufactured, accessed, measured and interpreted consistently. The approved inspection plan controls method and sampling.

Measurement resource	Typical use	Key limitation / control
Caliper / micrometer	Accessible linear sizes, diameters and thickness	Operator technique, contact force and feature geometry.
Height gauge / surface plate	Heights, steps and selected relational measurements	Requires stable datum contact and suitable part support.
Pin / plug / thread gauge	Functional hole and thread acceptance	Gauge class, wear and temperature must be controlled.
Optical / vision system	Small profiles, edges and non-contact measurements	Edge detection and surface condition can affect results.
CMM	GD&T, datum relationships, position, profile and complex geometry	Program, alignment, probing strategy, uncertainty and access must be approved.
Surface roughness tester	Ra and specified texture parameters	Cutoff, direction, location and evaluation length must match the requirement.

ACCEPT	REVIEW	REJECT
All controlled characteristics conform using the approved method. Release to the next operation or shipment.	Borderline result, method disagreement, inaccessible feature or uncertainty concern. Hold and escalate with evidence.	Confirmed nonconformance to drawing, approved plan or applicable default standard. Segregate and control.

Minimum evidence for a dimensional review

Evidence	Minimum requirement
Traceability	Part number, revision, order / lot, quantity affected and process stage.
Requirement	Drawing callout, datum references, specification and applicable default.
Result	Measured value, units, characteristic identifier and acceptance limit.
Method	Instrument, resolution, calibration status, fixture and measurement setup.
Disposition request	Re-measure, engineering clarification, use-as-is, rework or remake.