MECH 691T Engineering Metrology and Measurement Systems

Lecture 3

Instructor: N R Sivakumar

Outline

- Tolerancing and interchangeability
- Limits and tolerances
- Tolerance representation
- Tolerance stackup
- Theory and applications of tolerancing techniques in precision machining
- Tolerance analysis, and synthesis

Why Use GD&T?

- First and foremost: IT SAVES MONEY!
- Provides maximum production tolerances
- Helps achieve design intent
- Ensures interchangeability of mating parts
- Provides uniformity of specification and interpretation – reducing guesswork & controversy
- Maximizes quality

99% Quality Means…

- 20,000 lost pieces of mail per hour
- Unsafe drinking water 15 minutes per day
- 5,000 incorrect surgical procedures per week
- Two short or long landings at most major airports per day
- 200,000 wrong drug prescriptions per year
- No electricity for 7 hours per month

Tolerance Definition

"The total amount by which a given dimension may vary, or the difference between the limits"

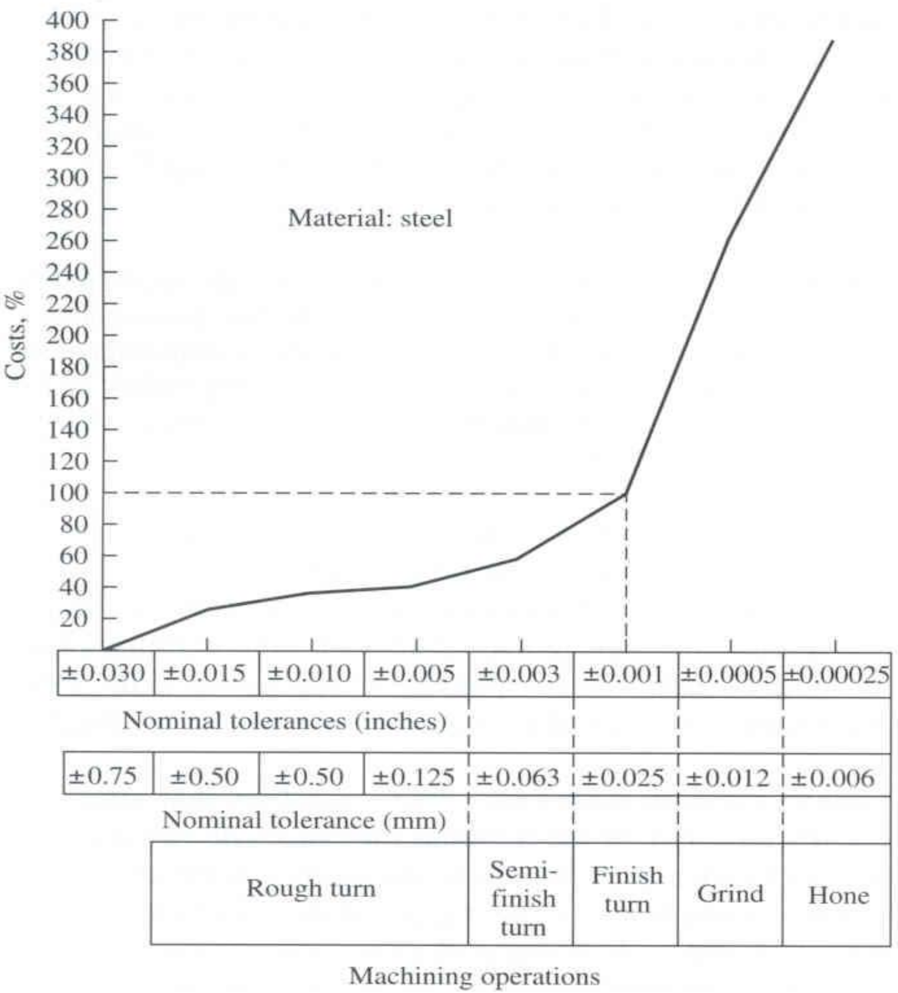
- ANSI Y14.5M-1982(R1988) Standard [R1.4]

Tolerance Types

- Linear Tolerance
  - Size
- Geometric Tolerance
  - Form
  - Profile
  - Location
  - Orientation
  - Runout

Tolerance vs. Mfg Process

- Nominal tolerances for steel
- Tighter tolerances => increase cost $



Limits and tolerances

Key Concepts

Actual Size - is the measured size

Basic Size - of a dimension is the theoretical size from which the limits for that dimension are derived by the application of the allowance and tolerance.

Design Size - refers to the size from which the limits of size are derived by the application of tolerances.

Limits of Size - are the maximum and minimum sizes permissible for a specific dimension.

Nominal Size - is the designation used for the purpose of general identification.

Limits and tolerances

Key Concepts

Tolerance - The tolerance on a dimension is the total permissible variation in the size of a dimension. The tolerance is the difference between the limits of size.

Bilateral Tolerance - With bilateral tolerance, variation is permitted in both directions from the specified dimension.

Unilateral Tolerance - With unilateral tolerance, variation is permitted in only one direction from the specified dimension.

Maximum Material Size - The maximum material size is the limit of size of a feature that results in the part containing the maximum amount of material. Thus it is the maximum limit of size for a shaft or an external feature, or the minimum limit of size for a hole or internal feature.

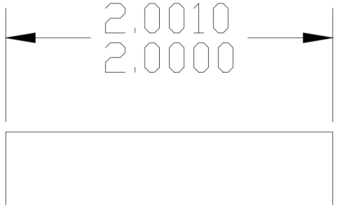
Limits and tolerances

Terminology

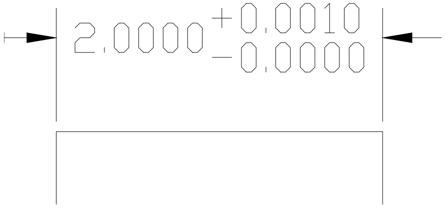
TERMINOLOGY	EXAMPLE	EXPLANATION
BASIC SIZE	1.500	
BASIC SIZE WITH TOLERANCE ADDED	1.500 ± .004 ←	HALF OF TOTAL TOLERANCE
LIMITS OF SIZE	1.504 1.496	LARGEST AND SMALLEST SIZES PERMITTED
TOLERANCE	.008	DIFFERENCE BETWEEN LIMITS OF SIZE

Representing Linear Tolerance

- General tolerance
  - Note: all decimals ±.02
- Limit dimensions



- Plus and minus dimensions
  - unilateral
  - bilateral



Fits and Allowances

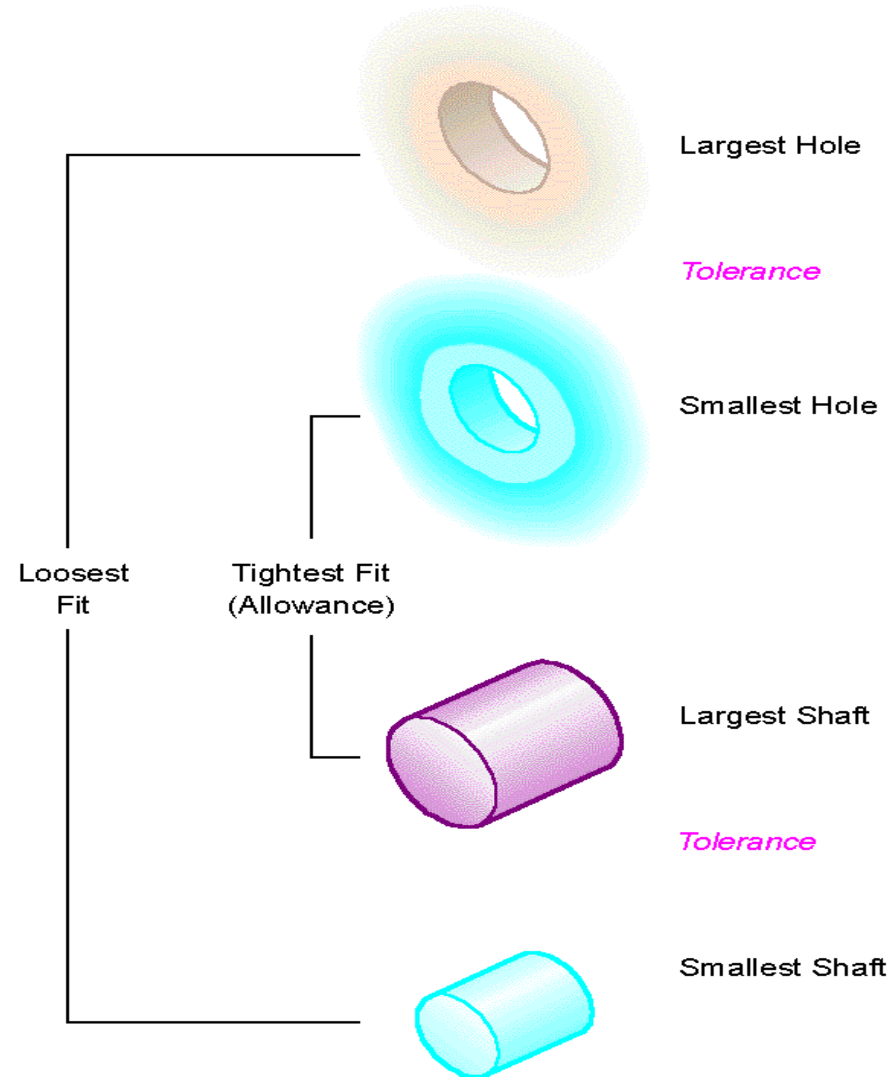
Fits

The fit between two mating parts is the relationship between them with respect to the amount of clearance or interference present when they are assembled. There are three basic types of fits: clearance, interference, and transition.

Clearance Fit - A fit between mating parts having limits of size so prescribed that a clearance always results in assembly.

Interference Fit - A fit between mating parts having limits of size so prescribed that an interference always results in assembly.

Determining Fits



Fits and Allowances

Allowance -intentional difference between the maximum material limits of mating parts.

Basic Size The size to which limits or deviations are assigned. - same for both members of a fit.

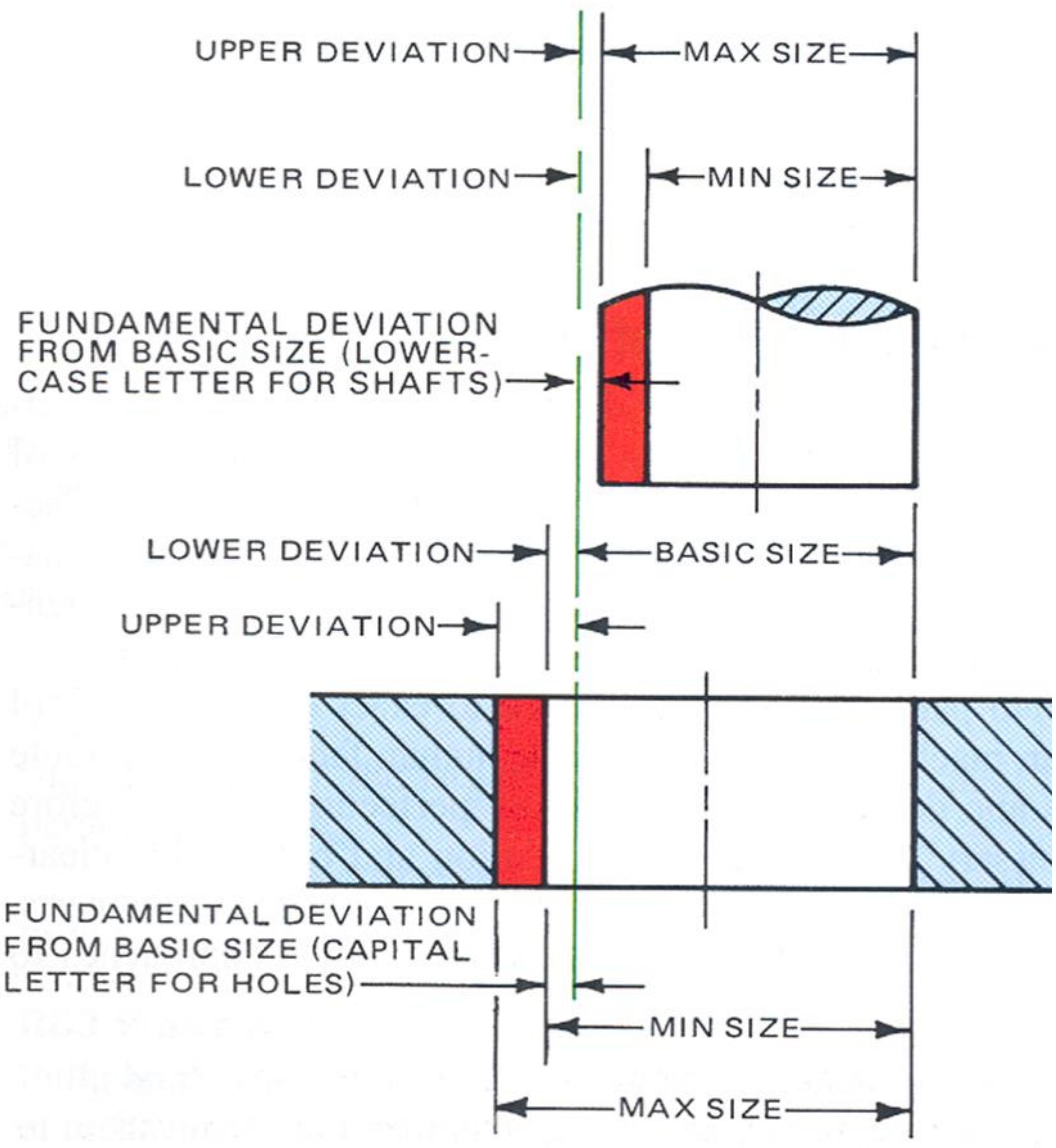
Deviation The difference between a size and corresponding basic size.

Upper Deviation maximum limit of size - basic size.

Lower Deviation minimum limit of size - basic size.

Tolerance maximum size - minimum size on a part. And the zone representing this is the **Tolerance Zone**

Fundamental Deviation The deviation closest to the basic size.



Fits and Allowances

Description of Fits (Values given in Handout)

Running and Sliding Fits - A special type of clearance fit. These are intended to provide a similar running performance, with suitable lubrication allowance, throughout the range of sizes.

Locational Fits - They may provide rigid or accurate location, as with interference fits, or some freedom of location, as with clearance fits. Accordingly, they are divided into three groups:
clearance fits, transition fits, and interference fits.

Locational clearance fits - are intended for parts that are normally stationary but that can be freely assembled or disassembled.

Locational transition fits - are a compromise between clearance and interference fits when accuracy of location is important but a small amount of either clearance or interference is permissible.

Locational interference fits - are used when accuracy of location is of prime importance and for parts requiring rigidity and alignment with no special requirements for bore pressure.

Fits and Allowances

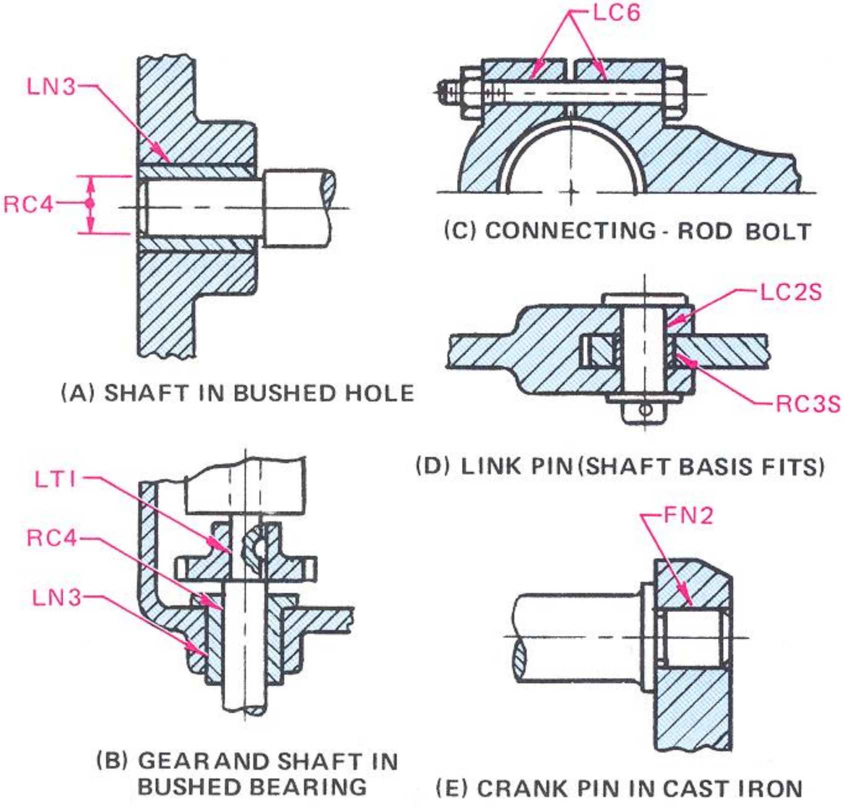
Description of Fits

Drive and force fits

It constitute a special type of interference fit, normally characterized by maintenance of constant bore pressures throughout the range of sizes.

Standard inch Fits

RC	Running and sliding fit
LC	Locational clearance fit
LT	Locational transition fit
LN	Locational interference fit
FN	Force or shrink fit

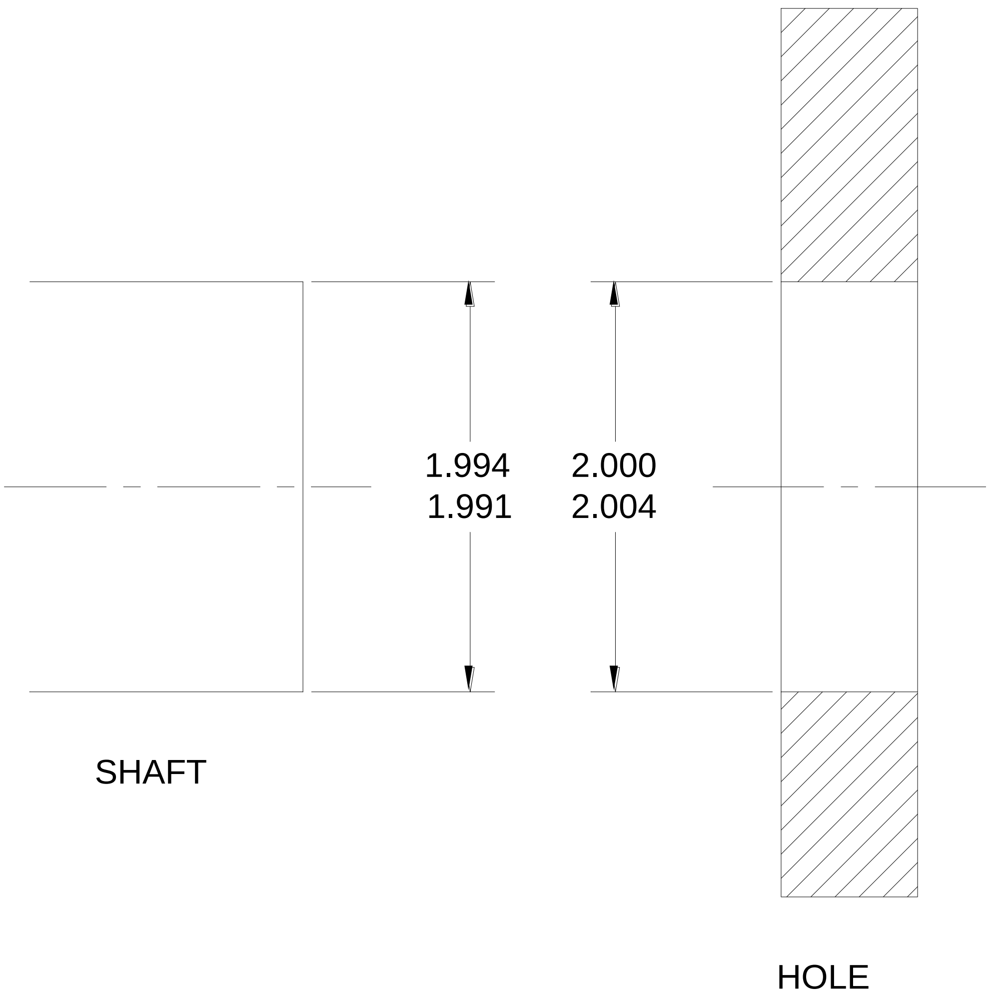


Typical design sketches showing classes of fits.

Hole Basis System

- Smallest possible hole (MMC) is the basic size
- Hole tolerance added to basic size for largest hole
- Allowance subtracted from basic size for largest shaft
- Tolerance of shaft subtracted from largest shaft for smallest shaft

Example - Hole Basis System



- Nominal size - 2”
- Basic size - 2.000”
- Hole tolerance - .004”
- Shaft tolerance - .003”
- Allowance - .006”

Fits and Allowances

Basic Hole System

In the basic hole system, which is recommended for general use, the basic size will be the design size for the hole, and the tolerance will be plus. The design size for the shaft will be the basic size minus the minimum clearance, or plus the maximum interference **(Refer Handout)**

	+.0020		+.0000
Hole Ø	**1.0000**	**Shaft Ø**	**.9975**
	-.0000		-.0012

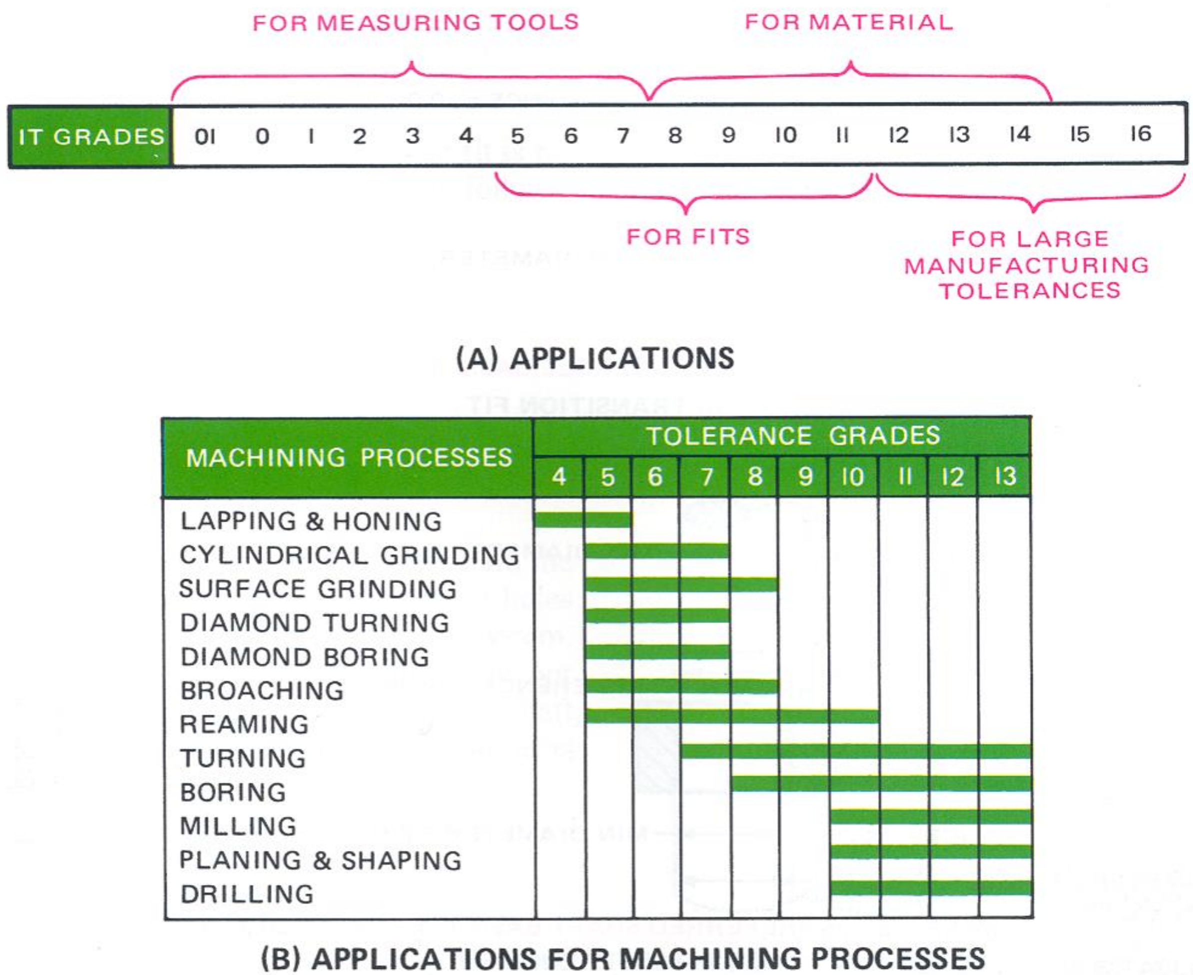
Basic Shaft System

Fits are sometimes required on a basic shaft system, especially when two or more fits are required on the same shaft.

	+.0020		+.0000
Hole Ø	**1.0025**	**Shaft Ø**	**1.0000**
	-.0000		-.0012

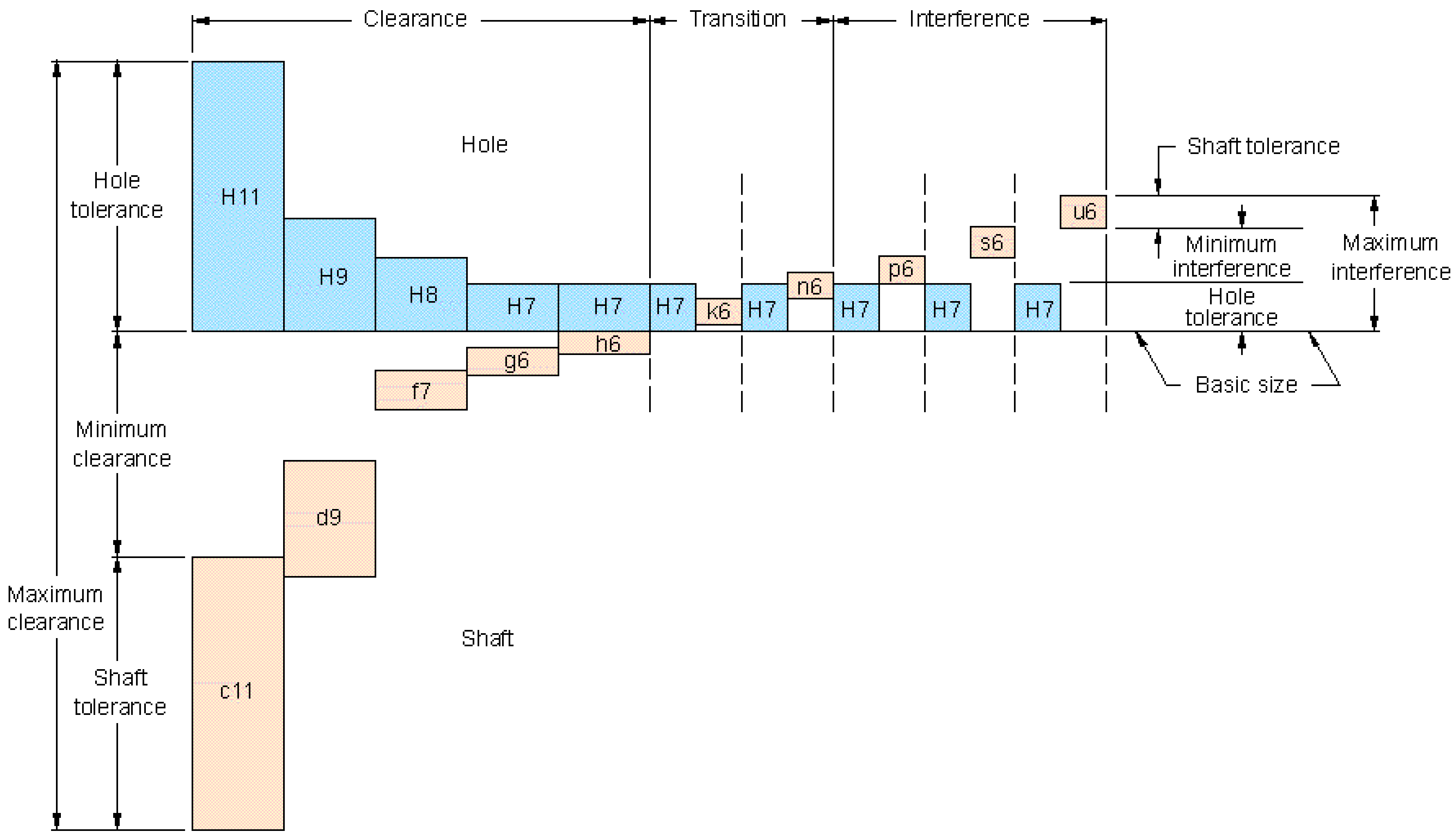
Fits and Allowances

Preferred Metric Limits and Fits

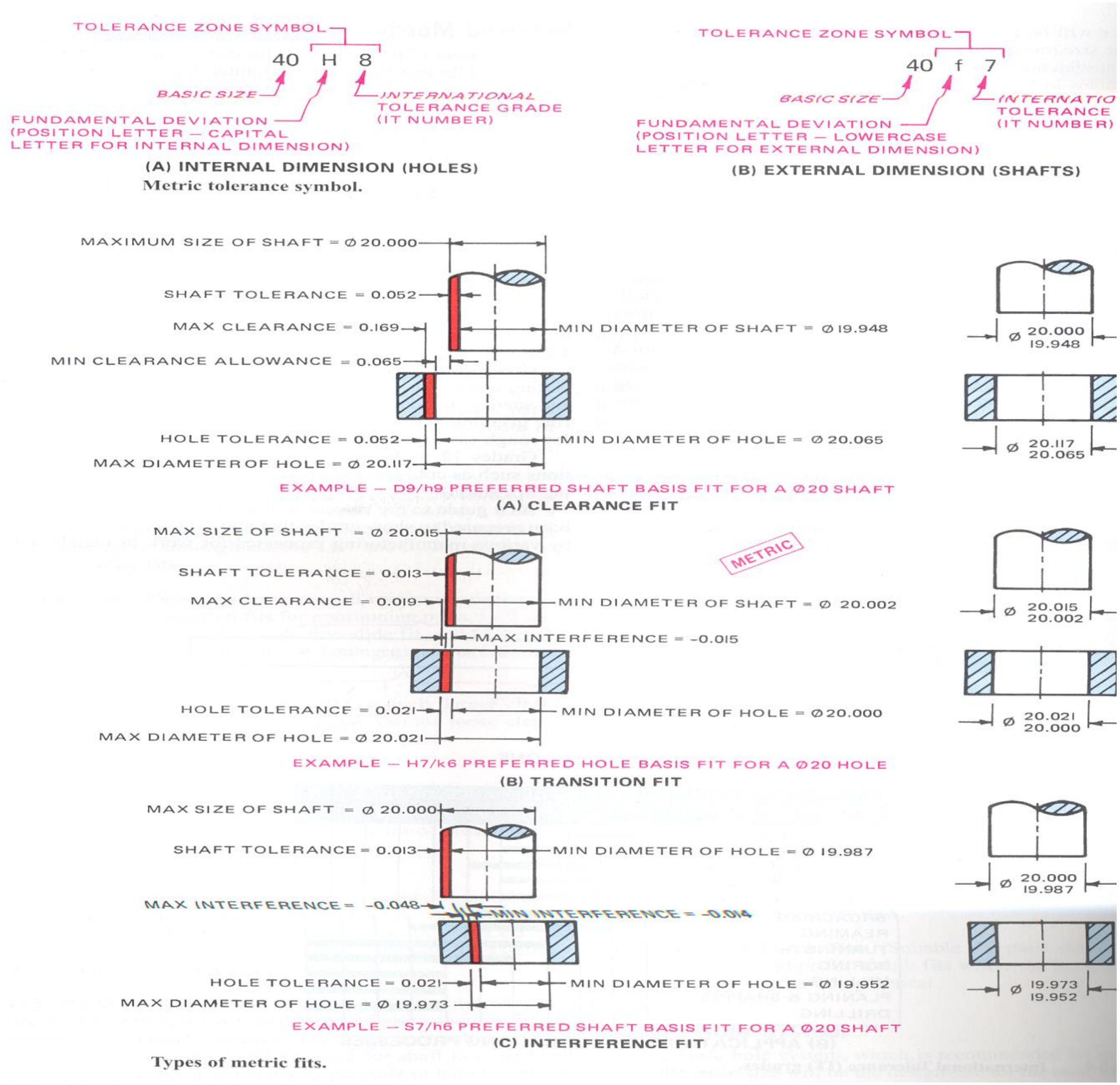


International Tolerance (IT) grades.

Hole Basis System

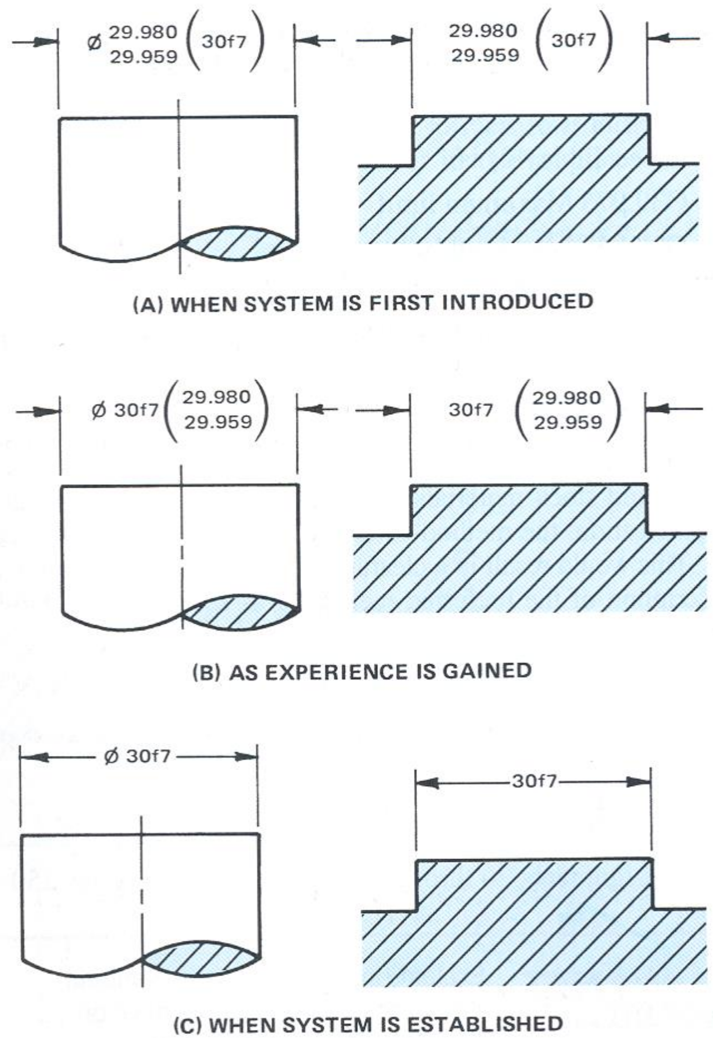


Metric Fits


TOLERANCE ZONE SYMBOL
40 H 8
BASIC SIZE
INTERNATIONAL TOLERANCE GRADE (IT NUMBER)
FUNDAMENTAL DEVIATION (POSITION LETTER — CAPITAL LETTER FOR INTERNAL DIMENSION)
(A) INTERNAL DIMENSION (HOLES)
Metric tolerance symbol.
TOLERANCE ZONE SYMBOL
40 f 7
BASIC SIZE
INTERNATIO TOLERANCE (IT NUMBER)
FUNDAMENTAL DEVIATION (POSITION LETTER — LOWERCASE LETTER FOR EXTERNAL DIMENSION)
(B) EXTERNAL DIMENSION (SHAFTS)
MAXIMUM SIZE OF SHAFT = Ø 20.000
SHAFT TOLERANCE = 0.052
MAX CLEARANCE = 0.169
MIN CLEARANCE ALLOWANCE = 0.065
MIN DIAMETER OF SHAFT = Ø 19.948
HOLE TOLERANCE = 0.052
MAX DIAMETER OF HOLE = Ø 20.117
MIN DIAMETER OF HOLE = Ø 20.065
Ø 20.000 19.948
Ø 20.117 20.065
EXAMPLE — D9/h9 PREFERRED SHAFT BASIS FIT FOR A Ø20 SHAFT
(A) CLEARANCE FIT
MAX SIZE OF SHAFT = Ø 20.015
SHAFT TOLERANCE = 0.013
MAX CLEARANCE = 0.019
MIN DIAMETER OF SHAFT = Ø 20.002
MAX INTERFERENCE = -0.015
METRIC
HOLE TOLERANCE = 0.021
MAX DIAMETER OF HOLE = Ø 20.021
MIN DIAMETER OF HOLE = Ø 20.000
Ø 20.015 20.002
Ø 20.021 20.000
EXAMPLE — H7/k6 PREFERRED HOLE BASIS FIT FOR A Ø20 HOLE
(B) TRANSITION FIT
MAX SIZE OF SHAFT = Ø 20.000
SHAFT TOLERANCE = 0.013
MIN DIAMETER OF HOLE = Ø 19.987
MAX INTERFERENCE = -0.048
MIN INTERFERENCE = -0.014
HOLE TOLERANCE = 0.021
MAX DIAMETER OF HOLE = Ø 19.973
MIN DIAMETER OF HOLE = Ø 19.952
Ø 20.000 19.987
Ø 19.973 19.952
EXAMPLE — S7/h6 PREFERRED SHAFT BASIS FIT FOR A Ø20 SHAFT
(C) INTERFERENCE FIT
Types of metric fits.

Fits and Allowances

Metric Tolerance symbol



Metric tolerance symbol application.

Fits and Allowances

Preferred Metric fits

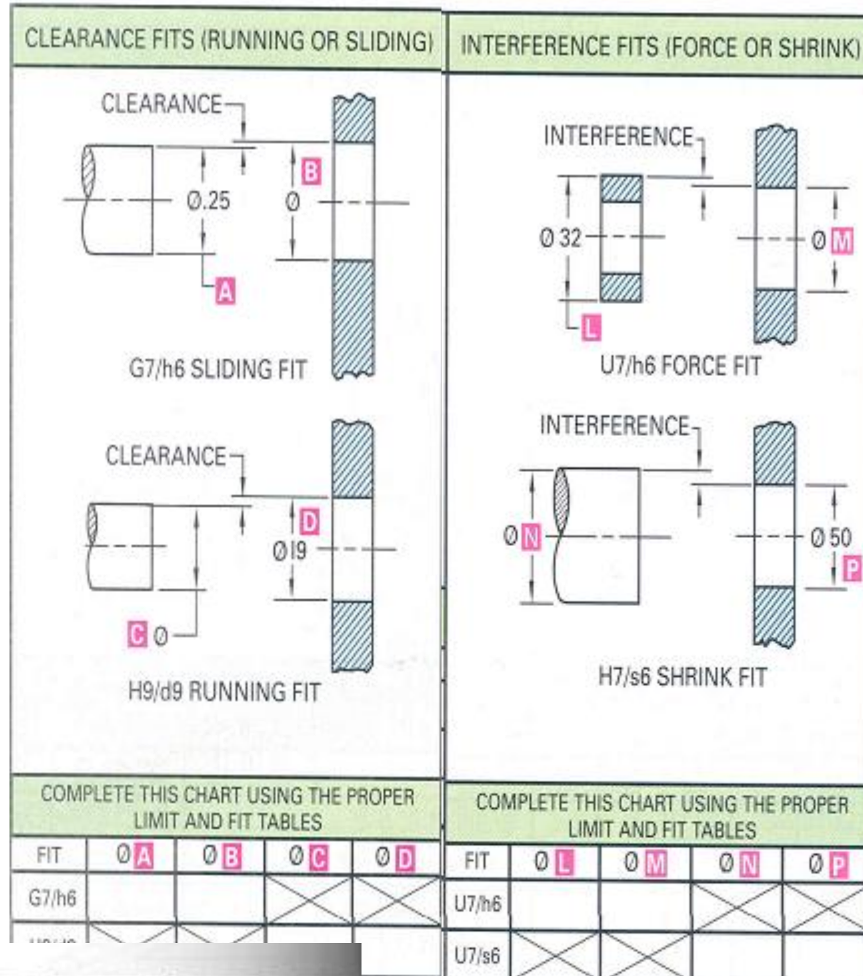
	ISO SYMBOL		
	HOLE BASIS	SHAFT BASIS	DESCRIPTION
CLEARANCE FITS	H11/c11	C11/h11	LOOSE RUNNING FIT FOR WIDE COMMERCIAL TOLERANCES OR ALLOWANCES ON EXTERNAL MEMBERS.
CLEARANCE FITS	H9/d9	D9/h9	FREE RUNNING FIT NOT FOR USE WHERE ACCURACY IS ESSENTIAL, BUT GOOD FOR LARGE TEMPERATURE VARIATIONS, HIGH RUNNING SPEEDS, OR HEAVY JOURNAL PRESSURES.
CLEARANCE FITS	H8/f7	F8/h7	CLOSE RUNNING FIT FOR RUNNING ON ACCURATE MACHINES AND FOR ACCURATE LOCATION AT MODERATE SPEEDS AND JOURNAL PRESSURES.
CLEARANCE FITS	H7/g6	G7/h6	SLIDING FIT NOT INTENDED TO RUN FREELY, BUT TO MOVE AND TURN FREELY AND LOCATE ACCURATELY.
TRANSITION FITS	H7/h6	H7/h6	LOCATIONAL CLEARANCE FIT PROVIDES SNUG FIT FOR LOCATING STATIONARY PARTS; BUT CAN BE FREELY ASSEMBLED AND DISASSEMBLED.
TRANSITION FITS	H7/k6	K7/h6	LOCATIONAL TRANSITION FIT FOR ACCURATE LOCATION, A COMPROMISE BETWEEN CLEARANCE AND INTERFERENCE.
TRANSITION FITS	H7/n6	N7/h6	LOCATIONAL TRANSITION FIT FOR MORE ACCURATE LOCATION WHERE GREATER INTERFERENCE IS PERMISSIBLE.
INTERFERENCE FITS	H7/p6	P7/h6	LOCATIONAL INTERFERENCE FIT FOR PARTS REQUIRING RIGIDITY AND ALIGNMENT WITH PRIME ACCURACY OF LOCATION BUT WITHOUT SPECIAL BORE PRESSURE REQUIREMENTS.
INTERFERENCE FITS	H7/s6	S7/h6	MEDIUM DRIVE FIT FOR ORDINARY STEEL PARTS OR SHRINK FITS ON LIGHT SECTIONS, THE TIGHTEST FIT USABLE WITH CAST IRON.
INTERFERENCE FITS	H7/u6	U7/h6	FORCE FIT SUITABLE FOR PARTS WHICH CAN BE HIGHLY STRESSED OR FOR SHRINK FITS WHERE THE HEAVY PRESSING FORCES REQUIRED ARE IMPRACTICAL.

↑ MORE CLEARANCE — MORE INTERFERENCE ↓

Description of preferred metric fits.

Fits and Allowances Example

Adobe Acrobat Document

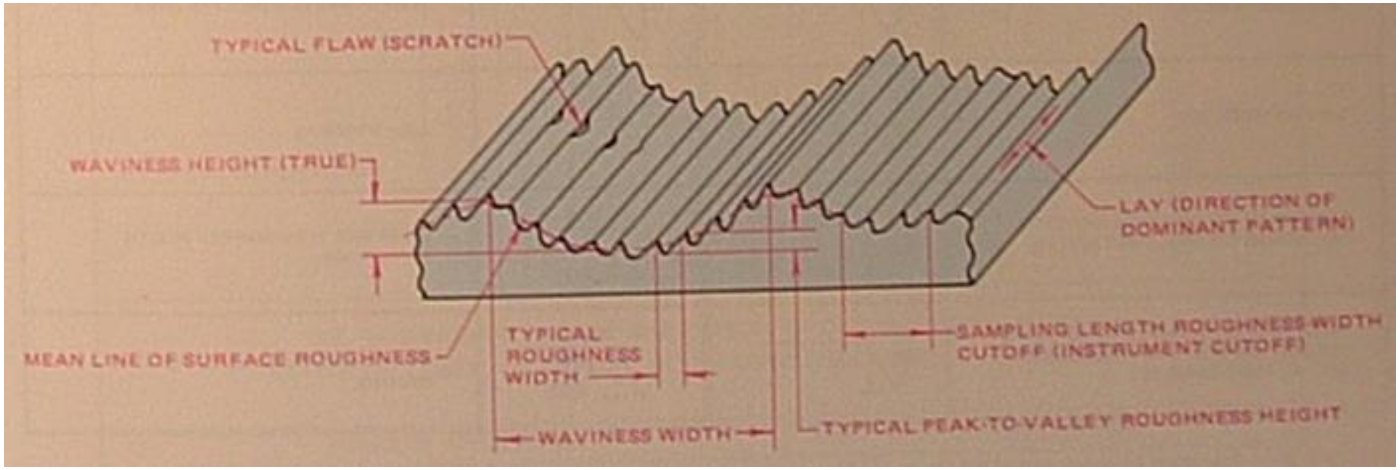


FIT	ØA	ØB	ØC	ØD
G7/h6	25.000 24.987	25.028 25.007		
H9/d9			18.935 18.883	19.052 19.000

FIT	ØL	ØM	ØN	Ø
U7/h6	32.000 31.987	31.960 31.939		
U7/s6			50.059 50.043	50.025 50.000

Surface Texture

Surface texture characterisitcs



Roughness - Roughness consists of the finer irregularities in the surface texture, usually including those that result from the inherent action of the production process. These include traverse feed marks and other irregularities within the limits of the roughness-width cutoff.

Roughness-Height Value - Roughness-height value is rated as the arithmetic average (AA) deviation expressed in micro inches or micrometers measured normal to the center line. The ISO and many European countries use the term CLA (center line average) in lieu of AA. Both have the same meaning.

Roughness Spacing - Roughness spacing is the distance parallel to the nominal surface between successive peaks or ridges that constitute the predominant pattern of the roughness. Roughness spacing is rated in inches or millimeters.

Surface Texture

Surface texture characterisitcs

Roughness Spacing - Roughness spacing is the distance parallel to the nominal surface between successive peaks or ridges that constitute the predominant pattern of the roughness. Roughness spacing is rated in inches or millimeters.

Roughness-Width Cutoff - The greatest spacing of repetitive surface irregularities is included in the measurement of average roughness height. Roughness-width cutoff is rated in inches or millimeters and must always be greater than the roughness width in order to obtain the total roughness-height rating.

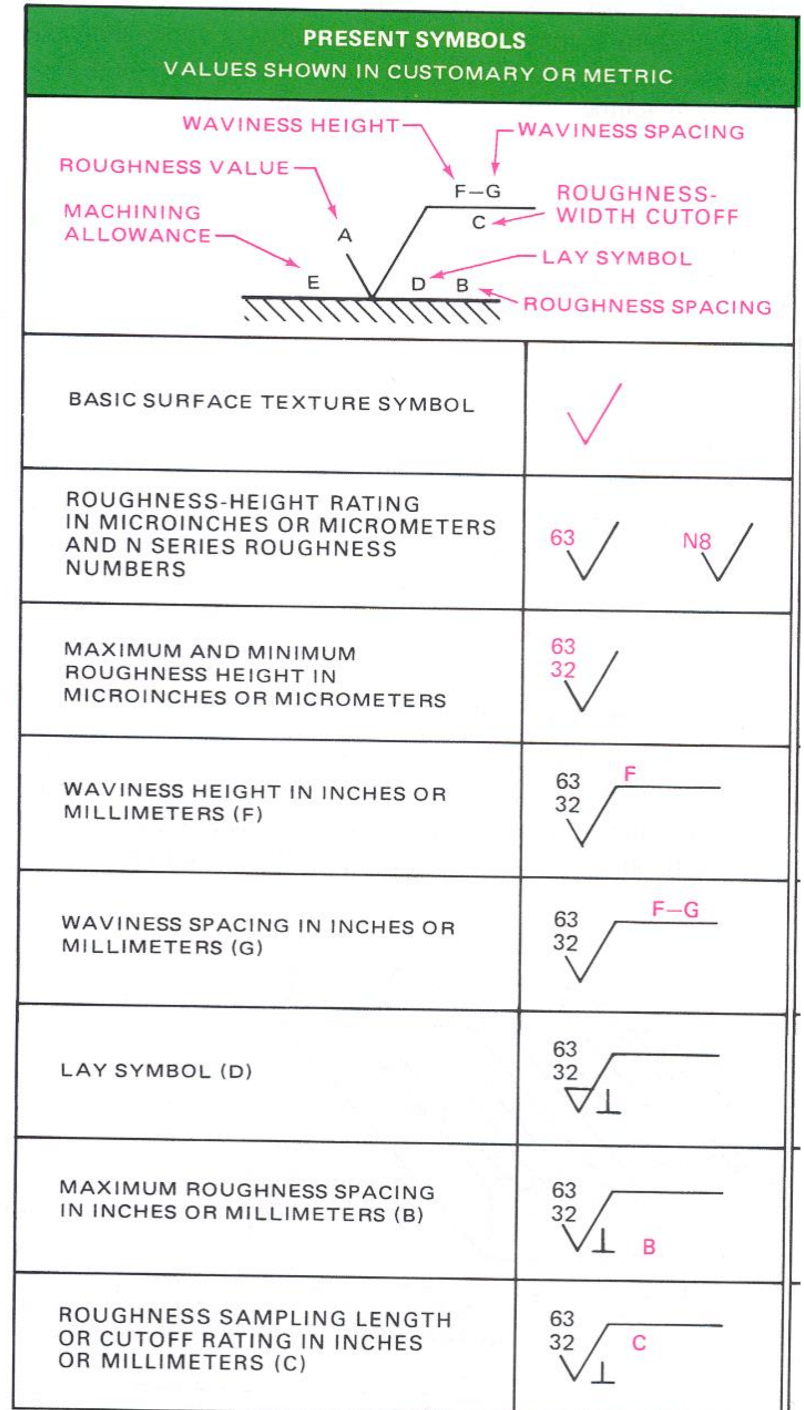
Waviness - Waviness is usually the most widely spaced of the surface texture components and normally is wider than the roughness-width cutoff.

Lay - The direction of the predominant surface pattern, ordinarily determined by the production method used, is the lay.

Flaws - Flaws are irregularities that occur at one place or at relatively infrequent or widely varying intervals in a surface. 0

Surface Texture

Surface texture symbols

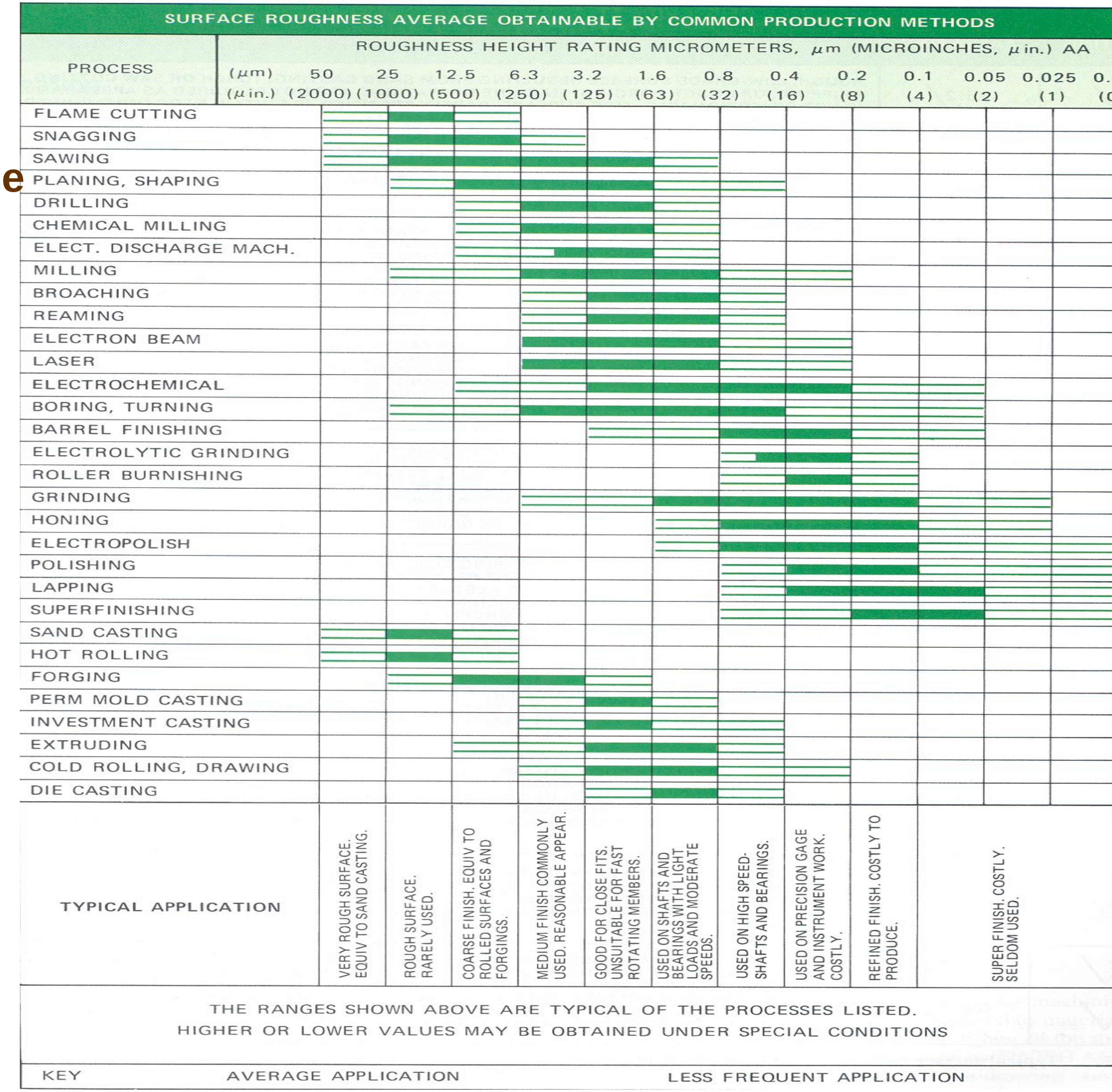


PRESENT SYMBOLS

VALUES SHOWN IN CUSTOMARY OR METRIC

BASIC SURFACE TEXTURE SYMBOL	
ROUGHNESS-HEIGHT RATING IN MICROINCHES OR MICROMETERS AND N SERIES ROUGHNESS NUMBERS	63 N8
MAXIMUM AND MINIMUM ROUGHNESS HEIGHT IN MICROINCHES OR MICROMETERS	63 32
WAVINESS HEIGHT IN INCHES OR MILLIMETERS (F)	63 32 F
WAVINESS SPACING IN INCHES OR MILLIMETERS (G)	63 32 F—G
LAY SYMBOL (D)	63 32 ⊥
MAXIMUM ROUGHNESS SPACING IN INCHES OR MILLIMETERS (B)	63 32 ⊥ B
ROUGHNESS SAMPLING LENGTH OR CUTOFF RATING IN INCHES OR MILLIMETERS (C)	63 32 C ⊥

Roughness range for common production methods



Typical surface Roughness height applications

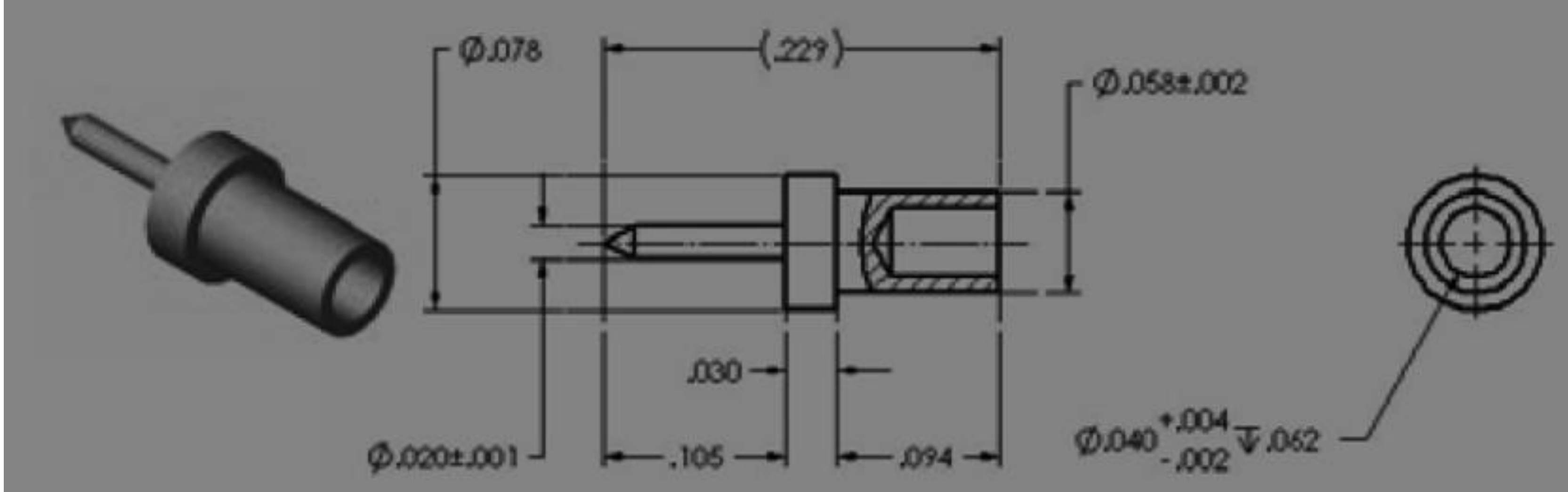
MICROINCHES AA RATING	MICROMETERS AA RATING	APPLICATION
1000√	25.2√	ROUGH, LOW-GRADE SURFACE RESULTING FROM SAND CASTING, TORCH OR SAW CUTTING, CHIPPING, OR ROUGH FORGING. MACHINE OPERATIONS ARE NOT REQUIRED AS APPEARANCE IS NOT OBJECTIONABLE. THIS SURFACE, RARELY SPECIFIED, IS SUITABLE FOR UNMACHINED CLEARANCE AREAS ON ROUGH CONSTRUCTION ITEMS.
500√	12.5√	ROUGH, LOW-GRADE SURFACE RESULTING FROM HEAVY CUTS AND COARSE FEEDS IN MILLING, TURNING, SHAPING, BORING, AND ROUGH FILING, DISC GRINDING, AND SNAGGING. IT IS SUIT-ABLE FOR CLEARANCE AREAS ON MACHINERY, JIGS, AND FIXTURES. SAND CASTING OR ROUGH FORGING PRODUCES THIS SURFACE.
250√	6.3√	COARSE PRODUCTION SURFACES, FOR UNIMPORTANT CLEARANCE AND CLEAN-UP OPERATIONS, RESULTING FROM COARSE SURFACE GRIND, ROUGH FILE, DISC GRIND, RAPID FEEDS IN TURN-ING, MILLING, SHAPING, DRILLING, BORING, GRINDING, ETC., WHERE TOOL MARKS ARE NOT OBJECTIONABLE. THE NATURAL SURFACES OF FORGINGS, PERMANENT MOLD CASTINGS, EXTRU-SIONS, AND ROLLED SURFACES ALSO PRODUCE THIS ROUGHNESS. IT CAN BE PRODUCED ECONOMICALLY AND IS USED ON PARTS WHERE STRESS REQUIREMENTS, APPEARANCE, AND CONDITIONS OF OPERATIONS AND DESIGN PERMIT.
125√	3.2√	THE ROUGHEST SURFACE RECOMMENDED FOR PARTS SUBJECT TO LOADS, VIBRATION, AND HIGH STRESS. IT IS ALSO PERMITTED FOR BEARING SURFACES WHEN MOTION IS SLOW AND LOADS LIGHT OR INFREQUENT. IT IS A MEDIUM COMMERCIAL MACHINE FINISH PRODUCED BY RELATIVELY HIGH SPEEDS AND FINE FEEDS TAKING LIGHT CUTS WITH SHARP TOOLS. IT MAY BE ECONOMICALLY PRODUCED ON LATHES, MILLING MACHINES, SHAPERS, GRINDERS, ETC., OR ON PERMANENT MOLD CASTINGS, DIE CASTINGS, EXTRUSIONS, AND ROLLED SURFACES.
63√	1.6√	A GOOD MACHINE FINISH PRODUCED UNDER CONTROLLED CONDITIONS USING RELATIVELY HIGH SPEEDS AND FINE FEEDS TO TAKE LIGHT CUTS WITH SHARP CUTTERS. IT MAY BE SPECIFIED FOR CLOSE FITS AND USED FOR ALL STRESSED PARTS, EXCEPT FAST-ROTATING SHAFTS, AXLES, AND PARTS SUBJECT TO SEVERE VIBRATION OR EXTREME TENSION. IT IS SATISFACTORY FOR BEARING SURFACES WHEN MOTION IS SLOW AND LOADS LIGHT OR IN-FREQUENT. IT MAY ALSO BE OBTAINED ON EXTRUSIONS, ROLLED SURFACES, DIE CASTINGS, AND PERMANENT MOLD CASTINGS WHEN RIGIDLY CONTROLLED.
32√	0.8√	A HIGH-GRADE MACHINE FINISH REQUIRING CLOSE CONTROL WHEN PRODUCED BY LATHES, SHAPERS, MILLING MACHINES, ETC., BUT RELATIVELY EASY TO PRODUCE BY CENTERLESS, CYLINDRICAL, OR SURFACE GRINDERS. ALSO, EXTRUDING, ROLLING, OR DIE CASTING MAY PRODUCE A COMPARABLE SURFACE WHEN RIGIDLY CONTROLLED. THIS SURFACE MAY BE SPECIFIED IN PARTS WHERE STRESS CONCENTRATION IS PRESENT. IT IS USED FOR BEARINGS WHEN MOTION IS NOT CONTINUOUS AND LOADS ARE LIGHT. WHEN FINER FINISHES ARE SPECIFIED, PRODUCTION COSTS RISE RAPIDLY; THEREFORE, SUCH FINISHES MUST BE ANALYZED CAREFULLY.
16√	0.4√	A HIGH-QUALITY SURFACE PRODUCED BY FINE CYLINDRICAL GRINDING, EMERY BUFFING, COARSE HONING, OR LAPPING. IT IS SPECIFIED WHERE SMOOTHNESS IS OF PRIMARY IMPORTANCE, SUCH AS RAPIDLY ROTATING SHAFT BEARINGS, HEAVILY LOADED BEARINGS, AND EXTREME TENSION MEMBERS.
8√	0.2√	A FINE SURFACE PRODUCED BY HONING, LAPPING, OR BUFFING. IT IS SPECIFIED WHERE PACK-INGS AND RINGS MUST SLIDE ACROSS THE DIRECTION OF THE SURFACE GRAIN, MAINTAINING OR WITHSTANDING PRESSURES, OR FOR INTERIOR HONED SURFACES OF HYDRAULIC CYLINDERS. IT MAY ALSO BE REQUIRED IN PRECISION GAGES AND INSTRUMENT WORK, OR SENSITIVE-VALUE SURFACES, OR ON RAPIDLY ROTATING SHAFTS AND ON BEARINGS WHERE LUBRICATION IS NOT DEPENDABLE.
4√	0.1√	A COSTLY REFINED SURFACE PRODUCED BY HONING, LAPPING, AND BUFFING. IT IS SPECIFIED ONLY WHEN THE REQUIREMENTS OF DESIGN MAKE IT MANDATORY. IT IS REQUIRED IN INSTRU-MENT WORK, GAGE WORK, AND WHERE PACKINGS AND RINGS MUST SLIDE ACROSS THE DIREC-TION OF SURFACE GRAIN, SUCH AS ON CHROME-PLATED PISTON RODS, ETC., WHERE LUBRICATION IS NOT DEPENDABLE.
2√ 1√	0.05√ 0.025√	COSTLY REFINED SURFACES PRODUCED ONLY BY THE FINEST OF MODERN HONING, LAPPING, BUFFING, AND SUPERFINISHING EQUIPMENT. THESE SURFACES MAY HAVE A SATIN OR HIGHLY POLISHED APPEARANCE DEPENDING ON THE FINISHING OPERATION AND MATERIAL. THESE SUR-FACES ARE SPECIFIED ONLY WHEN DESIGN REQUIREMENTS MAKE IT MANDATORY. THEY ARE SPECIFIED ON FINE OR SENSITIVE INSTRUMENT PARTS OR OTHER LABORATORY ITEMS, AND CERTAIN GAGE SURFACES, SUCH AS PRECISION GAGE BLOCKS.

Tolerance Types

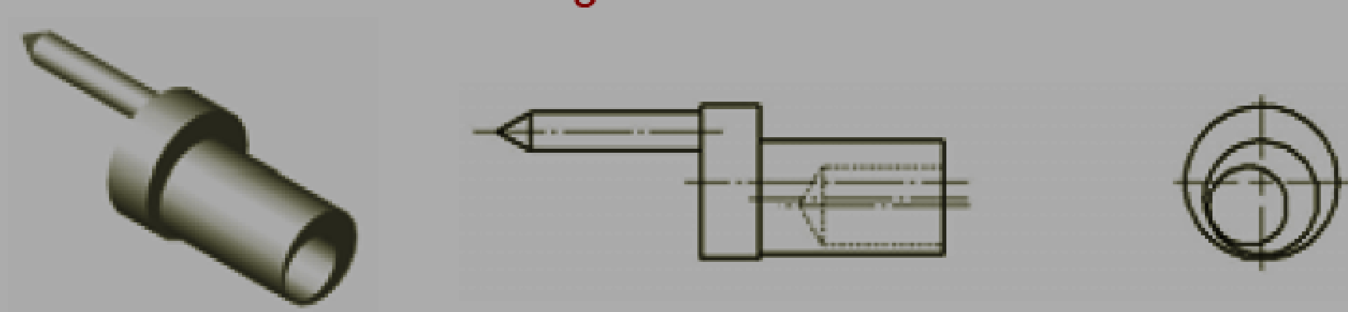
- Linear Tolerance
  - Size
- Geometric Tolerance
  - Form
  - Profile
  - Location
  - Orientation
  - Runout

Reason for GD & T

As drawn



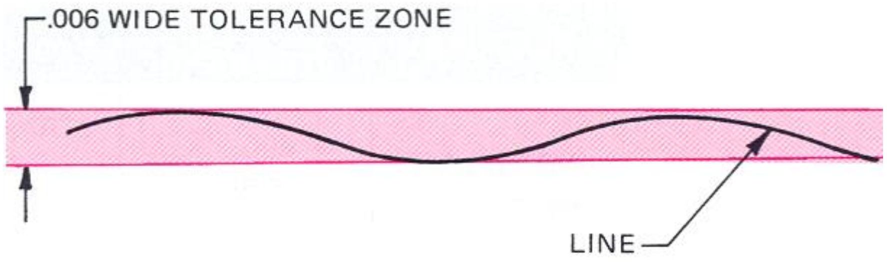
As manufactured –
Without geometric tolerance control



Geometric tolerancing

Rationale

- The tolerances and limits of size that are seen so far do not specifically control other variations – form, orientation, and position to some extent
- To meet functional requirement, geometric tolerances need to be added to control limits of size, but also to ensure that limits of form orientation and position are maintained
- A geometric tolerance is the maximum permissible variation of form, profile, orientation, location, and runout from that indicated or specified on a drawing.
- The tolerance value represents the width or diameter of the tolerance one within which the point, line or surface or the feature must lie.



. 16-2-1 Tolerance zone for straightness of a line.

Geometric tolerancing

Points line and surfaces

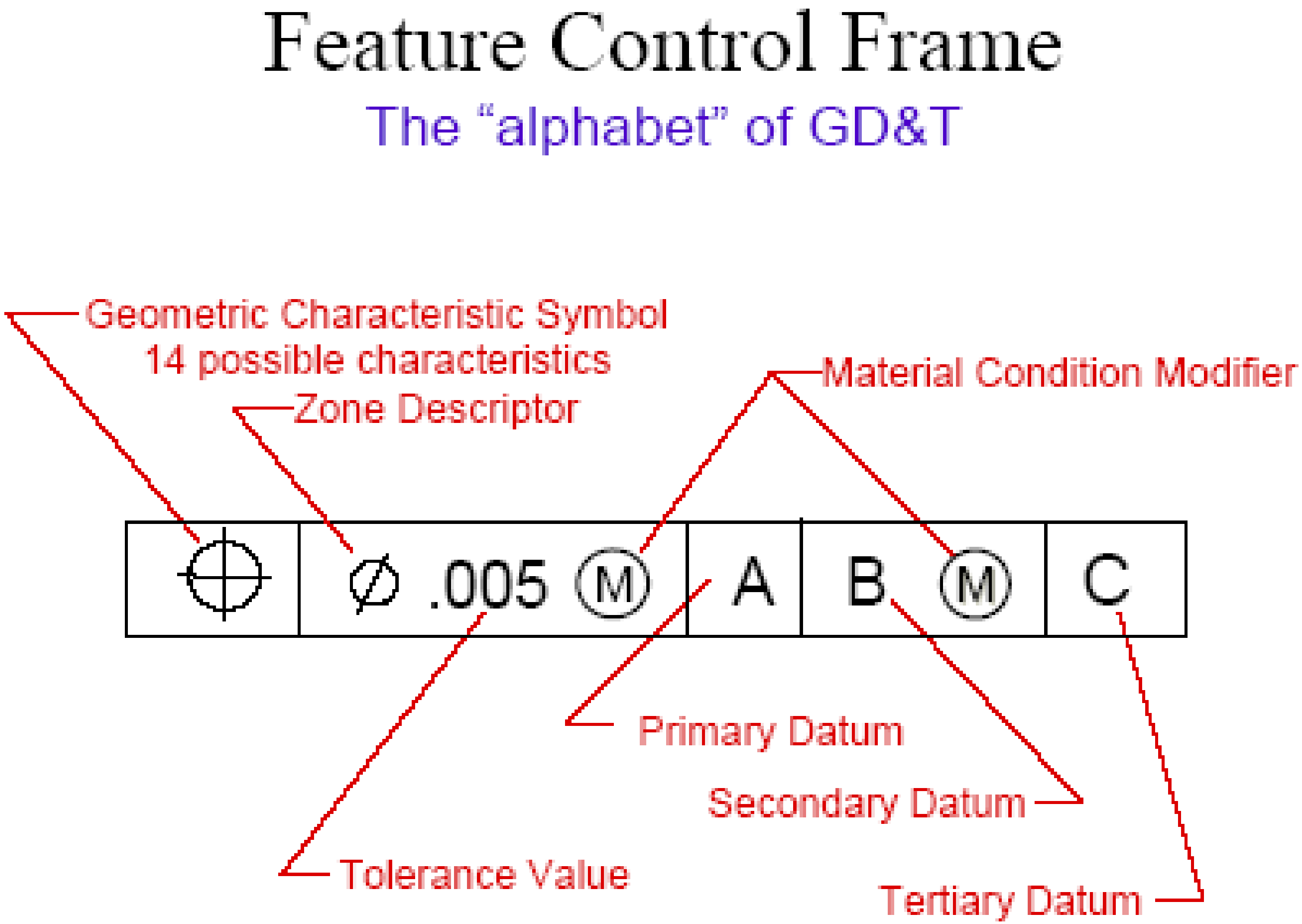
- Engineering surfaces may be flat, cylindrical conical etc or some irregular shape
- But line or surface is evaluated dimensionally by making measurements at various points along the length
- Geometric tolerances are concerned with points and lines, and surfaces are considered as series of lines in 2 or more directions
- Points have only position and no size, so position is the only thing requiring control.
- But lines must be controlled for from, orientation and location

FEATURE	TYPE OF TOLERANCE	CHARACTERISTIC	SYMBOL	SEE UNIT
INDIVIDUAL FEATURES	FORM	STRAIGHTNESS	⏤	16-2, 16-5
		FLATNESS	⏥	16-3
		CIRCULARITY (ROUNDNESS)	○	16-12
		CYLINDRICITY	⌭	
INDIVIDUAL OR RELATED FEATURES	PROFILE	PROFILE OF A LINE	⌒	16-13
		PROFILE OF A SURFACE	⌓	
RELATED FEATURES	ORIENTATION	ANGULARITY	∠	16-7, 16-8
		PERPENDICULARITY	⊥	
		PARALLELISM	∥	
	LOCATION	POSITION	⌖	16-9
		CONCENTRICITY	◎	16-14
		SYMMETRY	⌯	16-14
	RUNOUT	CIRCULAR RUNOUT	↗ *	16-14
		TOTAL RUNOUT	⌰ *	
SUPPLEMENTARY SYMBOLS		MAXIMUM MATERIAL CONDITION	Ⓜ	16-4
		LEAST MATERIAL CONDITION	Ⓛ	
		PROJECTED TOLERANCE ZONE	Ⓟ	16-9
		BASIC DIMENSION	[XX]	16-9, 16-11
		DATUM FEATURE	[A]	16-5
		DATUM TARGET	Ø.50 / A2	16-11

* MAY BE FILLED IN

Fig. 16-2-2 Geometric characteristic symbols.

Geometric Tolerances



Notes Vs. Symbols

- A symbol has uniform meaning
- Symbols are an international language
  - No translation necessary
- Symbols are more legible
  - Drawings are often copied w/ lower quality
- Symbols are compact – notes take more time & space

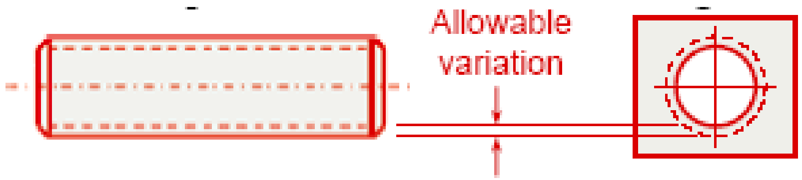
SURFACE B PERPENDICULAR
TO DATUM A WITHIN .001

=

⊥	.001	A

Material Condition

- Maximum Material Condition (MMC) Ⓜ
  - Largest possible external feature (e.g. shaft)
  - Smallest possible internal feature (e.g. hole)



- Least Material Condition (LMC) Ⓛ
  - Smallest possible external feature
  - Largest possible internal feature
- Regardless of Feature Size (RFS)
  - Default condition if not specified

Material Condition

Applicability of RFS, MMC and LMC

- If freedom of assembly in the mating parts is the key, MMC is leas preferred
- If in fig MMC is applied, both the pin and hole will be at .250in
- Theoretically, this will assemble provided there are no variations in form or orientation – and a perfect centre distance of 2in
- If the same is in LMC they are .247 and .253 this gives more play while assembly. Like the centre distance can be varied slightly or there can be form tolerances and orientation tolerance

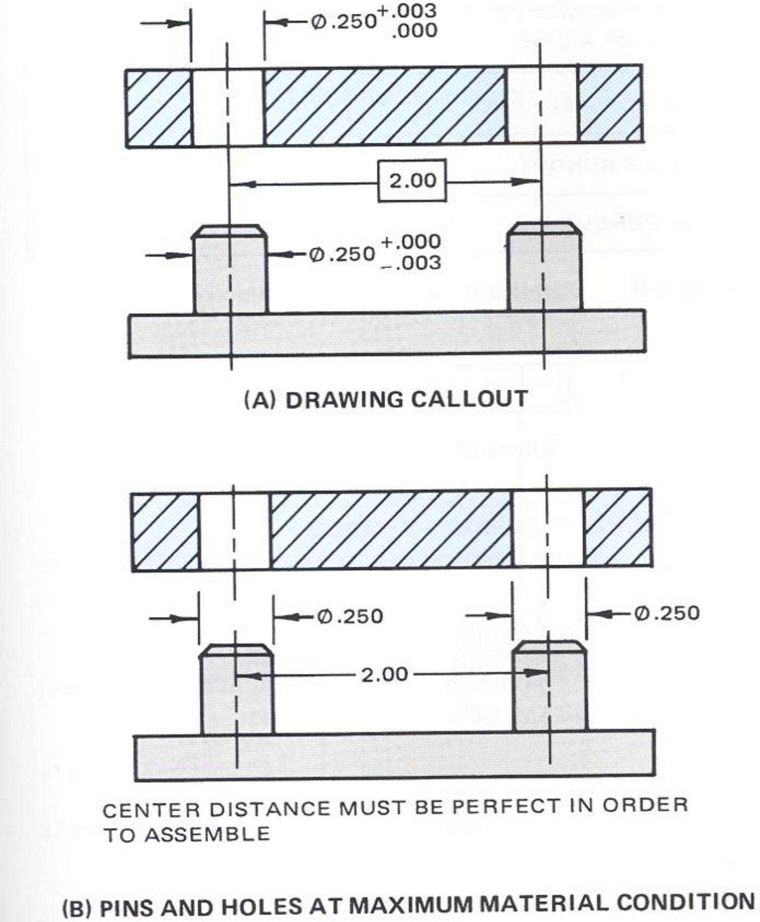


Fig. 16-4-6 Effect of location.

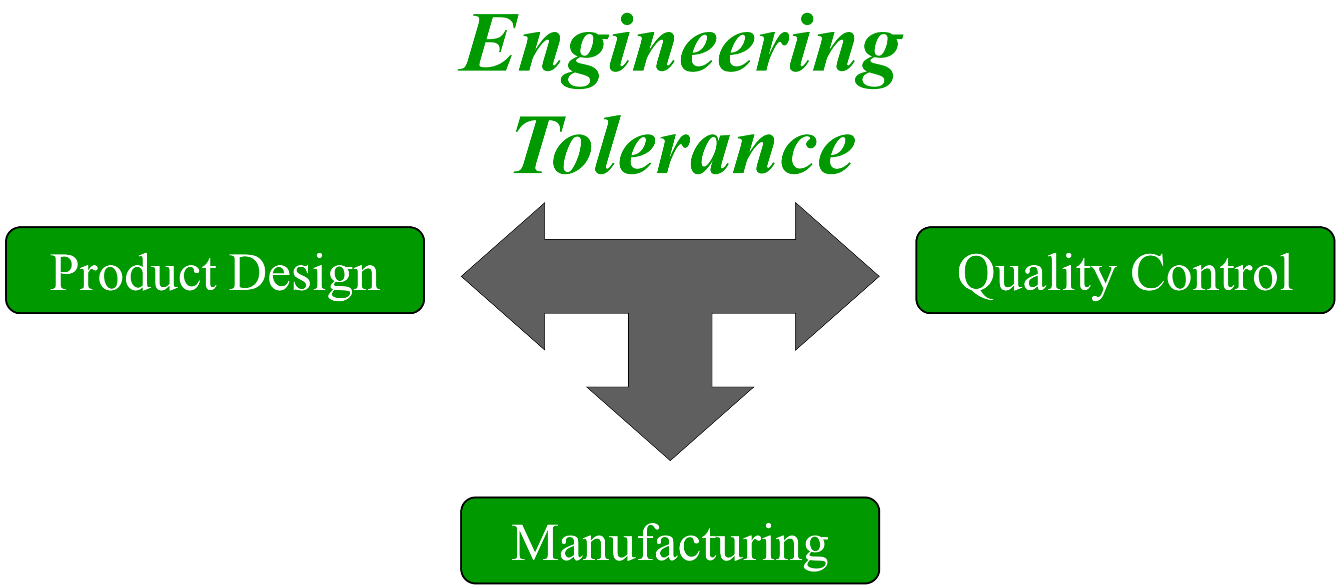
5 Categories of Control

Description	Type of Control	Geometric Characteristic	Symbol
Individual Features	Form	Straightness	⏤
		Flatness	⏥
		Circularity	○
		Cylindricity	⌭
Individual or Related	Profile	Profile of a Line	⌒
		Profile of a Surface	⌓
Related Features	Orientation	Angularity	∠
		Perpendicularity	⊥
		Parallelism	∥
	Location	Position	⌖
		Concentricity	◎
		Symmetry	⌯
	Runout	Circular Runout	↗
		Total Runout	⌰

Design Specs and Tolerance

- Develop from quest for production quality and efficiency
- Early tolerances support design's basic function
- Mass production brought interchangeability
- Integrate design and manufacturing tolerances

Affected Areas

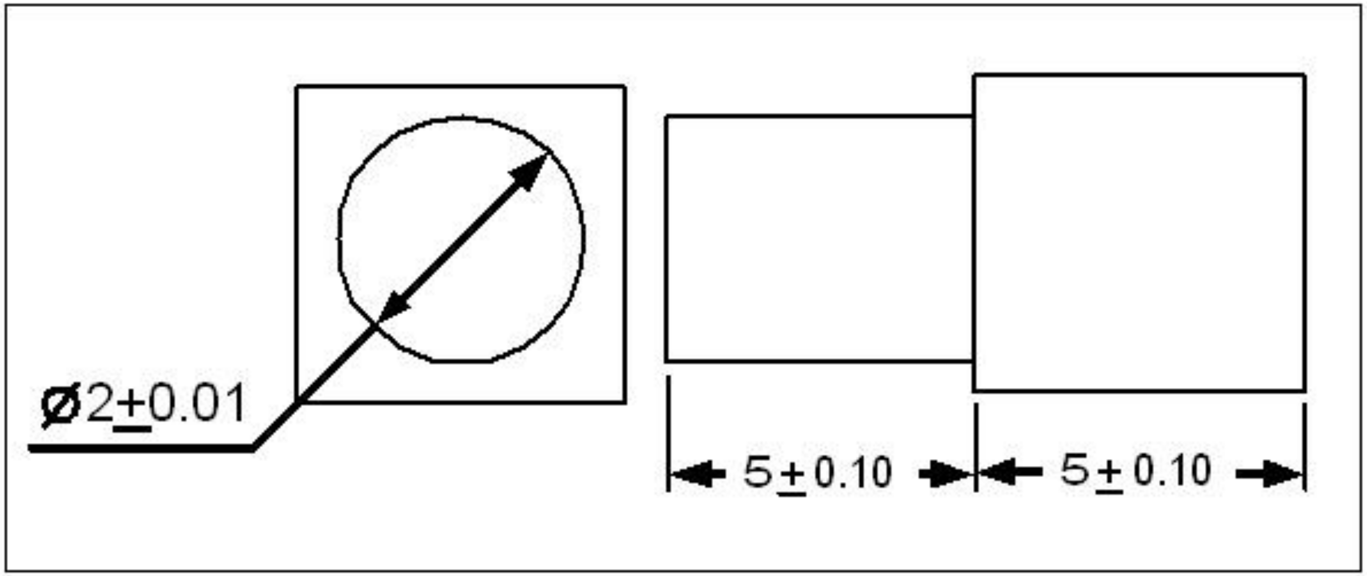


- Product Design - to design with consideration for the maximum possible working tolerances compatible with the functional design requirements.
- Manufacturing - to bringing the design into a physical entity with consideration for process methodologies, capabilities and economics.
- Quality control - measuring and evaluating the final product to ensure the integrity of the product tolerance.

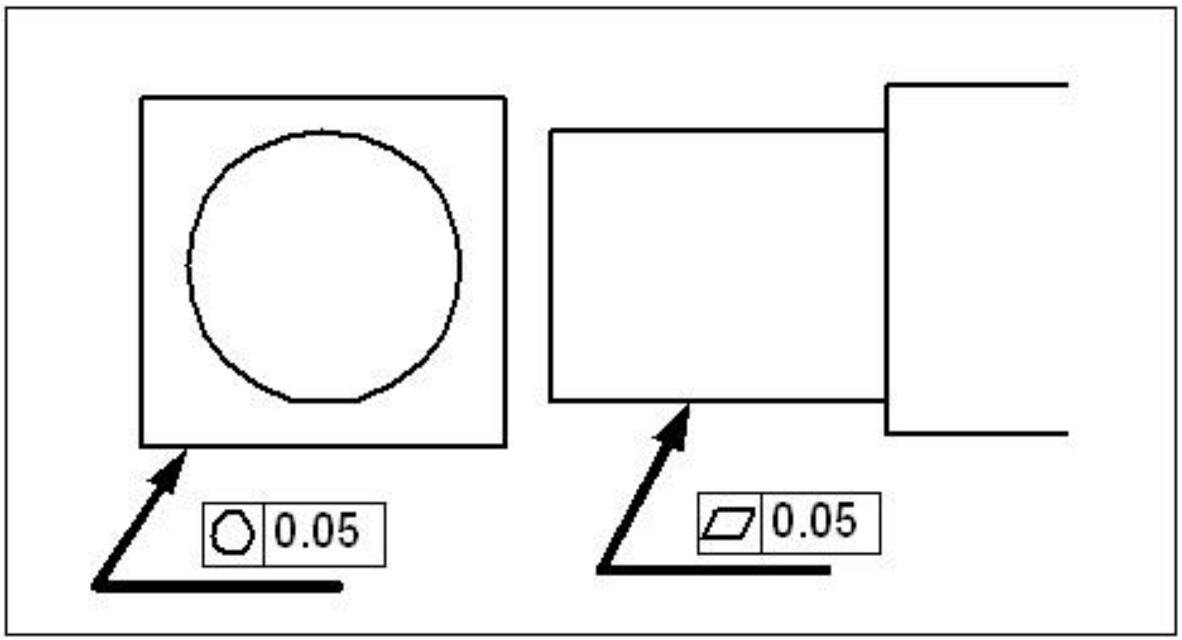
Geometric Dimensions

- GD&T are used by the designer to specify part requirements on a design.
- It is done through drawings using universal symbols and methods for specific functions and values mainly to
  - Accurately communicate the function of part
  - Provide uniform clarity in drawing delineation and interpretation
  - Provide maximum production tolerance

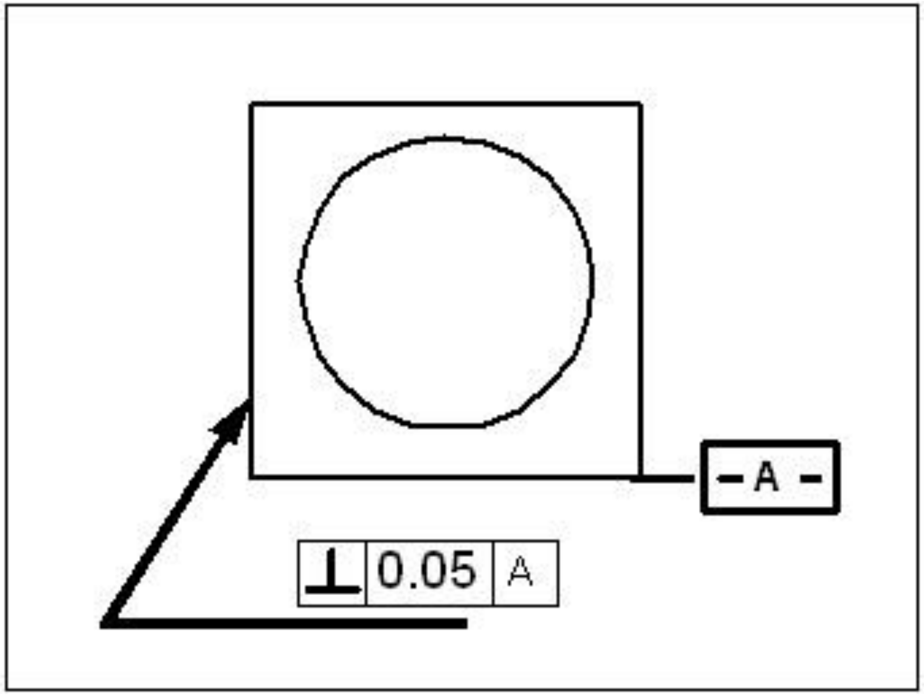
Size Tolerances



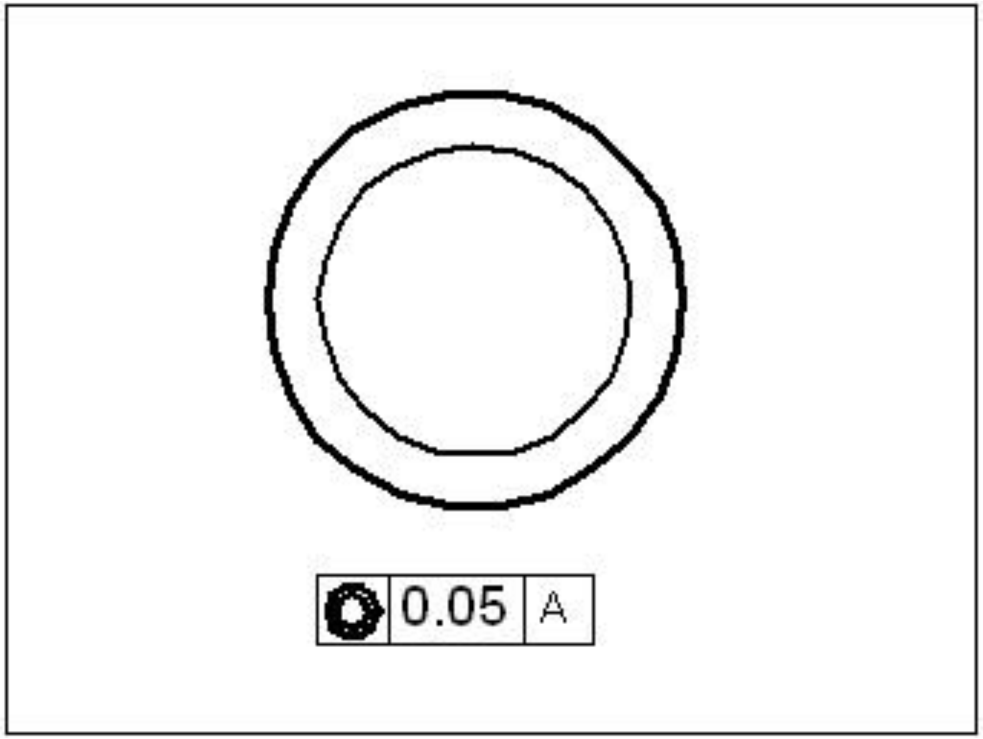
Form Tolerances



Orientation Tolerances



Location Tolerances



Positional tolerancing

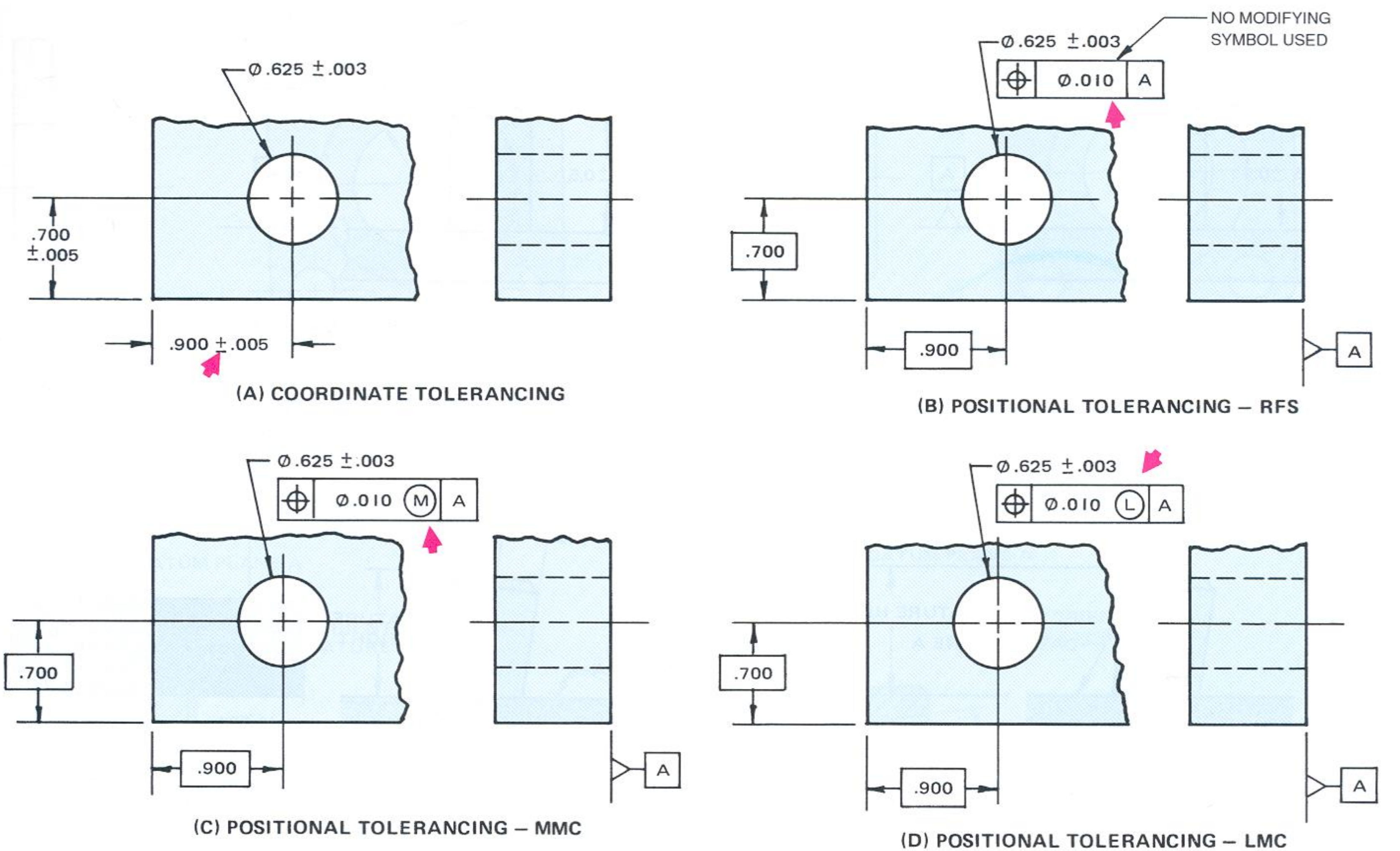
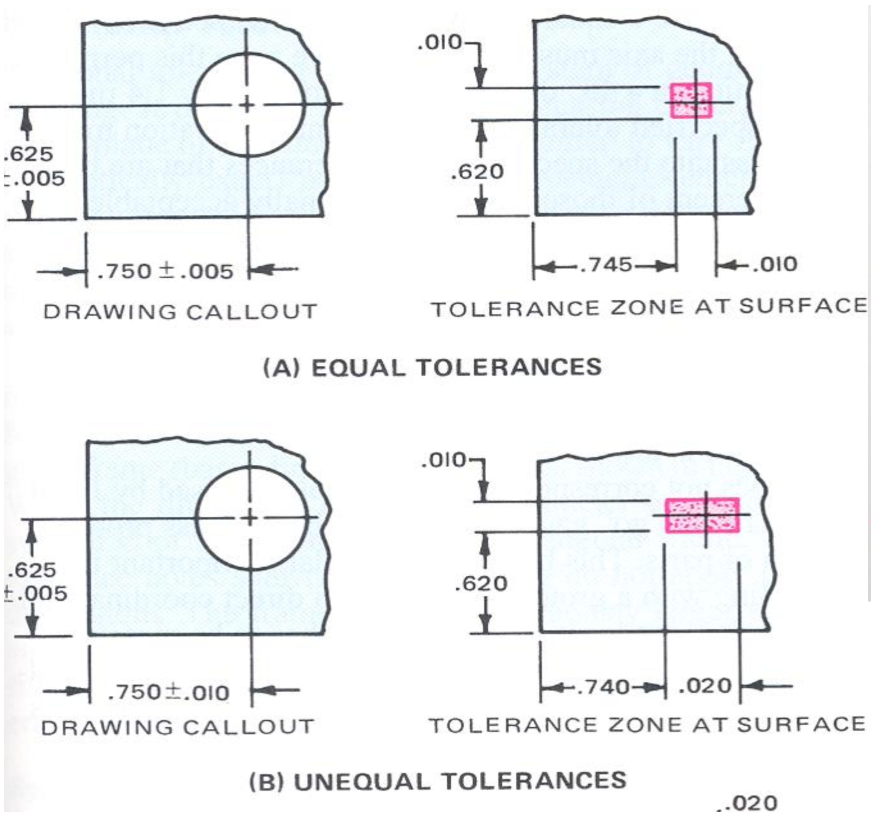


Fig. 16-9-1 Comparison of tolerancing methods.

Positional tolerancing

Coordinate Tolerancing

Coordinate dimensions and tolerances may be applied to the location of a single hole, as shown in Fig. 16-9-2. It should be noted that the tolerance zone extends for the full depth of the hole, that is, the whole length of the axis 16-9-3.



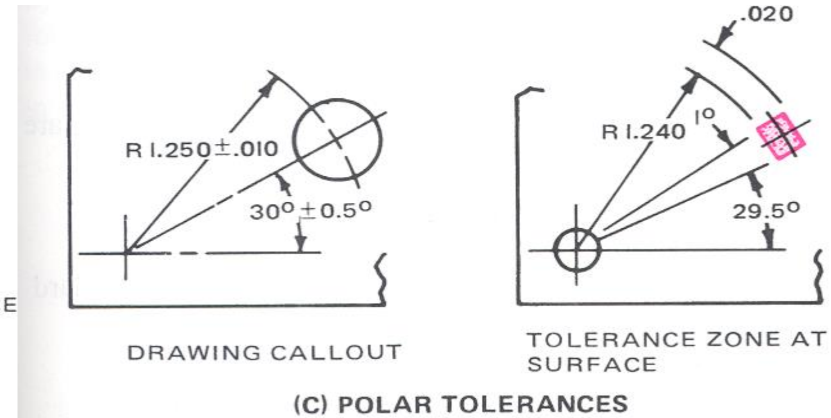
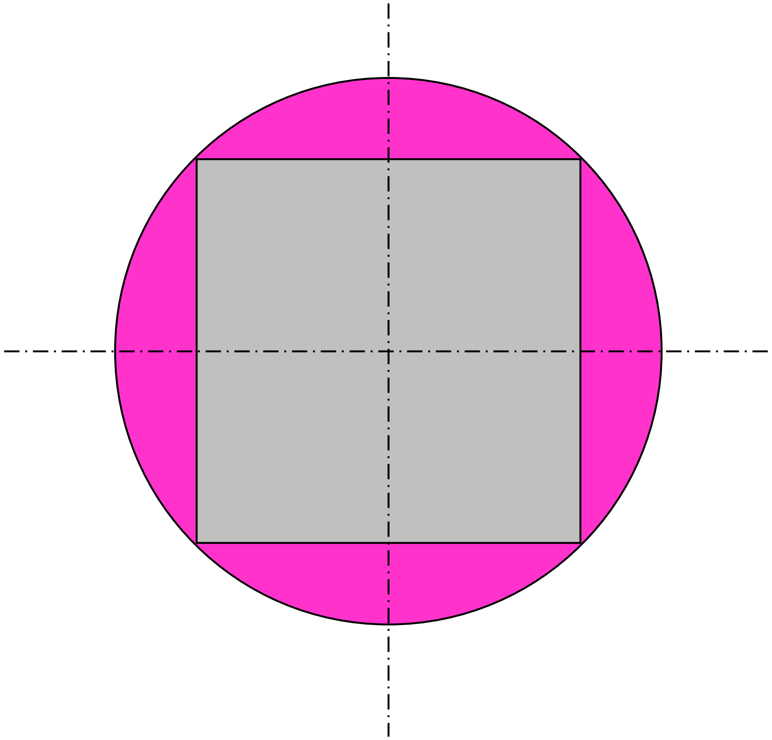


Fig. 16-9-2 Tolerance zones for coordinate tolerancing. (See also below.)

Positional tolerancing

Comparison between positional and coordinate tolerancing

- For the same tolerance values, the positional tolerance gives more room compared to coordinate tolerancing
- This is due the fact that in coordinate method the diagonal becomes responsible for limits.
- If we use the same limits in positional tolerancing we will get more parts accepted without any effect on assembly



Correlative tolerances

Runout

- Runout is a composite tolerance used to control the functional relationship of one or more features of a part to a datum axis.
- Each feature is within the runout tolerance when rotated around the datum axis
- It is the full indicator movement (FIM) in inspection terminology

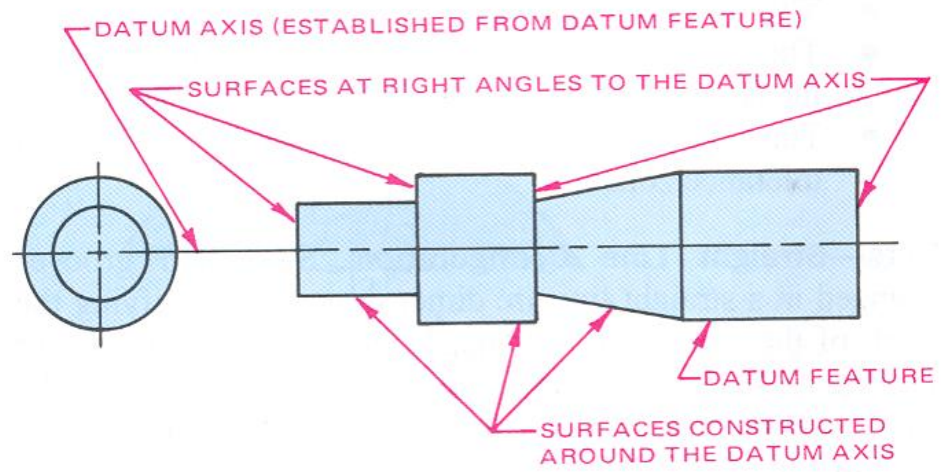


Fig. 16-14-13 Features applicable to runout tolerancing.

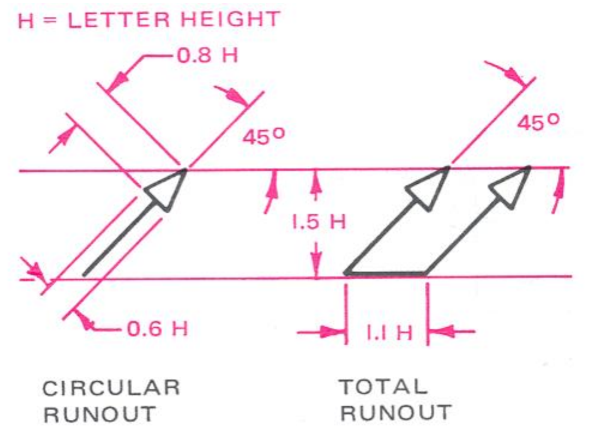


Fig. 16-14-14 Runout symbols.

Correlative tolerances

Circular Runout

- provides control of circular elements of a surface (it does not control in any other direction)
- The tolerance is usually applied at any usual measuring position when part is rotated 360°
- When applied to surfaces around the axis it controls circularity and coaxiality
- When done at surfaces 90° to axis it controls wobble at all positions
- When runout tolerance applies to specific portion, it is indicted by chain line and basic dimension

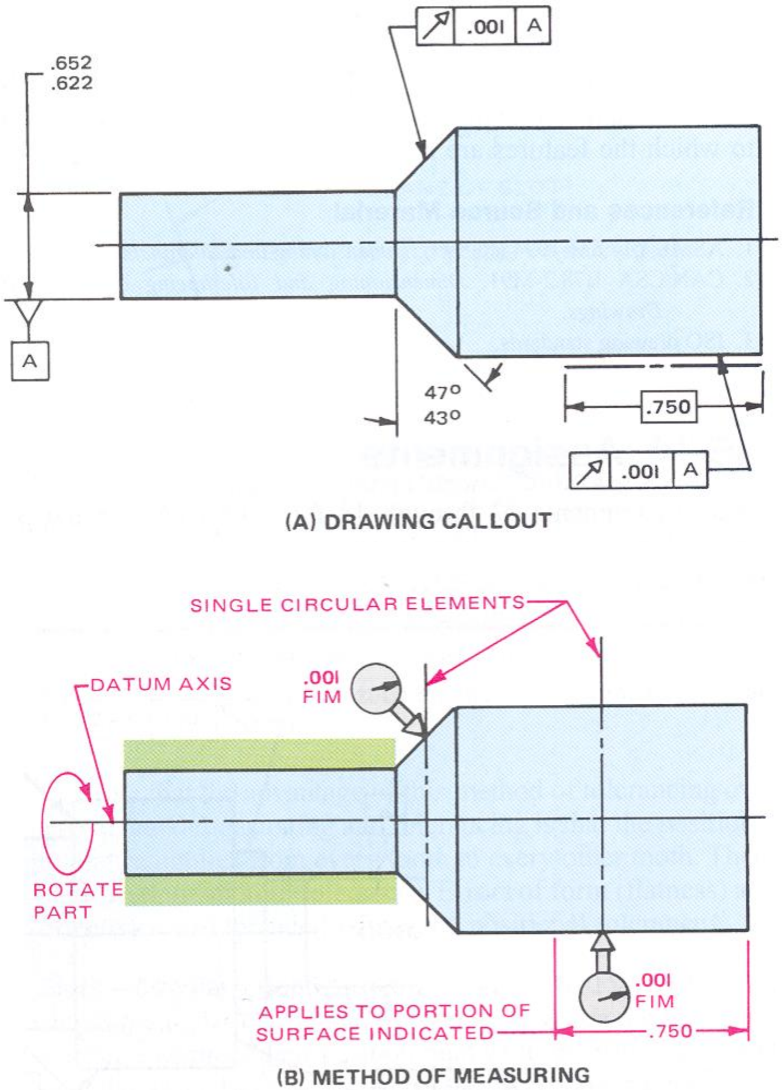
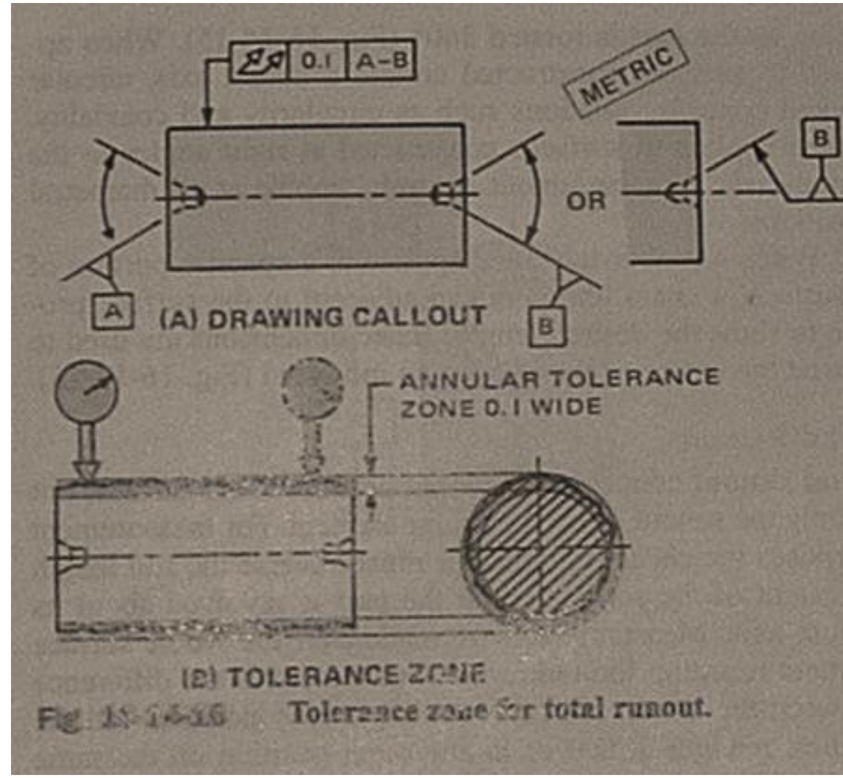


Fig. 16-14-15 Specifying circular runout relative to a datum diameter.

Correlative tolerances

Total Runout

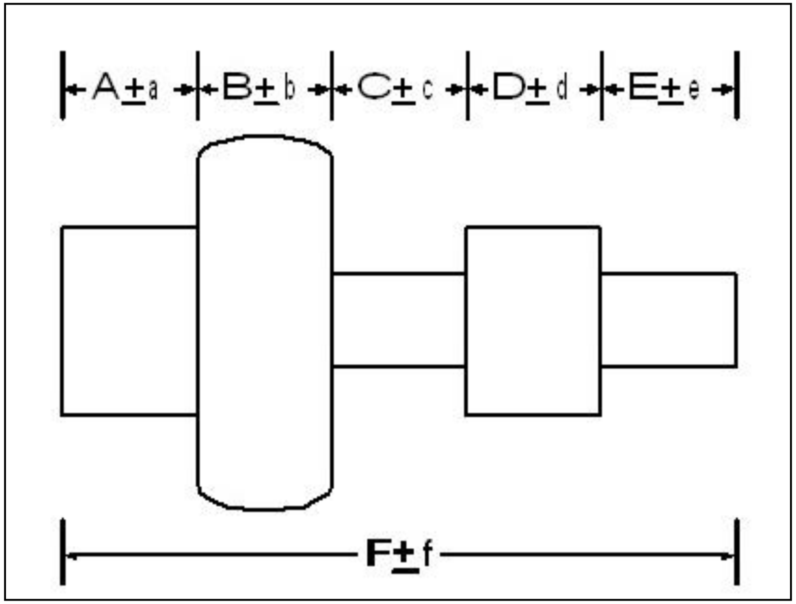
- provides control of entire surface and not only of each circular element
- The indicator is moved to extent of the surface while the part is rotated 360°
- Reading is taken along the surface without resetting the indicator. The difference between the maximum and minimum position of the indicator is the total runout



- The tolerance zone here is the space between the 2 cylinders separated by specified tolerances and coaxial with the datum axis
- Total runout is more costly to verify than circular runout and hence seldom used

Tolerance Buildup

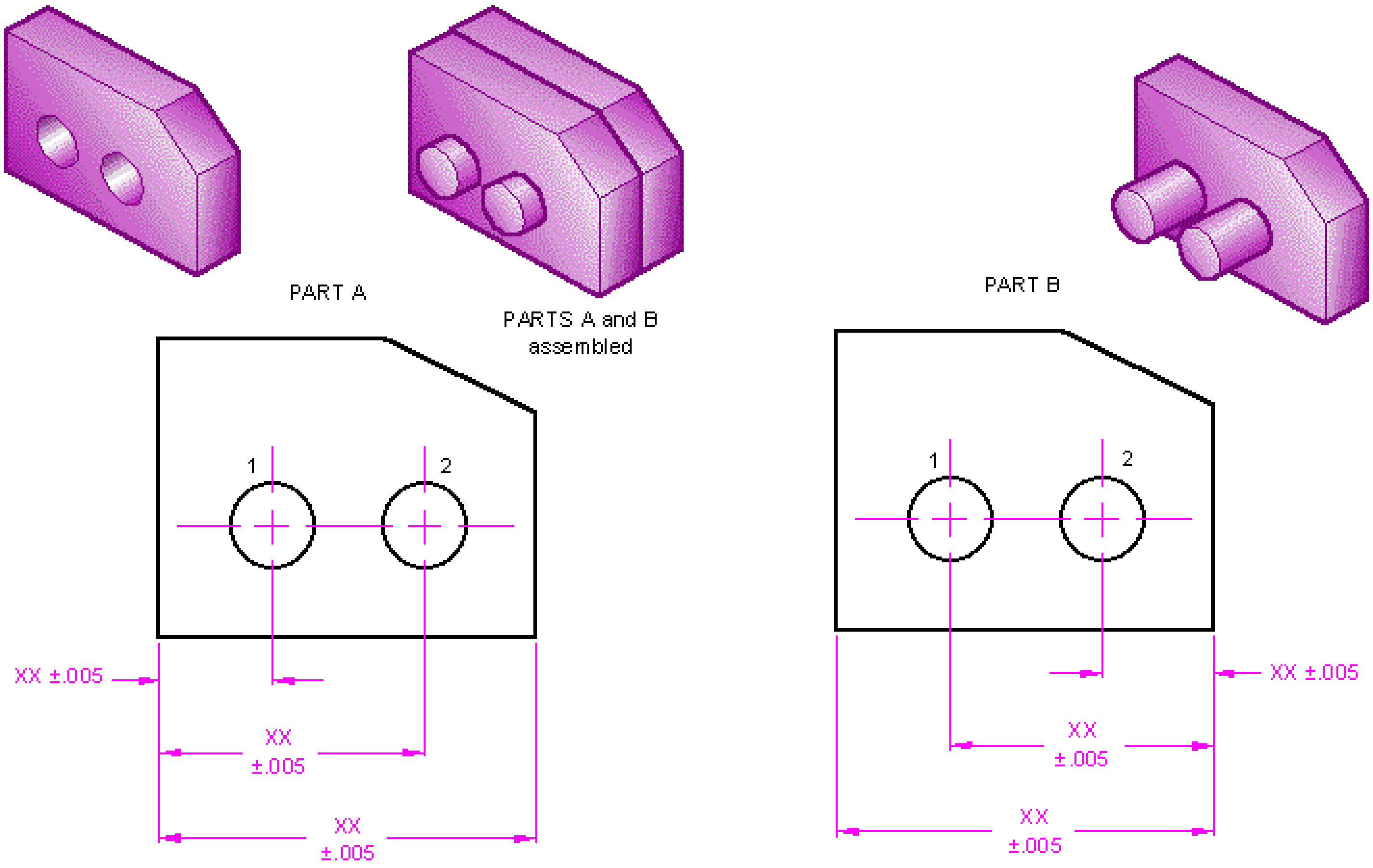
- During assembly, small variations in the part dimensions can multiply until the final assembled result is unacceptable from the original design.
- Though each part variation is small by themselves; however, with each added part, the errors can compound to a defected final part.
- Size tolerances is fairly straightforward. However, when added with form, location and/or orientation variances, then it becomes more difficult to predict dimensional fit.



Tolerance Stack-up

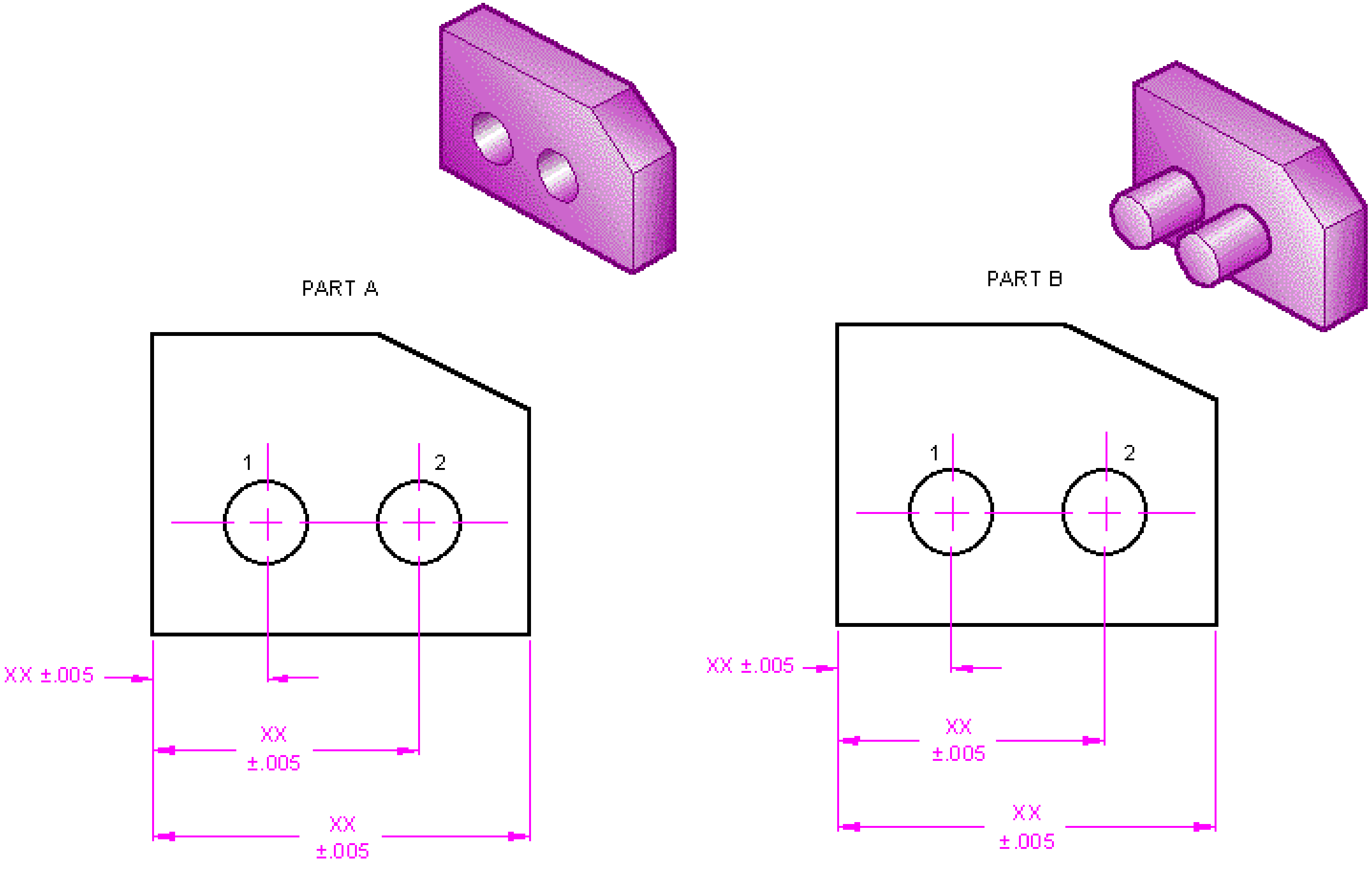
- Tolerances taken from the same direction from one reference are additive
- Tolerances taken to the same point in different directions are additive in both directions

Tolerance Stack-up



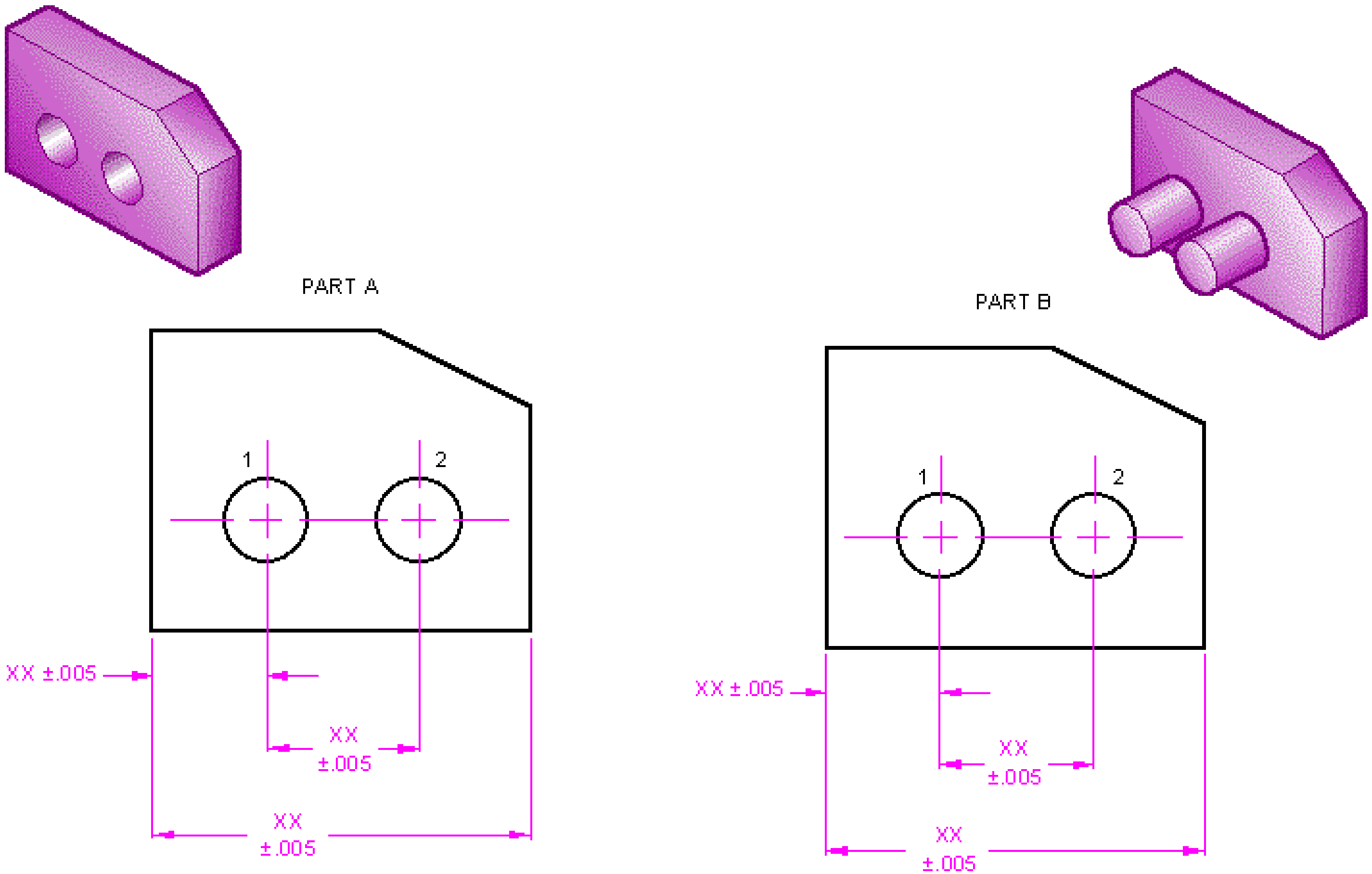
Case 1 - ±.020 between centers

Tolerance Stack-up



Case 2 - ±.010 between centers

Tolerance Stack-up



Case 3 - ±.005 between centers

Tolerance Analysis Methods

- There are many different approaches that are utilized in industry for tolerance analysis. The more tradition methods include:
  - Worst-Case analysis
  - Root Sum of Squares
  - Taguchi tolerance design

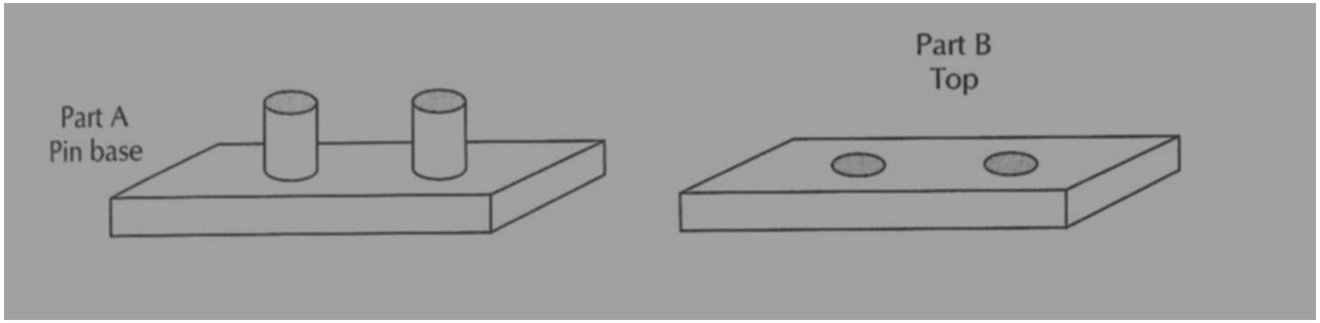
Worst-Case Methodology

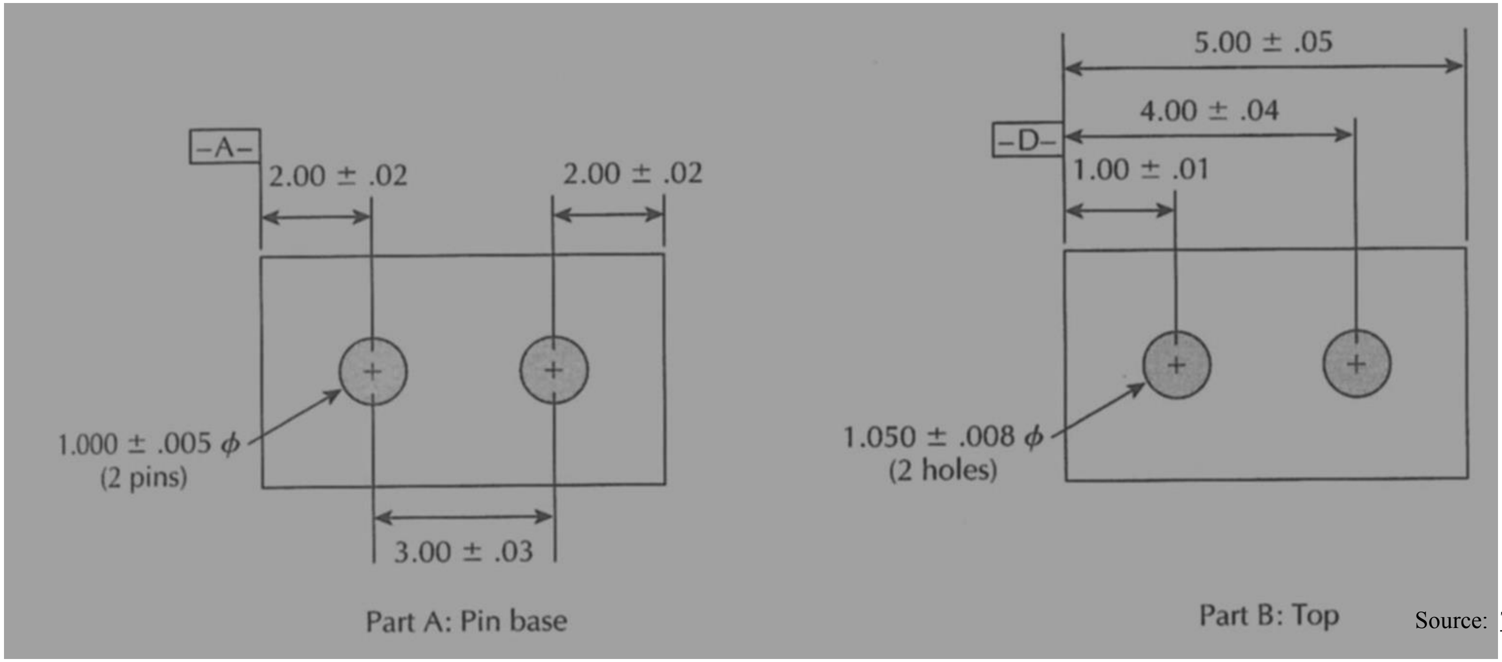
- This is not a statistical procedure but is used often for tolerance analysis and allocation
- Provides a basis to establish the dimensions and tolerances such that any combination will produce a functioning assembly
- Extreme or most liberal condition of tolerance buildup
- "…tolerances must be assigned to the component parts of the mechanism in such a manner that the probability that a mechanism will not function is zero…"

  \- Evans (1974)

Worst Case Scenario Example

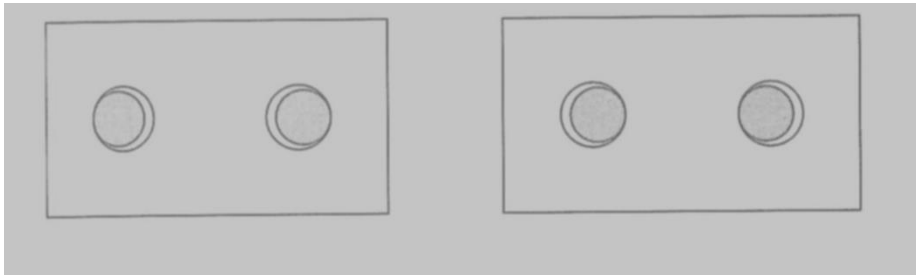
In this example, we see a mating hole and pin assembly.   The nominal dimensions are given in the second figure.



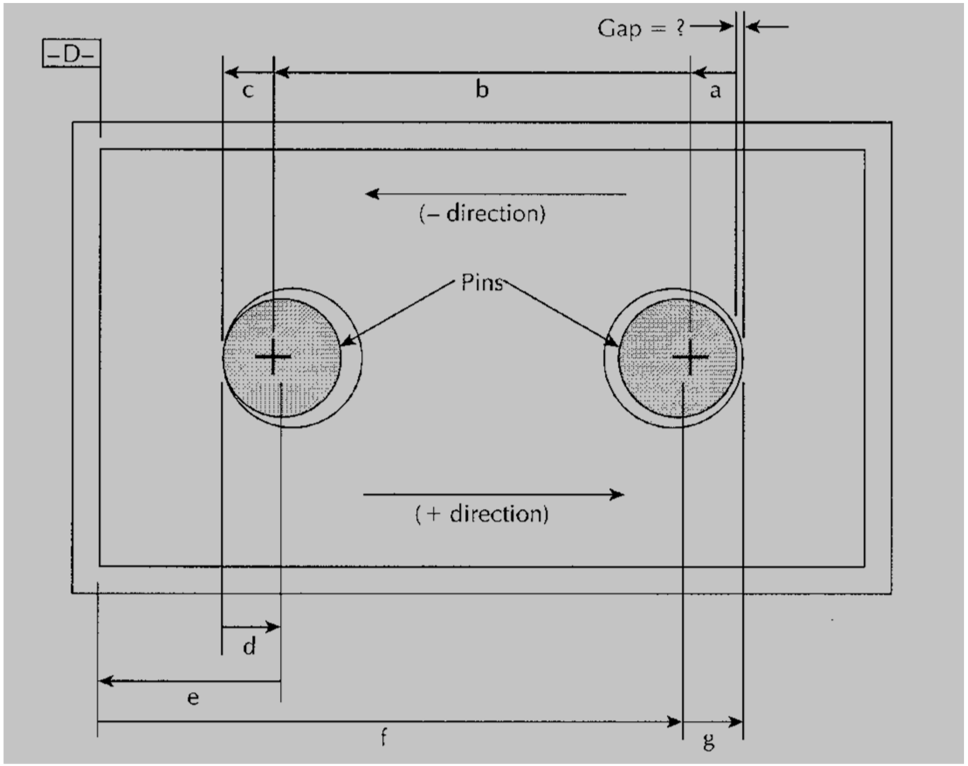


Source: Tolerance Design, pp 109-111

Worst Case Scenario Example



Here, we can see the two worst case situations where the pins are in the extreme outer edges or inner edges.



The analysis begins on the right edge of the right pin.

You should always try to pick a logical starting point for stack analysis.

Note that the stack up dimensions are summed according to their sign (the arrows are like displacement vectors).

Worst Case Scenario Example

a:	–1.00/2	±0.005/2	Part A: Pin base
b:	–3.0	±0.03	Part A: Pin base
c:	–1.00/2	±0.005/2	Part A: Pin base
d:	+1.05/2	±0.008/2	Part B: Top
e:	–1.0	±0.01	Part B: Top
f:	+4.0	±0.04	Part B: Top
g:	+1.05/2	±0.008/2	Part B: Top
	+0.05	±0.093	***for the Worst Case***

Contact point (c:, d:)

- Largest => 0.05 + 0.093 = 0.143
- Smallest => 0.05 - 0.093 = -0.043

From the stack up, we can determine the tolerance calculations shown in table.

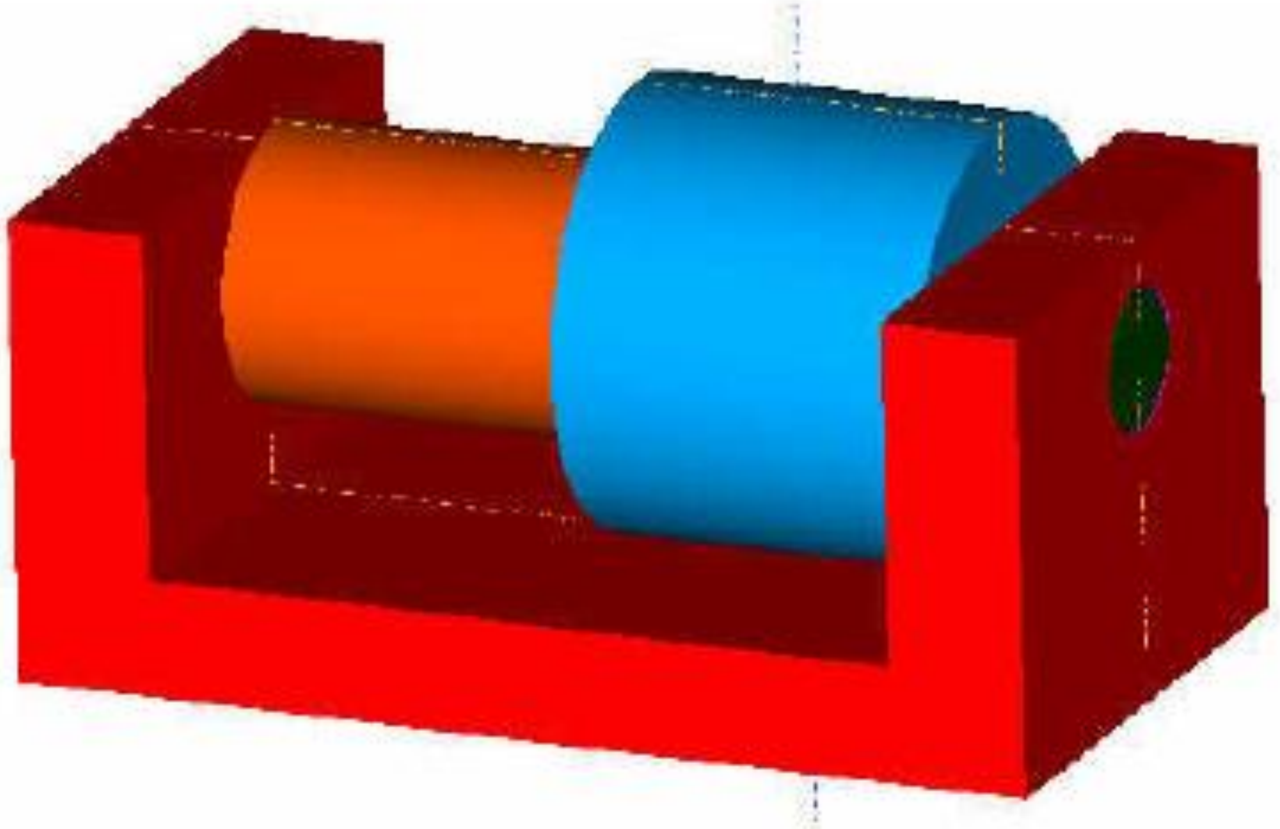
Analyzing the results, we find that there is a +0.05 nominal gap and +0.093 tolerance buildup for the worst case in the positive direction.

This gives us a total worst-case largest gap of +0.143. It gives us a worst case smallest gap of -0.043 which is an interference fit.

Thus, in this worst-case scenario, the parts will not fit and one needs to reconsider the dimension or the tolerance.

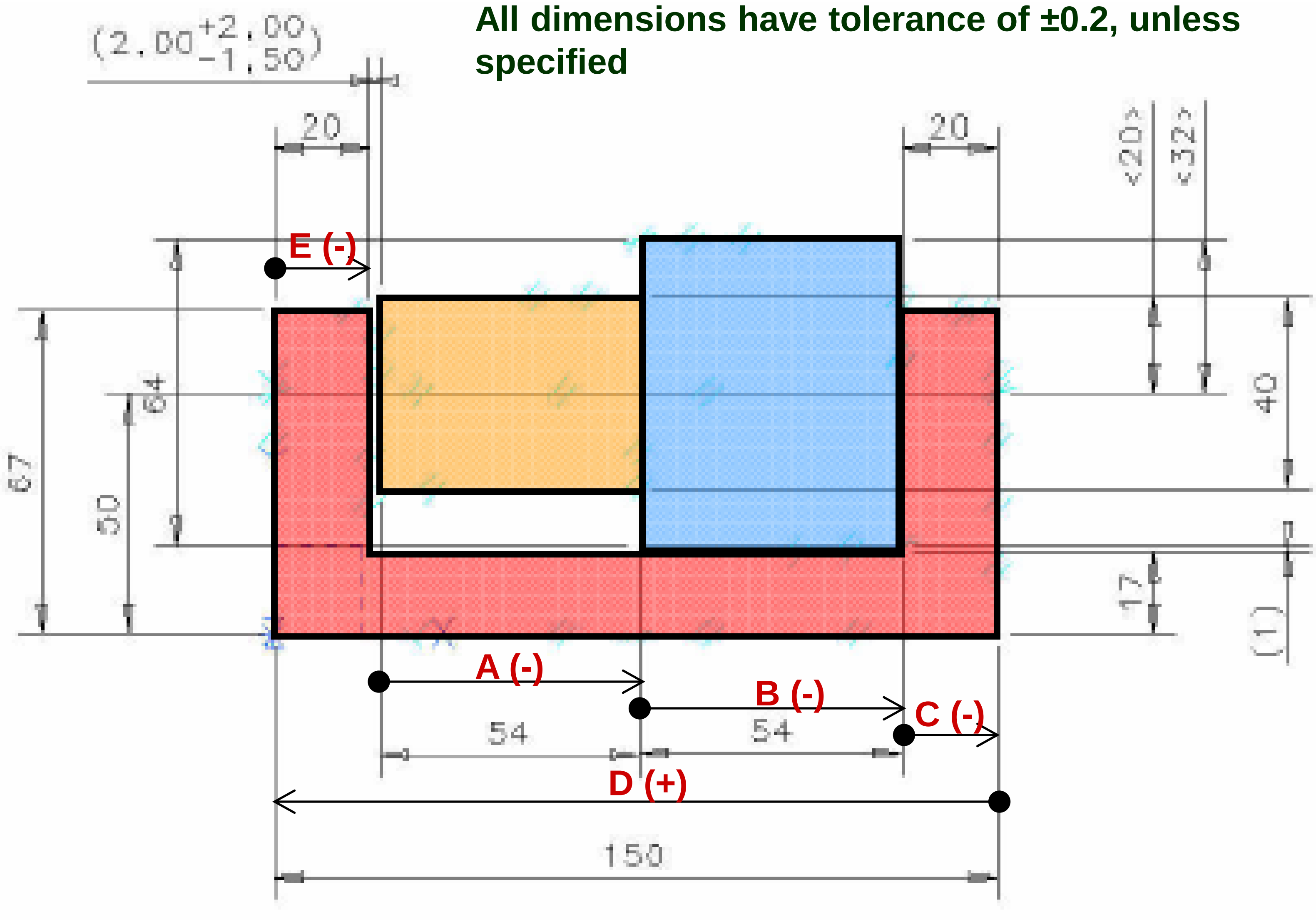
Calculating Stackups

Create Sketch or Obtain Cross-section



Calculating Stackups

All dimensions have tolerance of ±0.2, unless specified

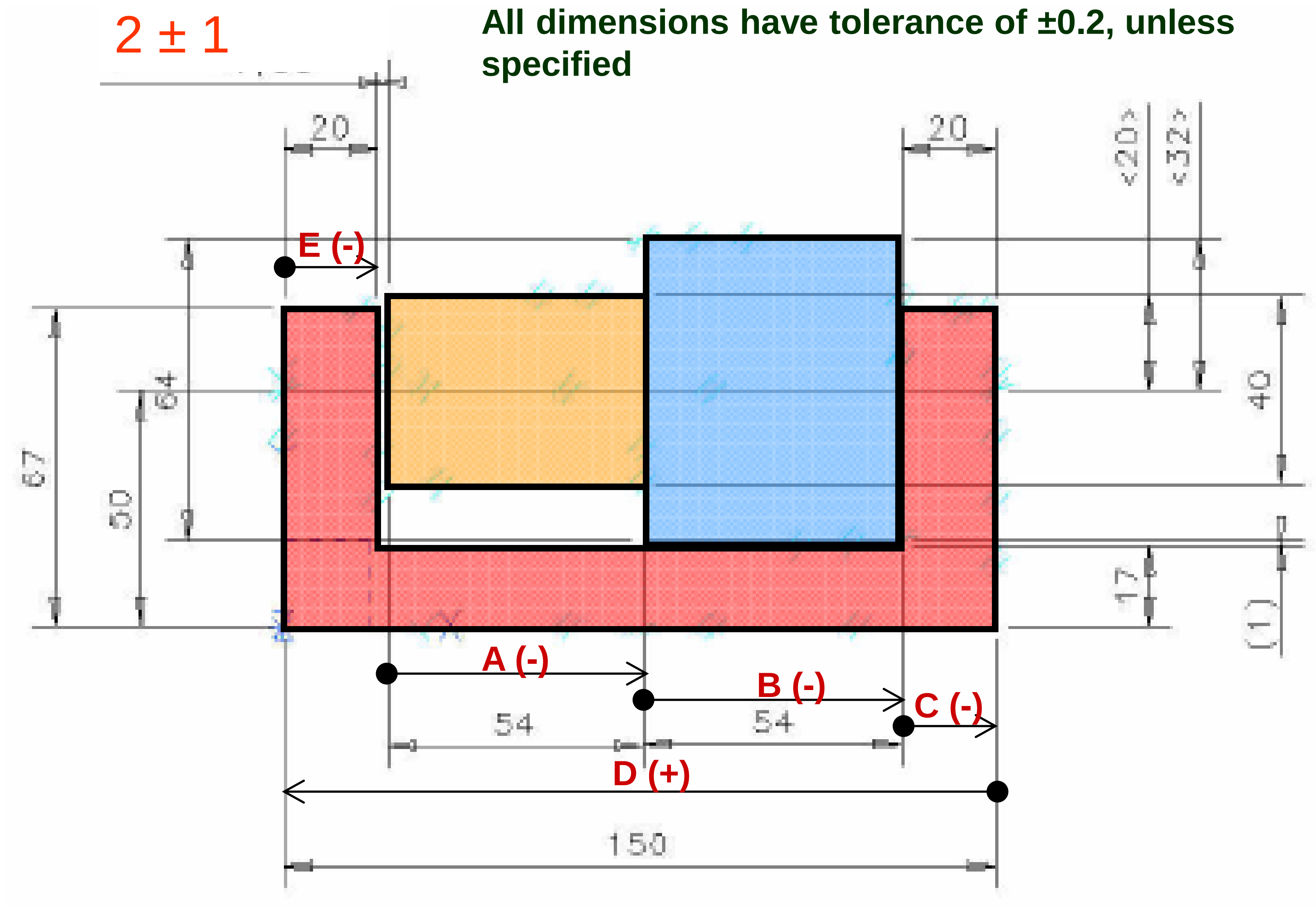


Calculating Stackups

Part	Name	Dim	Tol (+ve)	Tol (-ve)	Direction
Cylinder1	A	54	+0.2	-0.2	-ve
Cylinder2	B	54	+0.2	-0.2	-ve
Holder	C	20	+0.2	-0.2	-ve
Holder	D	150	+0.2	-0.2	+ve
Holder	E	20	+0.2	-0.2	-ve
Worst Case		2	1	-1	
Worst case (Max) = 2 + 1 = 3			**Worst case (Min) = 2 - 1 = 2**		

Calculating Stackups

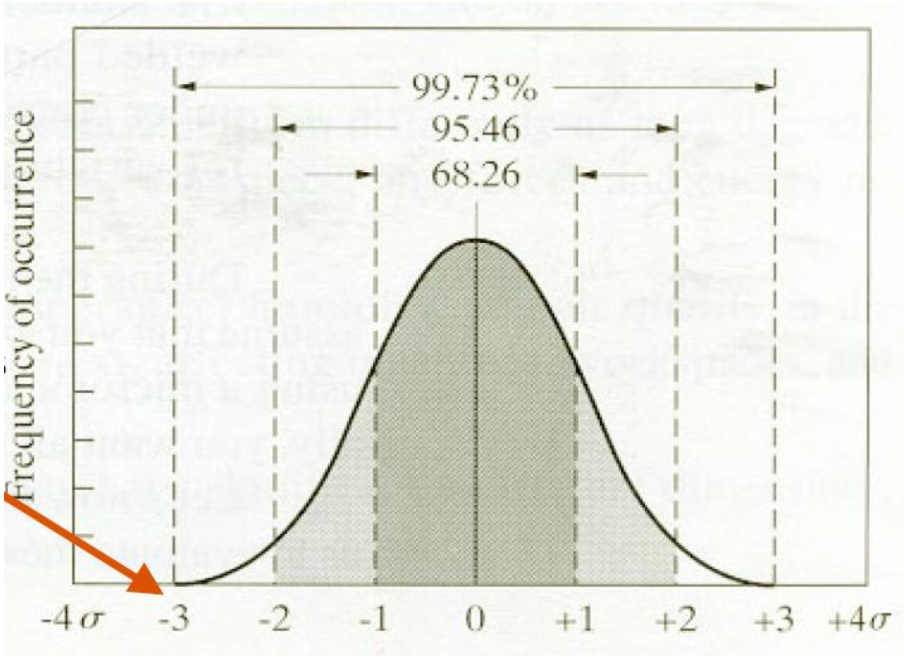
All dimensions have tolerance of ±0.2, unless specified



Statistical Principles

- Measurements require **precision** and **accuracy**.
- A group of measurements when plotted will usually follow a normal distribution or bell curve.
- The centre of the curve is the mean ($\overline{x}$),

and it's proximity to the true value is accuracy.

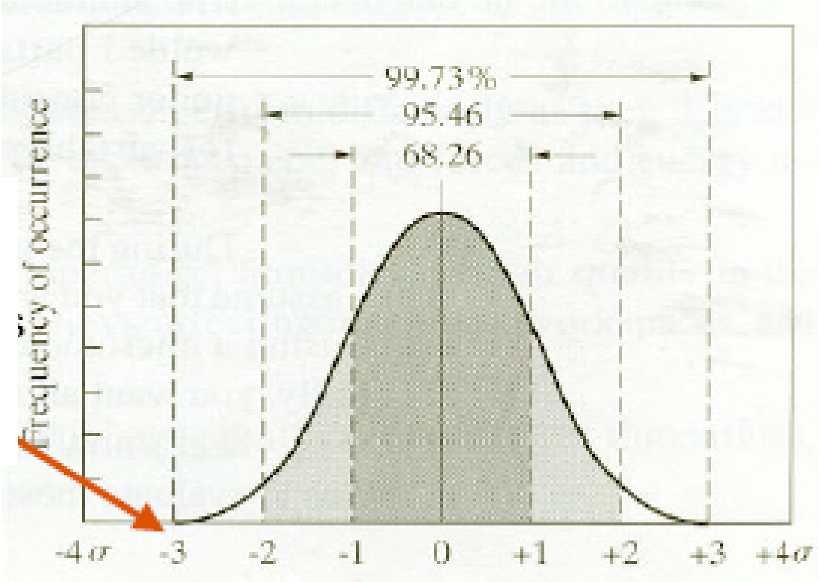


Statistical Principles

Standard Deviation:

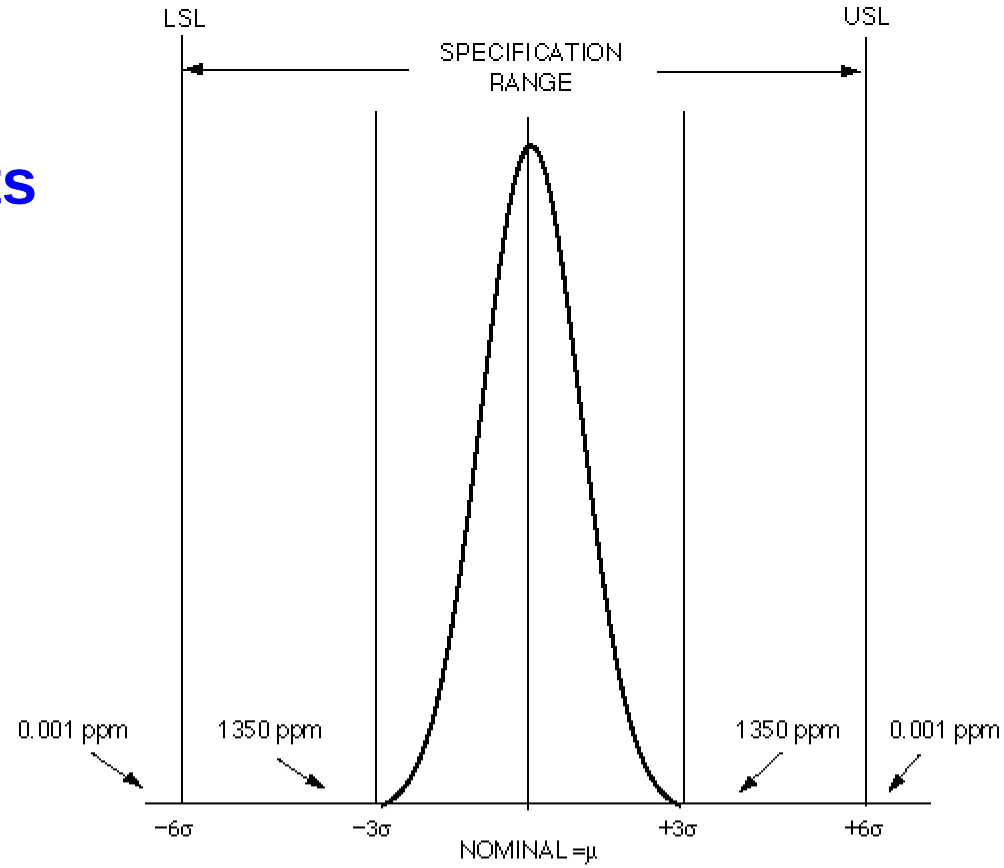
$$\sigma = \sqrt{\frac{\sum (x - \bar{x})^2}{n-1}}$$

- Standard deviation (σ) is common measure of precision.
- $\pm\ \sigma$, covers 68% of measurements
- $\pm\ 2\sigma$ covers 95% of measurements
- $\pm\ 3\sigma$ covers 99.73% of measurements
- The Industry standard once was, 27 in 10,000 parts can be defective; this is not acceptable anymore.



Statistical Principles

± 6σ translates to 2 parts per billion



Excerpted from The Complete Guide to the CQE *by Thomas Pyzdek. 1996. Tucson: Quality Publishing Inc.*