

REFERENCE

GD&T, Tolerances, and Fits: A Primer

General tolerances, the GD&T controls and feature control frame, material condition and bonus tolerance, and how to choose an ISO 286 fit, with achievable bands by process.

A tolerance is a promise about how much a dimension can vary and still work. Too loose and parts do not fit or seal; too tight and the price climbs for no functional gain. Geometric dimensioning and tolerancing (GD&T) and the ISO fit system are the shared language for stating those promises precisely, so a part means the same thing to the designer, the machinist, and the inspector. This primer covers the essentials: general tolerances, the GD&T controls, and how to pick a fit.

The standards behind this are public: ISO 2768 for general tolerances, ISO 286 for limits and fits, and ASME Y14.5 and ISO 1101 for GD&T symbols. Every achievable band we quote below is the same one the FabWorks DFM engine screens against per process.

General tolerances first

Most dimensions on a part do not need an individual tolerance. ISO 2768 assigns a default band by the size of the feature and a chosen class, so the drawing stays clean and only the critical dimensions carry a called-out tolerance.

ISO 2768 class	Character	Typical band on a small feature
Fine (f)	Precision	+/- 0.05 mm (0.002 in)
Medium (m)	General	+/- 0.1 mm (0.004 in)
Coarse (c)	Rough	+/- 0.2 to 0.3 mm (0.008 to 0.012 in)

ISO 2768 class	Character	Typical band on a small feature
Very coarse (v)	As-cut	+/- 0.5 mm (0.020 in) and up

Put a general class in the title block, then tighten only the features that mate, seal, or locate.

Achievable tolerance by process

Tolerance is not free, and what is achievable depends on the process. Asking a molded or cast feature to hold a machined tolerance forces a secondary machining operation.

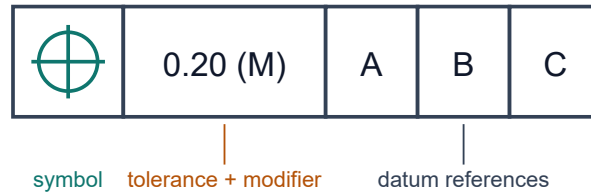
Process	Typical achievable (small feature)
CNC machining	+/- 0.025 to 0.05 mm (0.001 to 0.002 in), tighter with grinding
Sheet metal	+/- 0.2 mm (0.008 in)
Die casting	+/- 0.1 to 0.2 mm (0.004 to 0.008 in) as-cast
Injection molding	resin-dependent, roughly +/- 0.05 to 0.2 mm (0.002 to 0.008 in)
3D printing	+/- 0.1 to 0.3 mm (0.004 to 0.012 in) by technology

The GD&T controls

GD&T states tolerances as zones tied to datums, which controls form and relationship, not just size. The controls fall into groups:

- Form: flatness, straightness, circularity, cylindricity. No datum needed; controls a surface against itself.
- Orientation: perpendicularity, parallelism, angularity. Controls a feature relative to a datum.
- Location: position, concentricity, symmetry. Position is the workhorse: it holds a feature within a zone about its true location.
- Profile: profile of a line or surface. Controls a whole contour within a tolerance band.
- Runout: circular and total runout. Controls a rotating surface relative to an axis.

The feature control frame



Symbol, tolerance with any material-condition modifier, then the datum references, in order.

Each control is written in a feature control frame: the symbol, the tolerance zone with any modifier, and the datums it is measured from, in order. Datums are the reference faces or axes, labelled A, B, C, that fix how the part is held for measurement, so a stated datum scheme is what makes a tolerance repeatable.

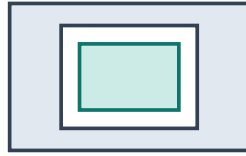
Material condition and bonus tolerance

Position tolerances often carry a material-condition modifier. At maximum material condition (MMC), a feature at its worst-case material size (a hole at its smallest, a pin at its largest) gets the stated tolerance, and any departure from that size adds bonus tolerance. This matches how parts actually assemble and can widen the usable tolerance without loosening the fit, so it lowers cost where it applies.

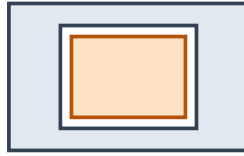
Fits: clearance, transition, interference

When a shaft goes in a hole, the fit is the intended looseness or tightness. ISO 286 defines it with a tolerance grade (the IT number, smaller is tighter) and a letter that positions the band, using a hole-basis system where the hole is the constant.

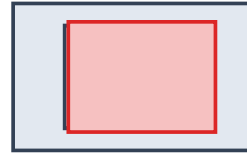
Clearance, transition, interference



clearance
H7/g6: slides



transition
H7/k6: located



interference
H7/p6: pressed

In the ISO 286 hole-basis system the shaft class sets the fit: g6 slides, k6 locates, p6 presses.

Fit	Example (hole basis)	Behavior
Clearance	H7/g6	Always has a gap, slides and rotates freely
Transition	H7/k6	Small gap or slight grip, locates precisely
Interference	H7/p6	Always tight, pressed or shrunk together, transmits load

Pick the fit from the function: a rotating shaft wants clearance, a located dowel wants transition, a permanently seated bushing wants interference. State it as the standard callout and the machinist and inspector both know exactly what to hold.

The checklist

1. A general ISO 2768 class in the title block.
2. Tight tolerances only on mating, sealing, and locating features.
3. Tolerances achievable in the chosen process, or secondary machining planned.
4. Datums defined for any geometric control.
5. Fits called out with the standard hole-basis notation.

How FabWorks checks this for you

Our DFM engine screens each toleranced feature against what the chosen process and material can hold, flags a callout tighter than the process supports, and points to the operation (reaming, grinding, machining a cast face) that would be needed. Each band is grounded in the ISO tolerance standards behind it, so the guidance is traceable and the quote reflects the real cost of the tolerances you asked for.