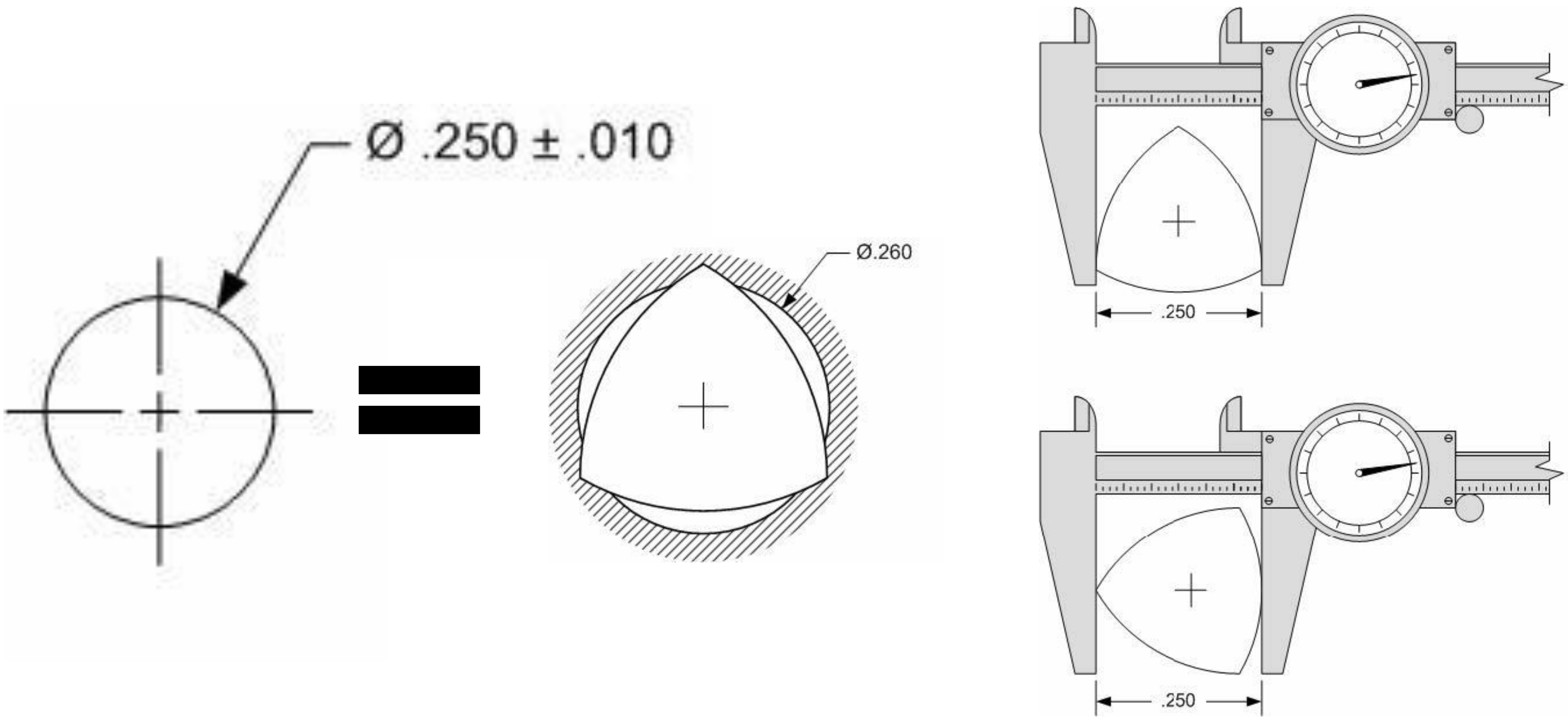
Vise Stop

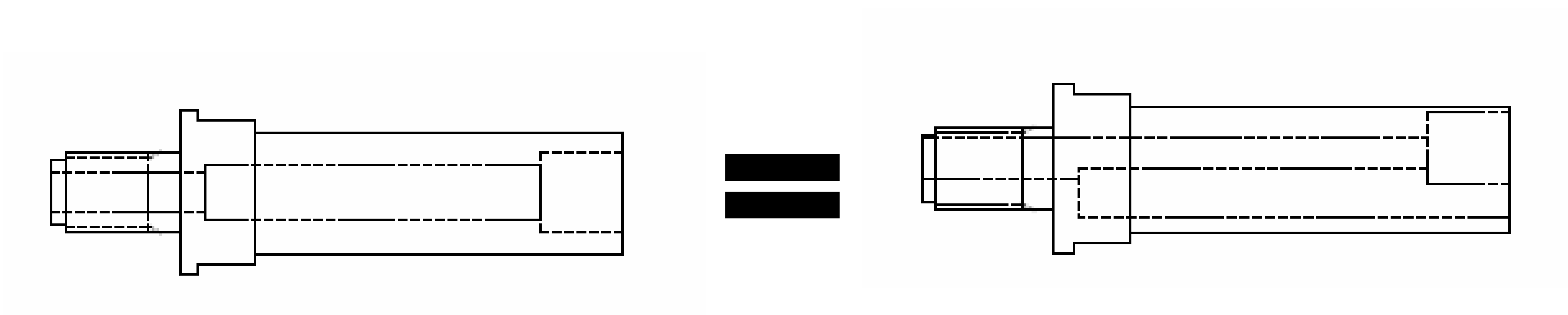
Geometric Dimensioning and Tolerancing

Life Without GD&T

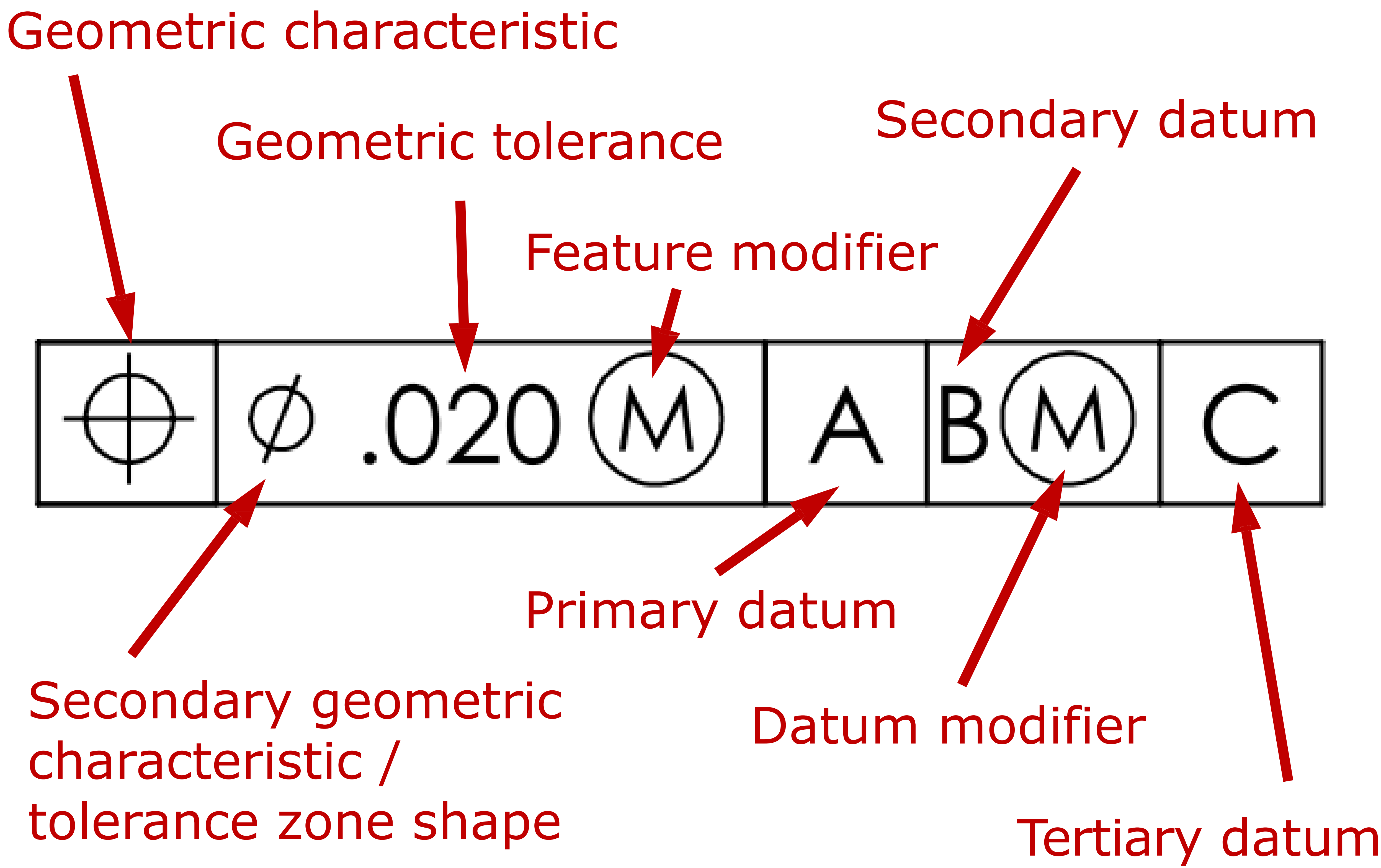


Univ. of Arizona. “What is Geometric Dimensioning & Tolerancing?”

Life Without GD&T



Feature Control Frame



Wabiszewski, Graham. “Engineering Drawings and GD&T.”
MEAM201: Machine Design and Manufacturing

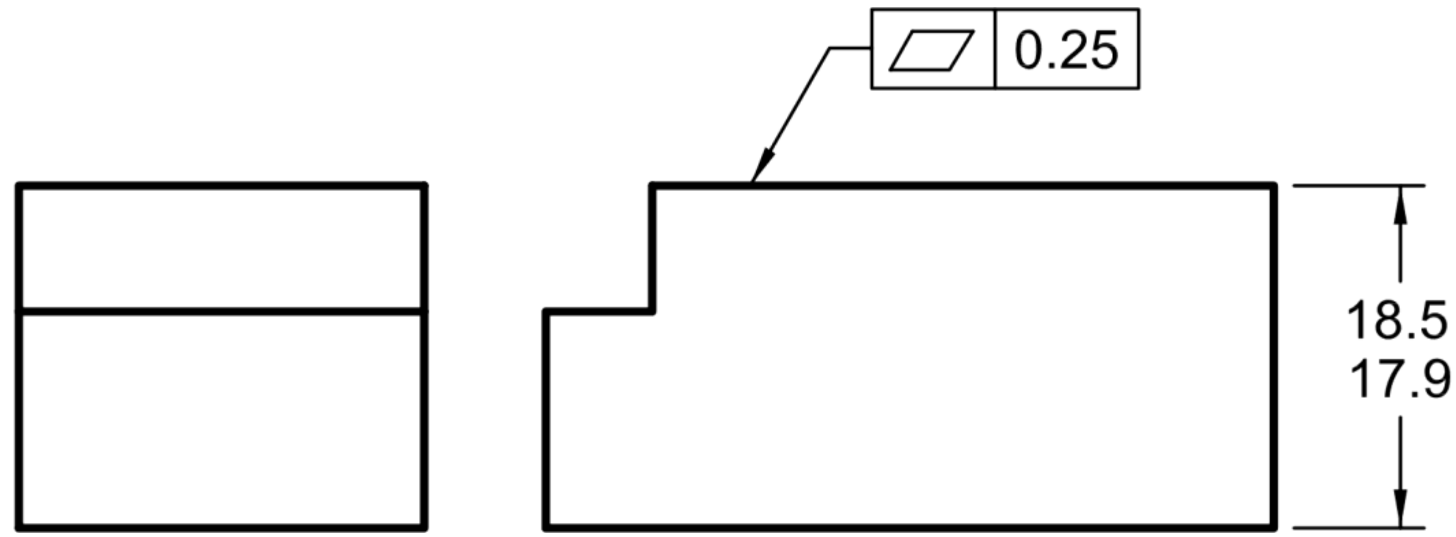
Geometric control symbols

<table>
<tr><th>Type</th><th colspan="2">Geometric Characteristics</th><th>Pertains To</th><th>Basic Dimensions</th><th>Feature Modifier</th><th>Datum Modifier</th></tr>
<tr><td rowspan="4">Form</td><td>⏤</td><td>Straightness</td><td rowspan="4">ONLY individual feature</td><td rowspan="4"></td><td rowspan="6">Modifier not applicable</td><td rowspan="4">NO datum</td></tr>
<tr><td>○</td><td>Circularity</td></tr>
<tr><td>⏥</td><td>Flatness</td></tr>
<tr><td>⌭</td><td>Cylindricity</td></tr>
<tr><td rowspan="2">Profile</td><td>⌒</td><td>Profile (Line)</td><td rowspan="2">Individual or related</td><td rowspan="2">Yes if related</td><td rowspan="6">RFS implied unless MMC or LMC is stated</td></tr>
<tr><td>⌓</td><td>Profile (Surface)</td></tr>
<tr><td rowspan="3">Orientation</td><td>∠</td><td>Angularity</td><td rowspan="6">ALWAYS related fea-ture(s)</td><td>Yes</td><td rowspan="4">RFS implied unless MMC or LMC is stated</td></tr>
<tr><td>⊥</td><td>Perpendicularity</td><td rowspan="2"></td></tr>
<tr><td>//</td><td>Parallelism</td></tr>
<tr><td rowspan="3">Location</td><td>⌖</td><td>Position</td><td>Yes</td></tr>
<tr><td>◎</td><td>Concentricity</td><td rowspan="4"></td><td rowspan="4">Only RFS</td><td rowspan="4">Only RFS</td></tr>
<tr><td>⌯</td><td>Symmetry</td></tr>
<tr><td rowspan="2">Runout</td><td>↗</td><td>Circular Runout</td><td rowspan="2"></td></tr>
<tr><td>⌰</td><td>Total Runout</td></tr>
</table>

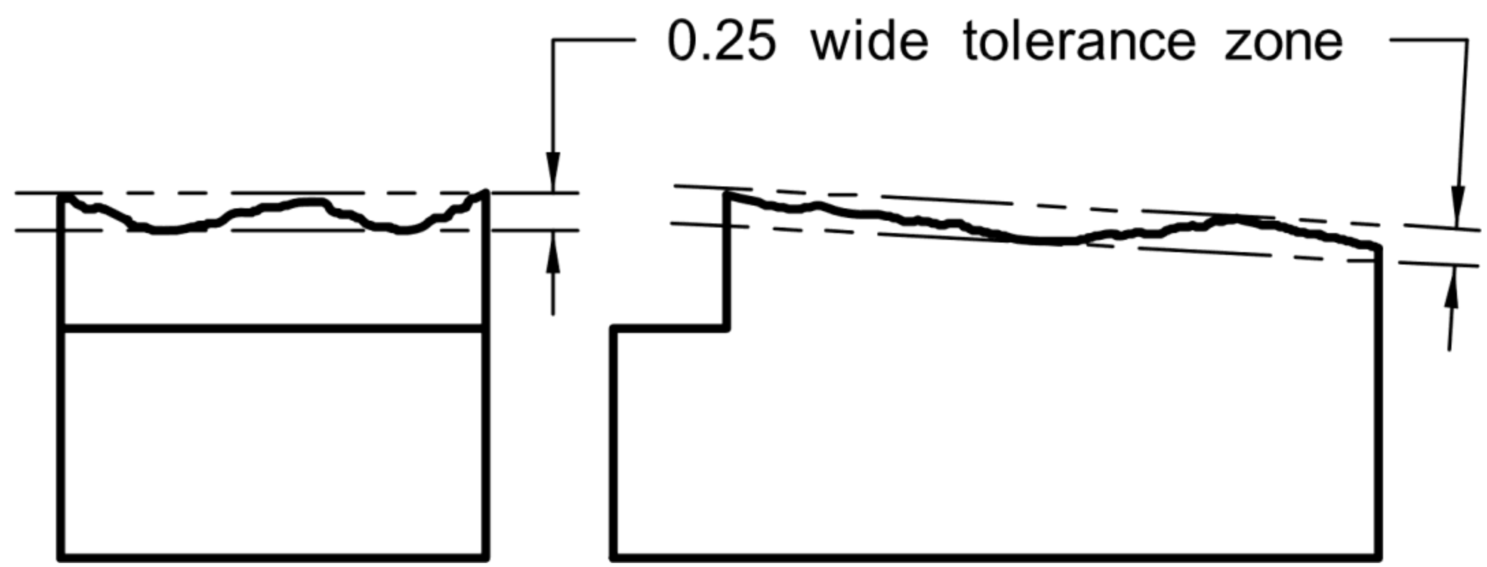
Adapted from Oberg’s Machinery Handbook

Form (flatness, ▱)

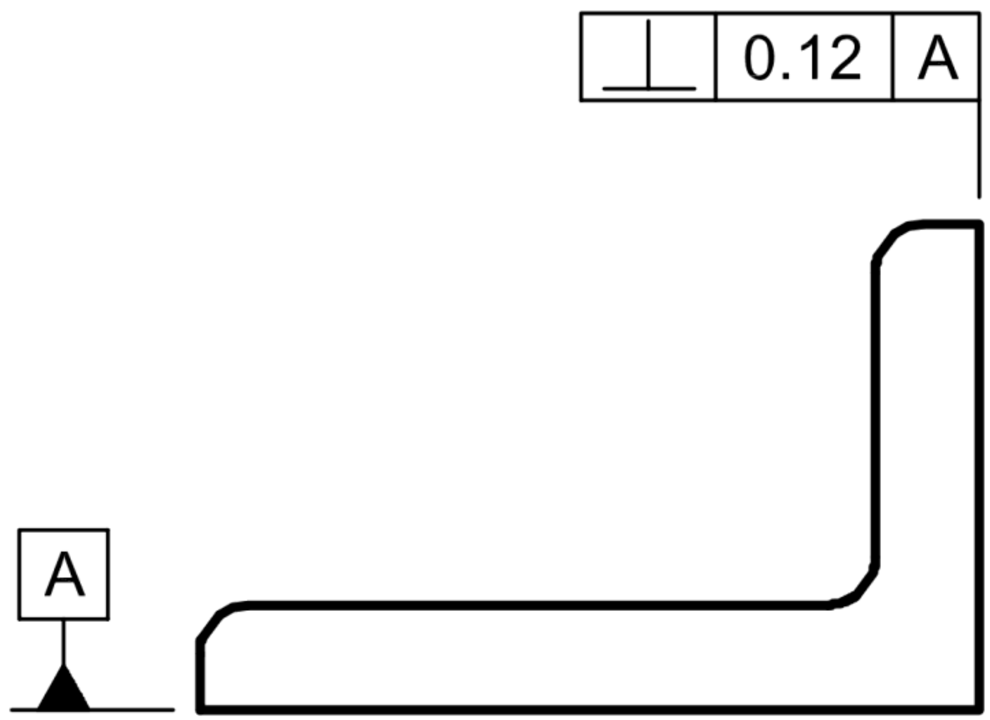
The distance (zone) between two parallel planes that just encompass all points on a surface



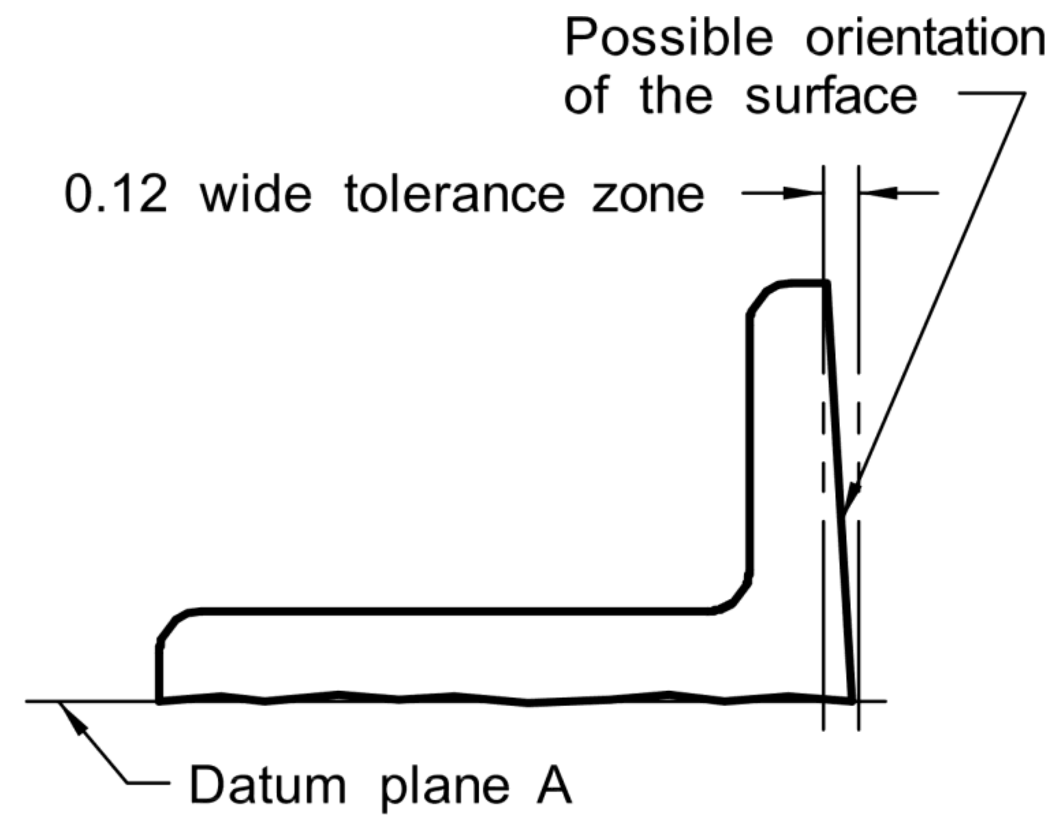
- means -



Orientation (perpendicularity, ⊥)



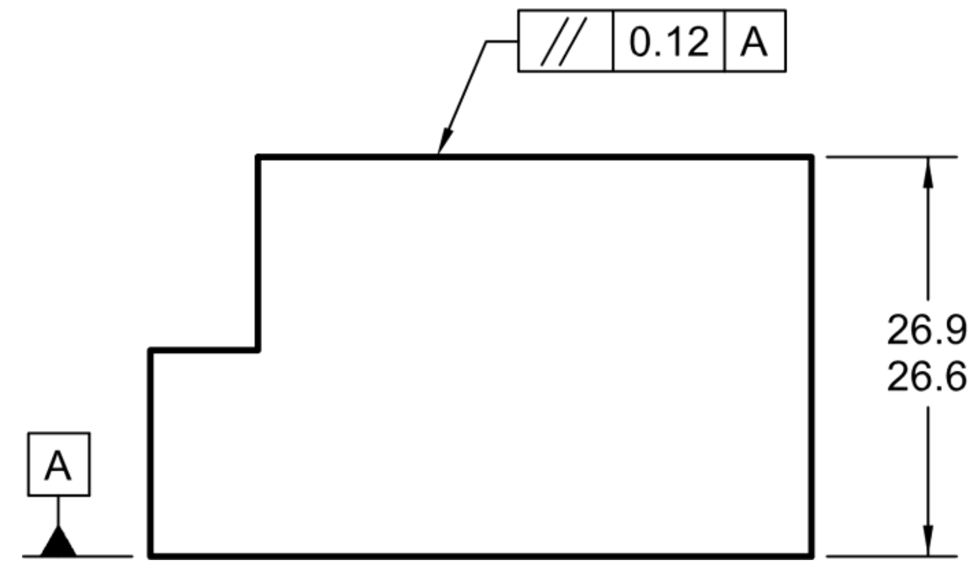
- means -



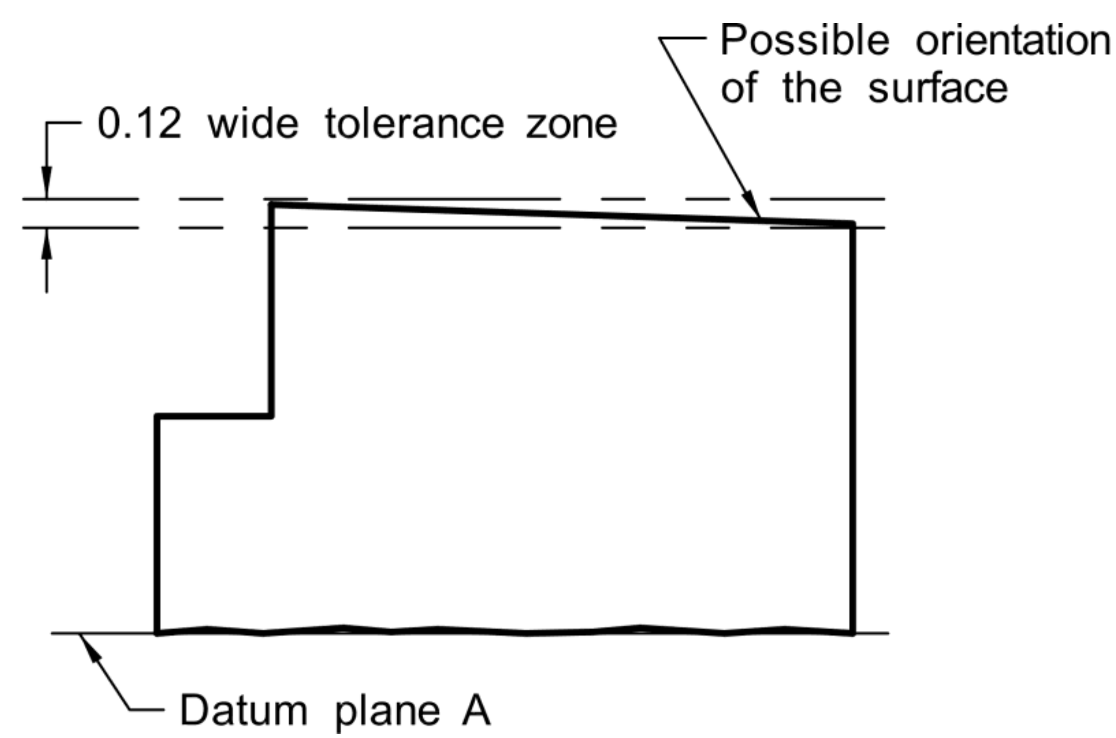
The distance (zone) between two lines parallel to each other and perpendicular to a datum placed such that all points on the evaluated surface <u>just</u> fall between the lines

Orientation (parallelism, //)

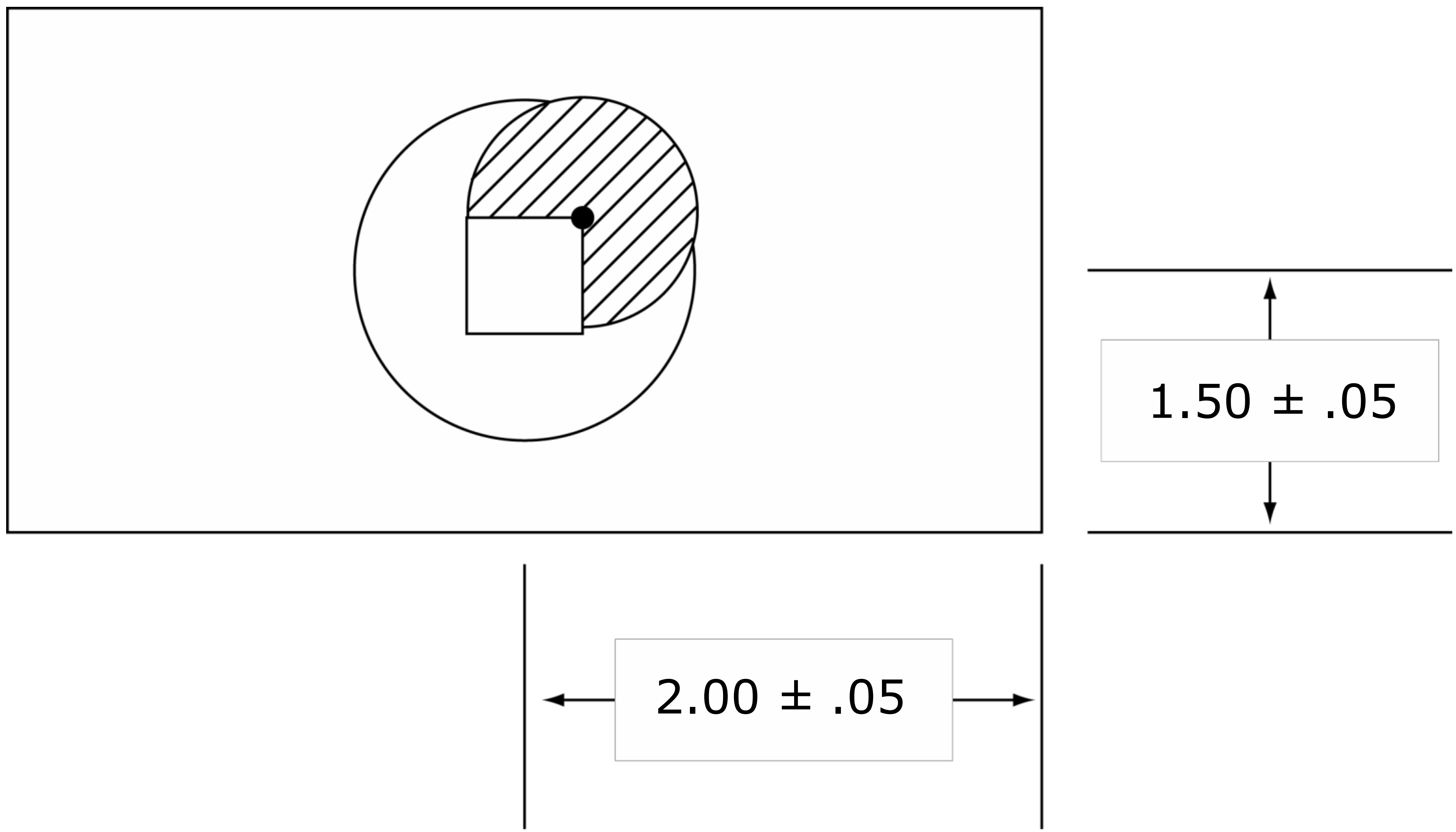
The distance (zone) between two lines parallel to each other and parallel to a datum placed such that all points on the evaluated surface just fall between the lines



- means -

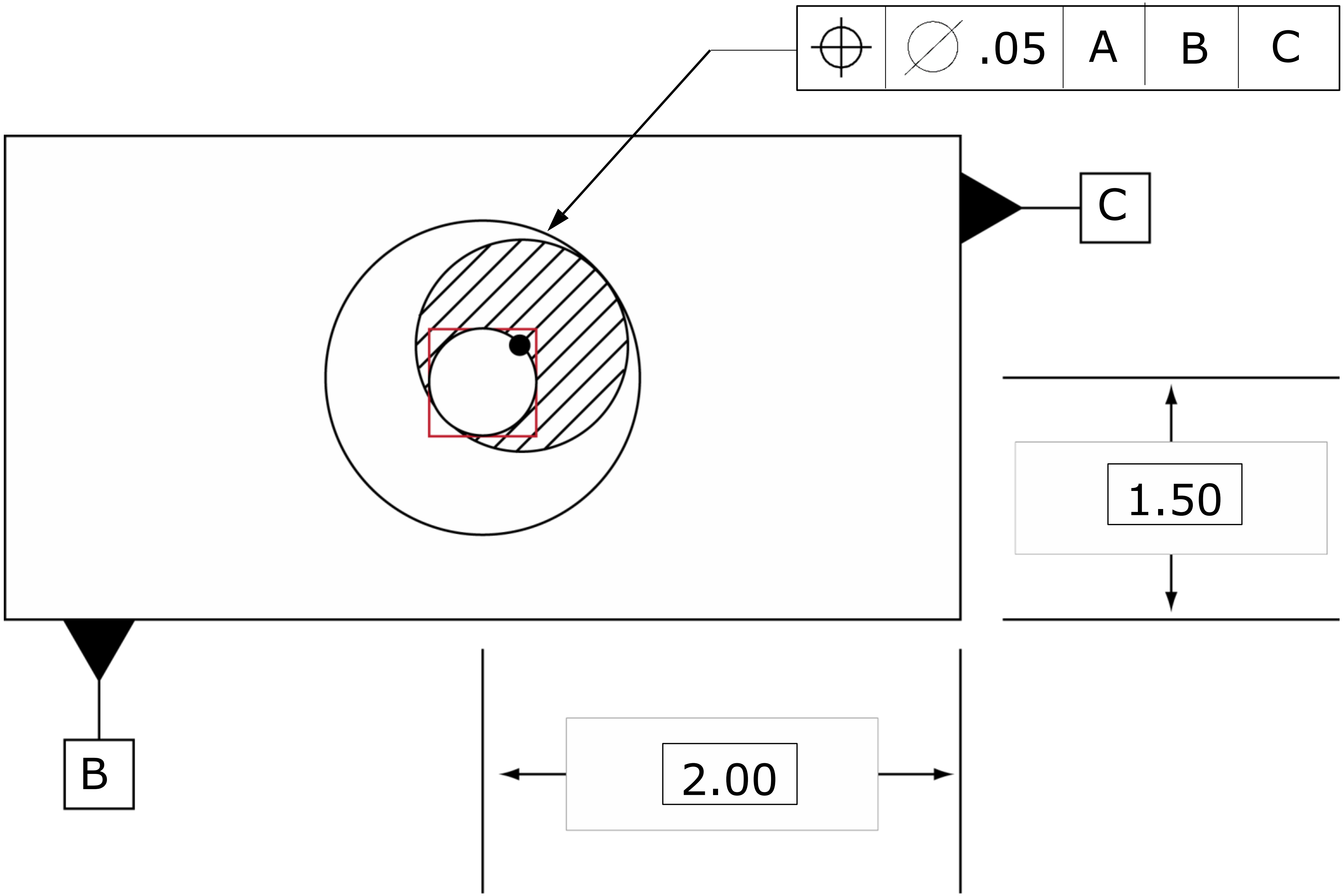


Hole/pin tolerances



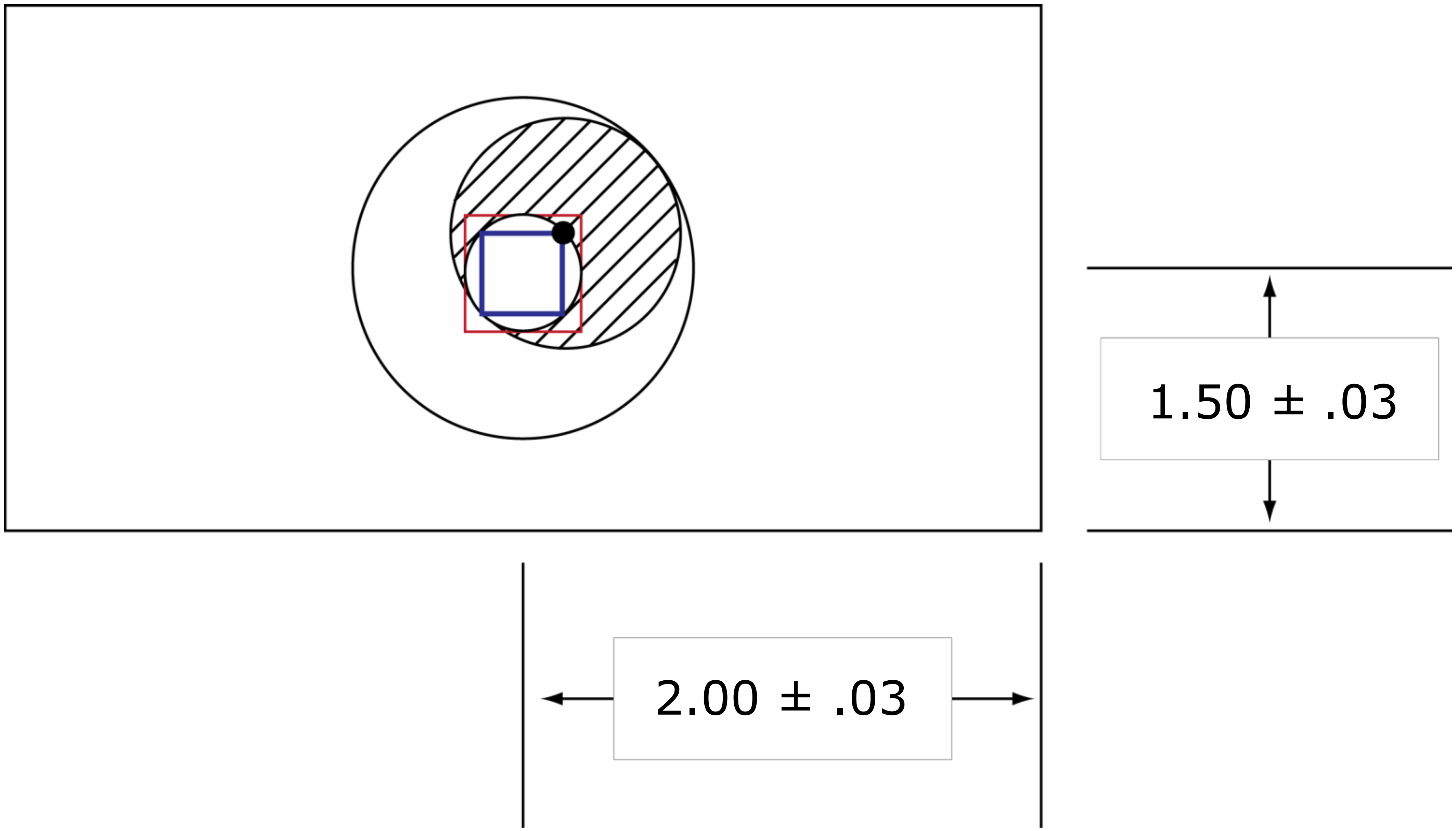
Locational tolerances based on coordinate dimensions infer a rectangular tolerance zone - care should be taken to ensure features do not overlap at their maximum extents (the corner of the tolerance zone)

Hole/pin tolerances



Instead, basic dimensions and a diametrical tolerance zone can be identified using a GD&T control reference frame

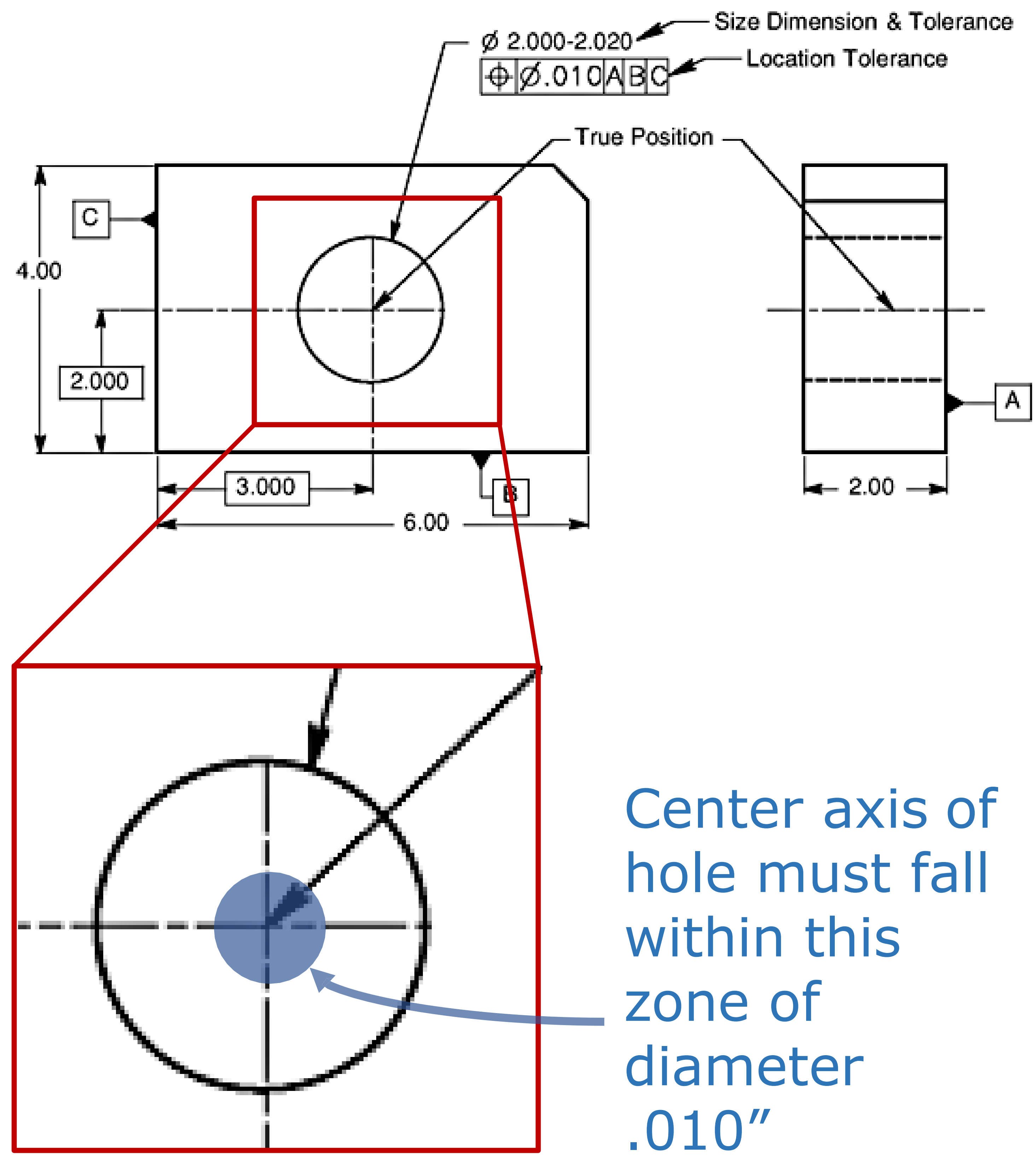
Hole/pin tolerances



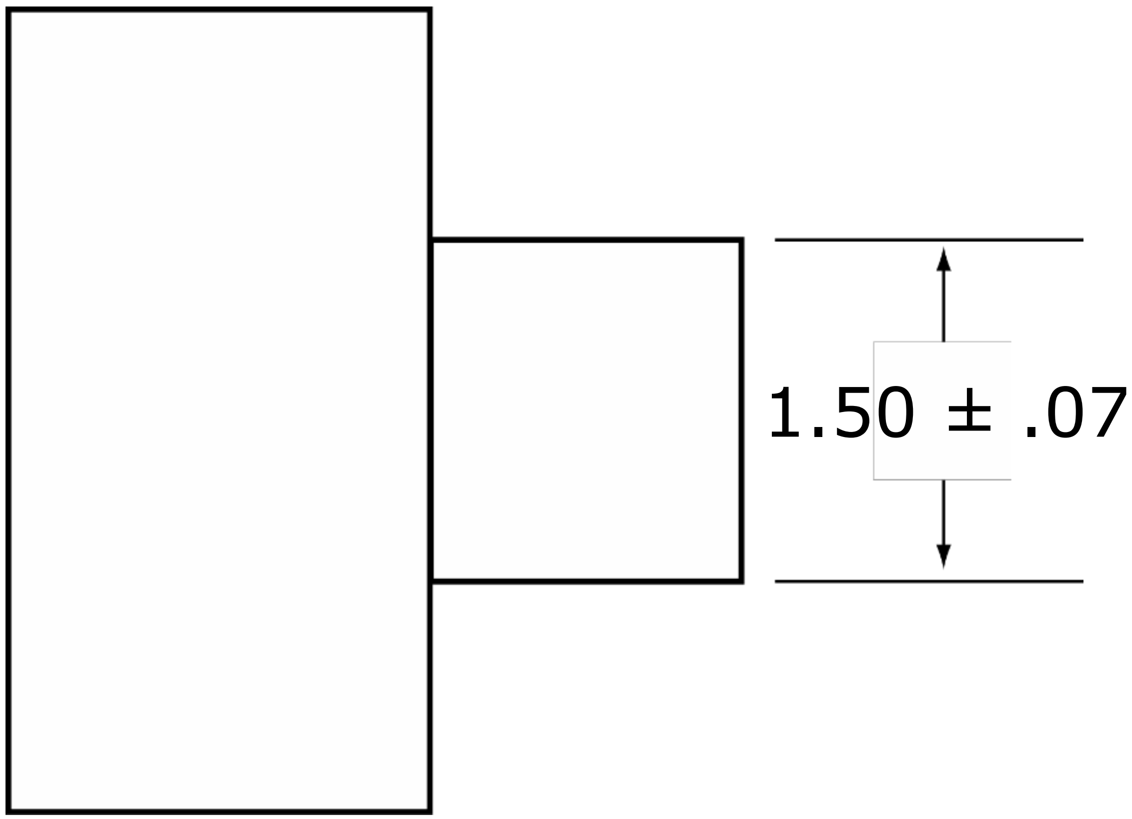
If locational tolerances based on coordinate dimensions are still preferred, a conservative figure of merit may be specified by the extents of a square inscribed in the circular tolerance zone

Location (position, ⌖)

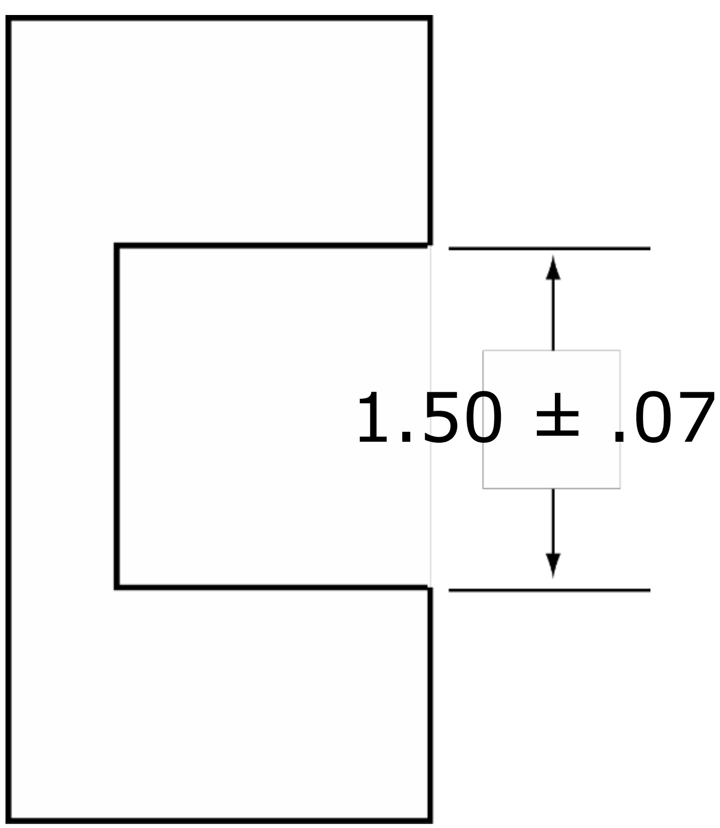
All center points of a cylindrical feature (hole or post) that fall within a cylinder of a prescribed diameter (tolerance) and centered at the true position of the cylindrical feature



MMC and LMC

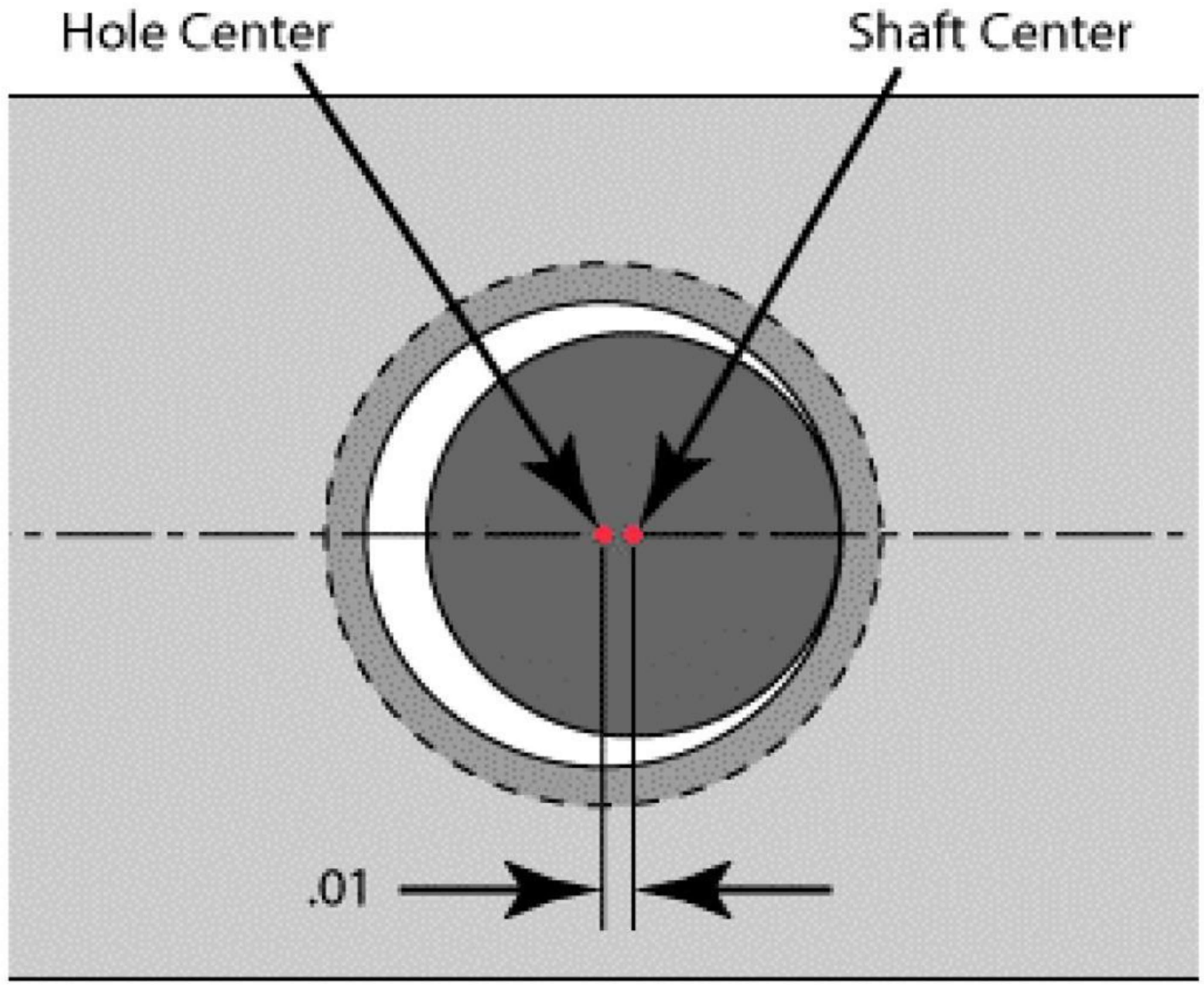


The <u>maximum material condition</u> (MMC, Ⓜ) describes a feature produced with the maximum amount of material – smallest hole size or largest shaft size



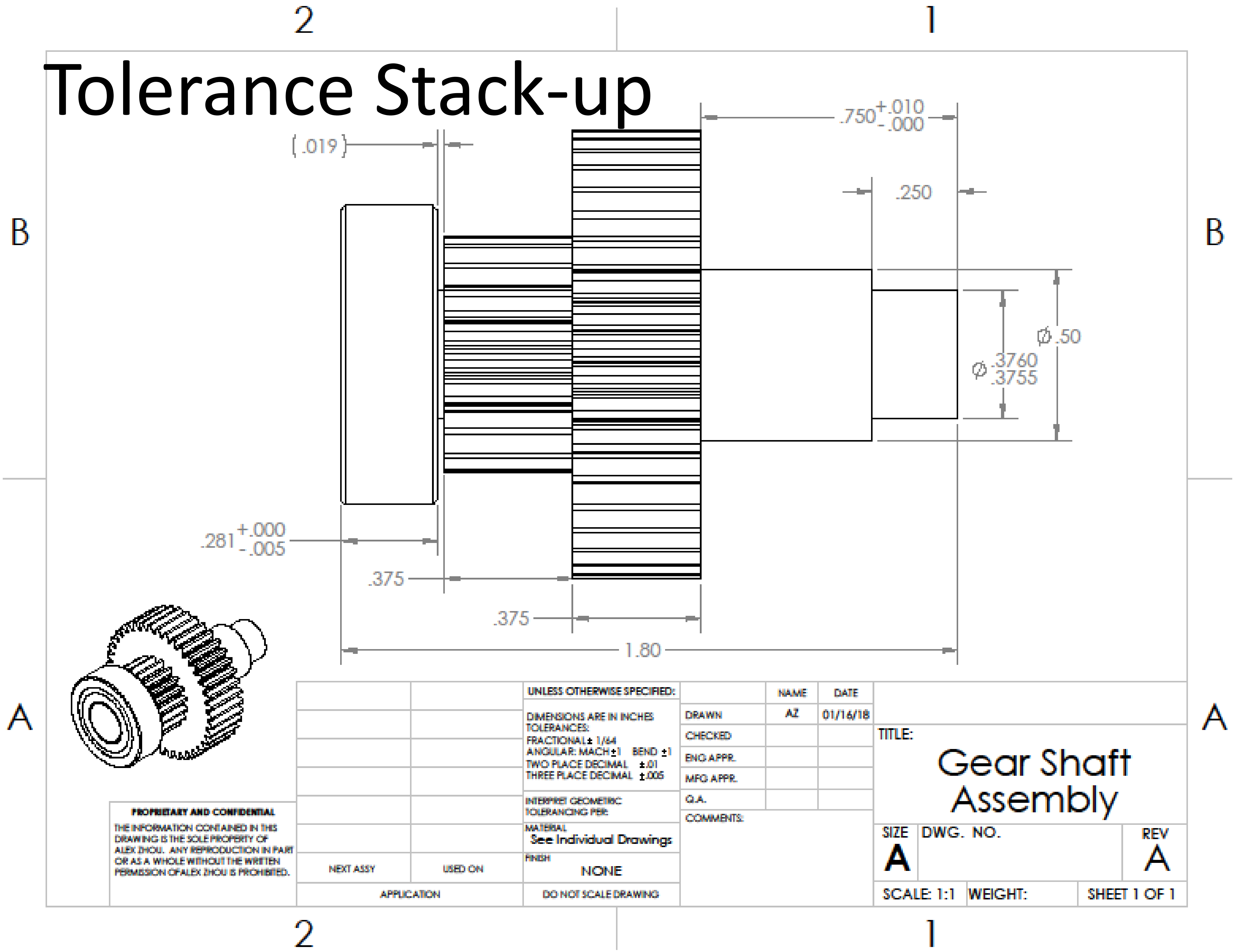
The <u>least material condition</u> (LMC, Ⓛ) describes a feature produced with the minimum amount of material – largest hole size or smallest shaft size

Bonus tolerance

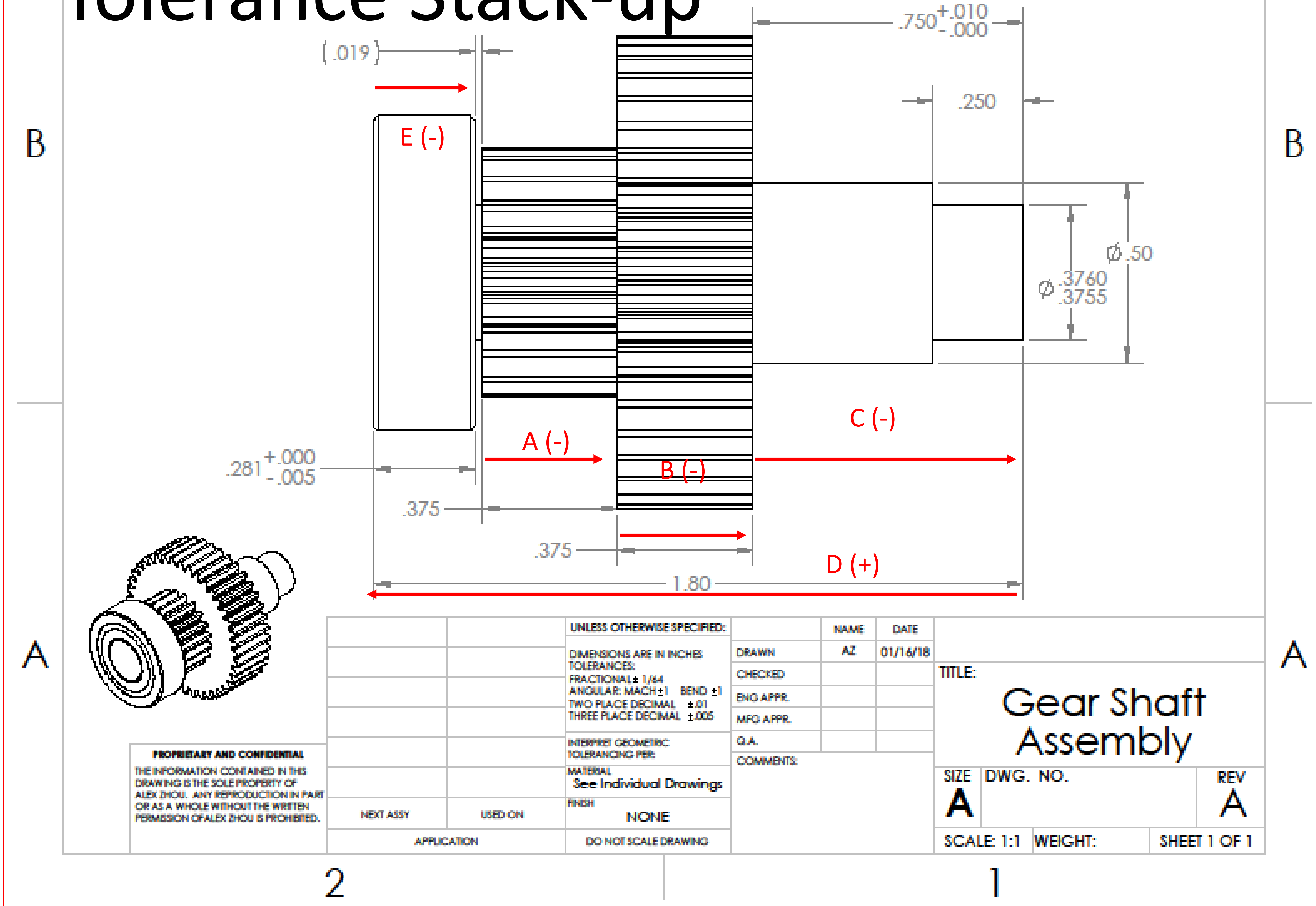


When positional tolerance is based on MMC, diametrical deviations from MMC yield corresponding increases in the diametrical tolerance zone

Tolerance Stack-up



Tolerance Stack-up

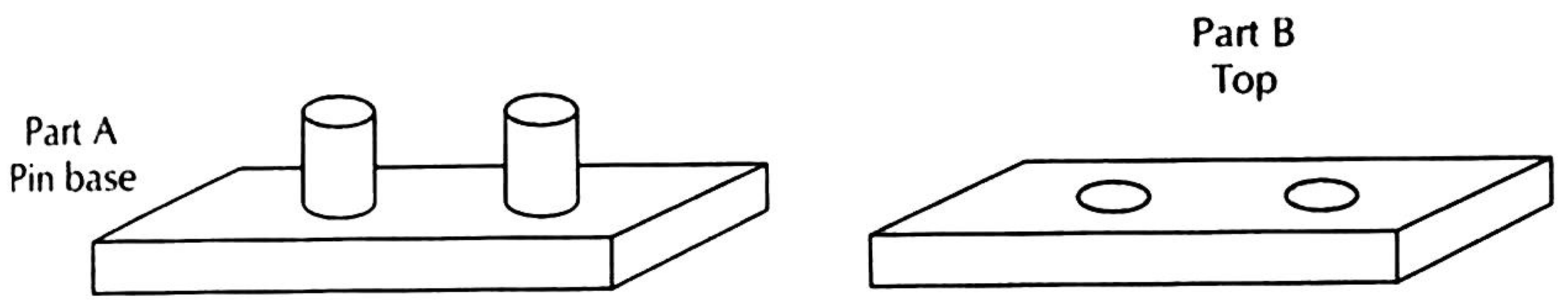


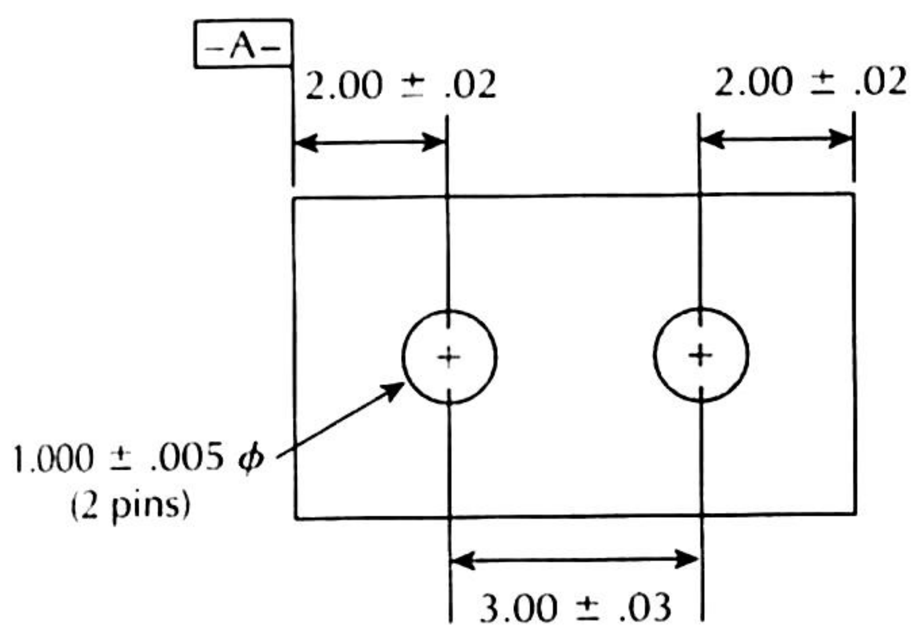
Tolerance Stack-up (cont'd)

Name	Dimension	Tolerance(+)	Tolerance(-)	Direction
A	0.375in	+.005in	-.005in	Negative
B	0.375in	+.005in	-.005in	Negative
C	0.750in	+.010in	-.000in	Negative
D	1.80in	+.01in	-.01in	Positive
E	0.281in	+.000in	-.005in	Negative
Worst Case	0.019in	+.030in	-.025in	
	Worst Case (Max) = +0.049in		Worst Case (Min) = -0.006in	

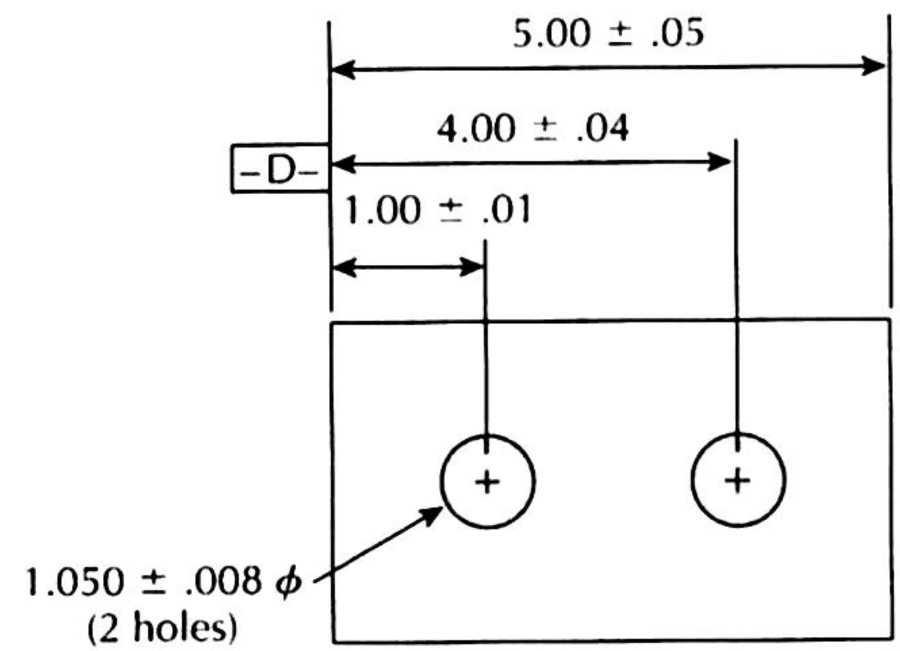
- We calculate that there is a 0.019" nominal gap.
- Worst case largest gap of 0.049".
- Worst case smallest gap of -0.006".
- This worst case scenario results in an interference fit. The parts will not fit and the dimensions/tolerances will need to be changed.

Another Worst Case Example



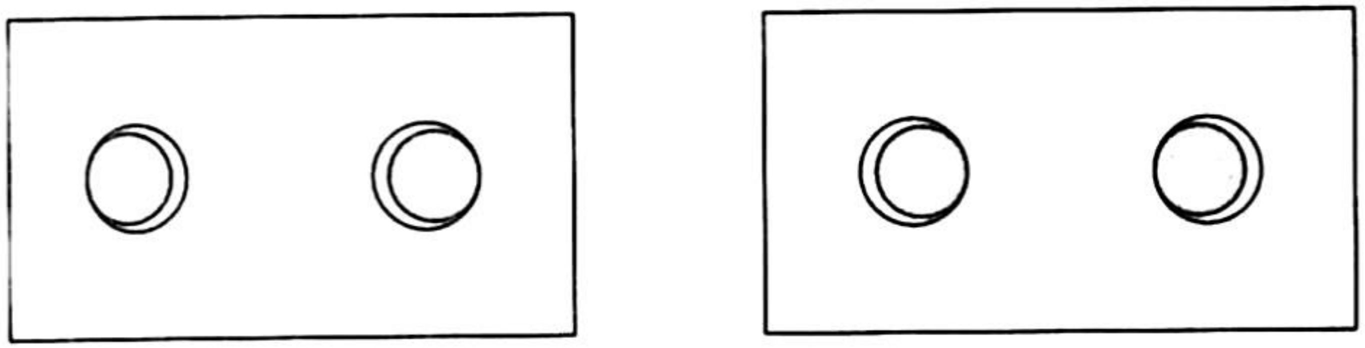


Part A: Pin base



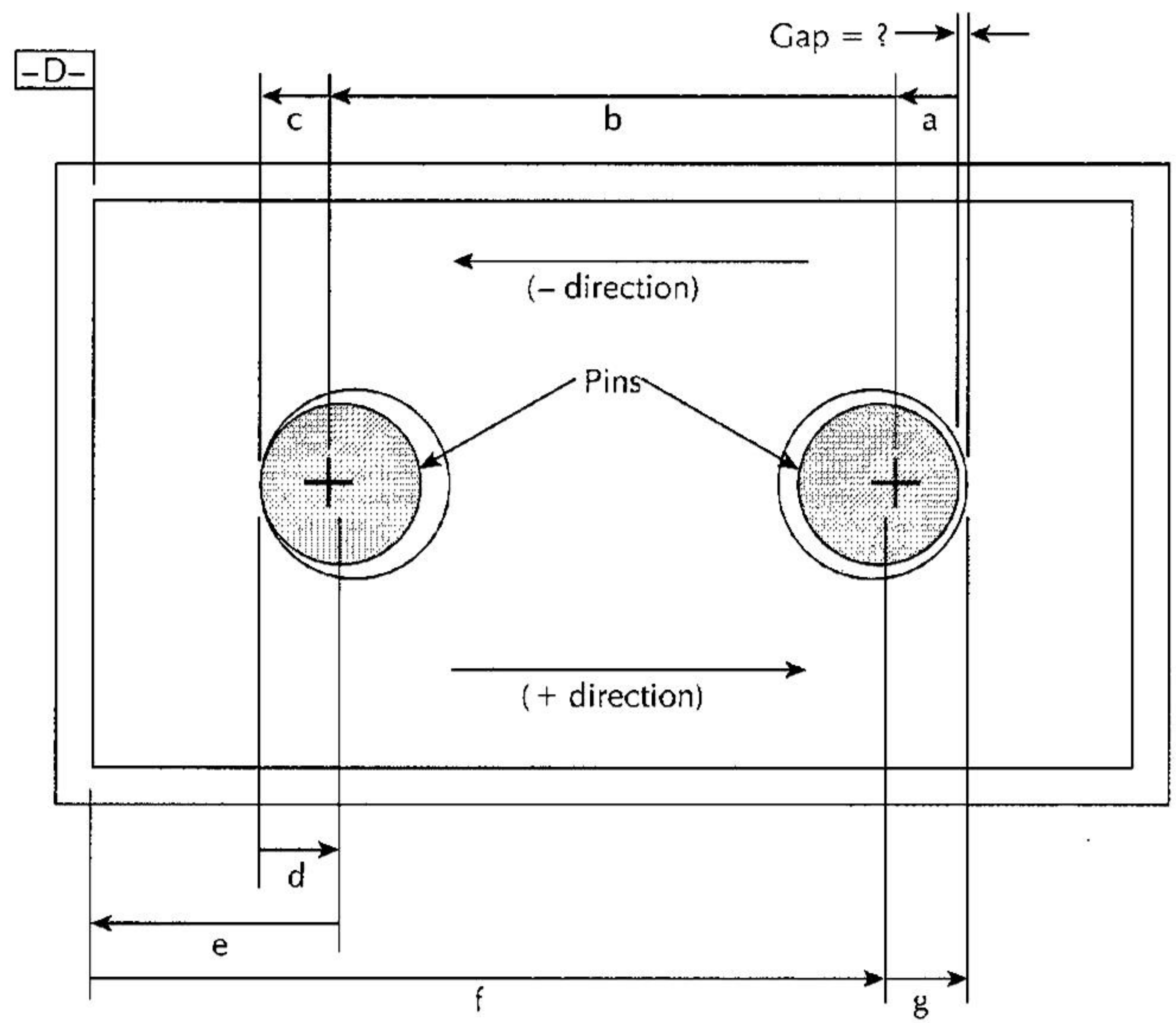
Part B: Top

Source: Tolerance Design, pp 109-111

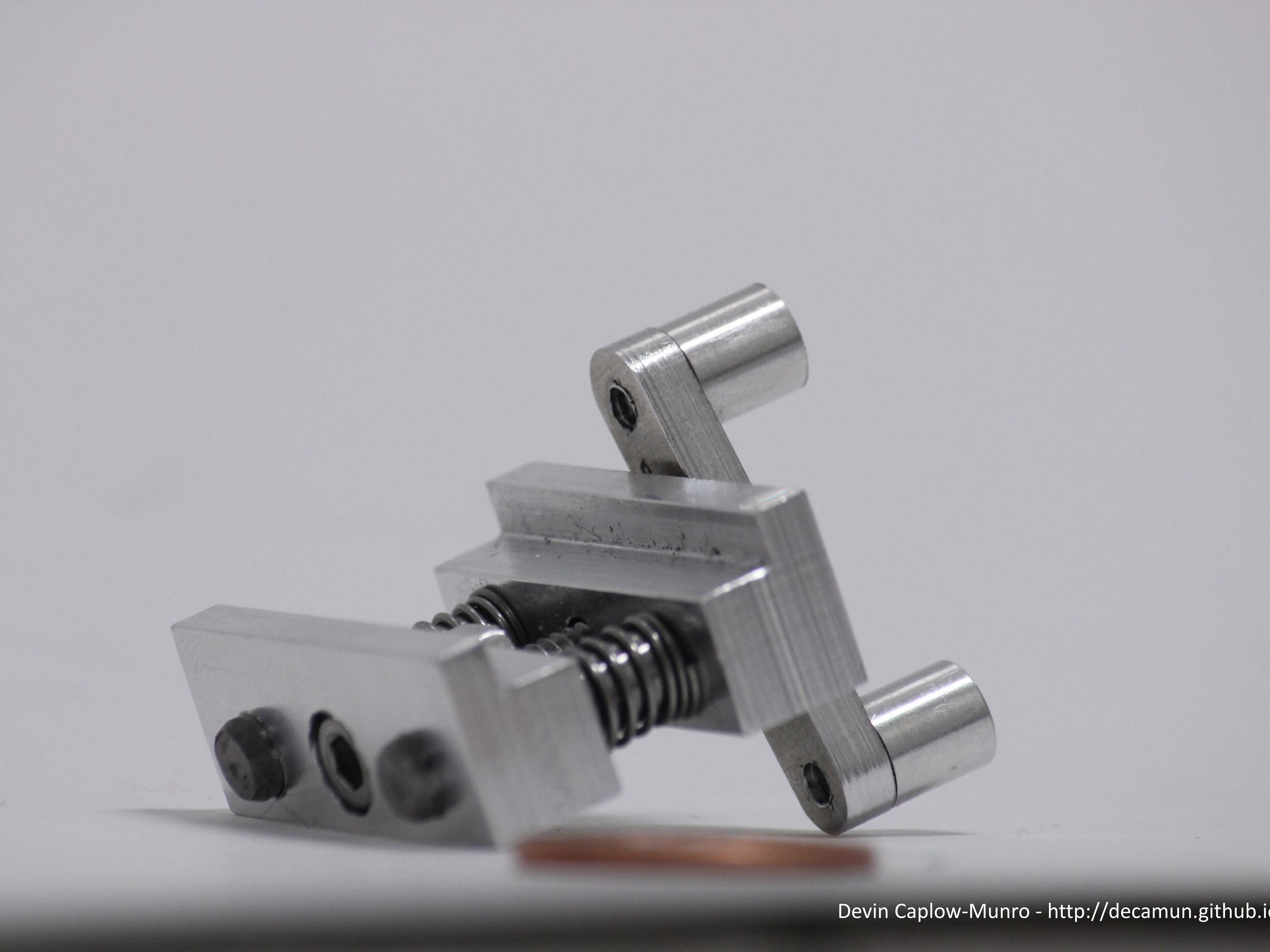


This diagram shows to two worst case scenarios.

This may be useful for a certain project…



Source: Tolerance Design, pp 109-111



Devin Caplow-Munro - http://decamun.github.io