

Principles of Engineering Drawing

Thayer Machine Shop

Drawing and Tolerancing

This Tutorial reviews the following principles:

Drawing: How to interpret and create engineering drawings

Dimensioning: How to communicate dimensions properly

Tolerancing: How to use geometric and dimensional tolerances to specify how much variation is acceptable during manufacture

Why Engineering Drawings?

- Engineering drawing is a formal and precise way of communicating information about the shape, size, features and precision of physical objects.
- Drawing is the universal language of engineering.
- Engineering drawing could be a complete course in itself, but we only have 80 minutes so...

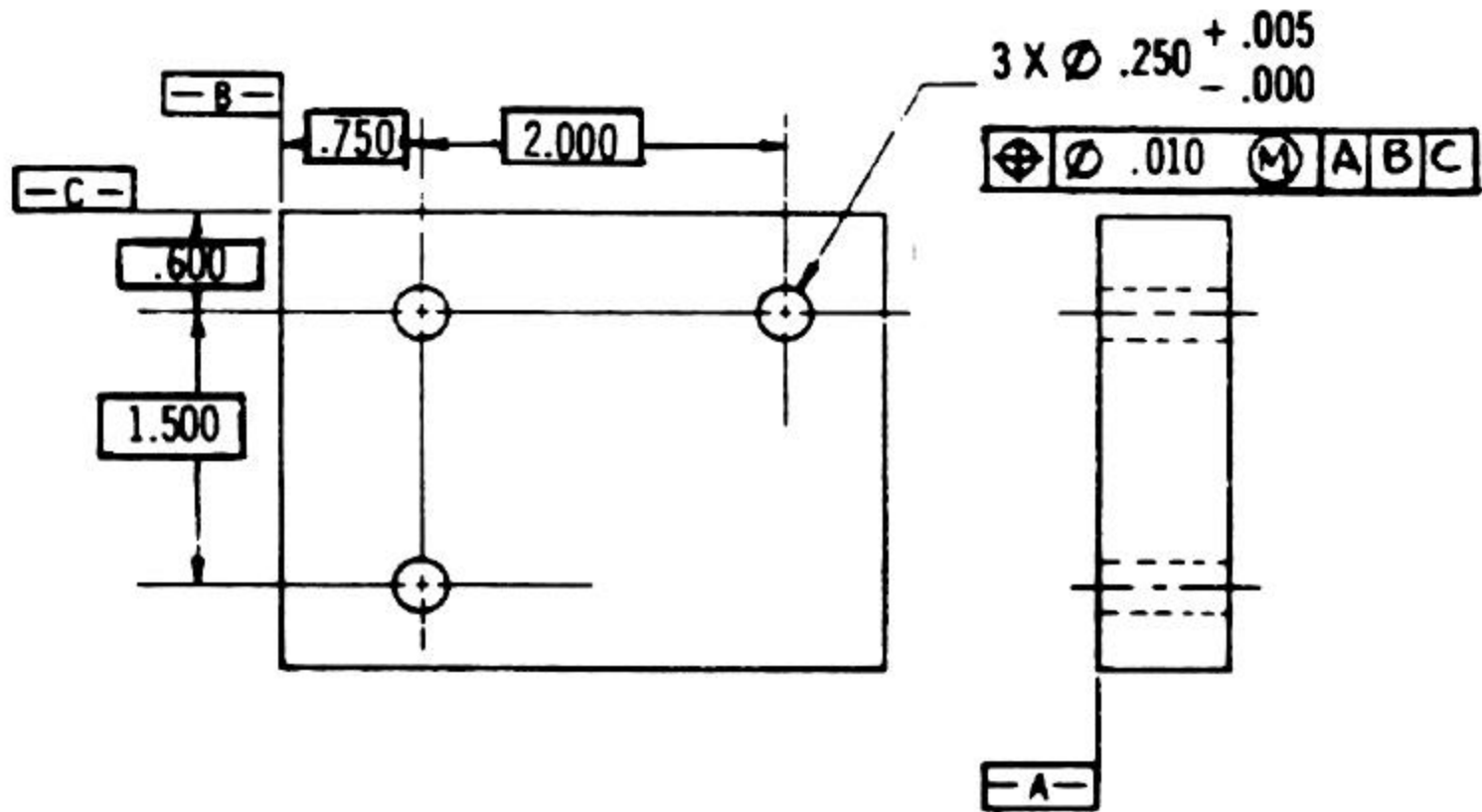
This is only going to cover the very basics.

Drawing Standards

- Just like written language has standards, the “grammar” of technical drawing is defined by...

the **ANSI Y14.5** or the **ISO** standard

- The ANSI standards must be understood to read a drawing.
- Lets look at a sample drawing...

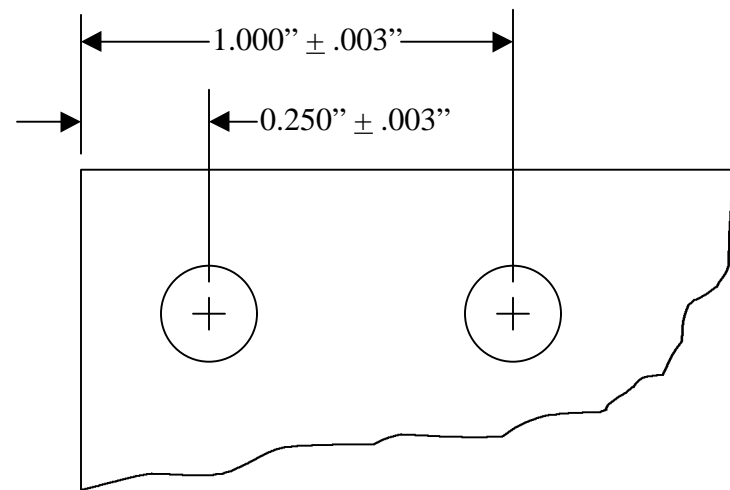
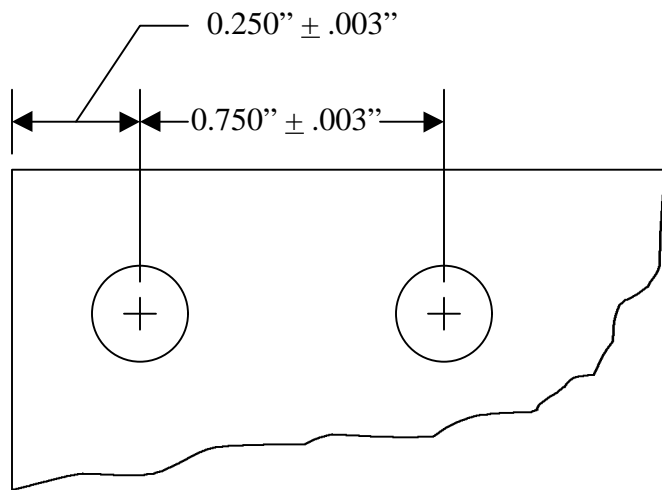


PSD Production System Design Laboratory Laboratory for Manufacturing and Productivity Massachusetts Institute of Technology	PART NAME: Left Mtg. Bracket	
	MATERIAL: AL6601-T6	
NOTES: Units in Inches	DRAWN BY: BC	
Deburr all edges	TOL:	QTY: 1
	REV:	DATE: 6- ⁵ 97

Basic Information Included in a Drawing

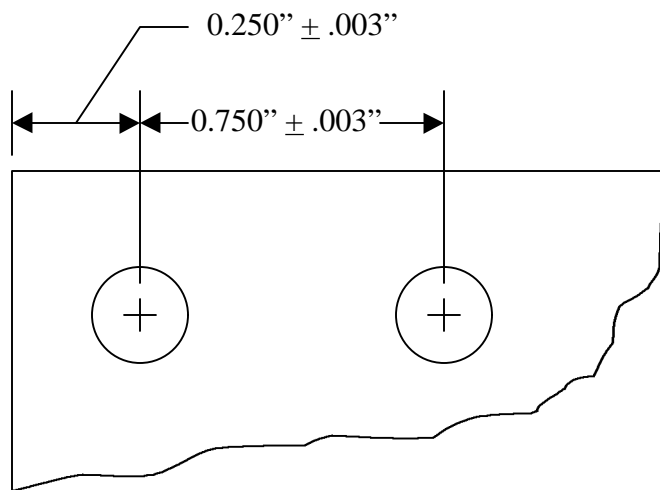
- **Projected Views:** Show as many sides as needed for completeness.
- **Cross Sections:** A view that is good for showing interior features.
- **Table:** Lower right corner, with material information, part name, designer etc.
and finally
- **DIMENSIONS!!!:** These are the most important and most complicated part of the drawing. There is more to it than just the numerical values!

Which is better?

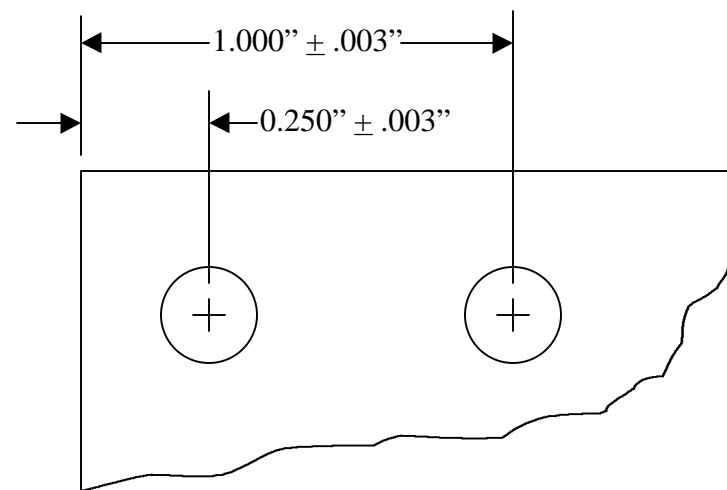


A Dimensioning Example, showing that placement should match intent

These drawings show bolts holes for mounting a flange onto a plate. When mounting the flange, the position of the holes with respect to each other is very important, or else the flange (or part) won't fit. It makes sense to dimension the distance between the holes, instead of the distances to the edge.



Dimension placement
matches intent



Dimension placement
does NOT match intent

Tolerances

(every part needs some)

There are two types of tolerances:

Dimensional Tolerances

and

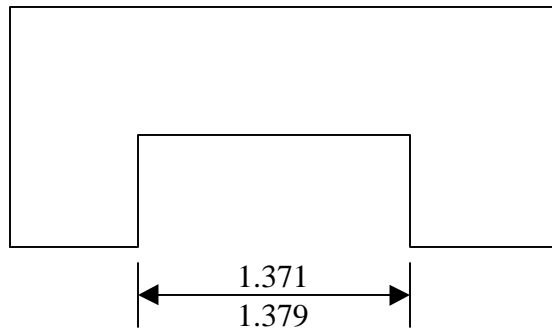
Geometric Tolerances

What's the difference?

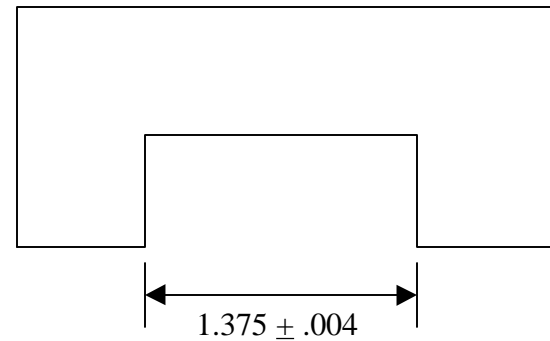
- Dimensional tolerances control _____.
- Geometric tolerances control _____ & _____.

Geometric tolerances affect dimensional tolerances!

Types of Dimensional Tolerances



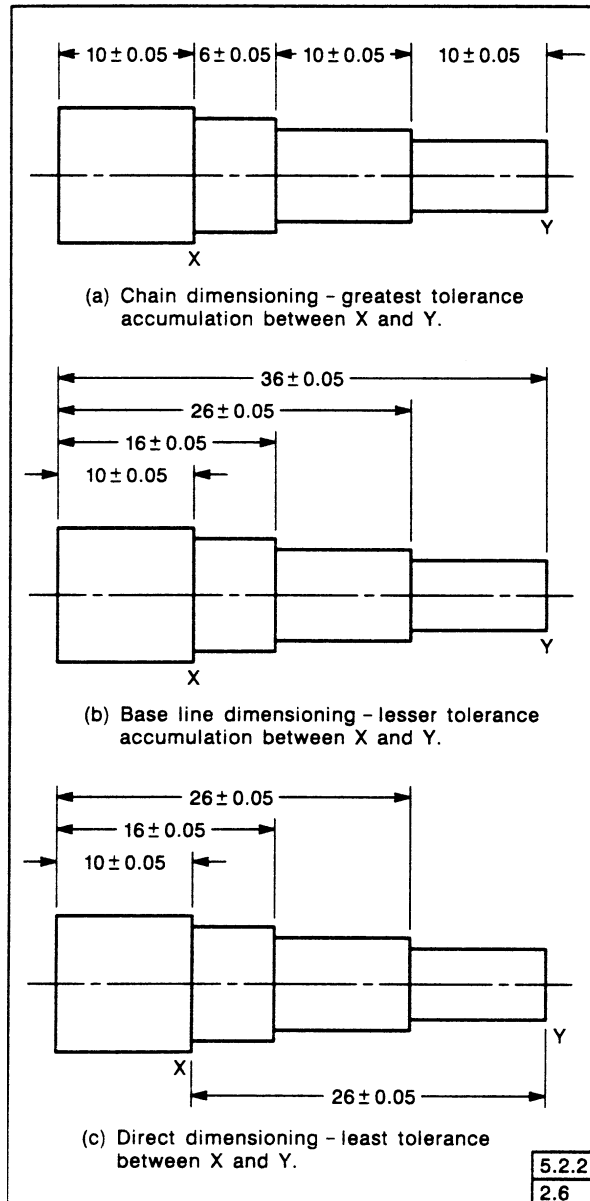
Limit Dimensioning



Plus & Minus Tolerancing

Both methods are acceptable.

Tolerance Accumulation



The distance between X and Y is a *critical dimension*. The total variation in the distance between X and Y depends on how the drawing is dimensioned.

How much tolerance is specified on the distance between X and Y in each example?

(a) \pm

(b) \pm

(c) \pm

Geometric Tolerancing

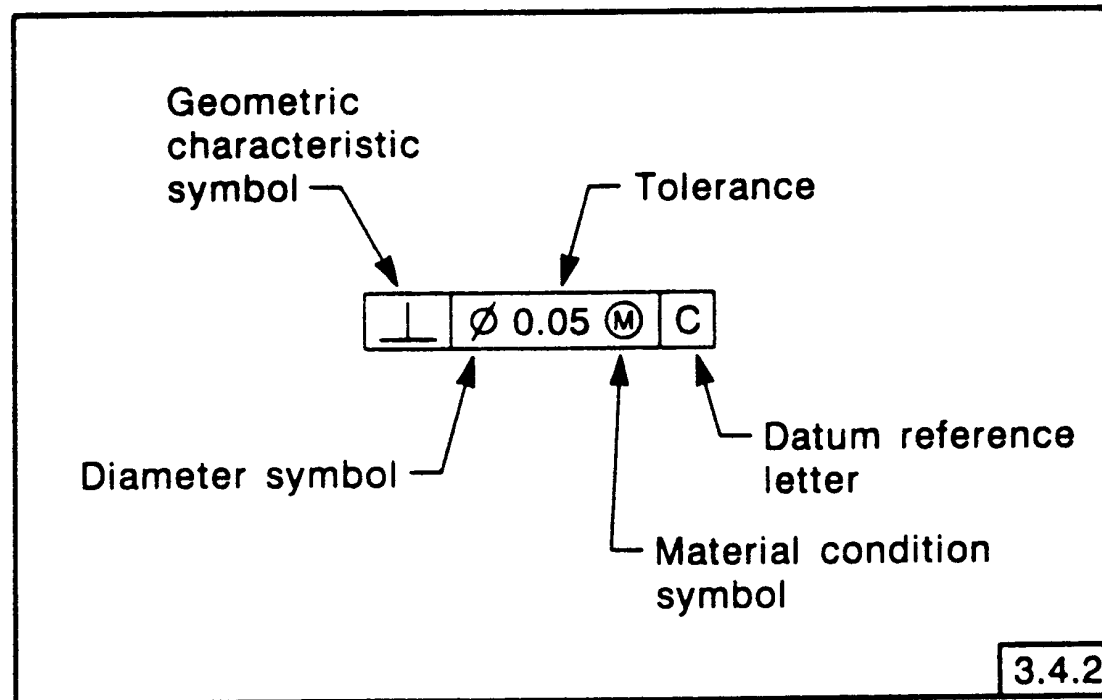
Geometric Tolerancing is used to specify the shape of features.

Things like:

- Straightness
- Flatness
- Circularity
- Cylindricity
- Angularity
- Profiles
- Perpendicularity
- Parallelism
- Concentricity
- And More...

Geometric Tolerances are shown on a drawing with a feature control frame.

The Feature Control Frame



This feature control frame is read as: “The specified feature must lie *perpendicular* within a tolerance zone of *0.05 diameter* at the *maximum material condition*, with respect to *datum axis C*. In other words, this places a limit on the amount of variation in perpendicularity between the feature axis and the datum axis. In a drawing, this feature control frame would accompany dimensional tolerances that control the feature size and position.

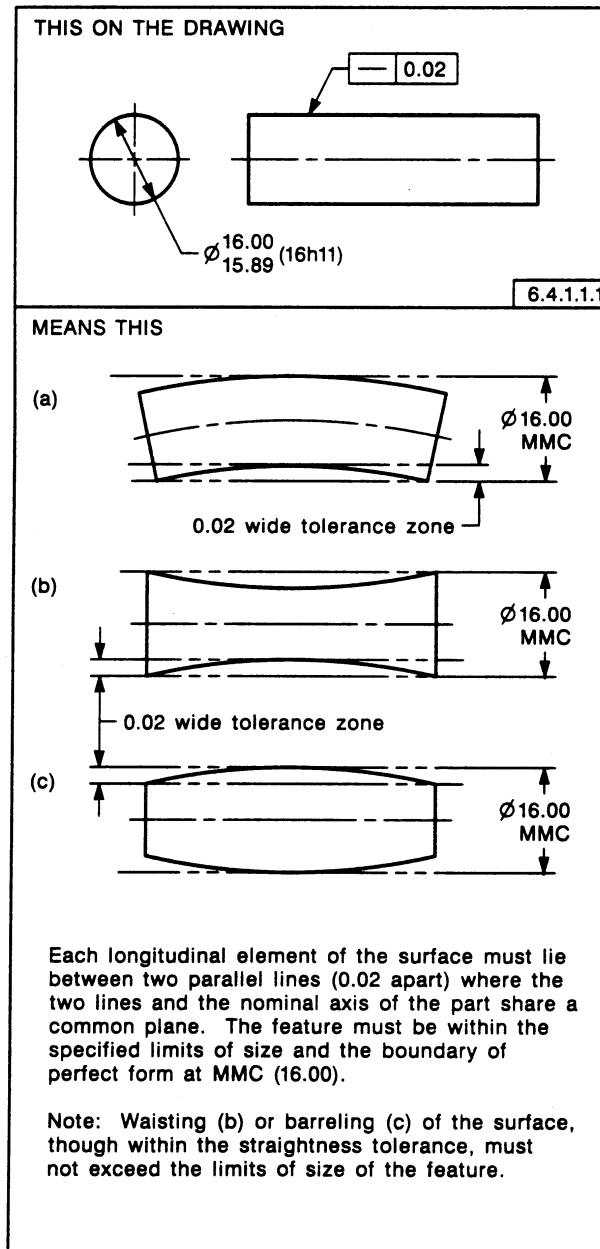
Geometric Characteristic Symbols

	TYPE OF TOLERANCE	CHARACTERISTIC	SYMBOL	SEE:
FOR INDIVIDUAL FEATURES	FORM	STRAIGHTNESS	—	6.4.1
		FLATNESS		6.4.2
		CIRCULARITY (ROUNDNESS)		6.4.3
		CYLINDRICITY		6.4.4
FOR INDIVIDUAL OR RELATED FEATURES	PROFILE	PROFILE OF A LINE		6.5.2 (b)
		PROFILE OF A SURFACE		6.5.2 (a)
FOR RELATED FEATURES	ORIENTATION	ANGULARITY		6.6.2
		PERPENDICULARITY		6.6.4
		PARALLELISM		6.6.3
	LOCATION	POSITION		5.2
		CONCENTRICITY		5.11.3
		SYMMETRY		5.13
	RUNOUT	CIRCULAR RUNOUT		6.7.1.2.1
		TOTAL RUNOUT		6.7.1.2.2
* ARROWHEADS MAY BE FILLED OR NOT FILLED				3.3.1

FIG. 3-1 GEOMETRIC CHARACTERISTIC SYMBOLS

A feature control frame gives information about geometric tolerances on the feature.

Straightness Example



Straightness at MMC

THIS ON THE DRAWING

6.4.1.1.2
4.5.4.1

MEANS THIS

Feature size	Diameter tolerance zone allowed
16.00	0.04
15.99	0.05
15.98	0.06
↓	↓
15.90	0.14
15.89	0.15

The derived median line of the feature actual local sizes must lie within a cylindrical tolerance zone of 0.04 diameter at MMC. As each actual local size departs from MMC, an increase in the local diameter of the tolerance cylinder is allowed which is equal to the amount of such departure. Each circular element of the surface must be within the specified limit of size.

ACCEPTANCE BOUNDARY

(a) The maximum diameter of the pin with perfect form is shown in a gage with a 16.04 diameter hole;

(b) with the pin at maximum diameter (16.00), the gage will accept the pin with up to 0.04 variation in straightness;

(c) with the pin at minimum diameter (15.89), the gage will accept the pin with up to 0.15 variation in straightness.

Meanings:

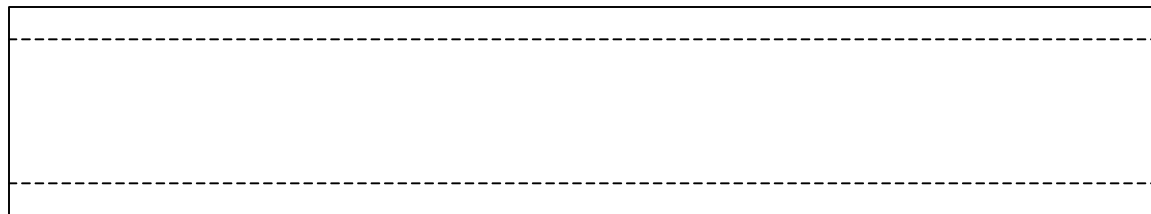
Demo

Sketch your observations:

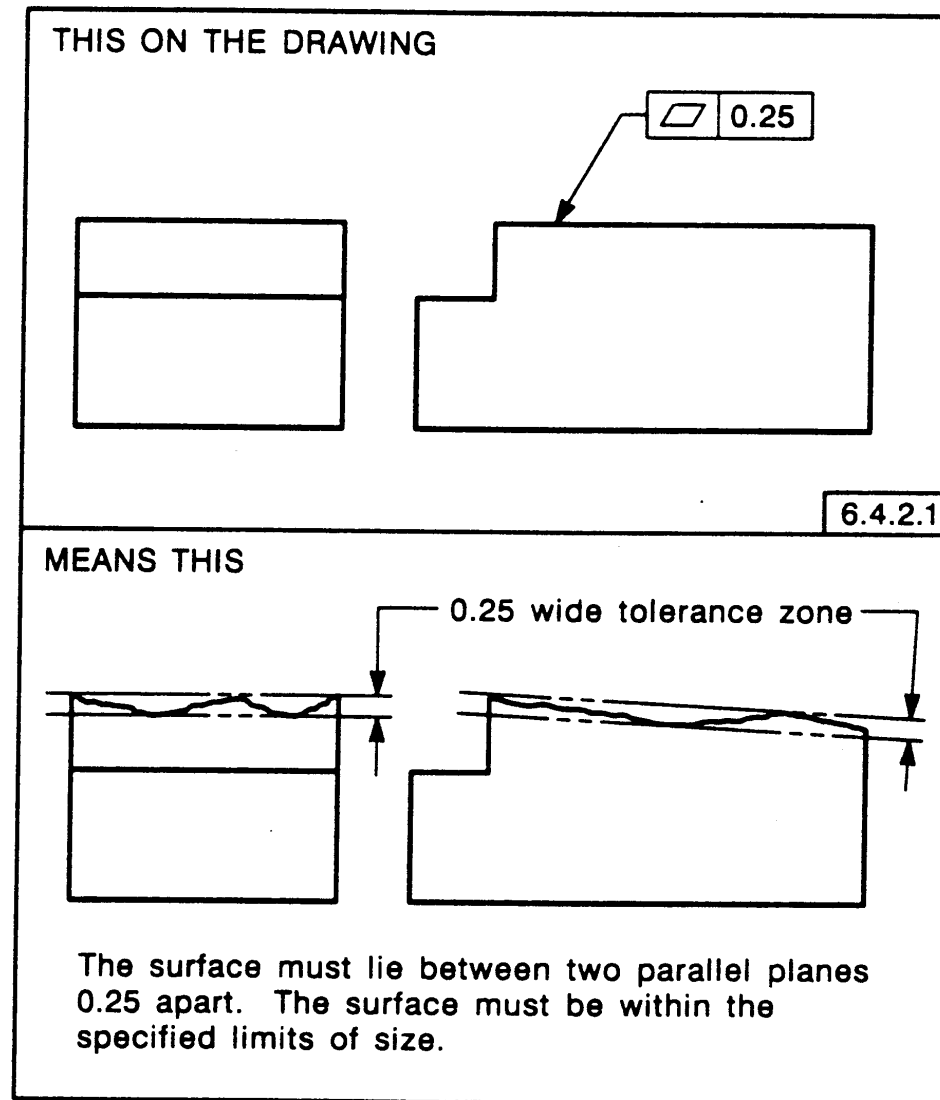
2 Rods: .375" diameter

 .750" diameter

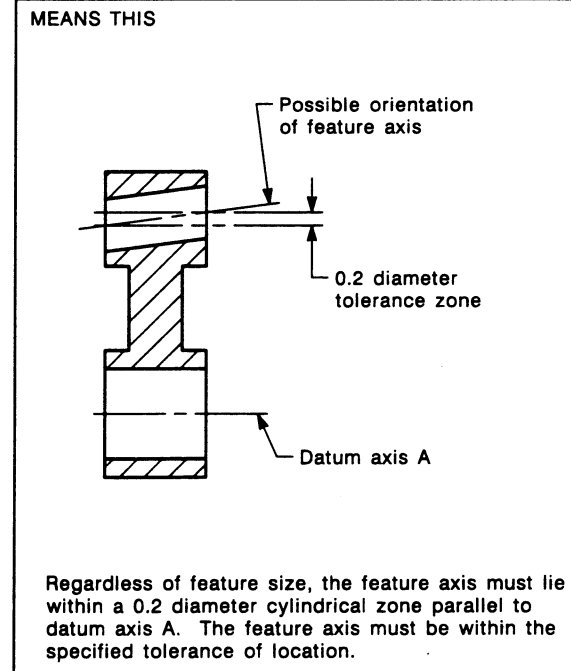
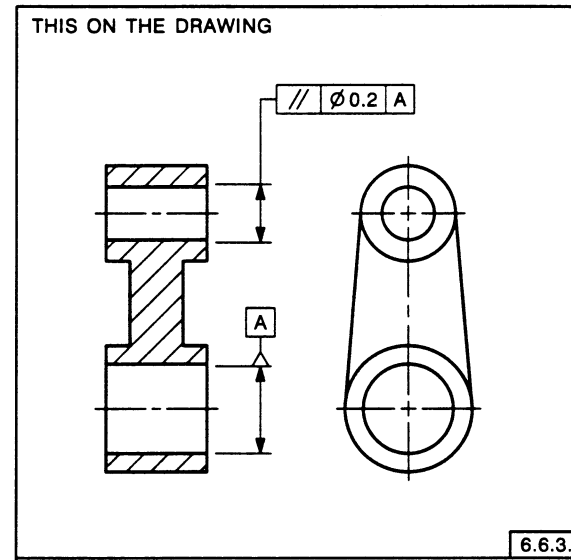
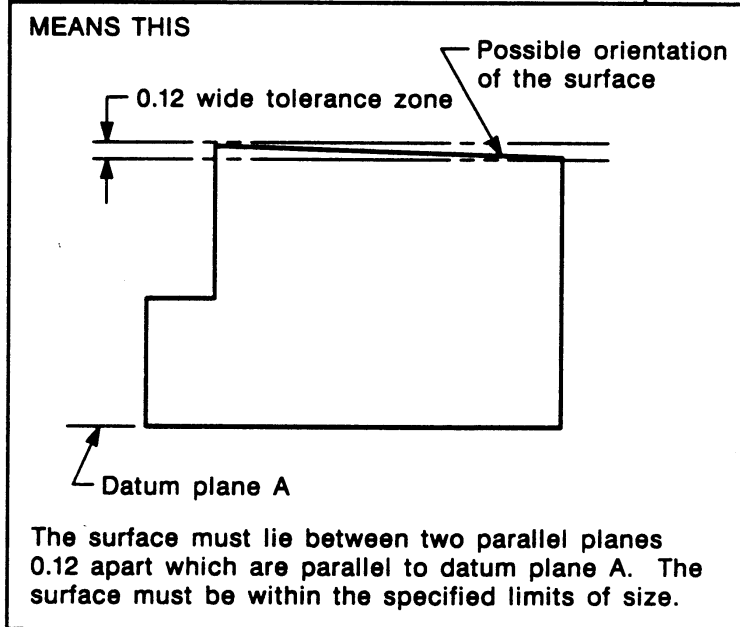
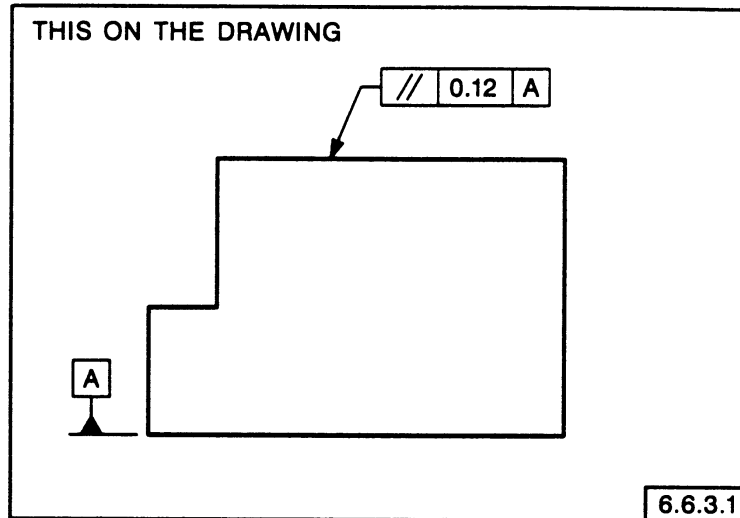
Tube with .755" hole



Flatness Examples

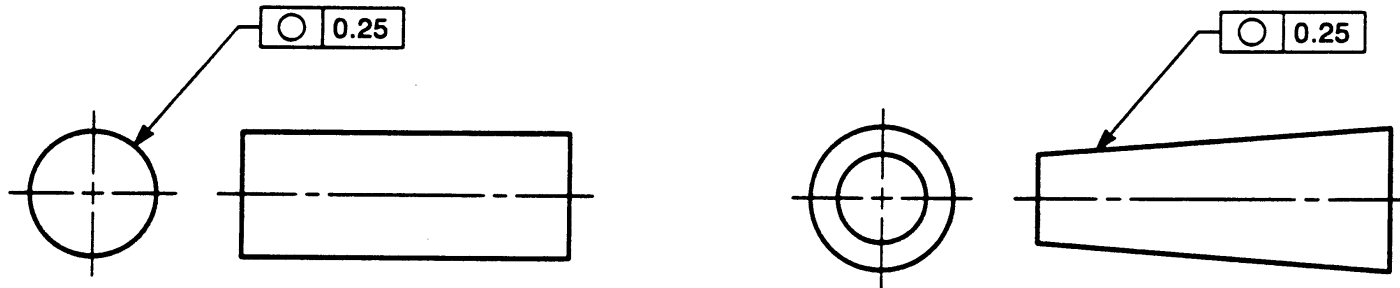


Parallelism Example



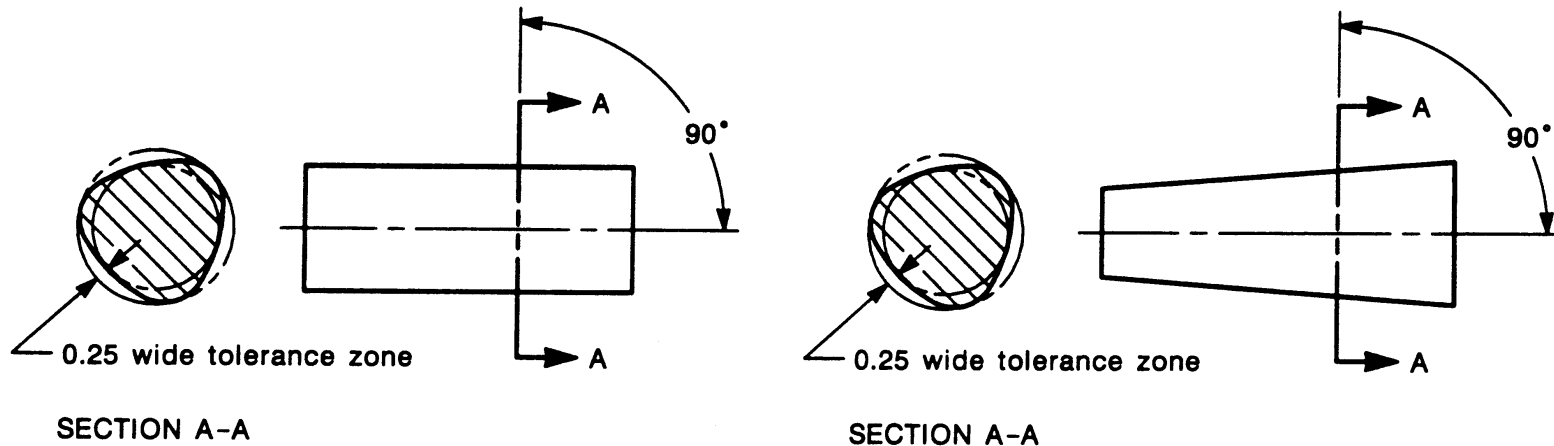
Circularity (Roundness) Example

THIS ON THE DRAWING



6.4.3.1

MEANS THIS



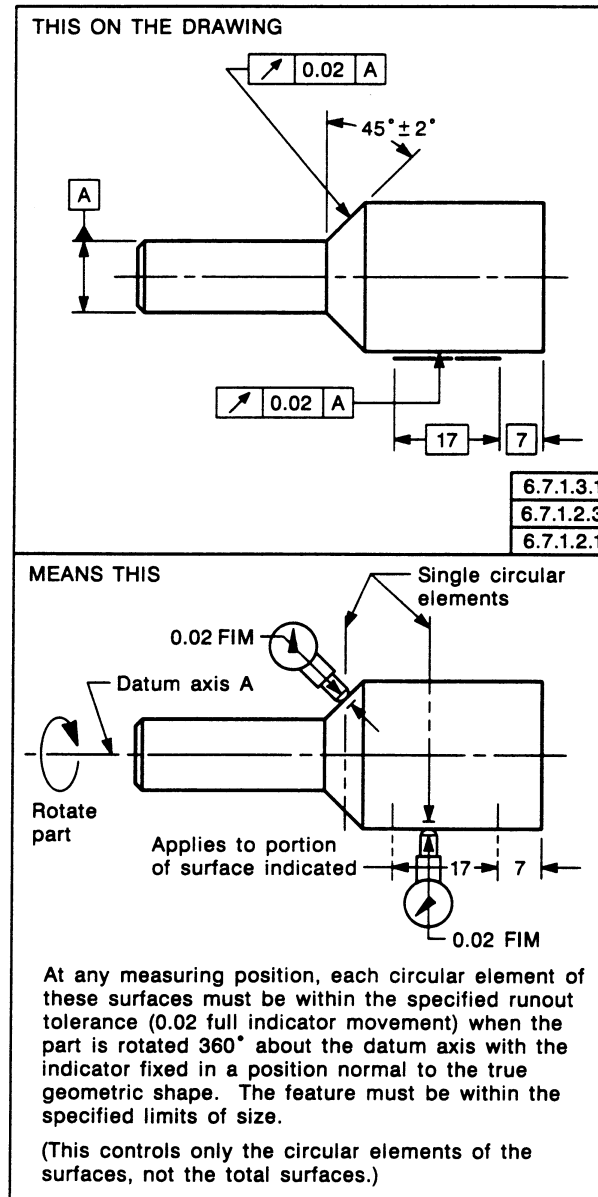
Each circular element of the surface in a plane perpendicular to an axis must lie between two concentric circles, one having a radius 0.25 larger than the other. Each circular element of the surface must be within the specified limits of size.

Circular & Total Runout

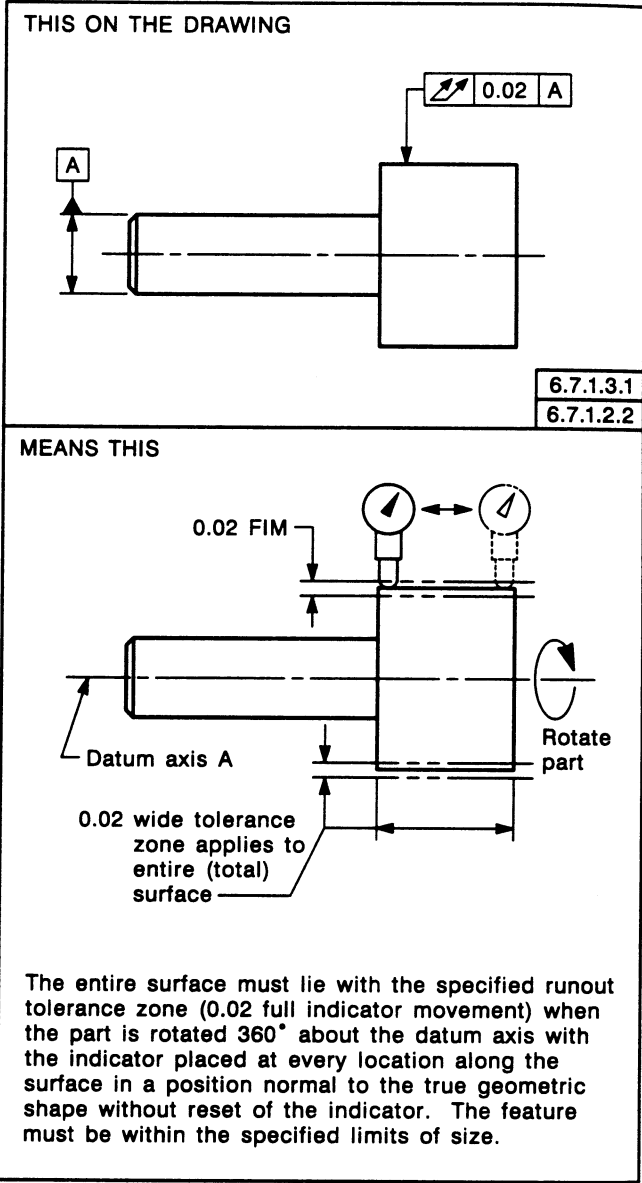
Runout is specified on cylindrical parts. It is measured by placing a gage on the part, and rotating the part through 360 degrees. The total variation is recorded as the runout.

- Circular runout is measured at one location.
- Total Runout is measured along the entire specified surface.

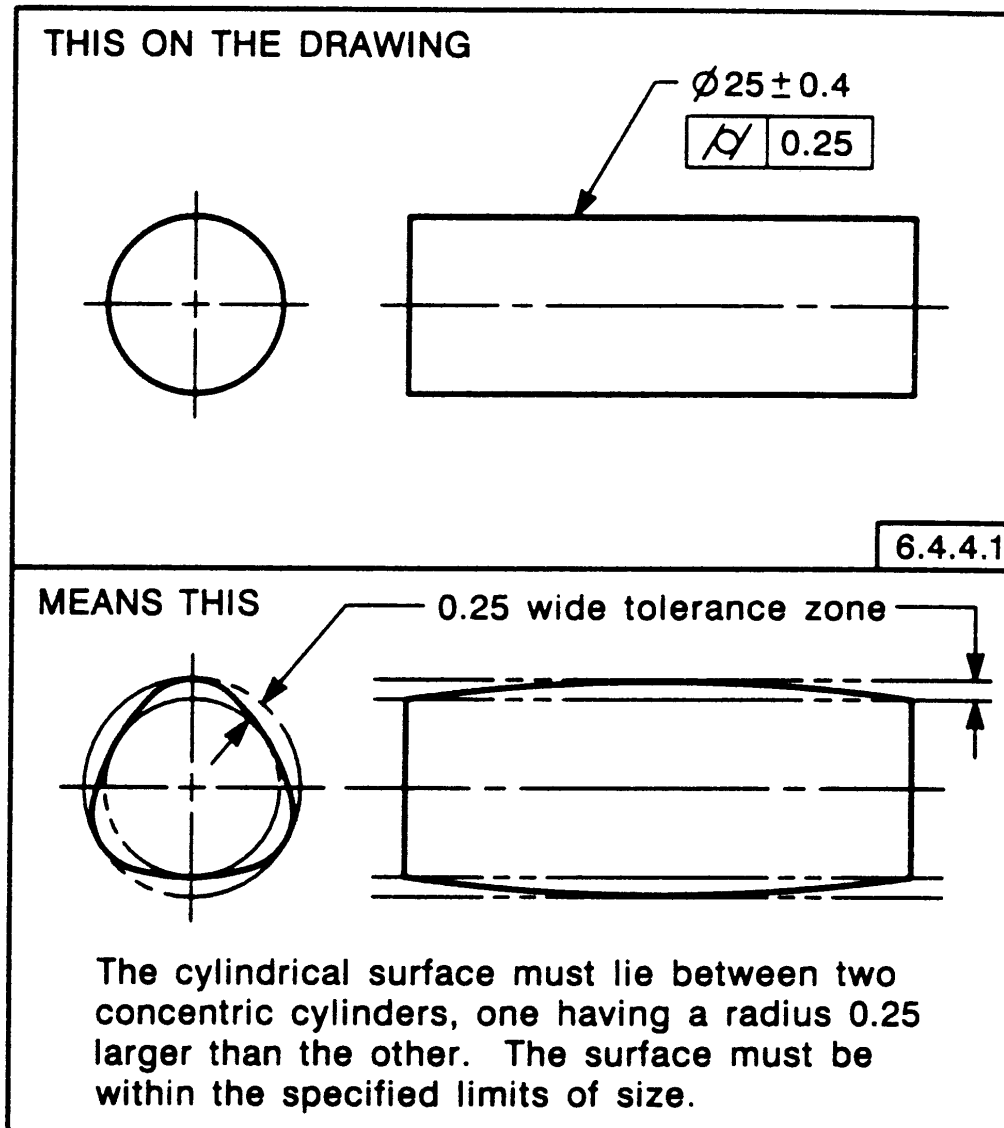
Circular Runout Example



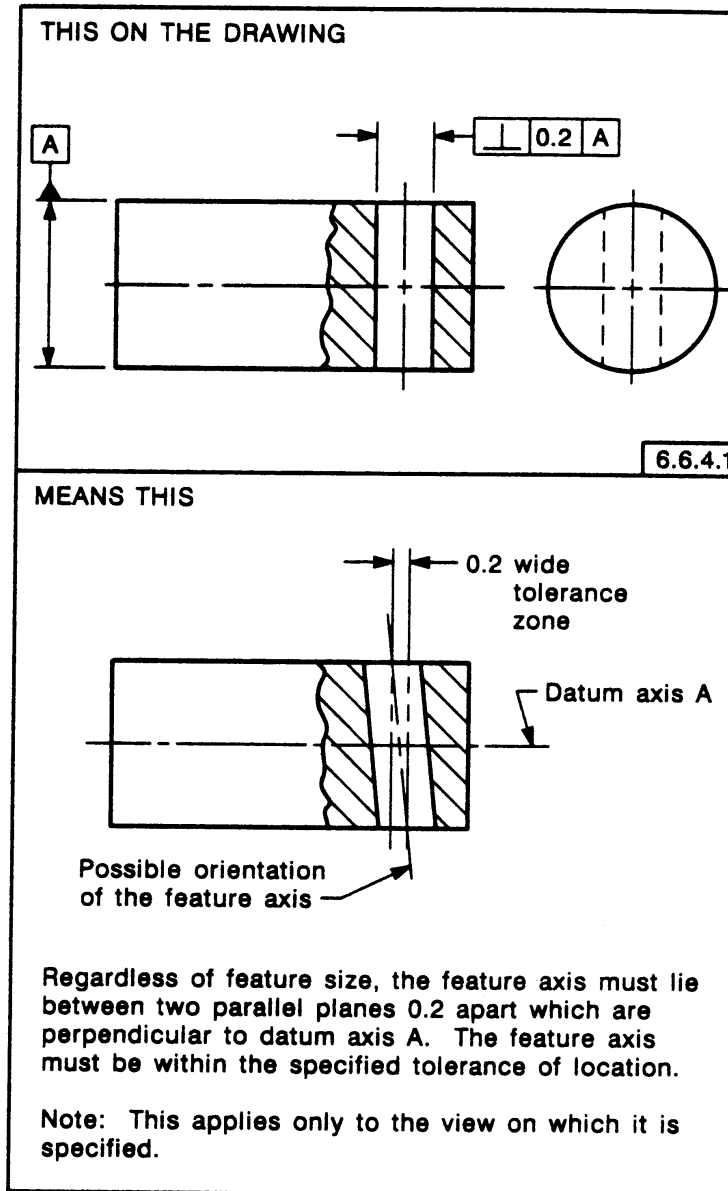
Total Runout Example



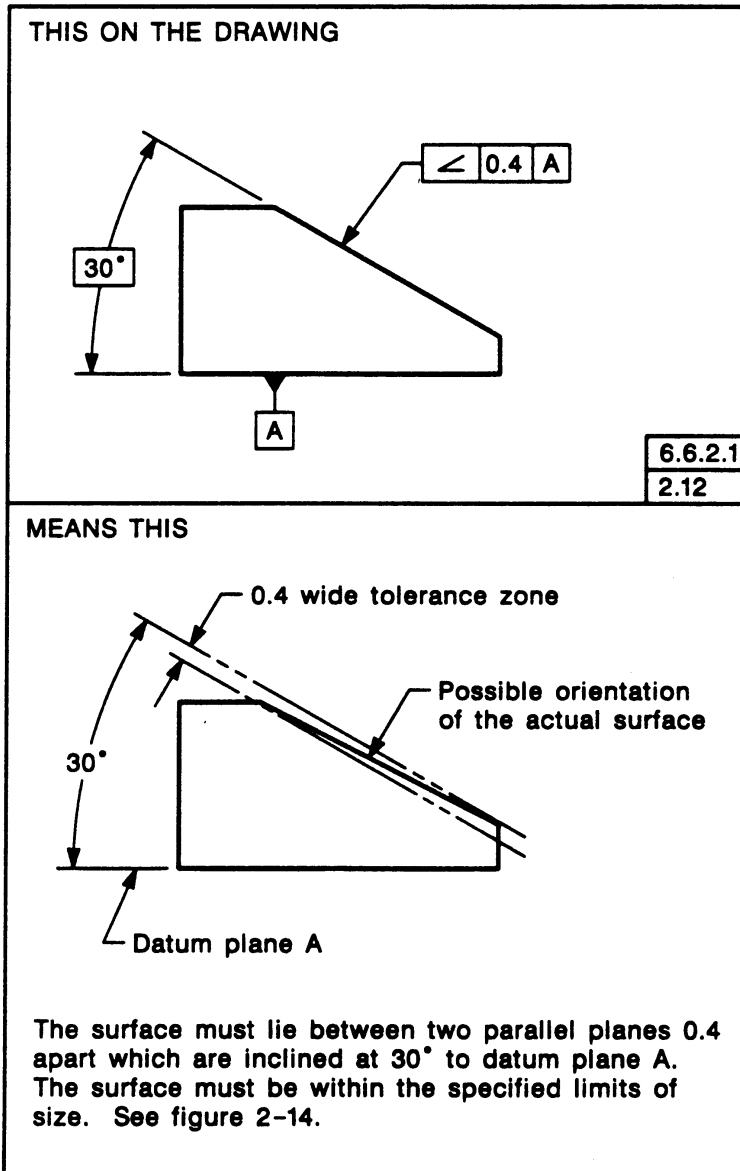
Cylindricity Example



Perpendicularity Example

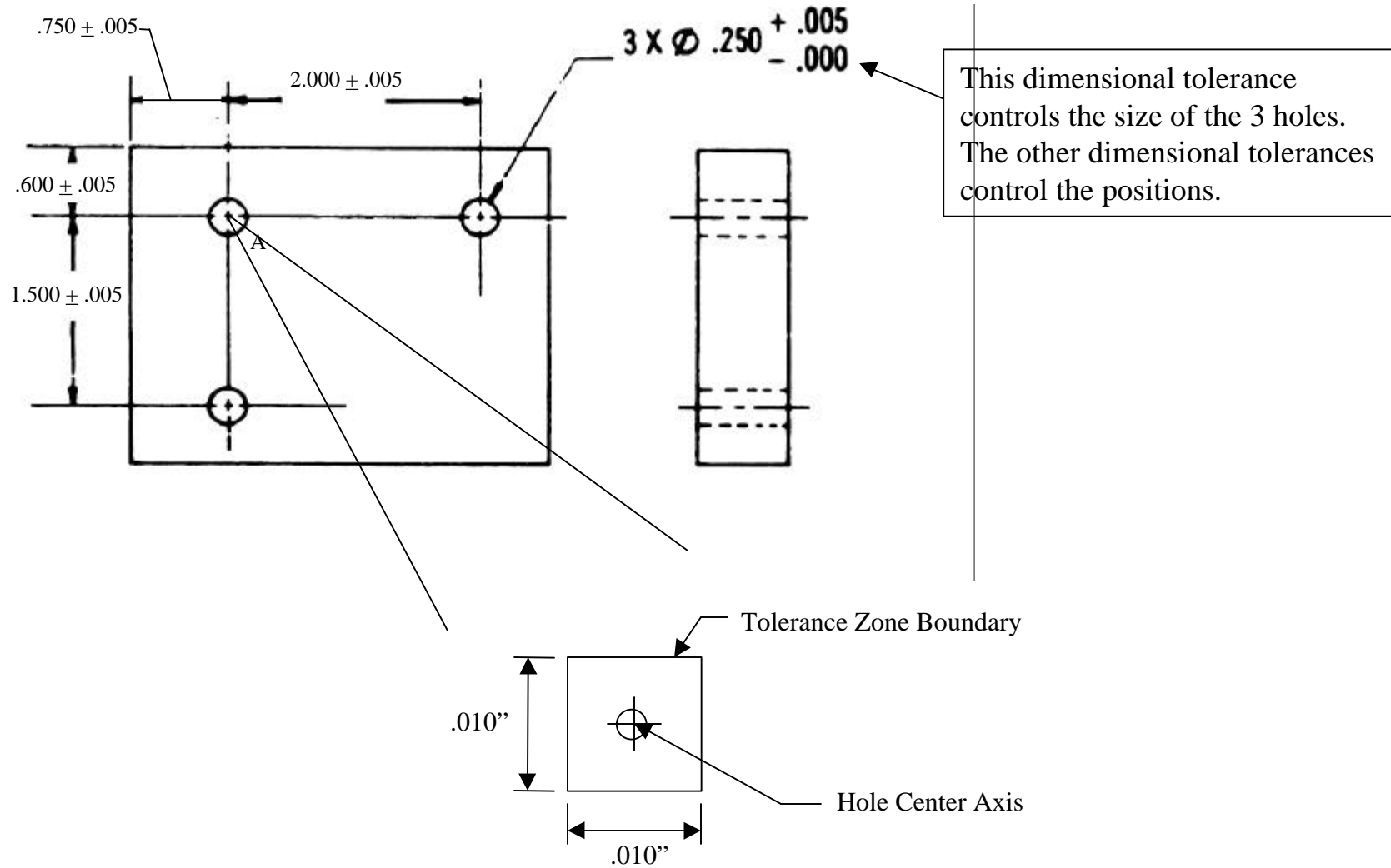


Angularity Example

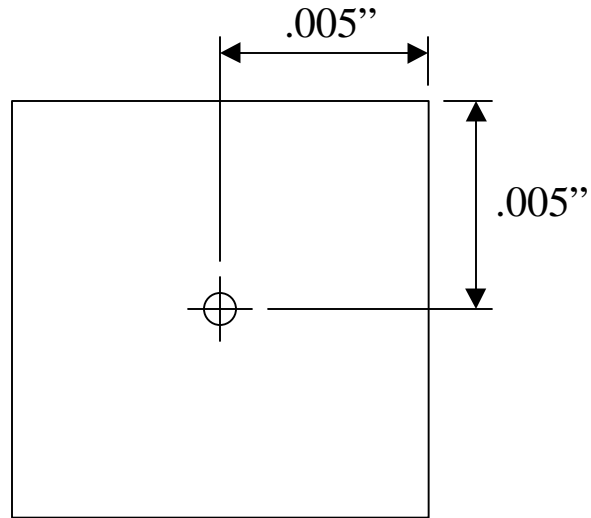


Measuring angularity is equivalent to measuring parallelism at an angle.

Conventional (Coordinate) Tolerancing

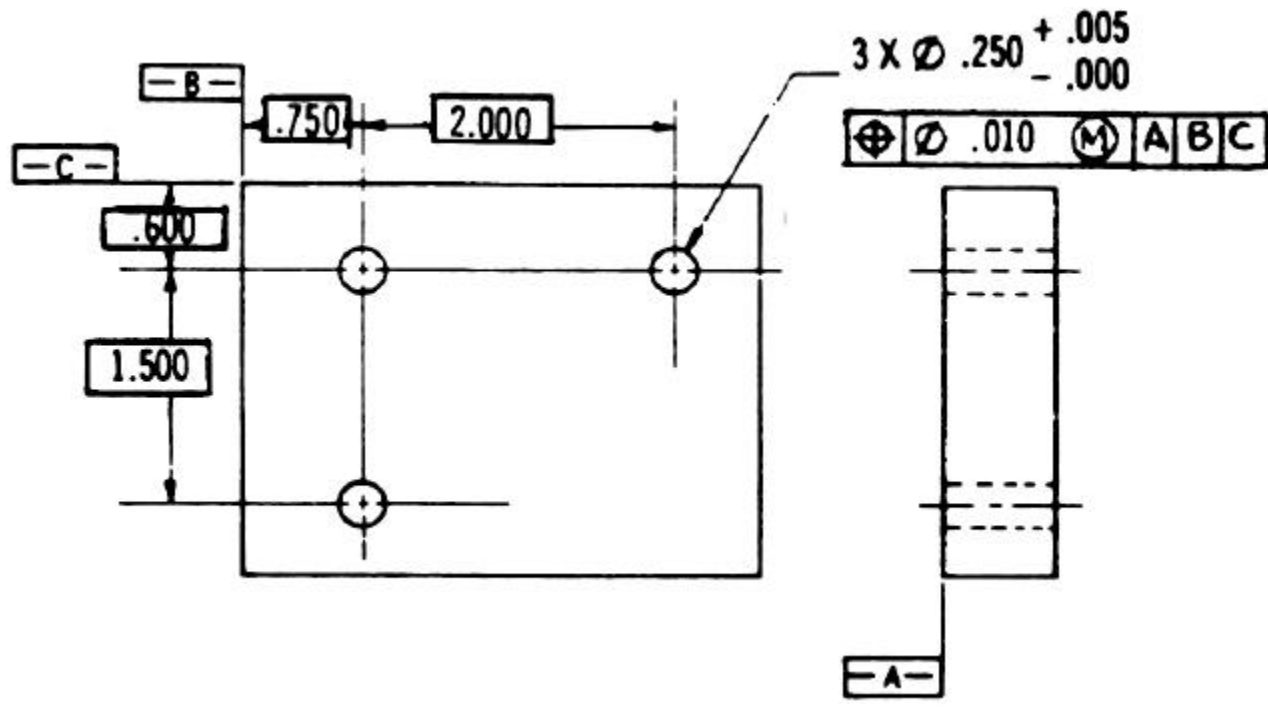


In the conventional tolerancing scheme, a hole center axis can reside anywhere in the square tolerance zone. The drawing may call out linear tolerances of $\pm.005''$, but...



By how much can the hole location deviate from spec?

Geometric Position Tolerancing



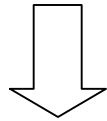
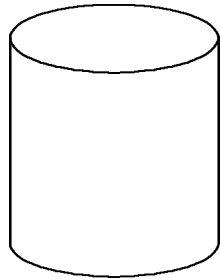
The Feature Control Frame is read like this:

“

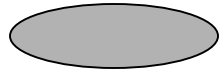
”

MMC vs. LMC

LARGEST
SHAFT

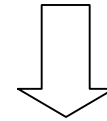
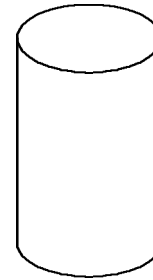


SMALLEST
HOLE

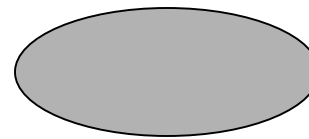


“Maximum Material Condition”

SMALLEST
SHAFT

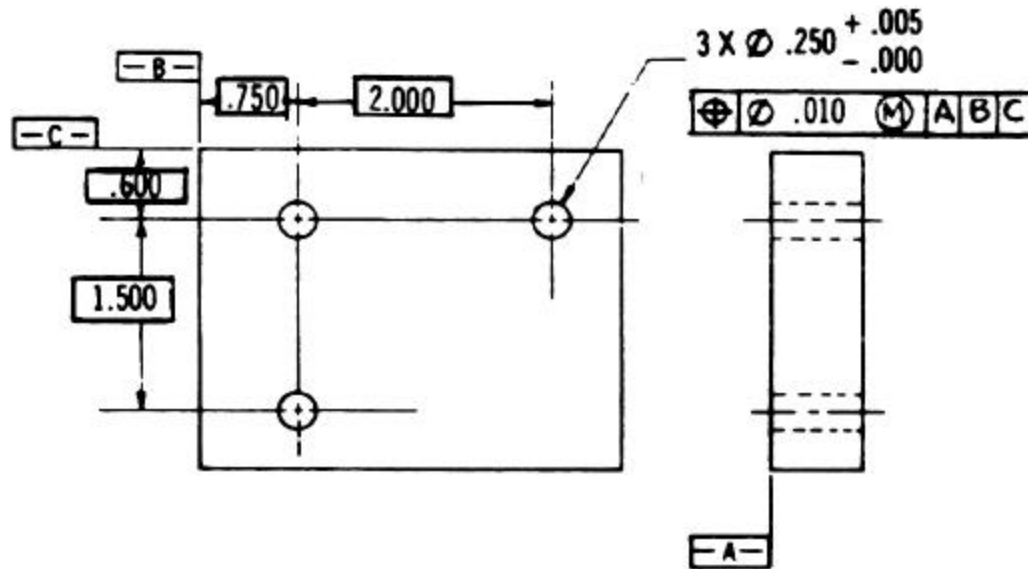


LARGEST
HOLE



“Least Material Condition”

Tolerance Zone Size



MMC of hole = .250

LMC of hole = .255

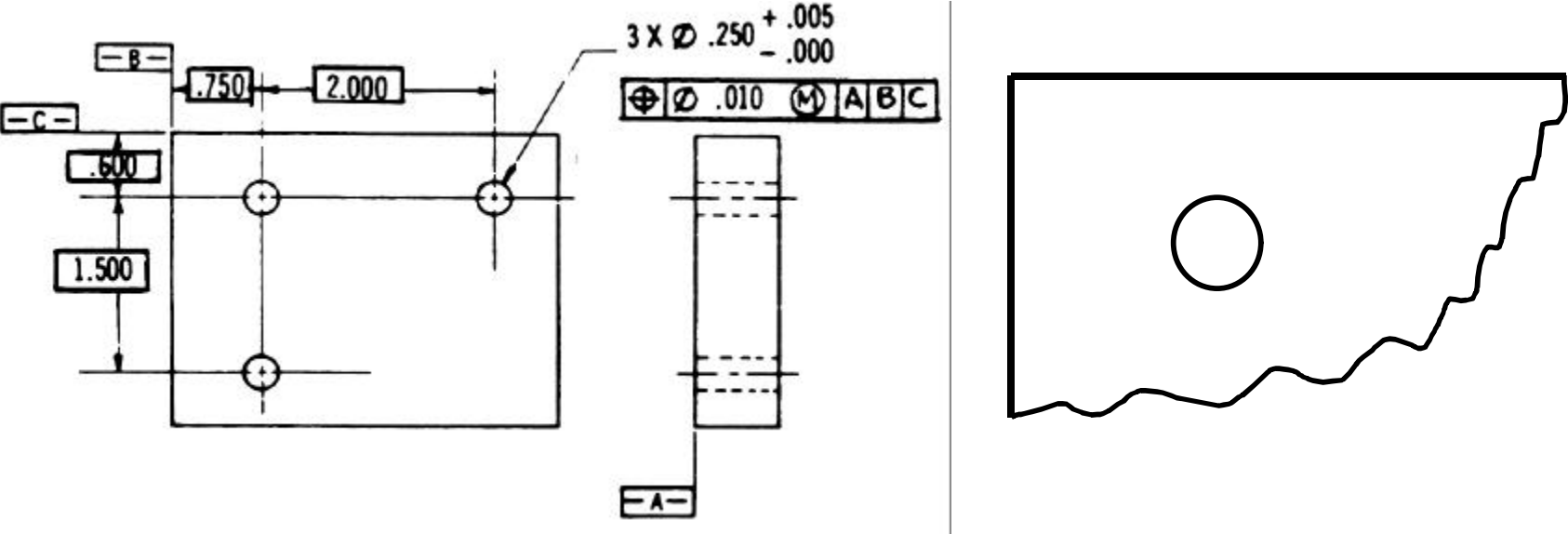
Hole diameter	Tolerance Zone diameter
.250 (MMC)	.010
.251	.011
.252	.012
.253	.013
.254	.014
.255 (LMC)	.015

This feature control frame specifies the tolerance zone as a circle of diameter .010 at MMC, centered according to the basic dimensions given. The size of the tolerance zone is dependent on the size of the hole.

A feature control frame can specify the size of the tolerance zone at MMC, LMC or RFS (regardless of feature size).

Determining Tolerance Zone Size

Does this feature meet the true position tolerance?

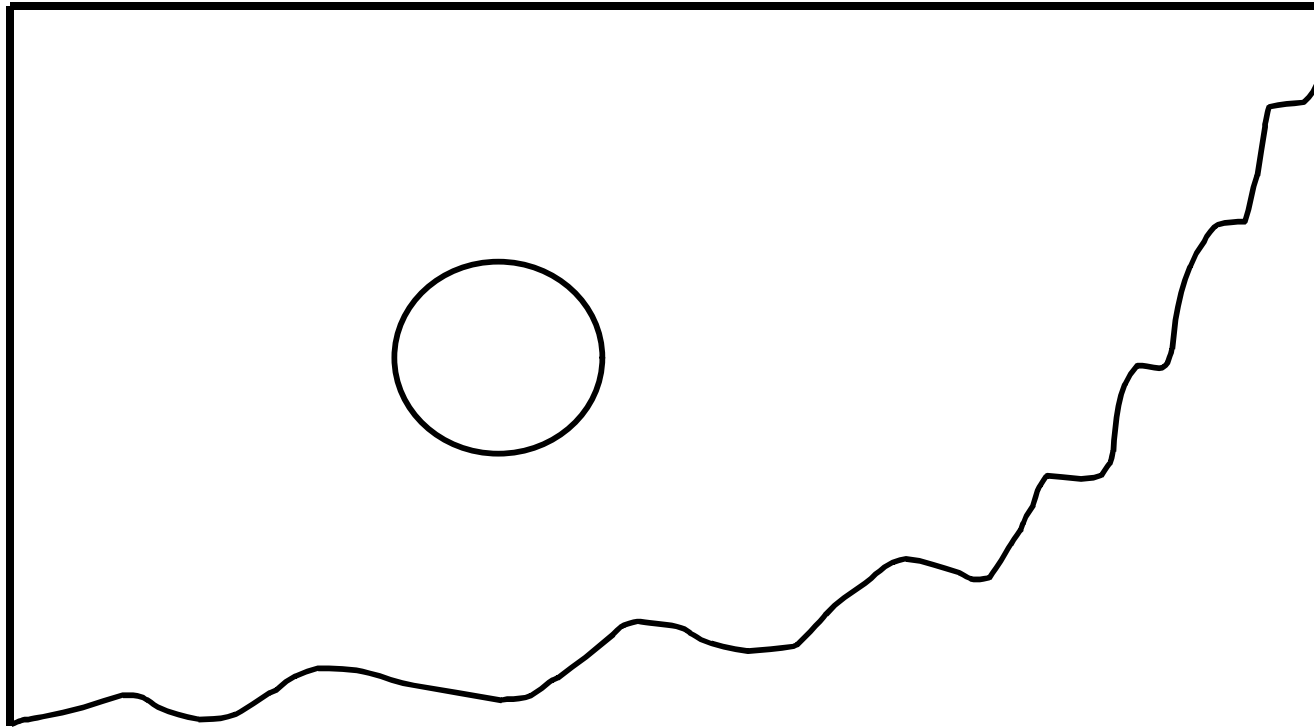


Step 1: What can we measure?

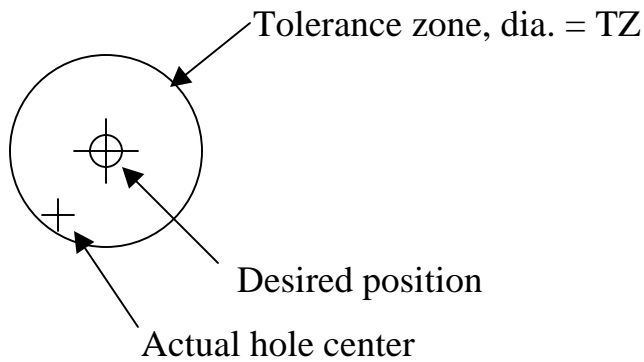
Step 2:

Calculate deviations in x and y directions.

How does this compare to the basic dimensions?

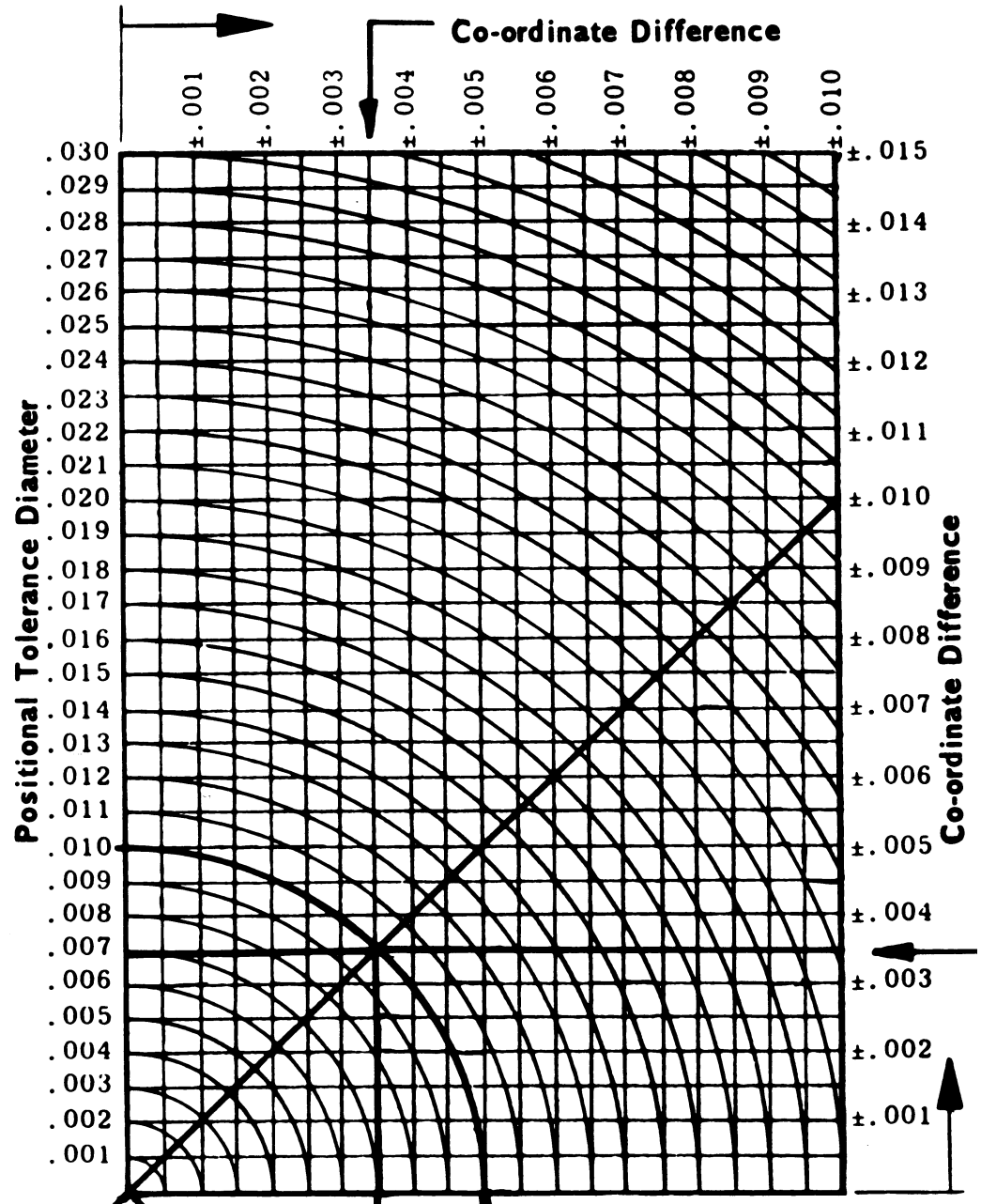


Step 3: Determining the True Position

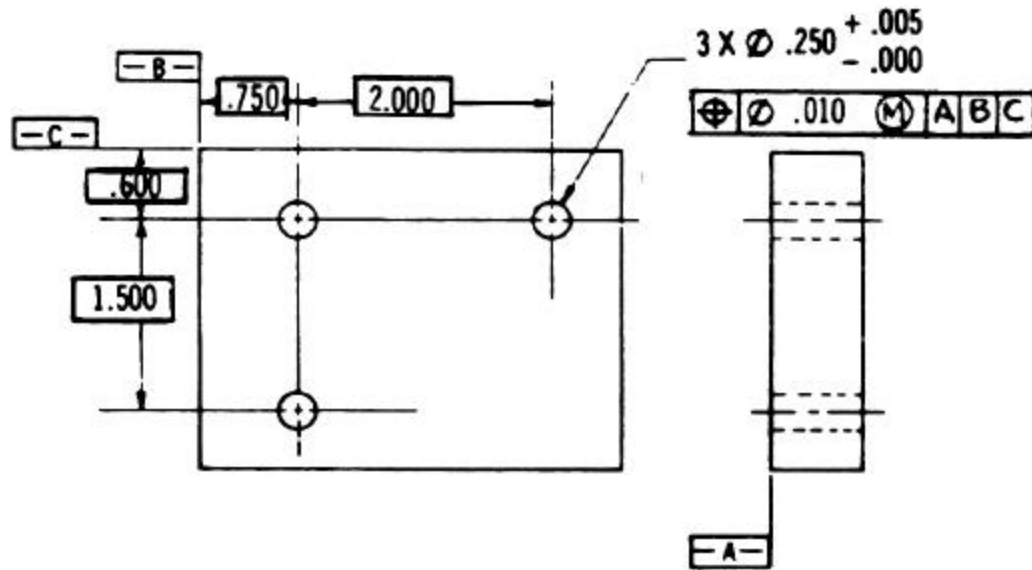


$$Z = 2\sqrt{x^2 + y^2}$$

A hole center that deviates from true position by x and y lies within a tolerance zone of diameter Z . If $Z > TZ$, the part is bad.

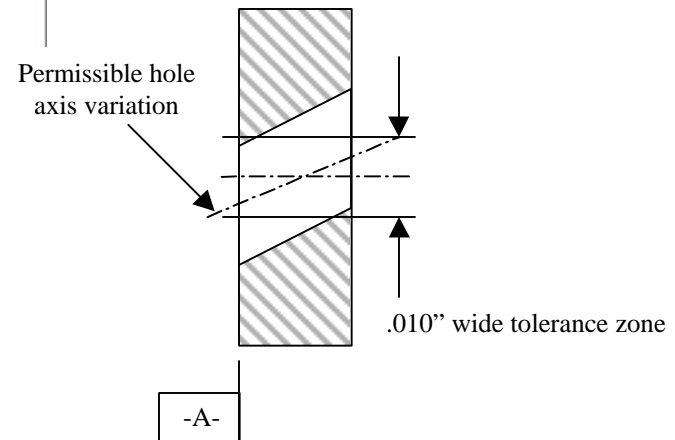


True Position and Perpendicularity



This feature control frame specifies the true position tolerance of the hole with respect to 3 datum planes. The order that the datum planes are listed in the feature control frame indicates the priority of each datum. Datums B and C provide reference for the x and y position of the hole center, and datum A controls the perpendicularity of the hole axis .

Referencing datum A means that the center axis of the hole must be perpendicular to datum plane A. The axis must intersect datum plane A inside the tolerance zone



Standard Fits

- Standard Fits are a way of specifying a fit between a hole and a shaft.
 - RC (1-9) Running or Sliding Clearance Fit
 - LC (1-11) Locational Clearance Fit
 - LT (1-6) Transition Clearance or Interference Fit
 - LN (1-3) Locational Interference Fit
 - FN (1-5) Force or Shrink Fit

We mention this here because it will be useful in dimension the parts of your yo-yo that must snap fit together.

Standard Fit Example

There is a nominal diameter of 1 inch for the shaft and hole on your yo-yo. You want a class FN2 fit. What should the dimensions and tolerances be for the shaft and the hole?

Table 48. Limits of Clearance for Running and Sliding Fits
(Limits are in thousandths of an inch on diameter)

Nominal size range, in.		Class								
Over	To	RC1	RC2	RC3	RC4	RC5	RC6	RC7	RC8	RC9
0.04	0.12	0.1	0.1	0.3	0.3	0.6	0.6	1.0	2.5	4.0
		0.45	0.55	0.8	1.1	1.4	1.8	2.6	5.1	8.1
0.12	0.24	0.15	0.15	0.4	0.4	0.8	0.8	1.2	2.8	4.5
		0.5	0.65	1.0	1.4	1.8	2.2	3.1	5.8	9.3
0.24	0.40	0.2	0.2	0.5	0.5	1.0	1.0	1.6	3.0	5.0
		0.6	0.85	1.3	1.7	2.2	2.8	3.9	6.6	10.7
0.40	0.71	0.25	0.25	0.6	0.6	1.2	1.2	2.0	3.5	6.0
		0.75	0.95	1.4	2.0	2.6	3.2	4.6	7.9	12.8
0.71	1.19	0.3	0.3	0.8	0.8	1.6	1.6	2.5	4.5	7.0
		0.95	1.2	1.8	2.4	3.2	4.0	5.7	10.0	15.5
1.19	1.97	0.4	0.4	1.0	1.0	2.0	2.0	3.0	5.0	8.0
		1.1	1.4	2.2	3.0	4.0	5.2	7.1	11.5	18.0
1.97	3.15	0.4	0.4	1.2	1.2	2.5	2.5	4.0	6.0	9.0
		1.2	1.6	2.6	3.6	4.9	6.1	8.8	13.5	20.5
3.15	4.73	0.5	0.5	1.4	1.4	3.0	3.0	5.0	7.0	10.0
		1.5	2.0	3.2	4.2	5.8	7.4	10.7	15.5	24.0
4.73	7.09	0.6	0.6	1.6	1.6	3.5	3.5	6.0	8.0	12.0
		1.8	2.3	3.6	4.8	6.7	8.5	12.5	18.0	28.0
7.09	9.85	0.6	0.6	2.0	2.0	4.0	4.0	7.0	10.0	15.0
		2.0	2.6	4.4	5.6	7.6	9.6	14.3	21.5	34.0
9.85	12.41	0.8	0.8	2.5	2.5	5.0	5.0	8.0	12.0	18.0
		2.3	2.9	4.9	6.5	9.0	11.0	16.0	25.0	38.0
12.41	15.75	1.0	1.0	3.0	3.0	6.0	6.0	10.0	14.0	22.0
		2.7	3.4	5.8	7.4	10.4	13.0	19.5	29.0	45.0
15.75	19.69	1.2	1.2	4.0	4.0	8.0	8.0	12.0	16.0	25.0
		3.0	3.8	7.2	9.0	13.0	16.0	22.0	32.0	51.0

Table 49. Limits of Interference for Force and Shrink Fits
(Limits are in thousandths of an inch on diameter)

Nominal size range, in.		Class				
Over	To	FN1	FN2	FN3	FN4	FN5
0.04	0.12	0.05	0.2	0.3	0.5
		0.5	0.85	0.95	1.3
0.12	0.24	0.1	0.2	0.4	0.7
		0.6	1.0	1.2	1.7
0.24	0.40	0.1	0.4	0.6	0.8
		0.75	1.4	1.6	2.0
0.40	0.56	0.1	0.5	0.7	0.9
		0.8	1.6	1.8	2.3
0.56	0.71	0.2	0.5	0.7	1.1
		0.9	1.6	1.8	2.5
0.71	0.95	0.2	0.6	0.8	1.4
		1.1	1.9	2.1	3.0
0.95	1.19	0.3	0.6	0.8	1.0	1.7
		1.2	1.9	2.1	2.3	3.3
1.19	1.58	0.3	0.8	0.8	1.5	2.0
		1.3	2.4	2.4	3.1	4.0
1.58	1.97	0.4	0.8	1.2	1.8	3.0
		1.4	2.4	2.8	3.4	5.0
1.97	2.56	0.6	0.8	1.3	2.3	3.8
		1.8	2.7	3.2	4.2	6.2
2.56	3.15	0.7	1.0	1.8	2.8	4.8
		1.9	2.9	3.7	4.7	7.2
3.15	3.94	0.9	1.4	2.1	3.6	5.6
		2.4	3.7	4.4	5.9	8.4
3.94	4.73	1.1	1.6	2.6	4.6	6.6
		2.6	3.9	4.9	6.9	9.4
4.73	5.52	1.2	1.9	3.4	5.4	8.4
		2.9	4.5	6.0	8.0	11.6
5.52	6.30	1.5	2.4	3.4	5.4	10.4
		3.2	5.0	6.0	8.0	13.6
6.30	7.09	1.8	2.9	4.4	6.4	10.4
		3.5	5.5	7.0	9.0	13.6
7.09	7.88	1.8	3.2	5.2	7.2	12.2
		3.8	6.2	8.2	10.2	15.8
7.88	8.86	2.3	3.2	5.2	8.2	14.2
		4.3	6.2	8.2	11.2	17.8
8.86	9.85	2.3	4.2	6.2	10.2	14.2
		4.3	7.2	9.2	13.2	17.8
9.85	11.03	2.8	4.0	7.0	10.0	16.0
		4.9	7.2	10.2	13.2	20.0
11.03	12.41	2.8	5.0	7.0	12.0	18.0
		4.9	8.2	10.2	15.2	22.0
12.41	13.98	3.1	5.8	7.8	13.8	19.8
		5.5	9.4	11.4	17.4	24.2
13.98	15.75	3.6	5.8	9.8	15.8	22.8
		6.1	9.4	13.4	19.4	27.2
15.75	17.72	4.4	6.5	9.5	17.5	25.5
		7.0	10.6	13.6	21.6	30.5
17.72	19.69	4.4	7.5	11.5	19.5	27.5
		7.0	11.6	15.6	23.6	32.5