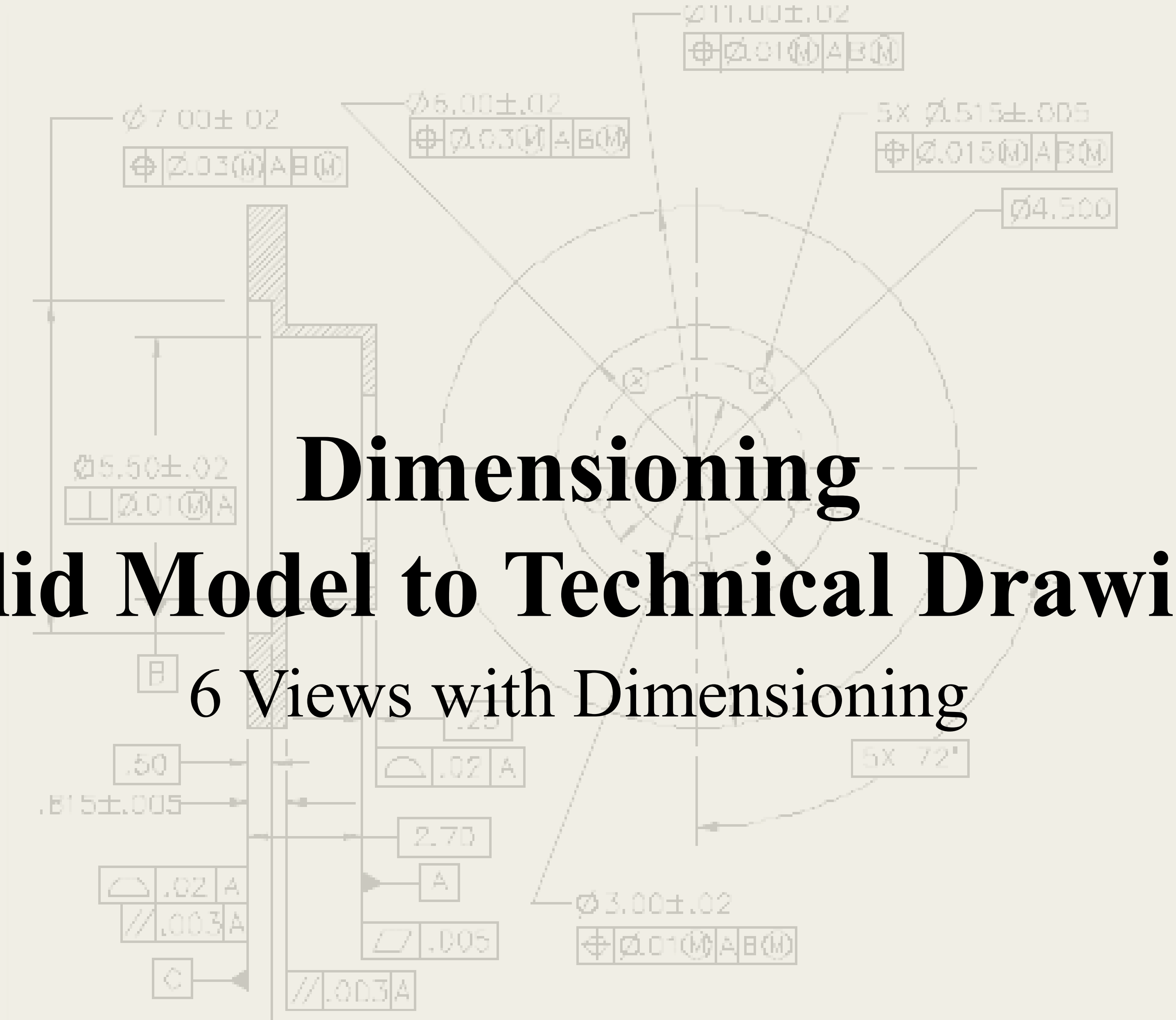
Introduction to Tolerancing and Dimensioning

Doing things right the first time adds nothing to the cost of your product of service. Doing things wrong is what costs money. – Philip B. Crosby

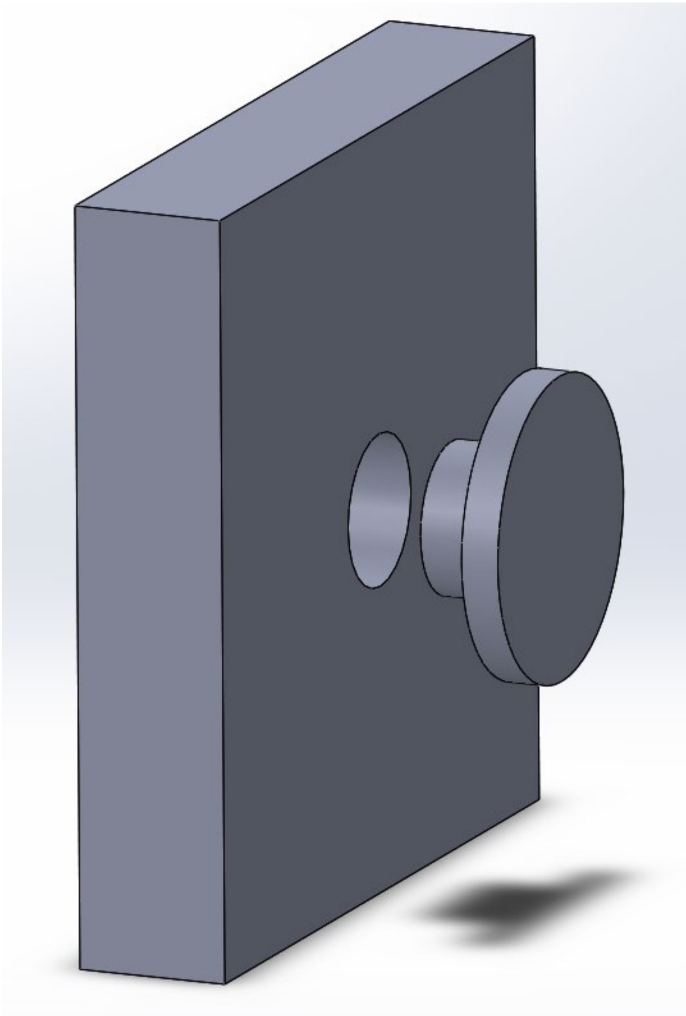
Collected by A. Trimble
Additional Slides Added by M. Nejhad & T. Sorensen

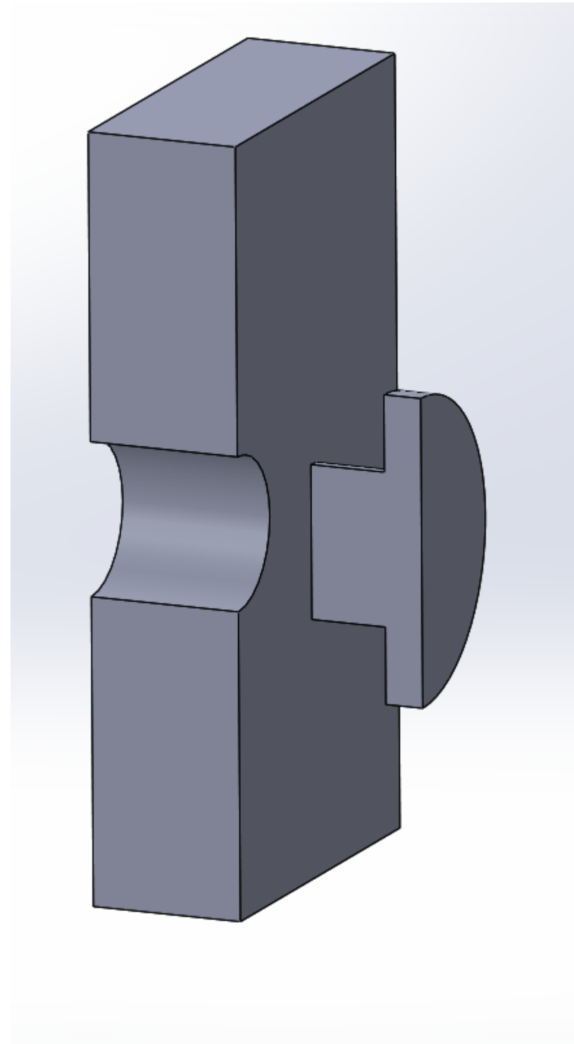


Dimensioning
Solid Model to Technical Drawing

6 Views with Dimensioning

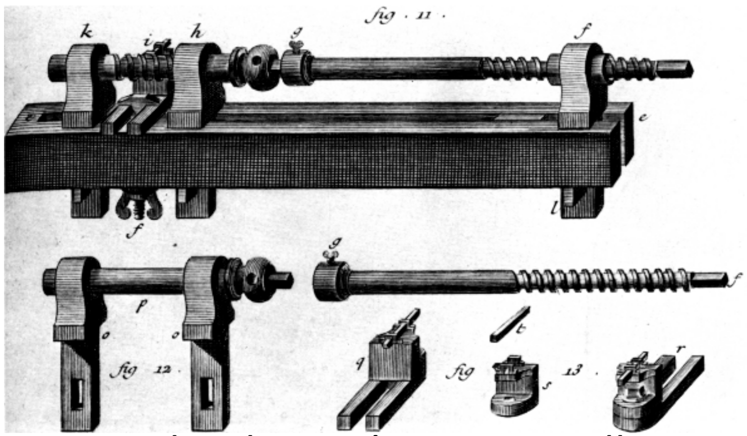
Tolerancing





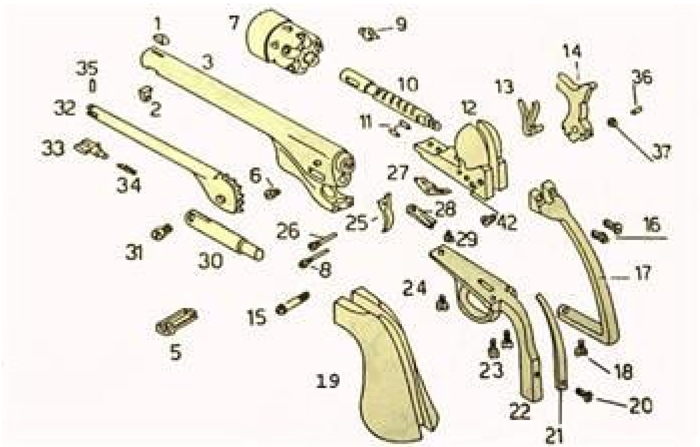
Why bother?

- Interchangeable Parts

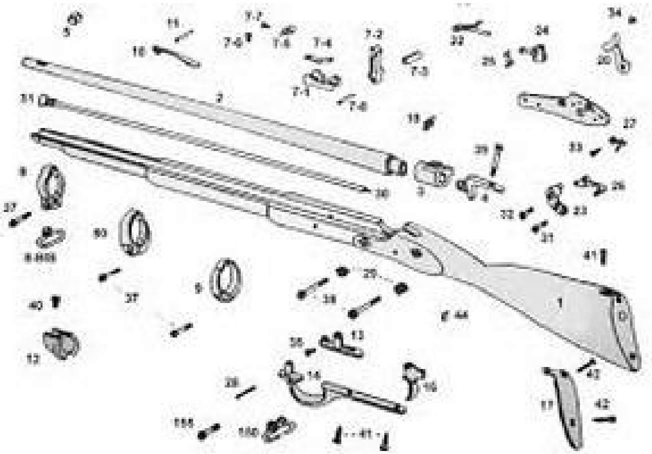


Eli Whitney's Cotton Mill

Early 1900's assembly line



Colt Peace Maker



Springfield 1856

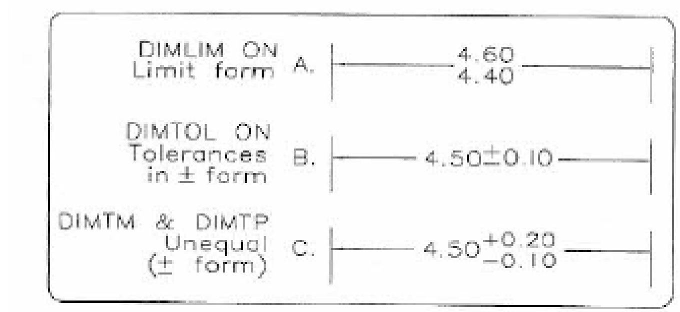
Tolerancing and Dimensioning Goals

- Tolerancing
  - Allow individual parts to assemble and properly operate together as a single system

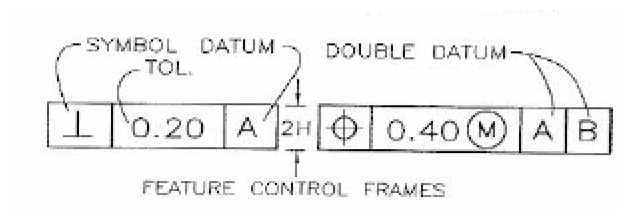
- Dimensioning
  - Exercise in Precise Communication
    - Typically a graphical representation of the part or assembly that provides a clear depiction and description of:
      - Geometry
        - » Shape
      - Size
        - » Dimensions
        - » Tolerances
        - » Description (material, finish, etc.)
      - Intent is clear
        - » Size and location of all features is mutual understood
          - Standards
            - ANSI
            - ISO

Tolerancing

- Motivation: Cannot make a part *exactly* the right size
  - A certain amount of variation on each dimension must be tolerated
- Example: Size limits
  - Unilateral/bilateral
  - Limit



  - Geometric



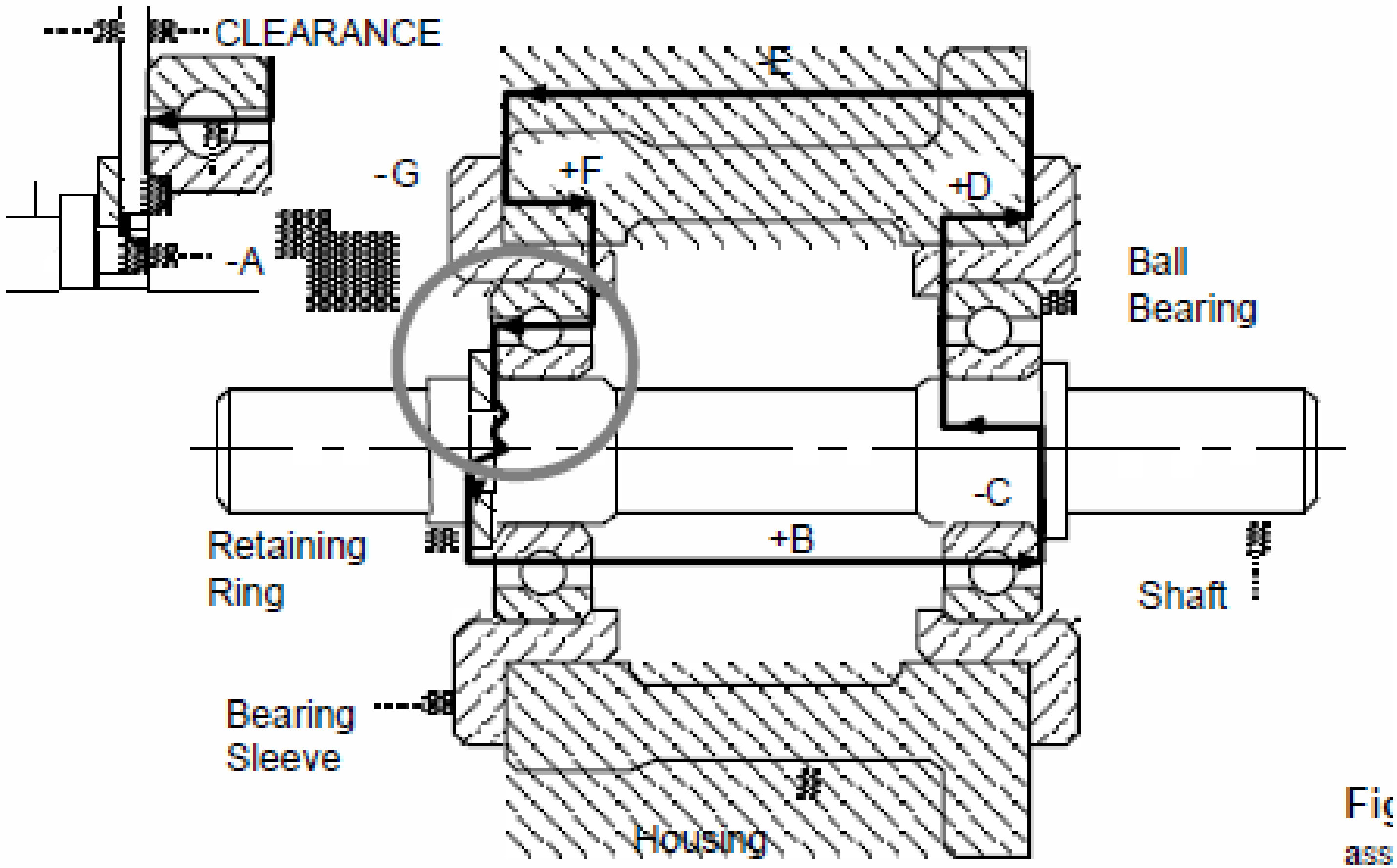
Considerations

but avoid "over-the-wall" thinking

- Engineering Driven
  - Any tolerance necessary to ensure the perceived functional requirements of a product – "over-the-wall"
- Process Driven
  - Based entirely on the capability of the manufacturing process – manufacturing dictates design requirements to engineering
- Inspection Driven
  - Expected measurement/gauging technique dictates dimensioning scheme and tolerancing

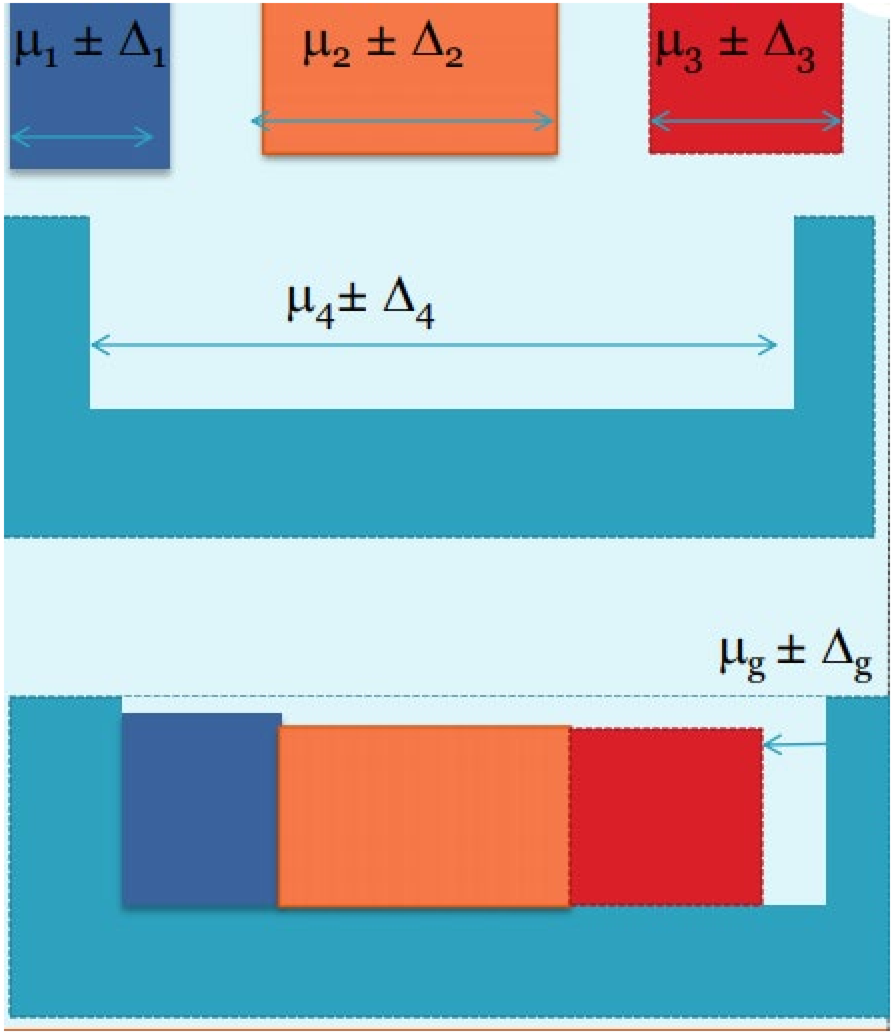
Engineering Driven

Tolerance Stackup process



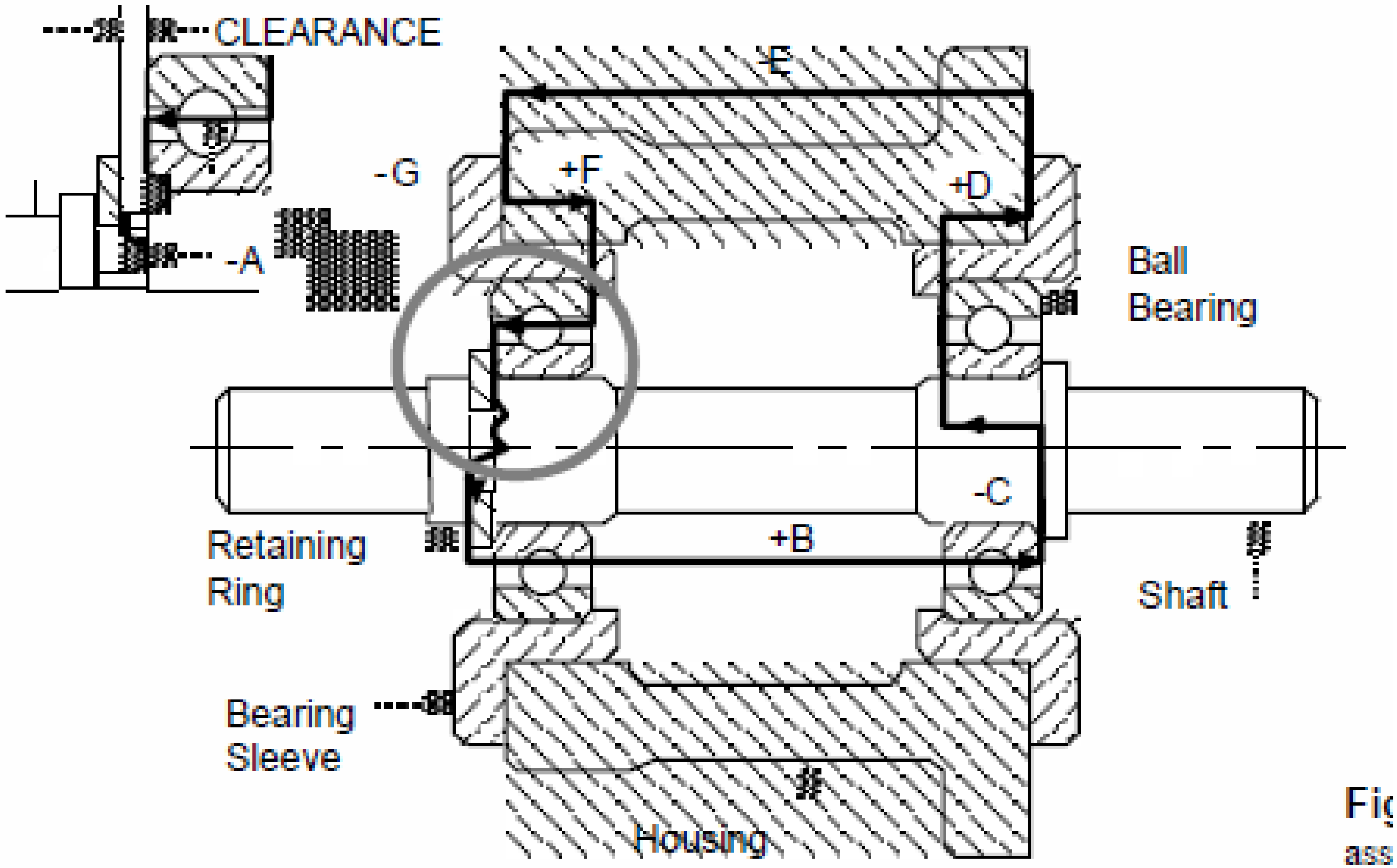
Tolerance Stack up

- Adding/subtracting tolerances
  - Worst Case
  - Good for:
    - Design verification
    - Catastrophic failure
    - Low number of parts (3-5)
- Statistical
  - Sum variance



Engineering Driven

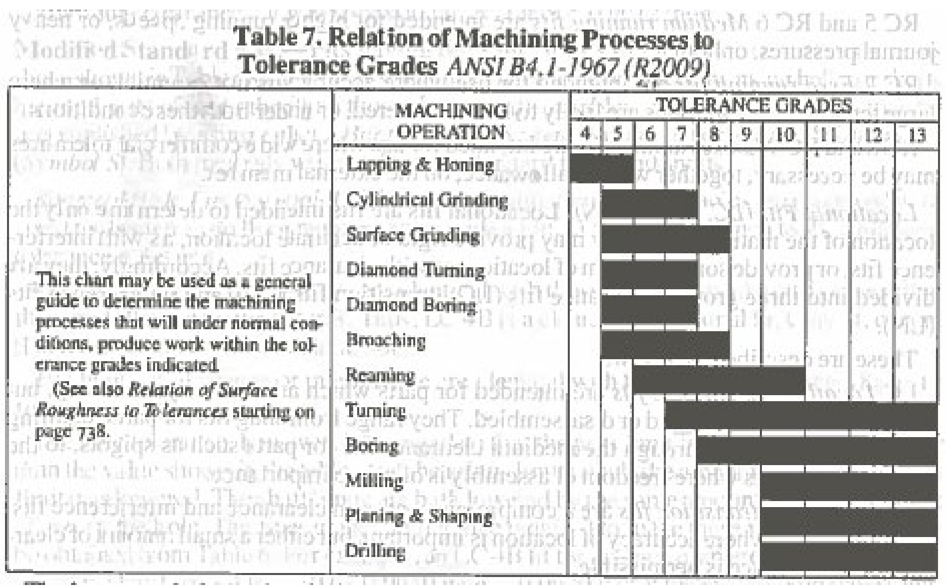
- Performance Requirements



Process Driven

- Tolerance Grades

	Measuring Tools									Material								
IT Grade	**01**	**0**	**1**	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**	**13**	**14**	**15**	**16**
							Fits							Large Manufacturing Tolerances				



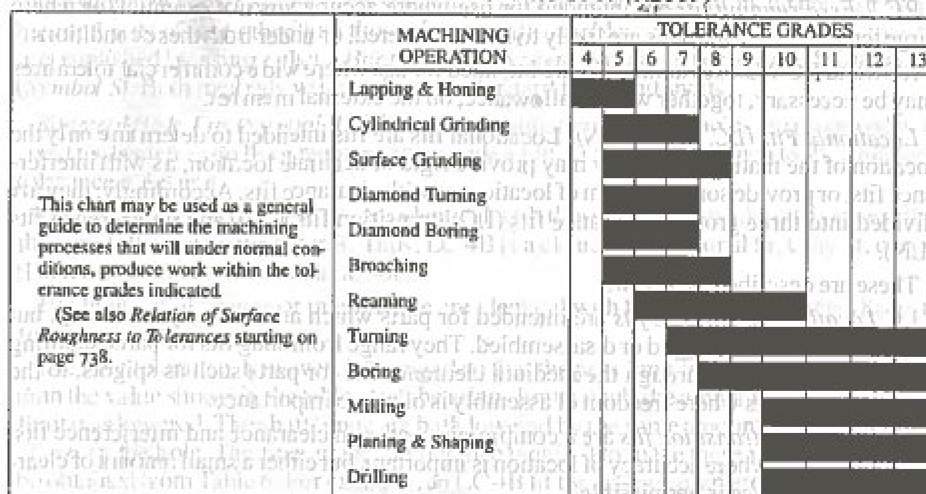
Process Driven

Table 6. ANSI Standard Tolerances *ANSI B4.1-1967 (R2009)*

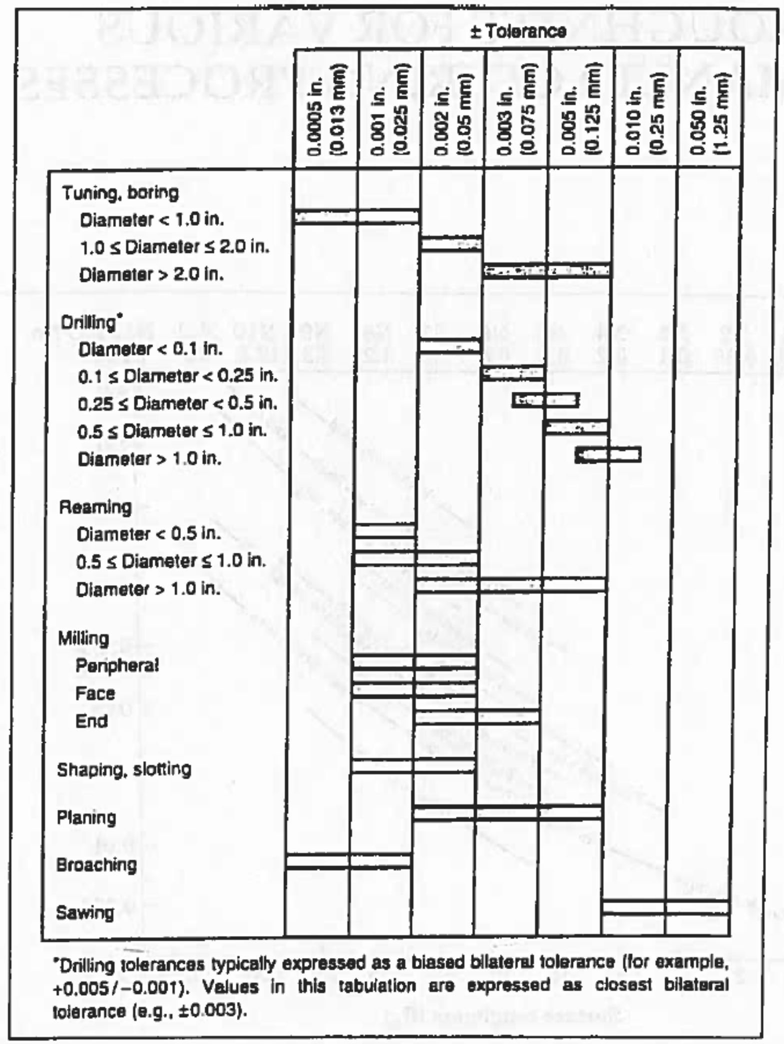
Nominal Size, Inches		Grade									
		4	5	6	7	8	9	10	11	12	13
Over	To	Tolerances in thousandths of an inch[a]									
0	0.12	0.12	0.15	0.25	0.4	0.6	1.0	1.6	2.5	4	6
0.12	0.24	0.15	0.20	0.3	0.5	0.7	1.2	1.8	3.0	5	7
0.24	0.40	0.15	0.25	0.4	0.6	0.9	1.4	2.2	3.5	6	9
0.40	0.71	0.2	0.3	0.4	0.7	1.0	1.6	2.8	4.0	7	10
0.71	1.19	0.25	0.4	0.5	0.8	1.2	2.0	3.5	5.0	8	12
1.19	1.97	0.3	0.4	0.6	1.0	1.6	2.5	4.0	6	10	16
1.97	3.15	0.3	0.5	0.7	1.2	1.8	3.0	4.5	7	12	18
3.15	4.73	0.4	0.6	0.9	1.4	2.2	3.5	5	9	14	22
4.73	7.09	0.5	0.7	1.0	1.6	2.5	4.0	6	10	16	25
7.09	9.85	0.6	0.8	1.2	1.8	2.8	4.5	7	12	18	28
9.85	12.41	0.6	0.9	1.2	2.0	3.0	5.0	8	12	20	30
12.41	15.75	0.7	1.0	1.4	2.2	3.5	6	9	14	22	35
15.75	19.69	0.8	1.0	1.6	2.5	4	6	10	16	25	40
19.69	30.09	0.9	1.2	2.0	3	5	8	12	20	30	50
30.09	41.49	1.0	1.6	2.5	4	6	10	16	25		
41.49	56.19	1.2	2.0	3	5	8	12	20	30		
56.19	76.39	1.6	2.5	4	6	10	16	25	40		
76.39	100.9	2.0	3	5	8	12	20	30	50		
100.9	131.9	2.5	4	6	10	16	25	40	60		
131.9	171.9	3	5	8	12	20	30	50	80		
171.9	200	4	6	10	16	25	40	60	100		

[a] All tolerances above heavy line are in accordance with American-British-Canadian ments.

Table 7. Relation of Machining Processes to Tolerance Grades *ANSI B4.1-1967 (R2009)*

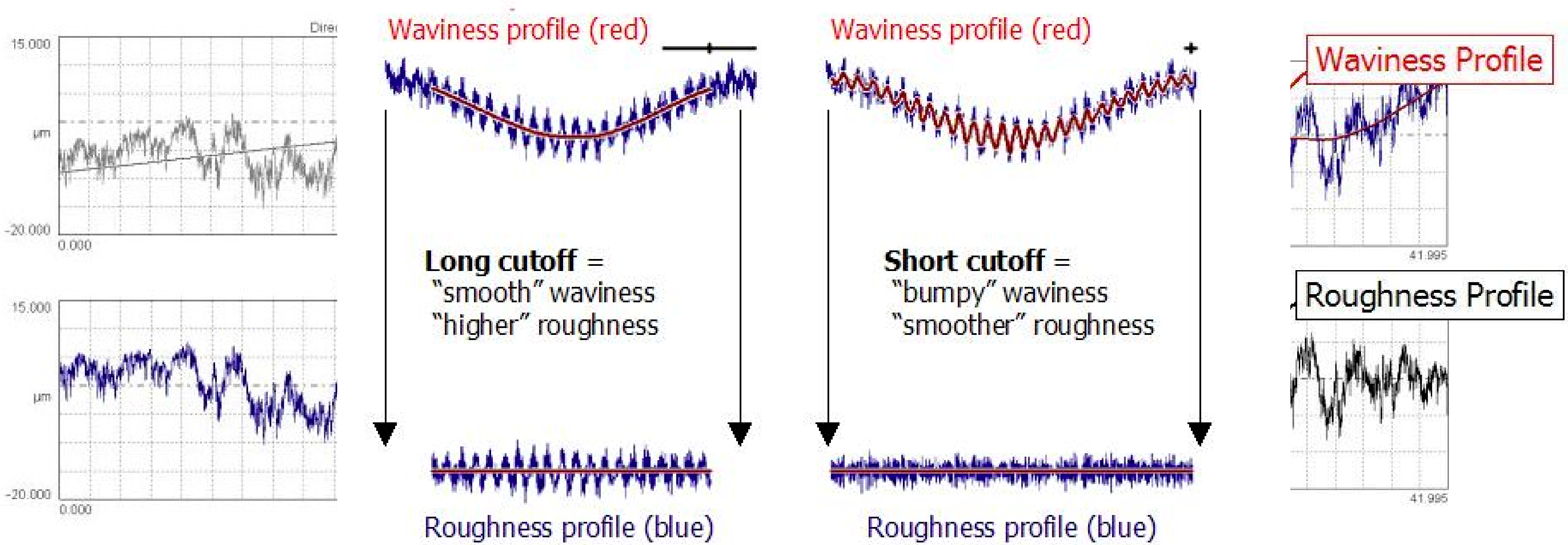


Process Driven

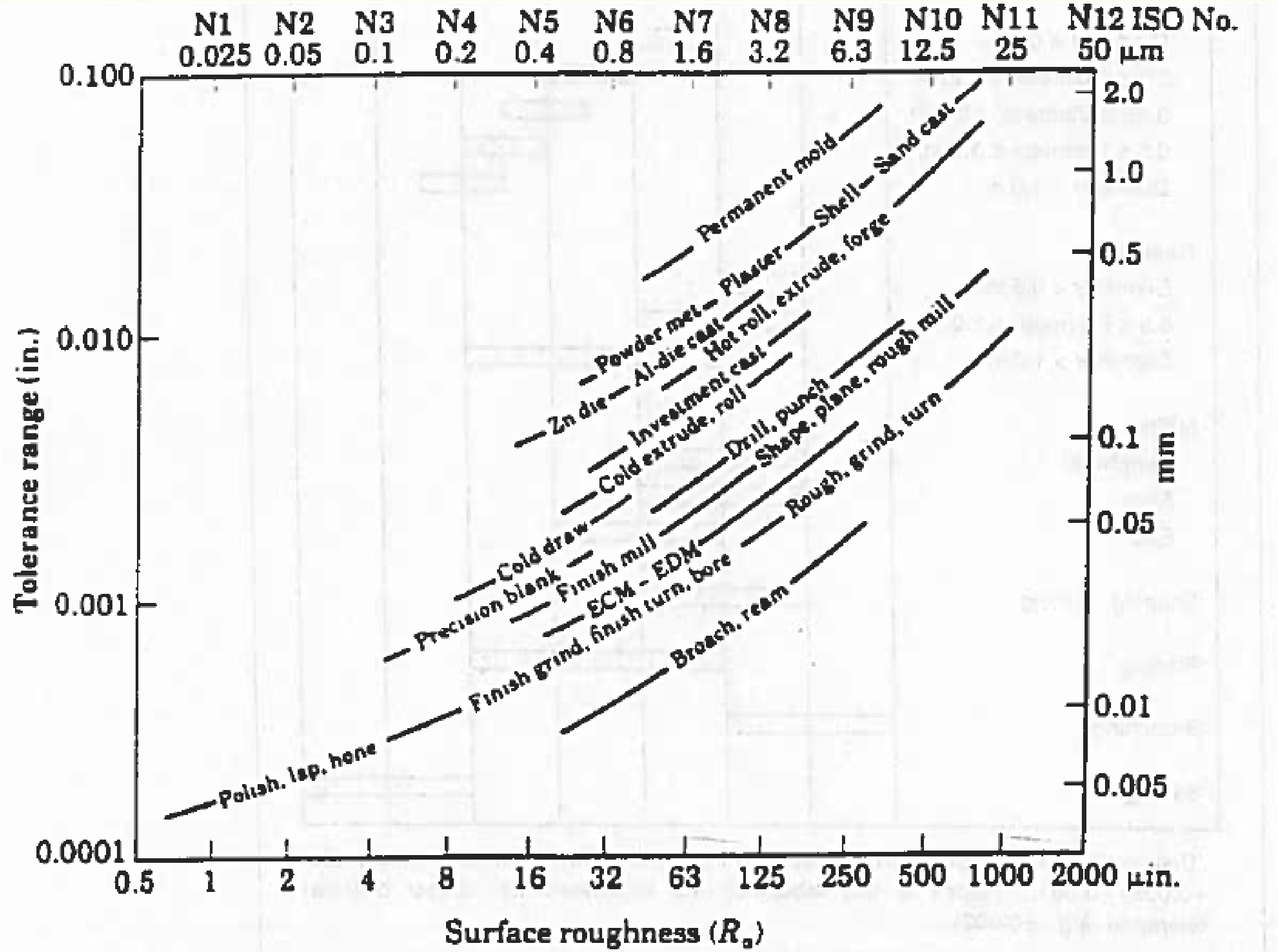


Surface Deviations

Form, Waviness, Roughness

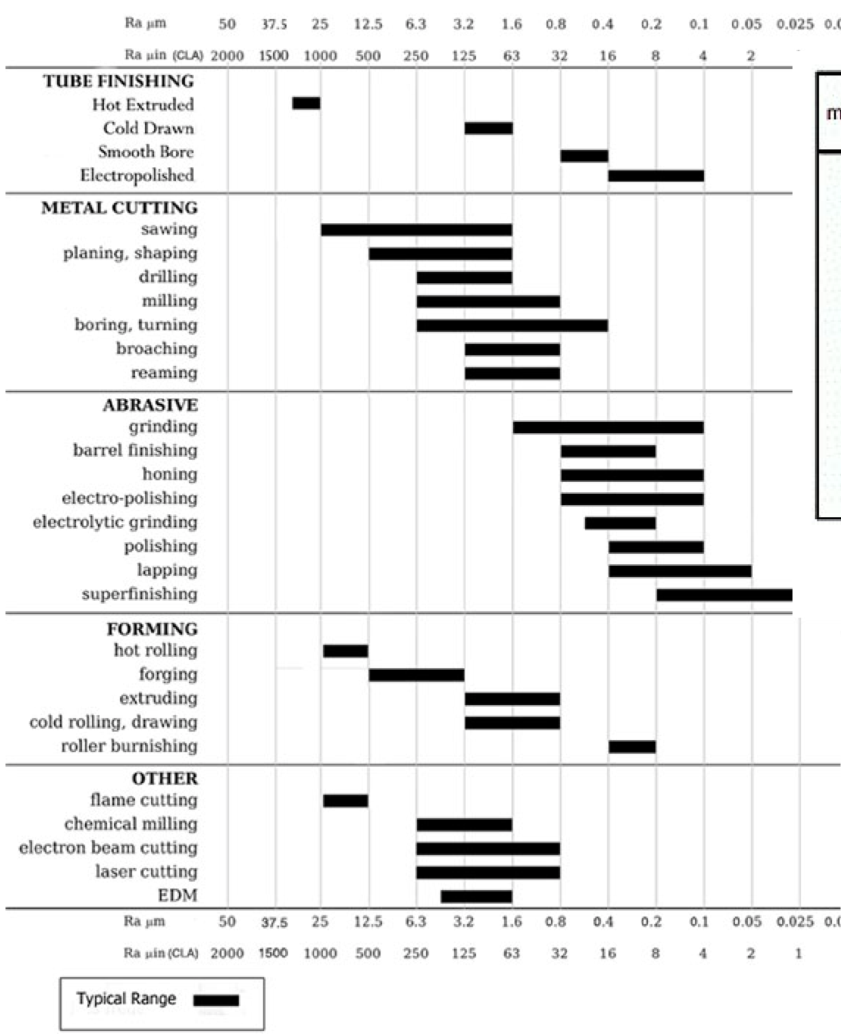


Process Driven



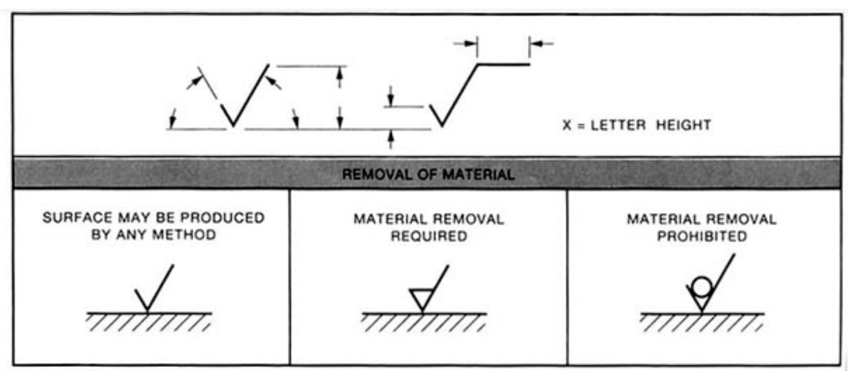
https://www.cnccookbook.com/surface-finish-chart-symbols-measure-calculators/

Indicative surface roughness comparisons

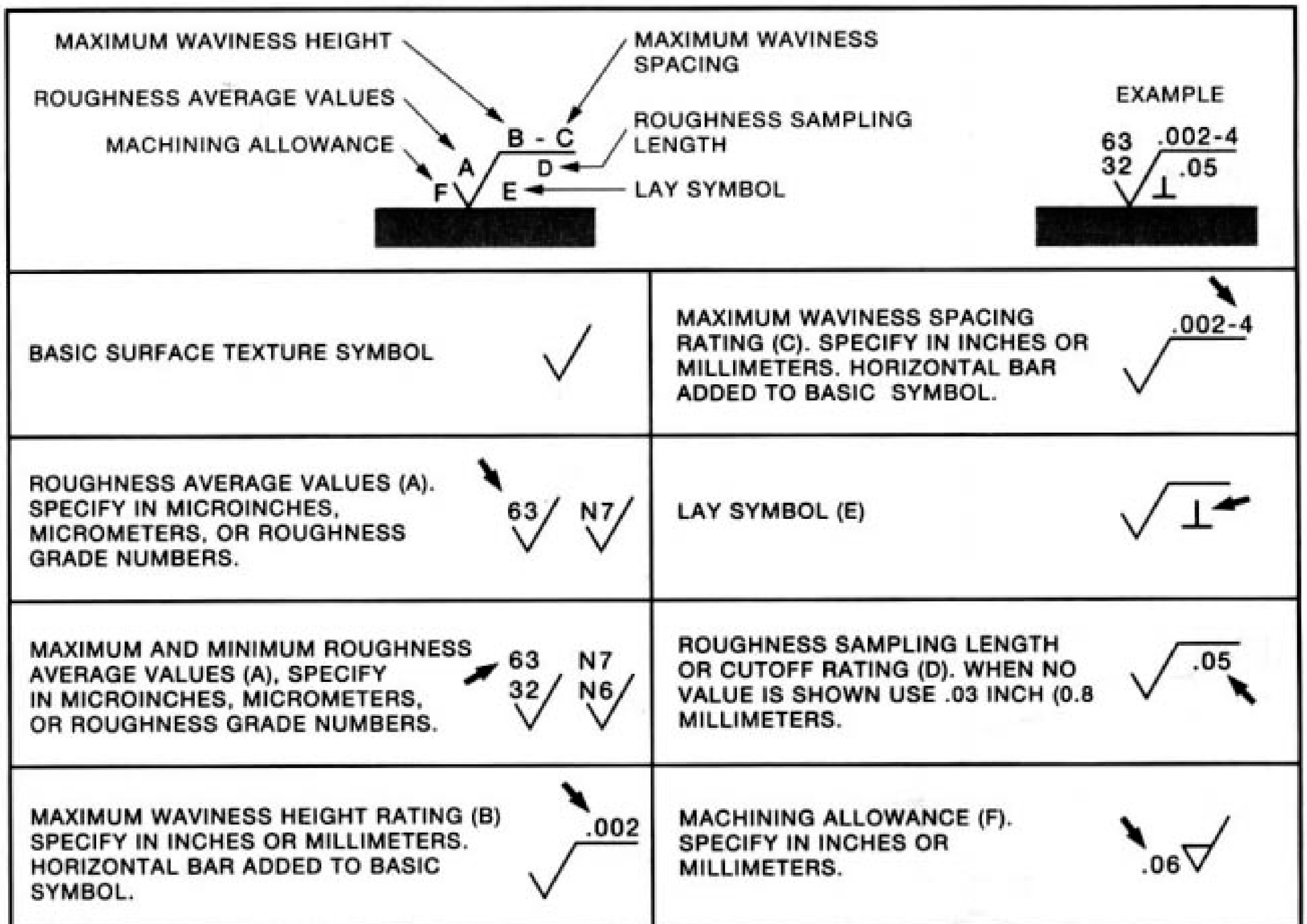


R_a micrometer µm	R_a micro-inch µ in	Roughness Grade Numbers (New)**	Roughness Grade Numbers (Old)***	R_t	∇(Ra)	Old Style	American standard
50	2000	N12	∇				
25	1000	N11					
12.5	500	N10					
6.3	250	N9	∇∇	32	6.3	32 ∇	250
3.2	125	N8		16	3.2	∇∇	125
1.6	63	N7		8	1.6	8 ∇∇	63
0.8	32	N6	∇∇∇	4	0.8	∇∇∇	32
0.4	16	N5		2	0.4	2 ∇∇∇	16
0.2	8	N4		1	0.2	∇∇∇∇	8
0.1	4	N3	∇∇∇∇	0.5	0.1	0.5 ∇∇∇∇	4
0.05	2	N2		0.25	0.05	0.25 ∇∇∇∇	2
0.025	1	N1					

Roughness Grade Numbers and Ra Measures…

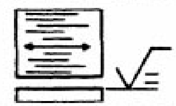
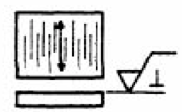
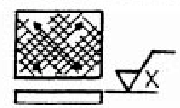
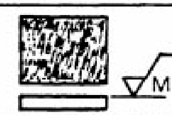
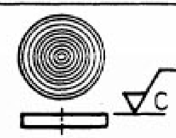
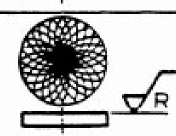
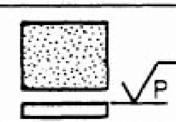


Surface Finish



NOTE: WAVINESS IS NOT USED IN ISO STANDARDS.

Surface Finish

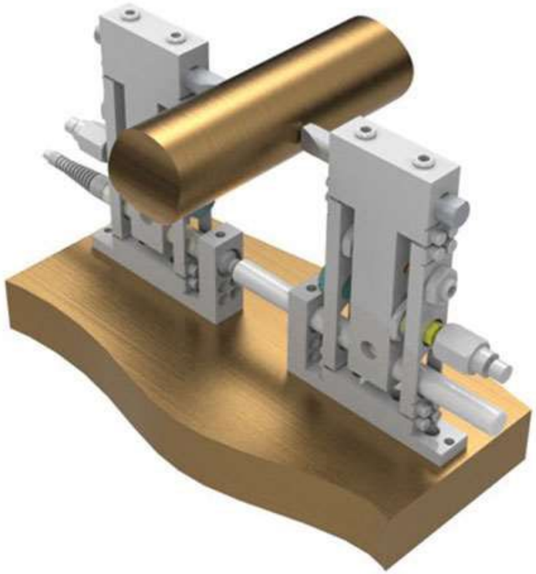
Lay Symbol	Meaning	Example Showing Direction of Tool Marks
=	Lay approximately parallel to the line representing the surface to which the symbol is applied.	
⊥	Lay approximately perpendicular to the line representing the surface to which the symbol is applied.	
X	Lay angular in both directions to line representing the surface to which the symbol is applied.	
M	Lay multidirectional.	
C	Lay approximately circular relative to the center of the surface to which the symbol is applied.	
R	Lay approximately radial relative to the center of the surface to which the symbol is applied.	
P[3]	Lay particulate, non-directional, or protuberant.	

Surface Finish

MICROMETERS RATING	MICROINCHES RATING	APPLICATION
25	1000	Rough, low grade surface resulting from sand casting, torch or saw cutting, chipping, or rough forging. Machine operations are not required because appearance is not objectionable. This surface, rarely specified, is suitable for unmachined clearance areas on rough construction items.
12.5	500	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 suitable for clearance areas on machinery, jigs, and fixtures. Sand casting or rough forging produces this surface.
6.3	250	Coarse production surface, for unimportant clearance and cleanup operation, resulting from coarse surface grind, rough file, disc grind, rapid feeds in turning, milling, shaping, drilling, boring, grinding, etc., where tool marks are not objectionable. The natural surfaces of forgings, permanent mold castings, extrusions, 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.
3.2	125	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, extrusion, and rolled surfaces.
1.6	63	A good machine finish produced under controlled conditions using relatively high speeds and fine feeds to take light cuts with sharp cuttings. 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 infrequent. It may also be obtained on extrusions, rolled surfaces, die castings and permanent mold casting when rigidly controlled.
0.8	32	A high-grade machine finish requiring close control when produced by lathes, shapers, milling machines, etc., but relatively easy to produced 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.
0.4	16	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 bearing and extreme tension members.
0.2	8	A fine surface produced by honing, lapping, or buffing. It is specified where packings and rings must slide across the direction of the surface grain, maintaining or withstanding pressures, or for interior honed surface of hydraulic cylinders. It may also be required in precision gauges and instrument work, or sensitive value surfaces, or on rapidly rotating shafts and on bearings where lubrication is not dependable.
0.1	4	A costly refined surface produced by honing, lapping and buffing. It is specified only when he design requirements make it mandatory. It is required in instrument work, gauge work, and where packing and rings must slide across the direction of surface grain such as on chrome plated piston rods, etc. where lubrication is not dependable.
0.05 0.025	2 1	Costly refined surfaces produced by only 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 surfaces are specified only when design requirements make it mandatory. They are specified on fine or sensitive instrument parts or other laboratory items, and certain gauge surfaces, such as precision gauge blocks.

Surface Finish

Inspection Driven



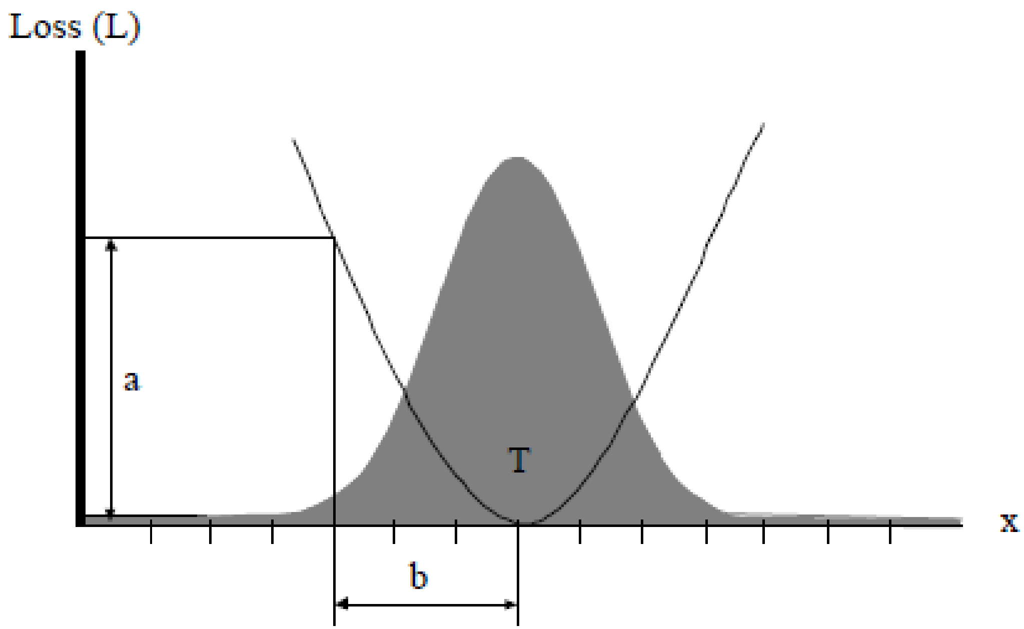
Gauging Techniques

Coordinate Measuring Machine

Taguchi – Customer perception missing

- Monetary losses occur with any deviation from the nominal. – Genichi Taguchi

$$L = \frac{a}{b^2}\left[(x - T)^2 + \sigma^2\right]$$



Introducing quality concepts at the design stage is more valuable than through inspection after manufacture

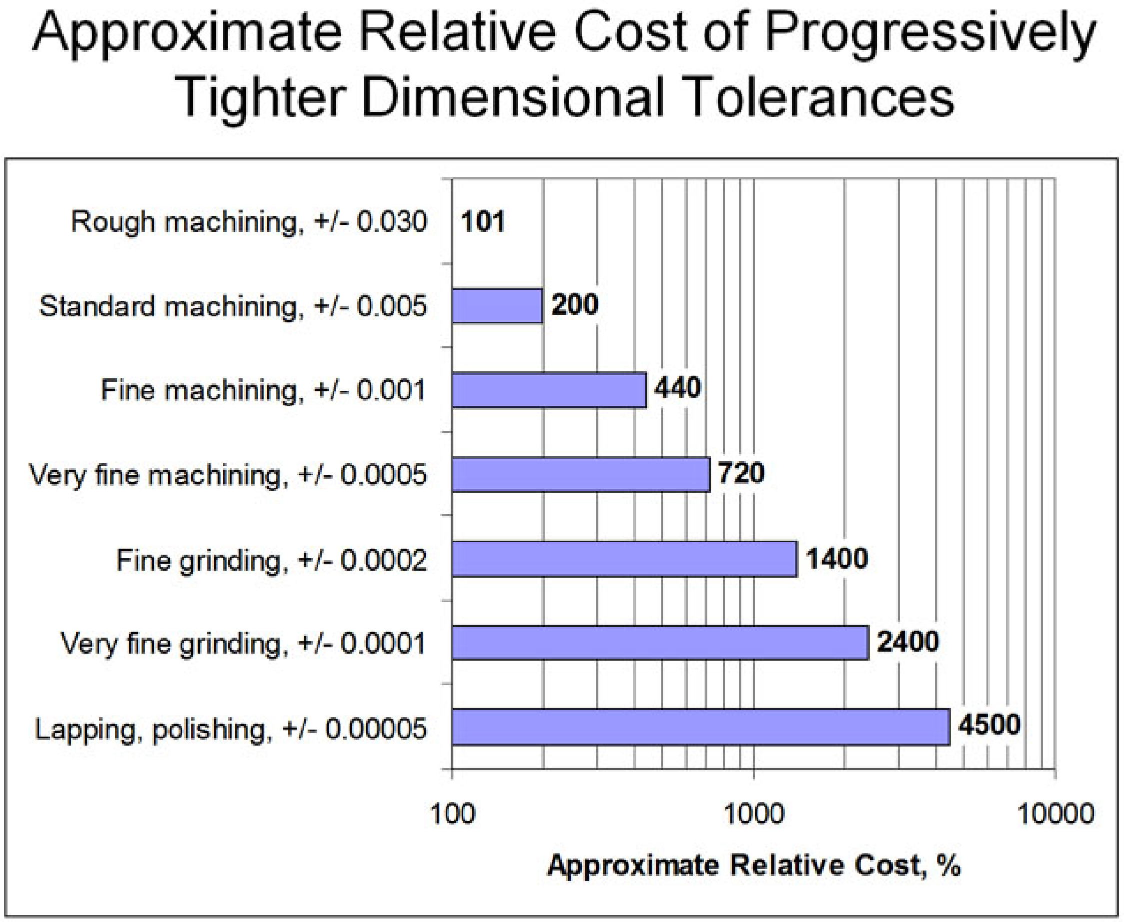
"Standard Rule-of-Thumb" Tolerances

1 decimal place	X.X	± 0.2"
2 decimal places	X.XX	± 0.01"
3 decimal places	X.XXX	± 0.005"
4 decimal places	X.XXXX	± 0.0005"

But be careful with this table, if you are working on a part that measures 3.5" nominal the best tolerance you can expect from milling is 0.005"

- Prescribe the largest tolerances you can afford
  - Budget trade-off
    - Tighter tolerances = more performance (usually)
    - Looser tolerances = less expensive manufacturing (usually)
    - But of course there is Taguchi…
  - Generally, most parts require only a few features to be held to high accuracy

Tolerances/Surface Finish & Costs



N.E.Woldman, Machinability and Machining of Metals [9]

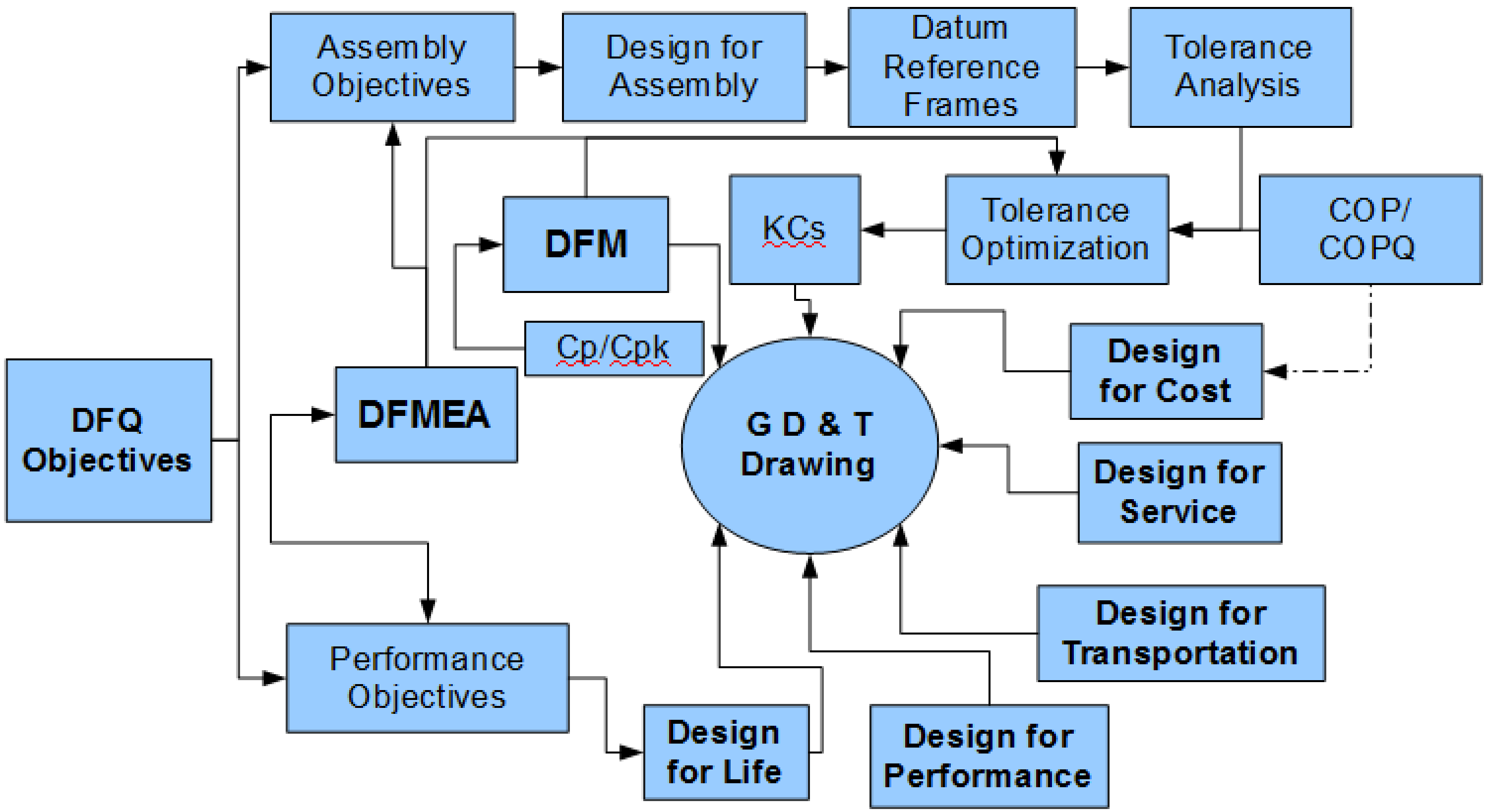
Surface Finish →

Quality

- Meets the need of the customer and thereby provides product satisfaction
- Freedom from deficiencies – absence of defects.

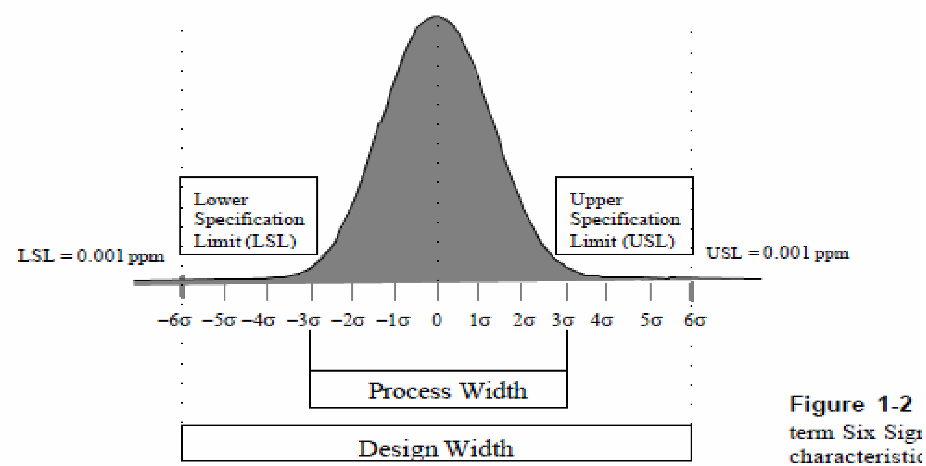
- Standards
  - ISO 9000 series

Design for Quality



Six Sigma

- There is a direct correlation between the number of product defects, wasted operating costs, and the level of customer satisfaction.
- **The Six Sigma measures the capability of the process to perform *defect-free* work.**
  - DFQ Objective: defects per unit
    - Component
    - Piece of Material
    - Line of Code
    - Administrative form
    - Time Frame
    - Distance



Single critical-to-quality (CTQ) characteristic

Cp

- $C_p \equiv$ Capability index or Concurrent Engineering Index: Design / Manufacturing
  - $C_p = \frac{Spec\ Limits}{Process\ Capability} = \frac{USL - LSL}{\pm a\sigma} \Rightarrow \sigma = \frac{T}{aC_p}$
    - Automotive: $C_p = 1.33$

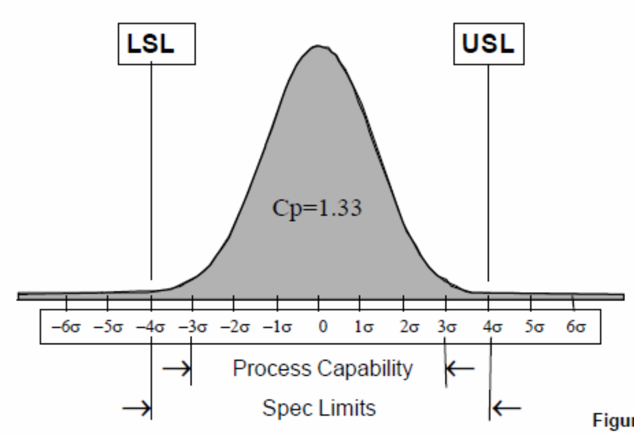


Figure 10-15

Ford motor company 305 BOSS:
Rod and main bores manufacturing tolerances ±.0003”.

- Crankshaft
  - Diameter ≈ 2" ± 0.0003“
  - $\Rightarrow Process\ Capability = 0.00025$“

Cp

- $C_p \equiv$ Capability index or Concurrent Engineering Index: Design / Manufacturing
  - $C_p = \dfrac{Spec\ Limits}{Process\ Capability} = \dfrac{USL - LSL}{\pm a\sigma}$
    - Automotive: $C_p = 1.33$

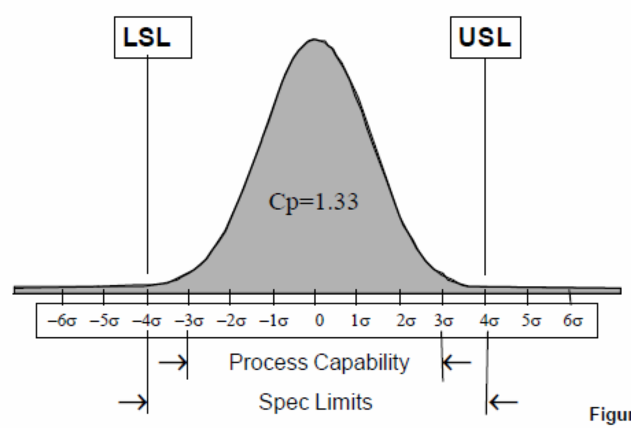


Figure 10-15

Cpk

- $C_{pk} \equiv$ Process capability index adjusted for centering
- $C_{pk} = C_p(1 - k)$
  - $k \equiv$ ratio of the amount the center has moved off target divided by the amount from the center to the nearest specification limit

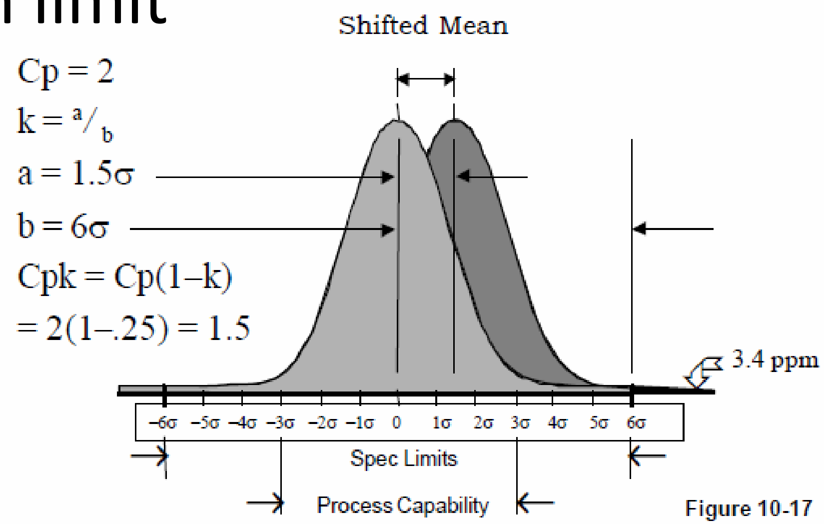


Figure 10-17 Cp and Cpk at Six Sigma

Some Statistical Quality measures

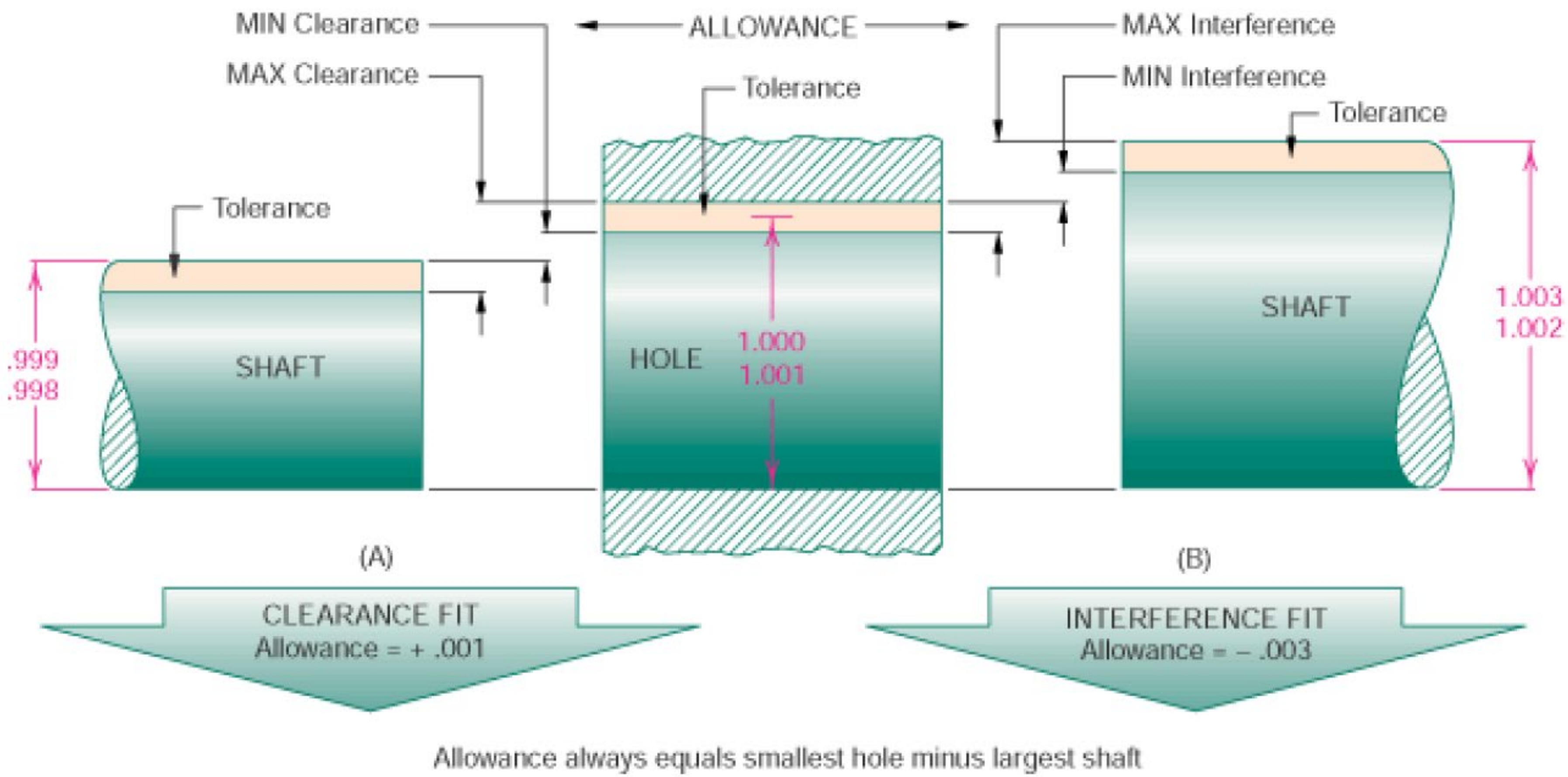
- $C_p = \frac{U-L}{6\sigma}$, measure of the spread of the population about the average
- $C_{pk} = \min(C_{pl}, C_{pu})$, measure of both the location and spread of the population
  - $C_{pl} = \frac{\mu-L}{3\sigma}$
  - $C_{pu} = \frac{U-\mu}{3\sigma}$
- $C_c = \max(C_{cl}, C_{cu})$, measure of the location of the average of the population form the target value
  - $C_{cl} = \frac{\tau-\mu}{\tau-L}$
  - $C_{cu} = \frac{\mu-\tau}{U-\tau}$
- $C_{pm} = \frac{U-L}{6\sqrt{\sigma^2+(\mu-\tau)^2}}$, root-mean-square (RMS) deviation index (closely related to a Taguchi quadratic cost function)

Six Sigma

Sigma	Defects per Million	Cost of Poor Quality	
6 Sigma	3.4	<10% of sales	World Class
5 Sigma	233	10-15% of sales	
4 Sigma	6210	15-20% of sales	Industry Average
3 Sigma	66,807	20-30% of sales	
2 Sigma	308,537	30-40% of sales	Noncompetitive
1 Sigma	690,000		

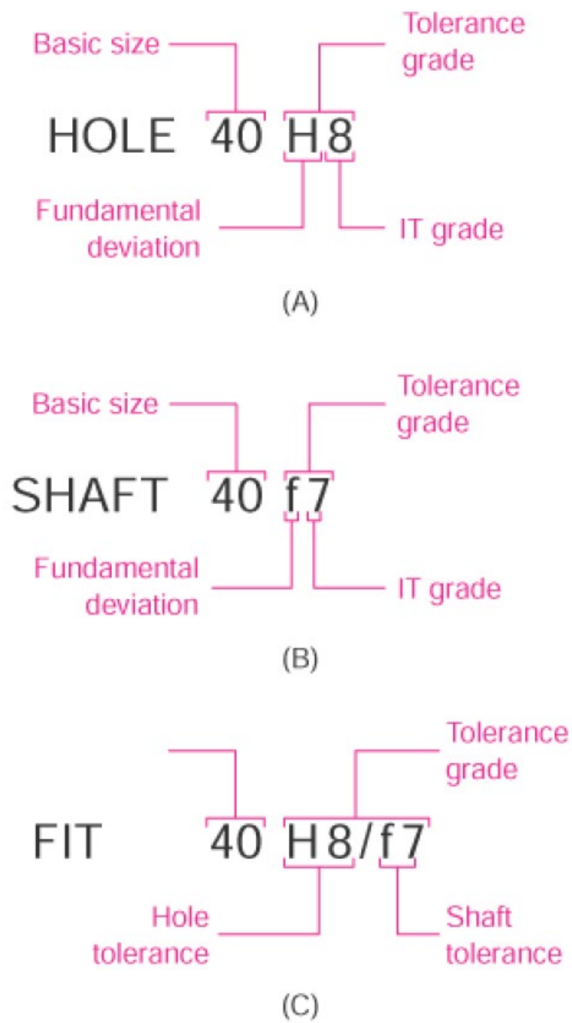
Fits

-

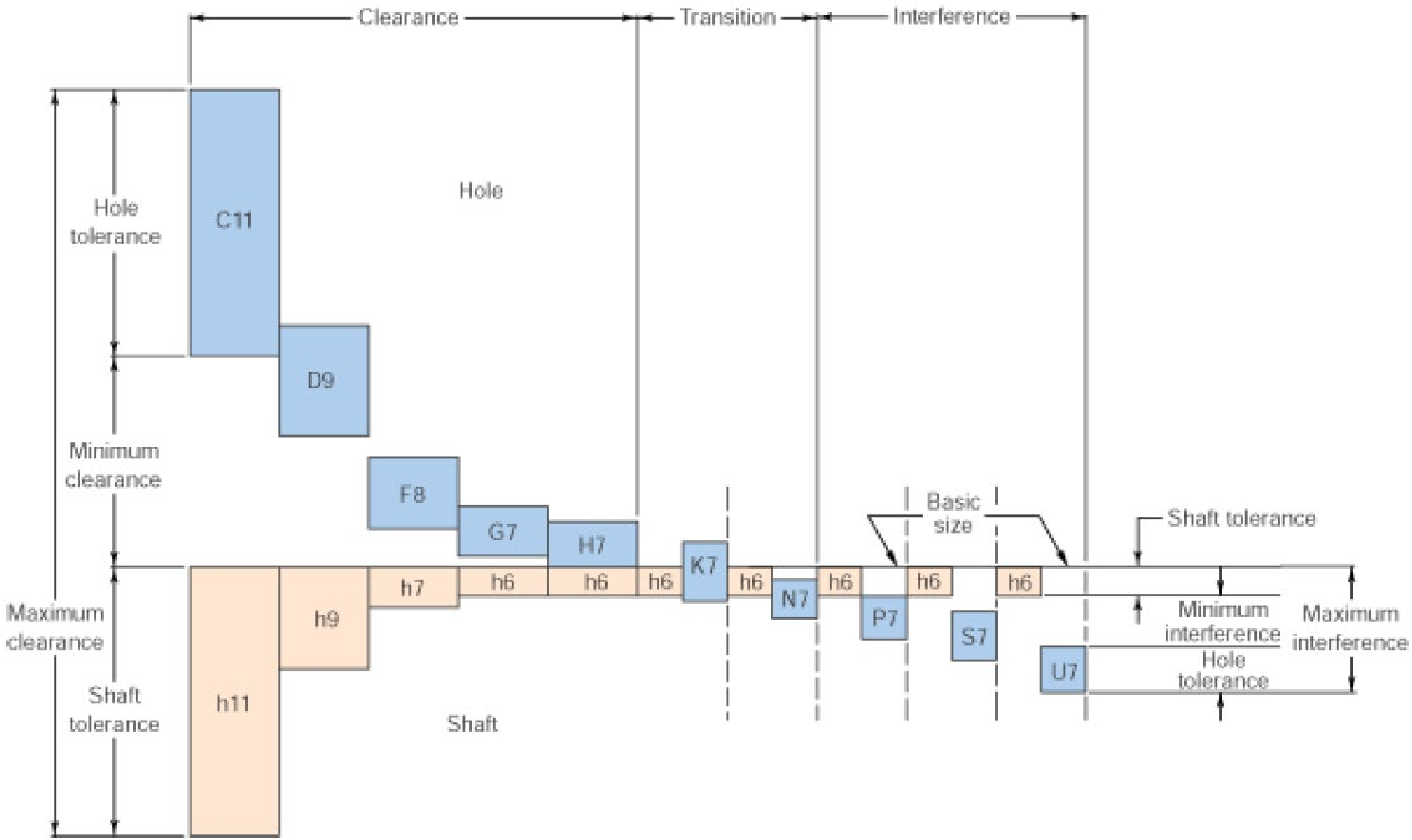


MIN Clearance
ALLOWANCE
MAX Interference
MAX Clearance
Tolerance
MIN Interference
Tolerance
Tolerance
SHAFT
.999
.998
HOLE
1.000
1.001
SHAFT
1.003
1.002
(A)
(B)
CLEARANCE FIT
Allowance = + .001
INTERFERENCE FIT
Allowance = – .003
Allowance always equals smallest hole minus largest shaft

Fits General

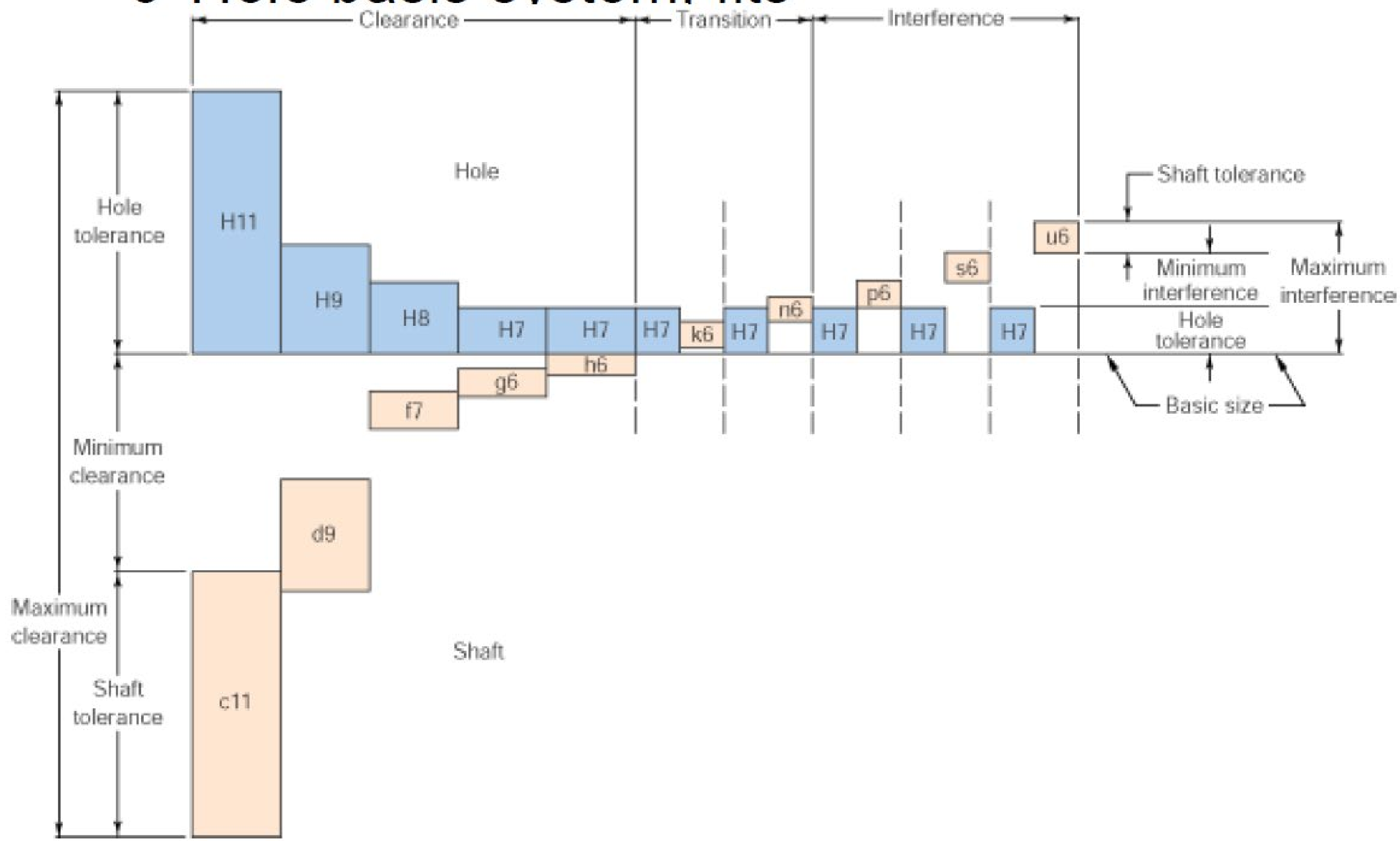


Fits Shaft Driven (Shaft basis System)



Fits Hole Driven

- Hole basis system: fits



Fits ANSI

Table 8a. American National Standard Running and Sliding Fits *ANSI B4.1-1967 (R2009)*

Nominal Size Range, Inches		Class RC 1			Class RC 2			Class RC 3			Class RC 4		
			Standard Tolerance Limits			Standard Tolerance Limits			Standard Tolerance Limits			Standard Tolerance Limits	
Over	To	Clearance[a]	Hole H5	Shaft g4	Clearance[a]	Hole H6	Shaft g5	Clearance[a]	Hole H7	Shaft f6	Clearance[a]	Hole H8	Shaft f7
		Values shown below are in thousandths of an inch											
0	0.12	0.1	+0.2	−0.1	0.1	+0.25	−0.1	0.3	+0.4	−0.3	0.3	+0.6	−0.3
		0.45	0	−0.25	0.55	0	−0.3	0.95	0	−0.55	1.3	0	−0.7
0.12	0.24	0.15	+0.2	−0.15	0.15	+0.3	−0.15	0.4	+0.5	−0.4	0.4	+0.7	−0.4
		0.5	0	−0.3	0.65	0	−0.35	1.2	0	−0.7	1.6	0	−0.9
0.24	0.40	0.2	+0.25	−0.2	0.2	+0.4	−0.2	0.5	+0.6	−0.5	0.5	+0.9	−0.5
		0.6	0	−0.35	0.85	0	−0.45	1.5	0	−0.9	2.0	0	−1.1
0.40	0.71	0.25	+0.3	−0.25	0.25	+0.4	−0.25	0.6	+0.7	−0.6	0.6	+1.0	−0.6
		0.75	0	−0.45	0.95	0	−0.55	1.7	0	−1.0	2.3	0	−1.3
0.71	1.19	0.3	+0.4	−0.3	0.3	+0.5	−0.3	0.8	+0.8	−0.8	0.8	+1.2	−0.8
		0.95	0	−0.55	1.2	0	−0.7	2.1	0	−1.3	2.8	0	−1.6
1.19	1.97	0.4	+0.4	−0.4	0.4	+0.6	−0.4	1.0	+1.0	−1.0	1.0	+1.6	−1.0
		1.1	0	−0.7	1.4	0	−0.8	2.6	0	−1.6	3.6	0	−2.0
1.97	3.15	0.4	+0.5	−0.4	0.4	+0.7	−0.4	1.2	+1.2	−1.2	1.2	+1.8	−1.2
		1.2	0	−0.7	1.6	0	−0.9	3.1	0	−1.9	4.2	0	−2.4
3.15	4.73	0.5	+0.6	−0.5	0.5	+0.9	−0.5	1.4	+1.4	−1.4	1.4	+2.2	−1.4
		1.5	0	−0.9	2.0	0	−1.1	3.7	0	−2.3	5.0	0	−2.8
4.73	7.09	0.6	+0.7	−0.6	0.6	+1.0	−0.6	1.6	+1.6	−1.6	1.6	+2.5	−1.6
		1.8	0	−1.1	2.3	0	−1.3	4.2	0	−2.6	5.7	0	−3.2
7.09	9.85	0.6	+0.8	−0.6	0.6	+1.2	−0.6	2.0	+1.8	−2.0	2.0	+2.8	−2.0
		2.0	0	−1.2	2.6	0	−1.4	5.0	0	−3.2	6.6	0	−3.8
9.85	12.41	0.8	+0.9	−0.8	0.8	+1.2	−0.8	2.5	+2.0	−2.5	2.5	+3.0	−2.5
		2.3	0	−1.4	2.9	0	−1.7	5.7	0	−3.7	7.5	0	−4.5
12.41	15.75	1.0	+1.0	−1.0	1.0	+1.4	−1.0	3.0	+2.2	−3.0	3.0	+3.5	−3.0
		2.7	0	−1.7	3.4	0	−2.0	6.6	0	−4.4	8.7	0	−5.2
15.75	19.69	1.2	+1.0	−1.2	1.2	+1.6	−1.2	4.0	+2.5	−4.0	4.0	+4.0	−4.0
		3.0	0	−2.0	3.8	0	−2.2	8.1	0	−5.6	10.5	0	−6.5

Fits ISO

Table 1. Description of Preferred Fits

	ISO SYMBOL		DESCRIPTION
	Hole Basis	Shaft Basis	
Clearance Fits	H11/c11	C11/h11	*Loose running* fit for wide commercial tolerances or allowances on external members.
	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.
	H8/f7	F8/h7	*Close Running* fit for running on accurate machines and for accurate moderate speeds and journal pressures.
	H7/g6	G7/h6	*Sliding fit* not intended to run freely, but to move and turn freely and locate accurately.
	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.
	H7/n6	N7/h6	*Locational transition* fit for more accurate location where greater interference is permissible.
Interference Fits	H7/p6[a]	P7/h6	*Locational interference* fit for parts requiring rigidity and alignment with prime accuracy of location but without special bore pressure requirements.
	H7/s6	S7/h6	*Medium drive* fit for ordinary steel parts or shrink fits on light sections, the tightest fit usable with cast iron.
	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.

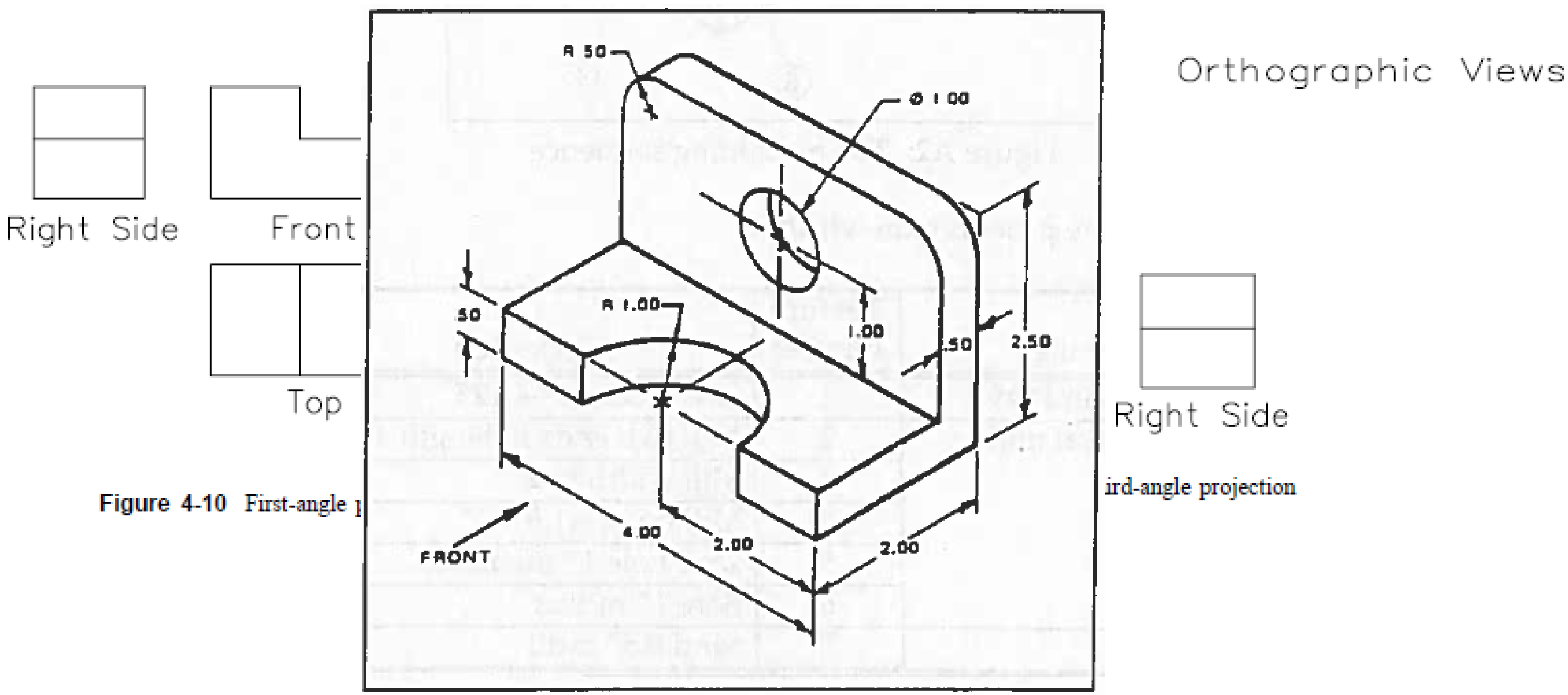
↑ More Clearance

More Interference ↓

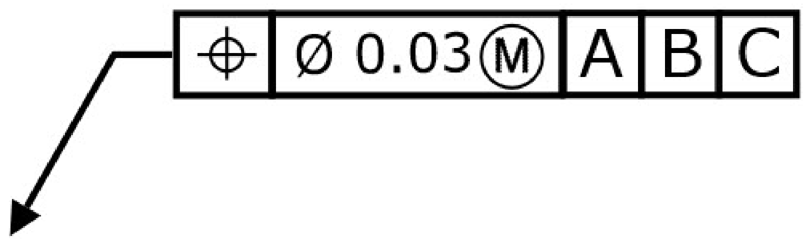
[a]Transition fit for basic sizes in range from 0 through 3 mm.

Dimensioning

- Complete information about both *size* and *shape*
  - Size: Dimensions
  - Shape: Drawings (Usually orthographic)



GD&T

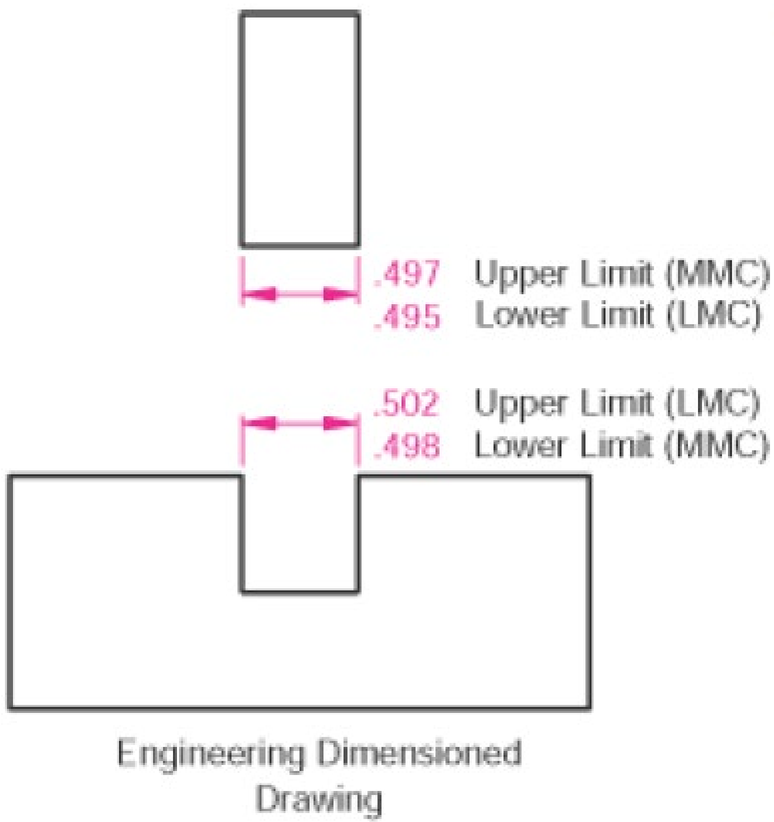


Definitions

- Maximum Material Condition (MMC) Ⓜ
  - The dimension tolerance limit that produces a part that contains the most amount of material for that dimension.

- Least Material Condition (LMC) Ⓛ
  - The dimension tolerance limit that produces a part that contains the least amount of material possible for that dimension.

Definitions



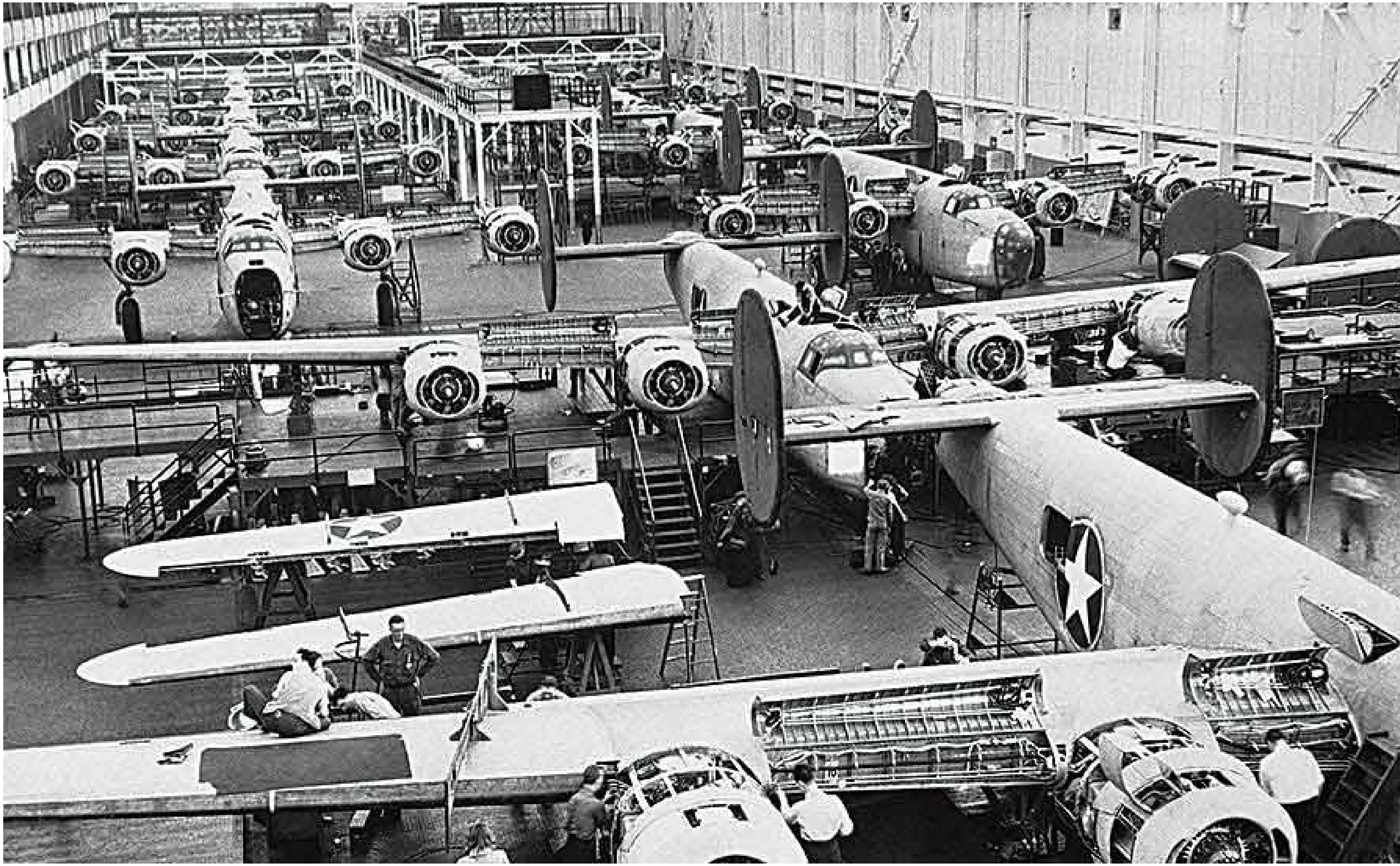
Definitions

- Regardless of Feature Size (RFS) Ⓢ
  - Tolerances apply to a geometric feature regardless of its size. These sizes range from MMC to LMC.

Tolerancing Fits

- Clearance
  - Mating parts always have space or clearance when assembled
- Interference
  - Mating parts always interfere when assembled
- Transition
  - Mating parts will sometimes be interference and sometimes be clearance when assembled

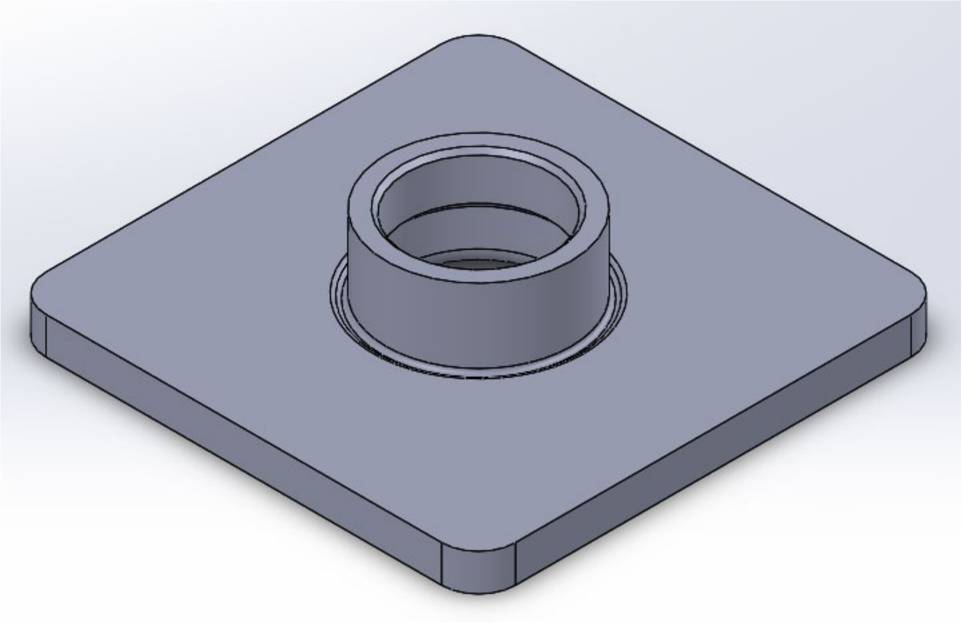
Why?



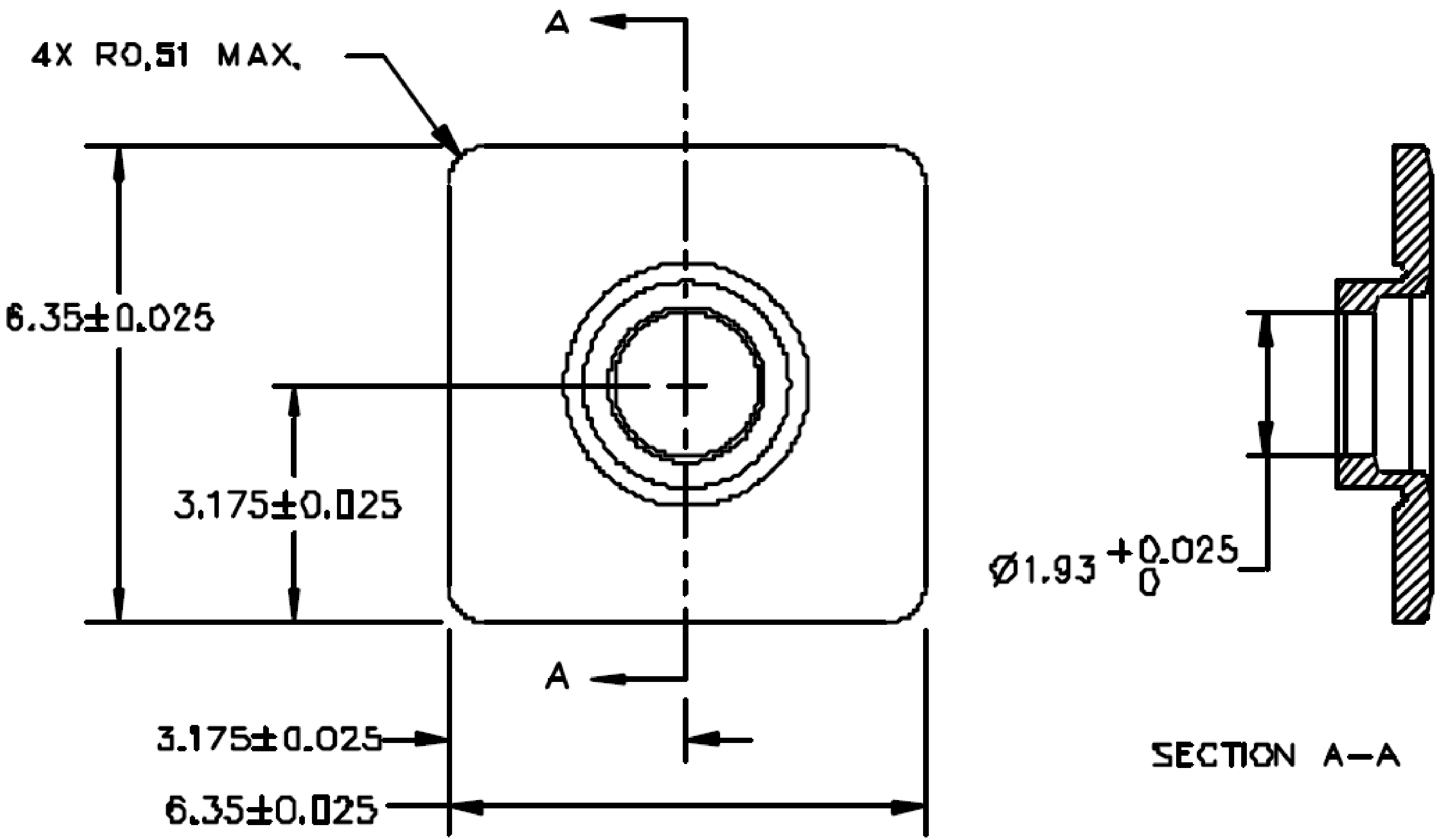
Motivation

Design Intent

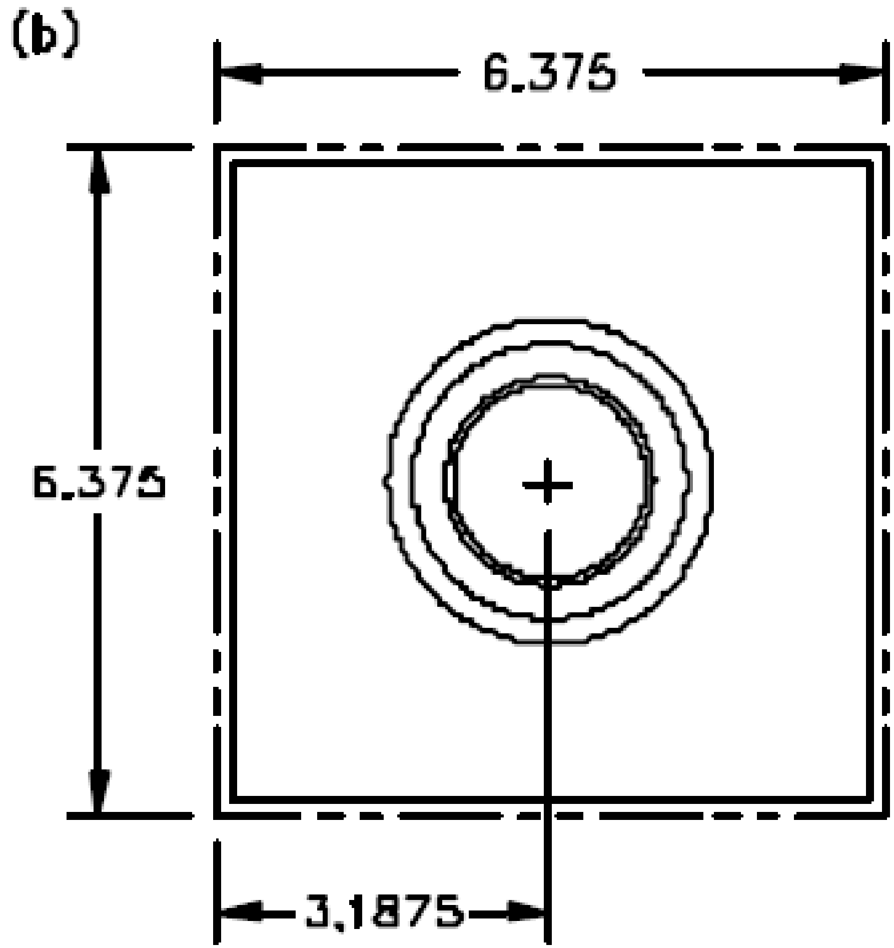
- External boundary 6.35 mm ± 0.025 mm "square"
- Hub inside diameter on "center" of the square within ± 0.025 mm



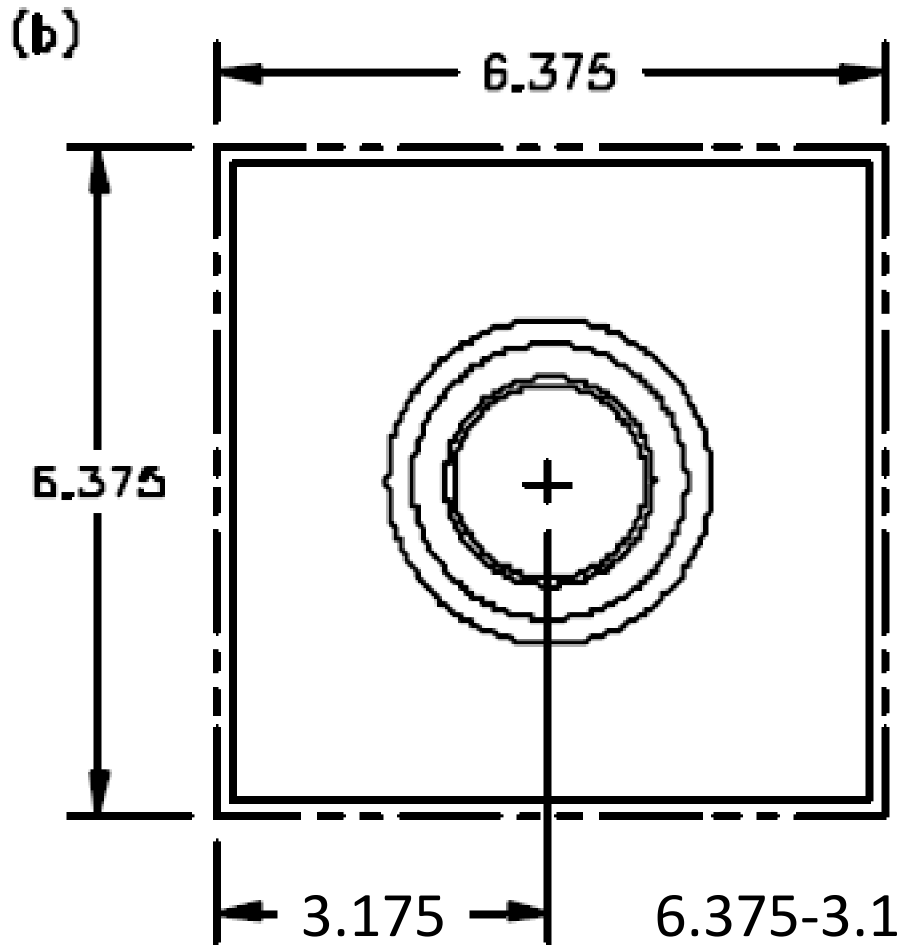
Linear Strategy



Linear Strategy

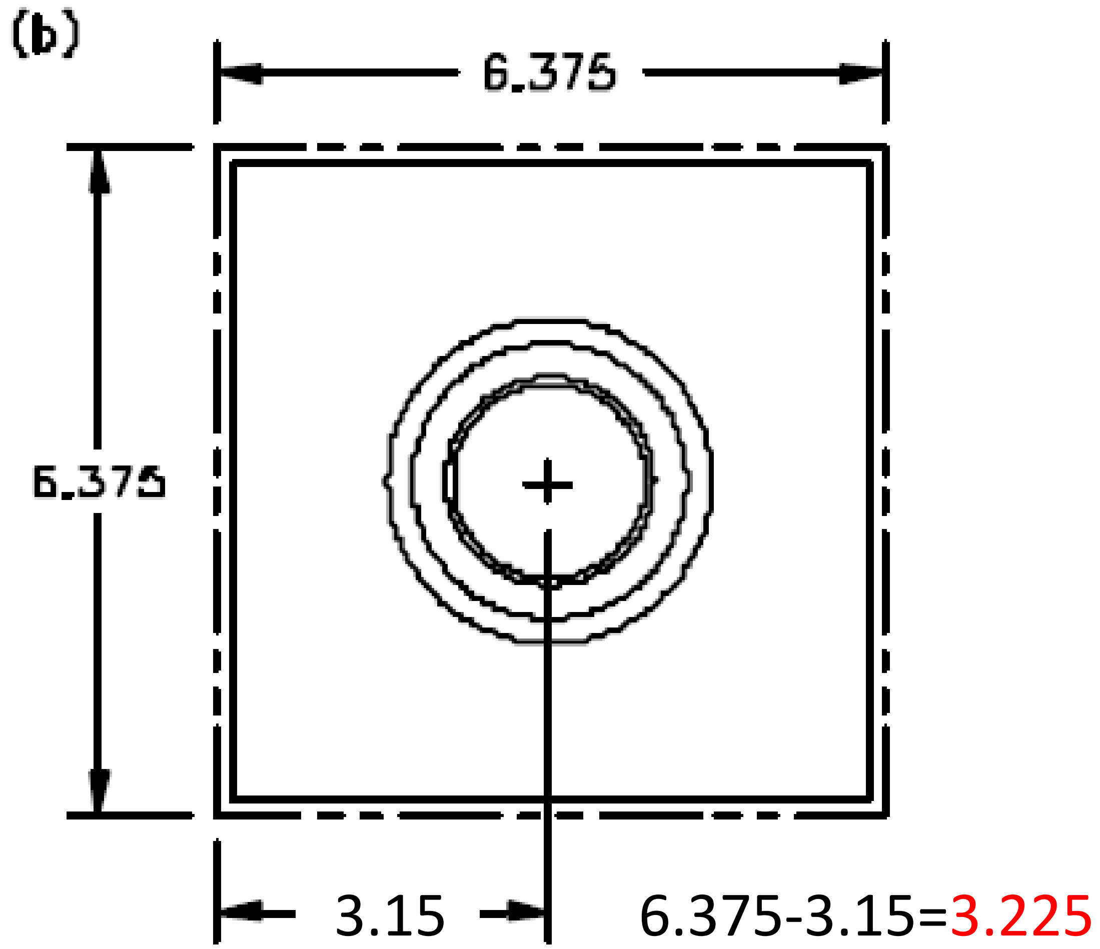


Linear Strategy

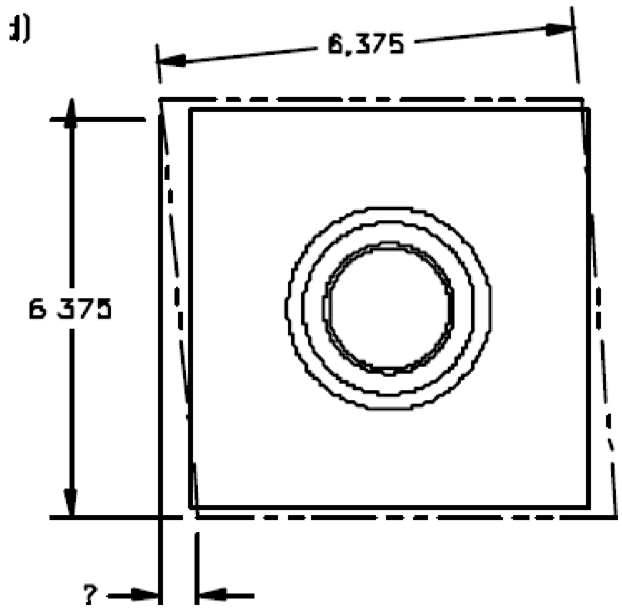


6.375-3.175=3.2

Linear Strategy



Linear Strategy



- **ANSI:** Rule #1 (Taylor Principle) - When only a size tolerance is specified for an individual feature of size the form of this feature shall not extend beyond a boundary (envelope) of perfect form at maximum material condition (MMC).

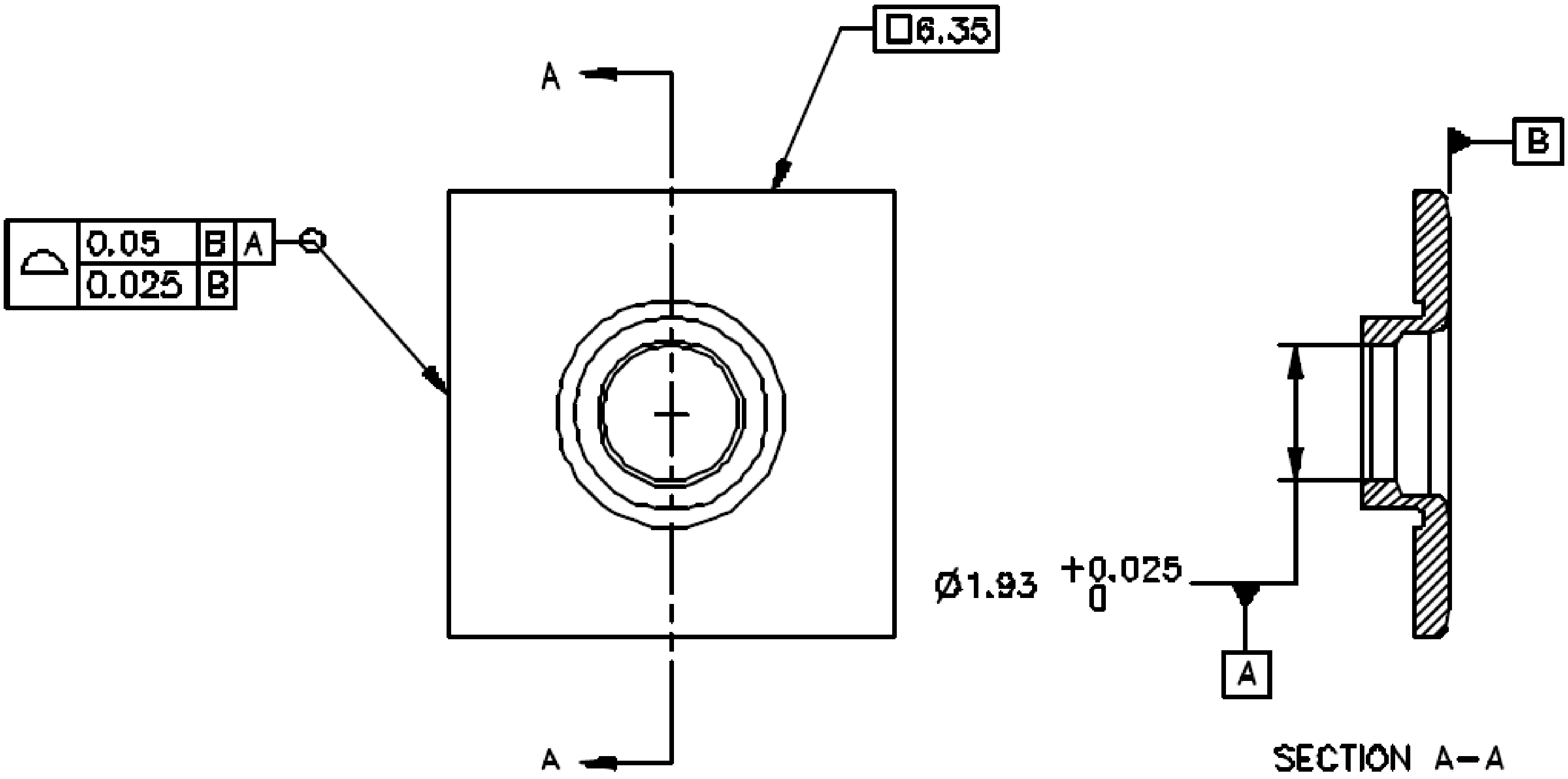
- **ISO:** Principle of Independence

Linear Strategy

- Paragraph 2.7.3 of Y14.5 addresses the “relationship between individual features,” and states:

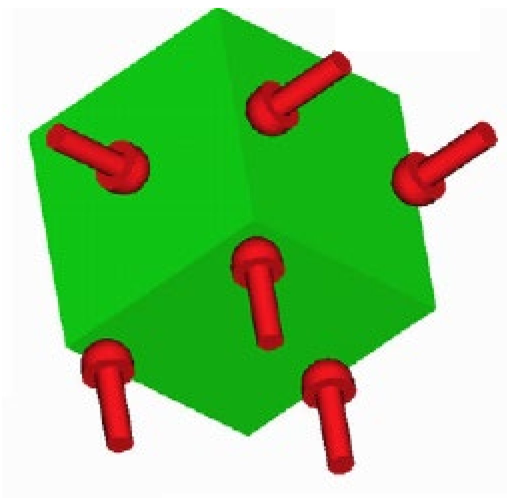
  The limits of size do not control the orientation or location relationship between individual features. Features shown perpendicular, coaxial, or symmetrical to each other must be controlled for location or orientation to avoid incomplete drawing requirements.

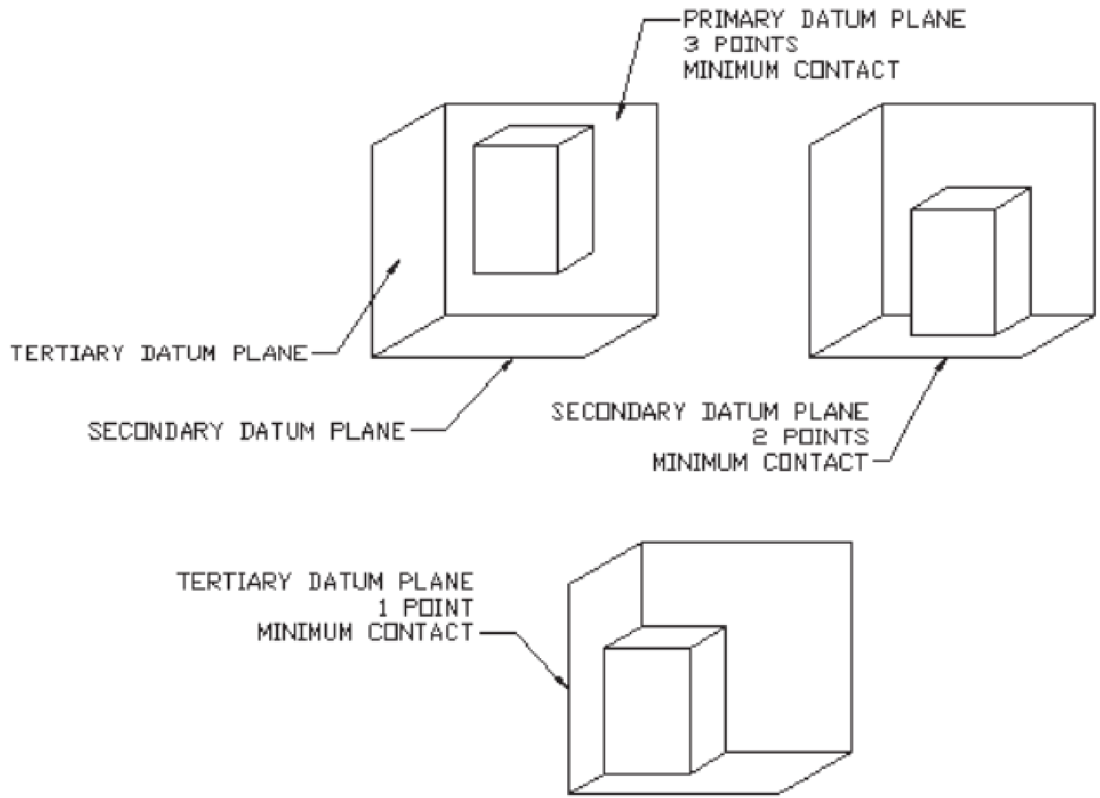
Full Geometric



Datum

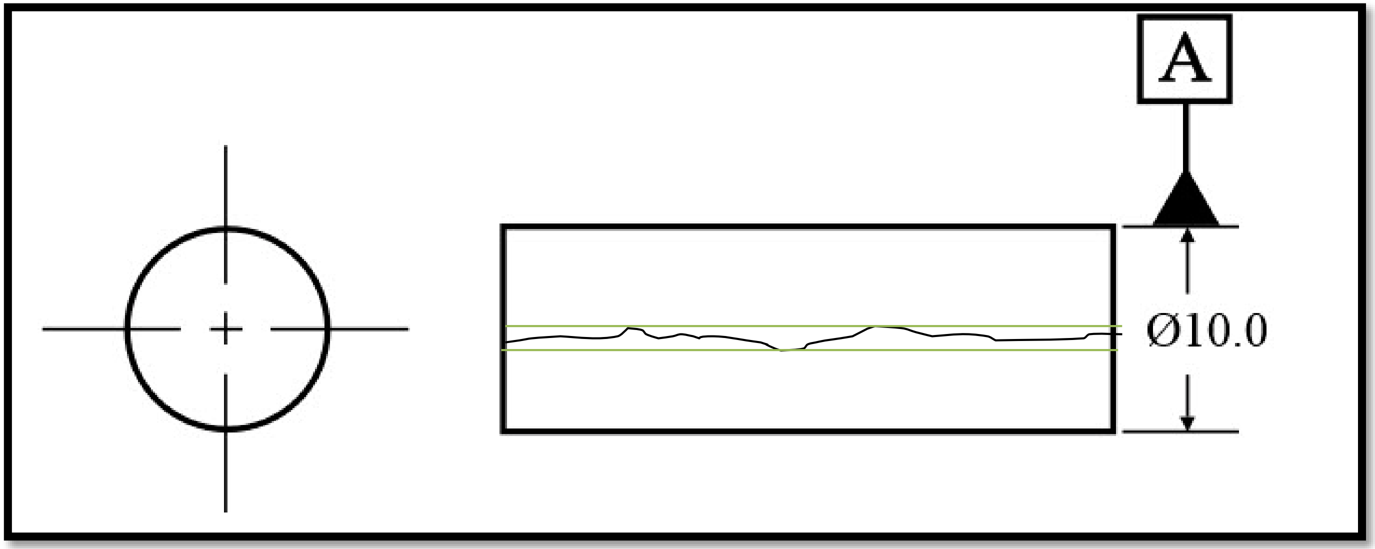
6 DOF



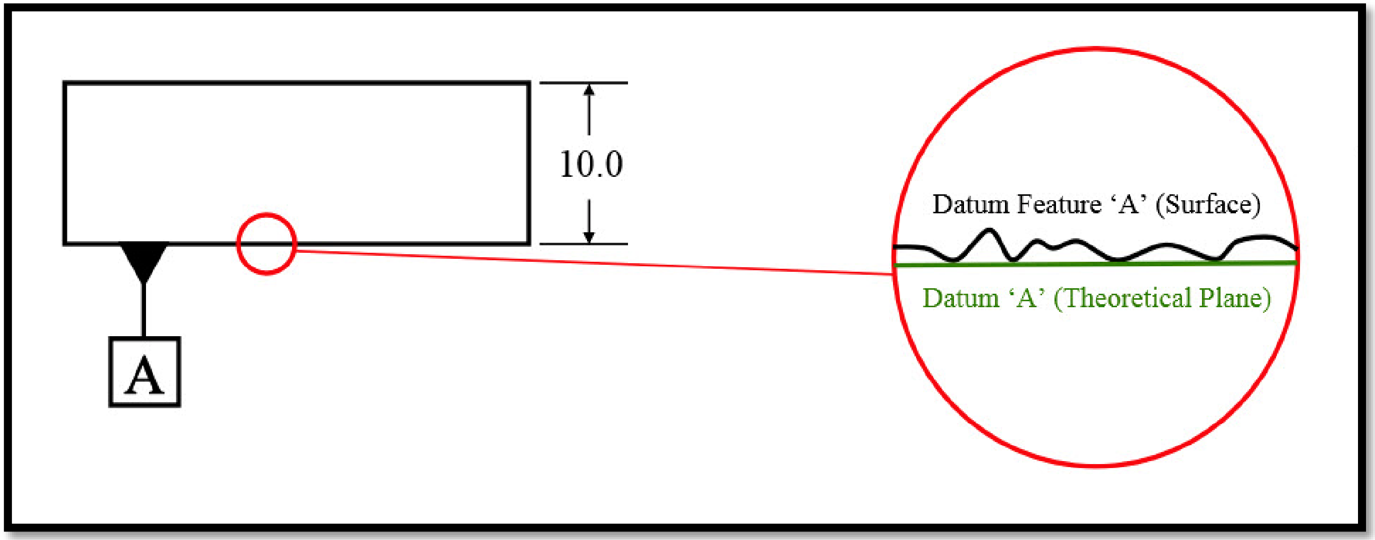

PRIMARY DATUM PLANE
3 POINTS
MINIMUM CONTACT
TERTIARY DATUM PLANE
SECONDARY DATUM PLANE
SECONDARY DATUM PLANE
2 POINTS
MINIMUM CONTACT
TERTIARY DATUM PLANE
1 POINT
MINIMUM CONTACT

Theory Vs Reality

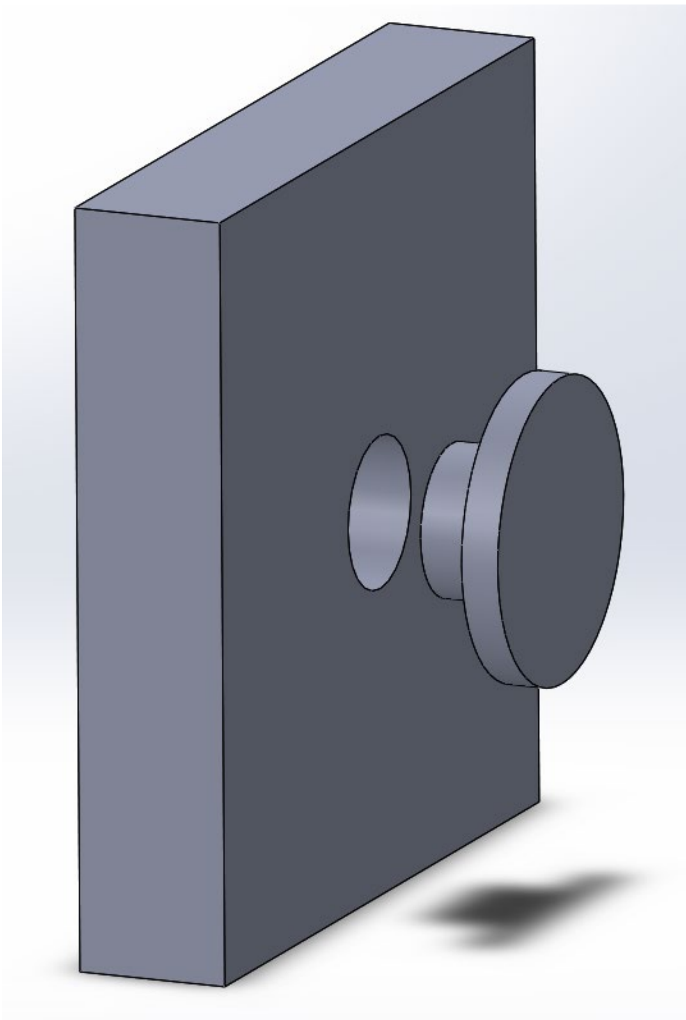
Axis

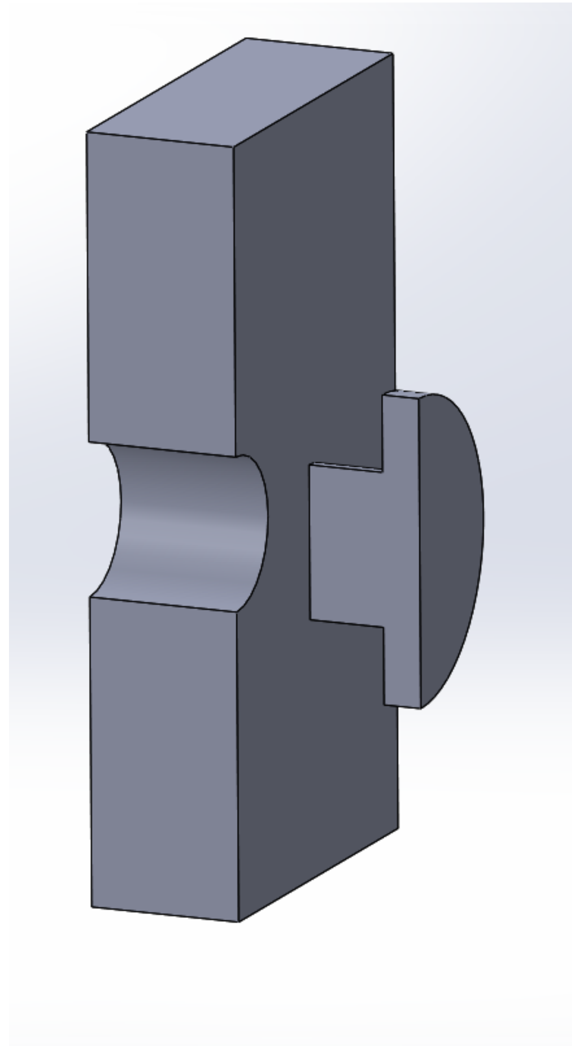


Plane

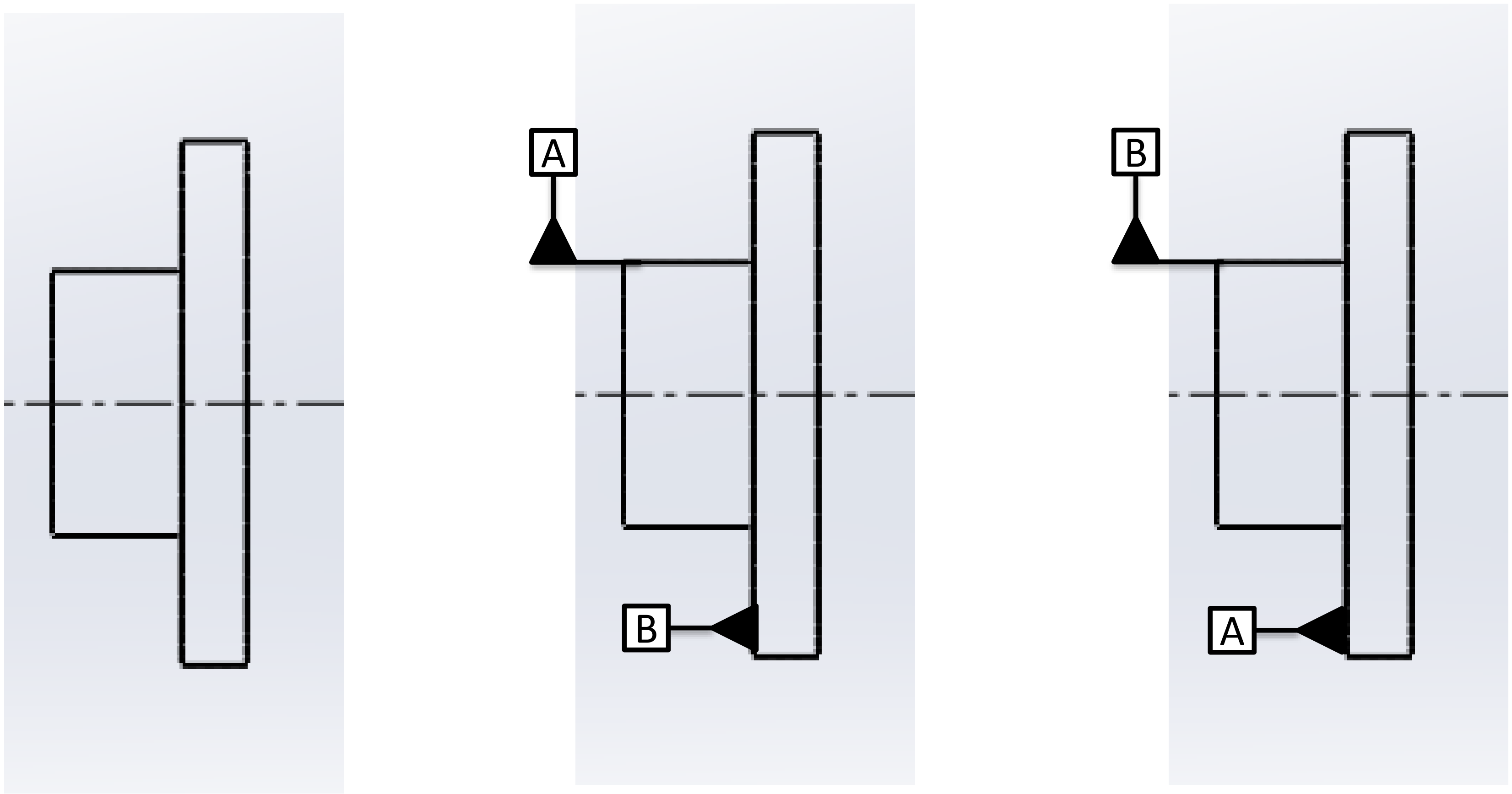


Datums and Design Intent

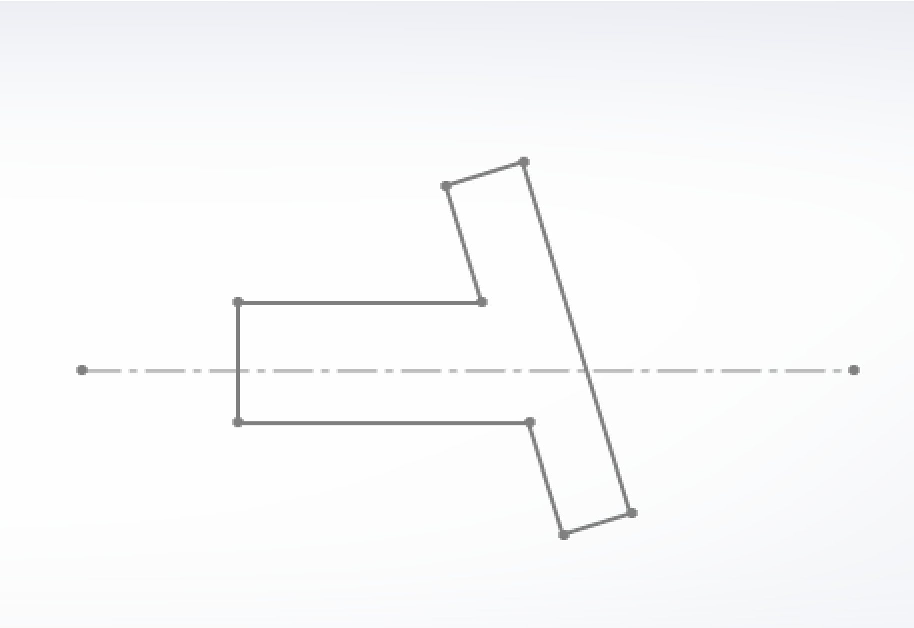




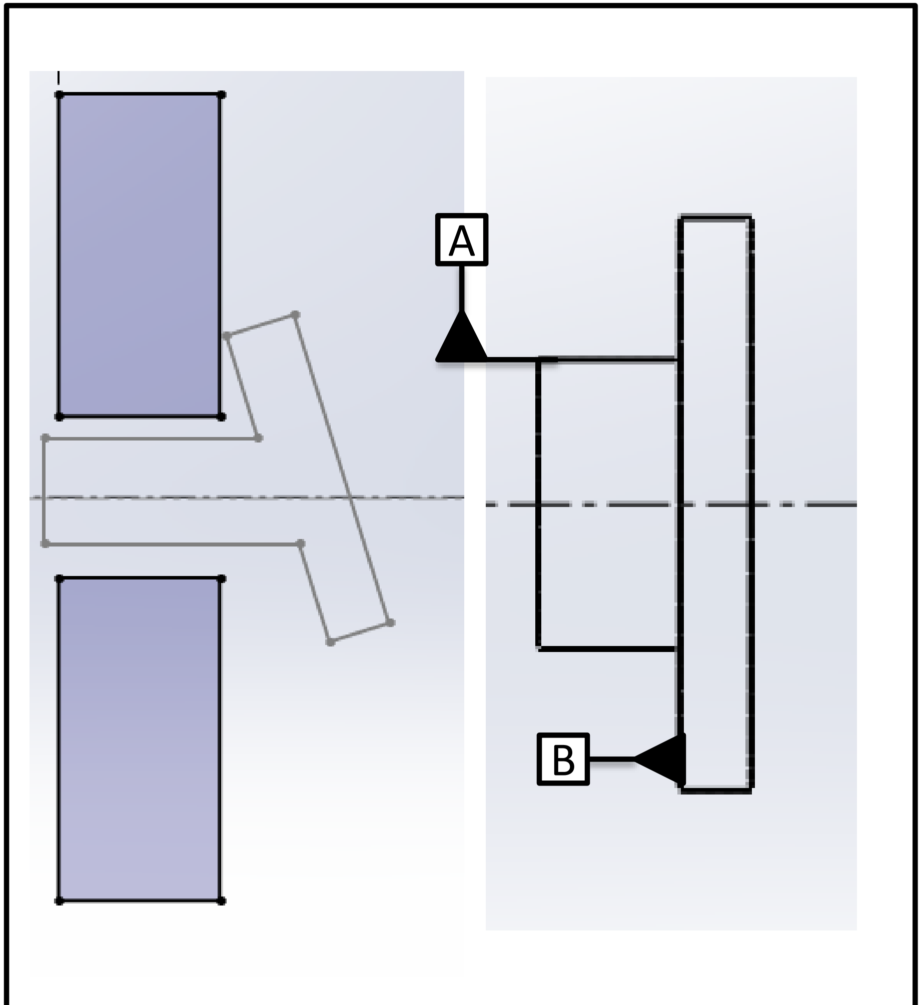
Datum Choices

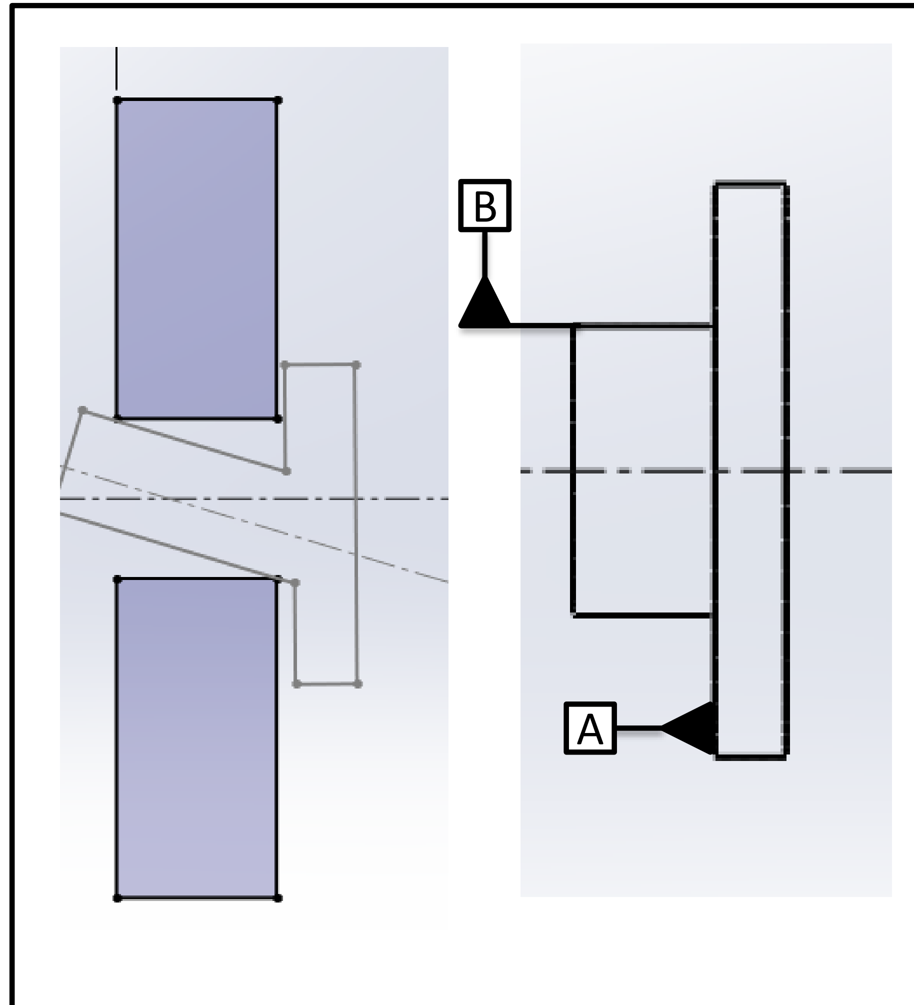


As built



Datum Intent

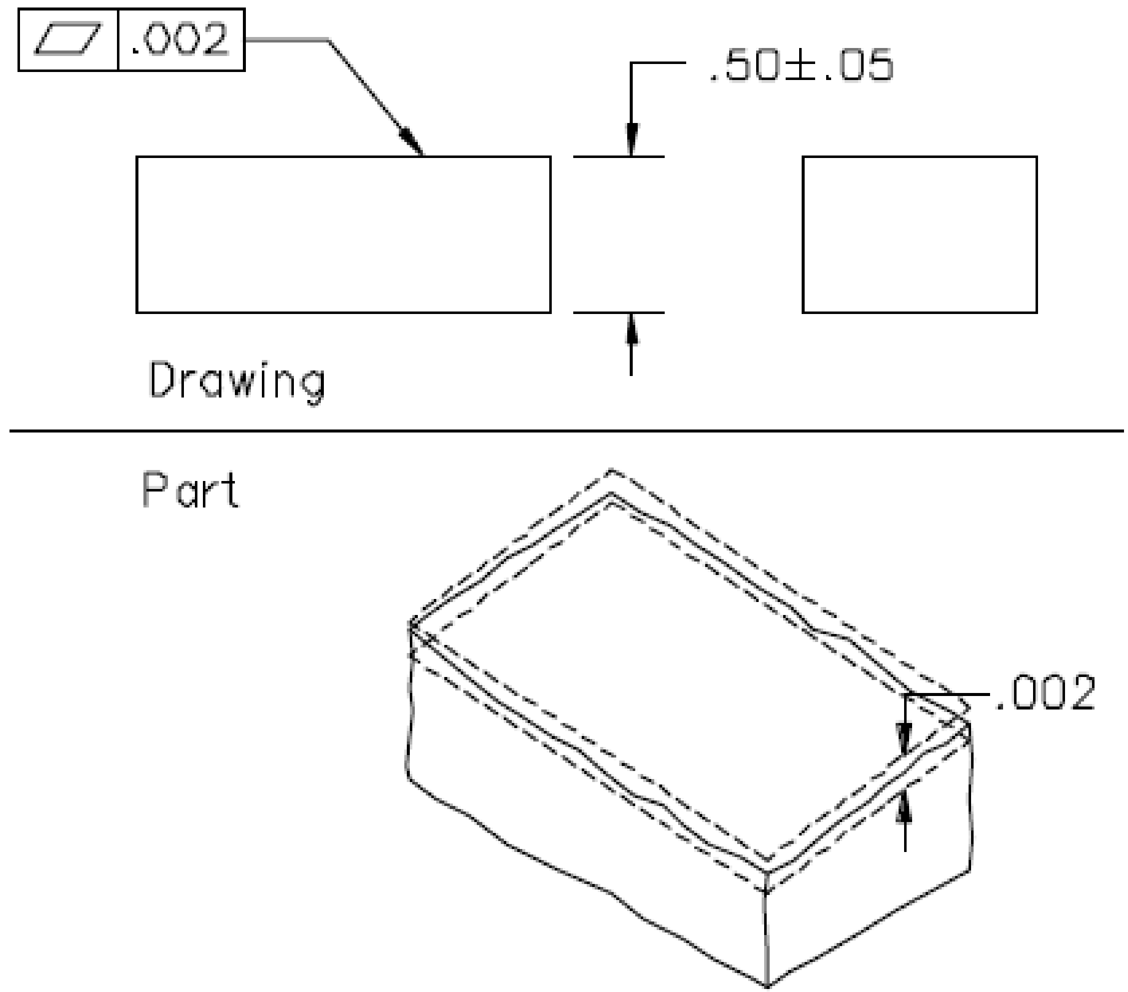




Some Controls

CHARACTERISTIC	SYMBOL	TYPE OF FEATURE CONTROLLED	FEATURE CONTROL FRAME PLACEMENT OPTIONS (SEE LEGEND)	BOUNDARY/TOL ZONE SHAPE MODIFIER	TOLERANCE MODIFIABLE TO MMC OR LMC	NUMBER OF DATUM REFERENCES ALLOWED	MMC/LMC ALLOWED FOR DATUM REFERENCE(S)	BASIC DIMENSIONS REQ'D
STRAIGHTNESS	—	CYL-SURFACE ELEMENTS	b			0		
		CYL-DERIVED MEDIAN LINE	a, d	Ø	✓	0		
		PLANE-LINE ELEMENTS	b, c			0,1		
FLATNESS	⏥	PLANE	b, c			0		
		WIDTH-DERIVED MEDIAN PLANE	a, d		✓	0		
CIRCULARITY	○	REVOLUTE, SPHERE	a, b, d			0		
CYLINDRICITY	⌭	CYLINDER	a, b, d			0		
PROFILE OF A LINE	⌒	ALL	b			0–3	✓	✓
PROFILE OF A SURFACE	⌓	REVOLUTE	b			0–3	✓	✓
		OTHER (NON-REVOLUTE)	b			0–3	✓	✓
		COPLANARITY OF PLANES	b			0		
PERPENDICULARITY PARALLELISM	⊥ //	PLANE (INCL LINE ELEMENTS)	b, c			1–3	✓	
		CYLINDER	a, d	Ø	✓	1–3	✓	
		WIDTH	a, d		✓	1–3	✓	
		REVOLUTE-RADIAL ELEMENT	b, c			1–3	✓	
ANGULARITY	∠	PLANE (INCL LINE ELEMENTS)	b ,c			1–3	✓	✓
		CYLINDER	a, d	Ø	✓	1–3	✓	✓
		WIDTH	a, d		✓	1–3	✓	✓
		REVOLUTE-RADIAL ELEMENT	b, c			1–3	✓	✓
POSITION	⌖	CYLINDER	a, d	Ø	✓	1–3	✓	✓
		WIDTH	a, d		✓	1–3	✓	✓
		SPHERE	a, d	SØ	✓	1–3	✓	✓
CONCENTRICITY	◎	ALL NON-SPHERICAL	a, b, d	Ø		1–3		
		SPHERE	a, b, d	SØ		1–3		
SYMMETRY	⌯	OPPOSED POINTS	a, d			1–3		
CIRCULAR RUNOUT	↗	REVOLUTE	a, b, d			1–2		
TOTAL RUNOUT	⌰	CYLINDER	a, b, d			1–2		
		PLANE PERP TO AXIS	b, c			1–2		

Flatness



Circularity

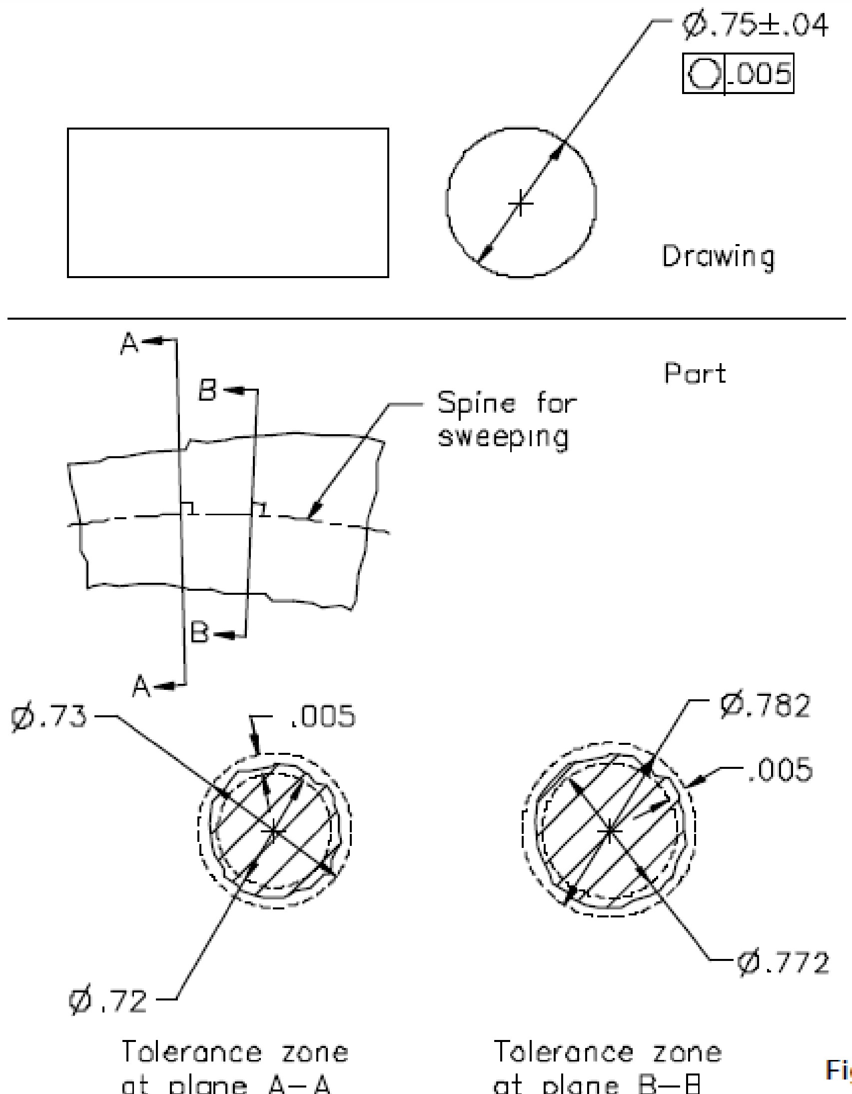
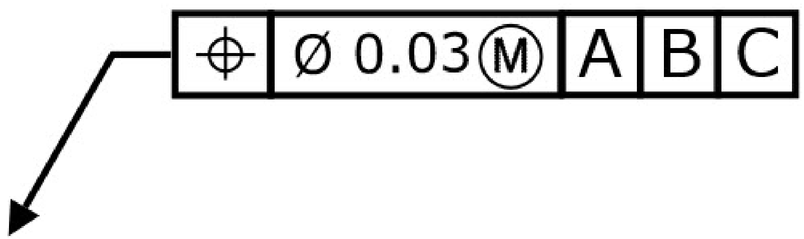
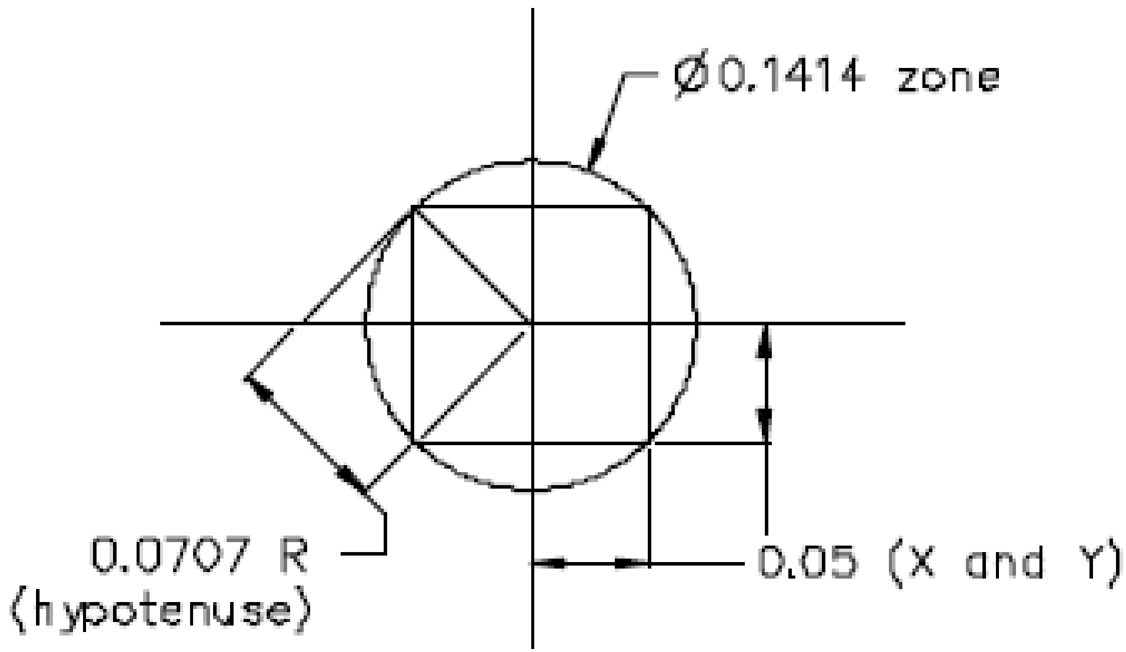


Fig
non

GD&T



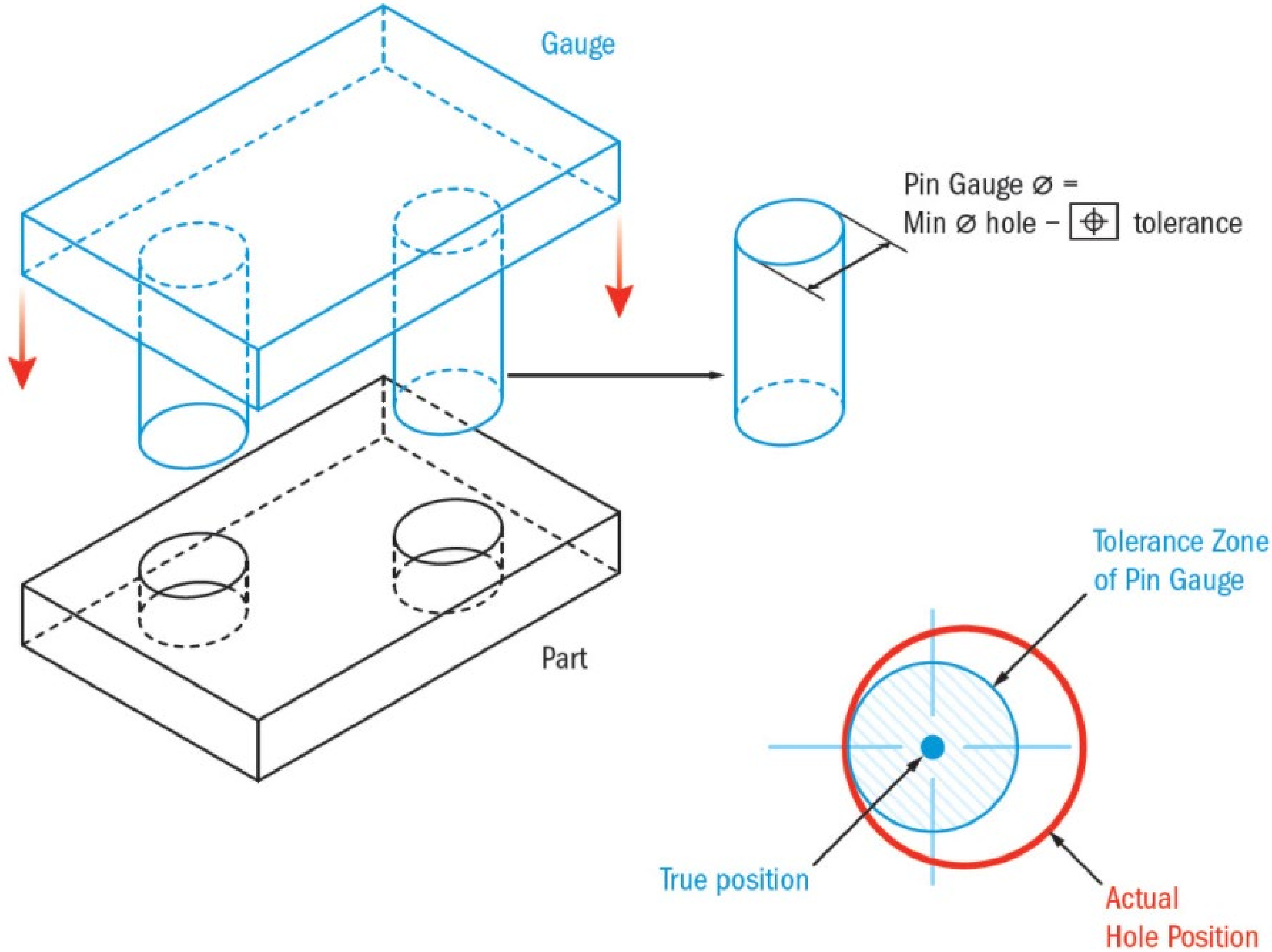
Tolerance Zone



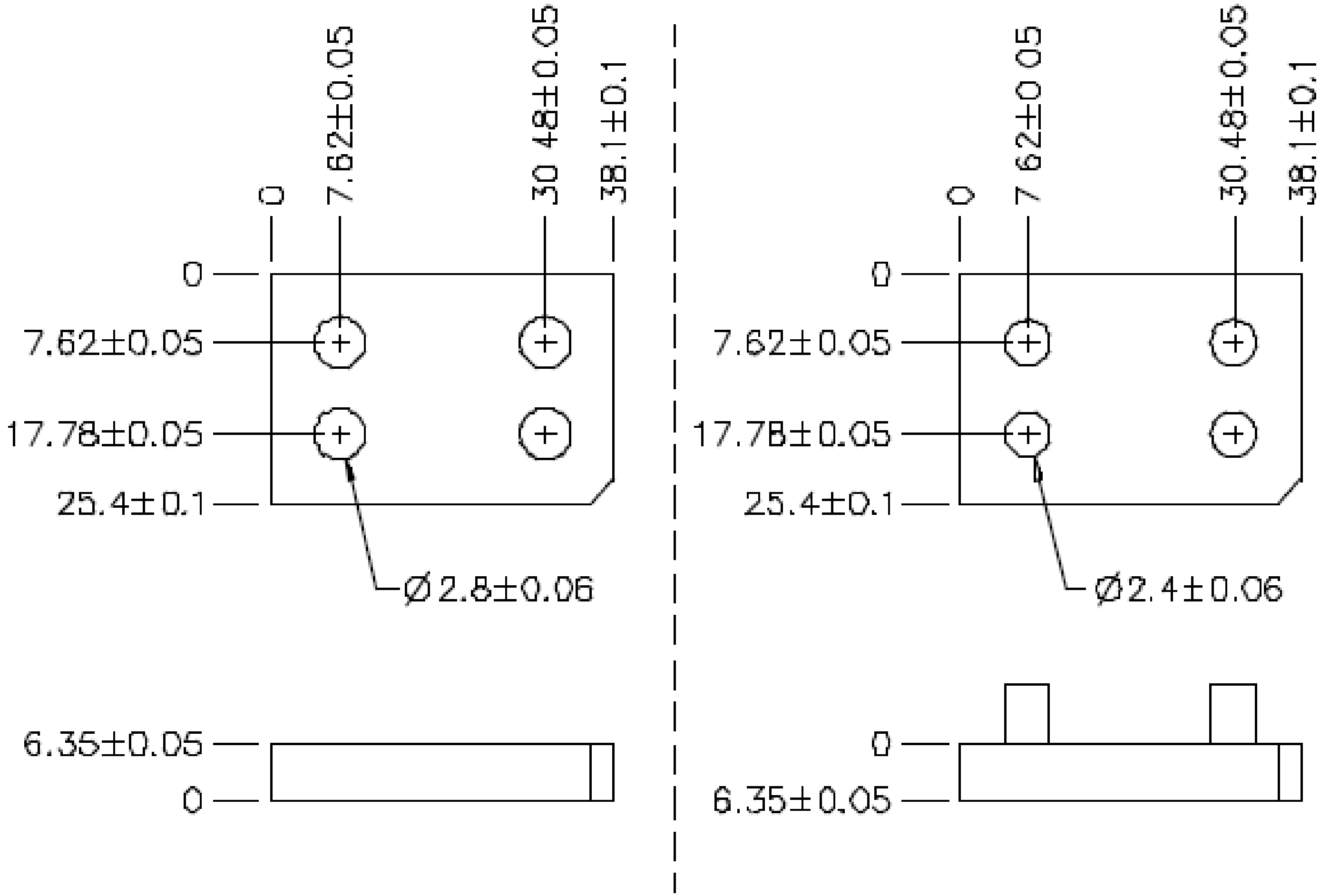
Location

- **<u>4 Pins & 4 Holes</u>**
- Pins: 2.4 +/-0.06 = **2.46 & 2.34** (Δ = 0.12)
- Holes: 2.8 +/-0.06 = **2.86 & 2.74** (Δ = 0.12)
- Square Size = 0.05 ➔ Dia = 0.07070 (x2 = **0.14**)
- 0.14 + 0.12 = **0.26**

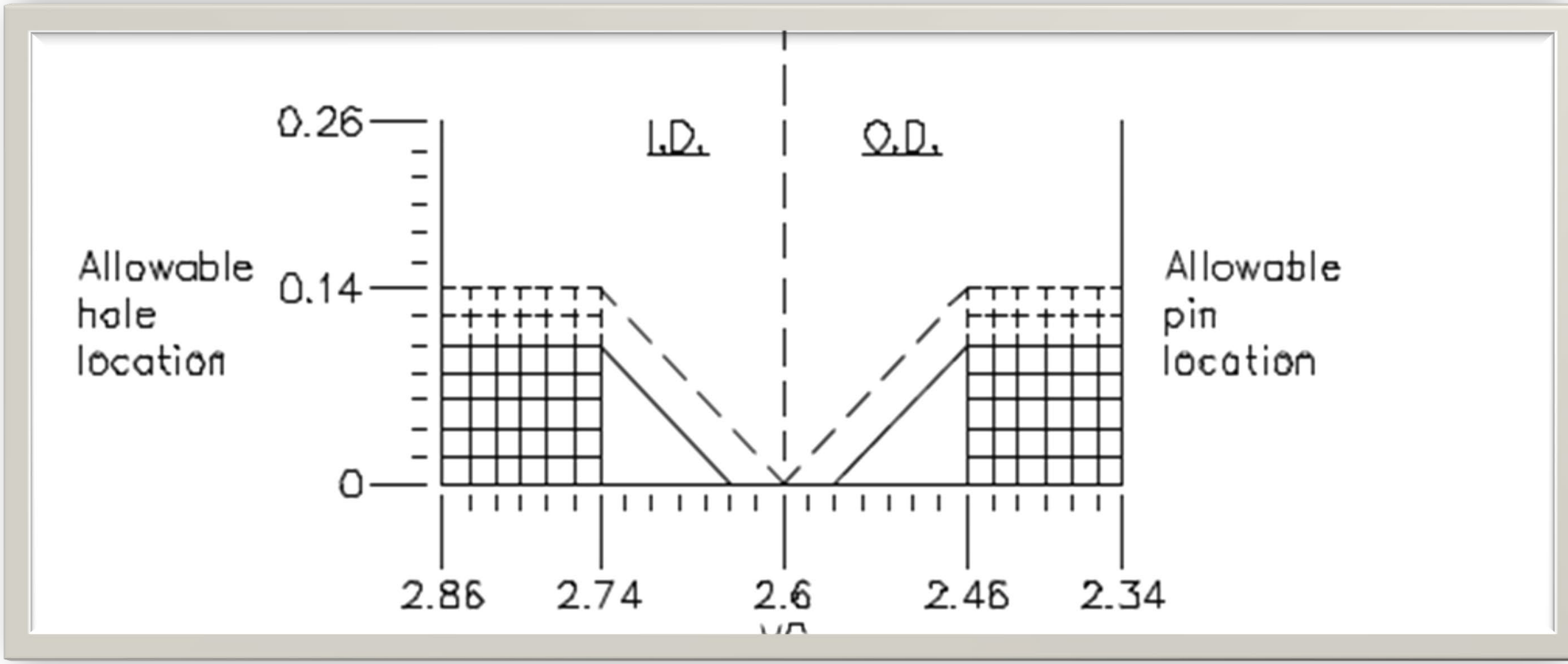
Location


Gauge
Pin Gauge ⌀ =
Min ⌀ hole – ⌖ tolerance
Part
Tolerance Zone
of Pin Gauge
True position
Actual
Hole Position

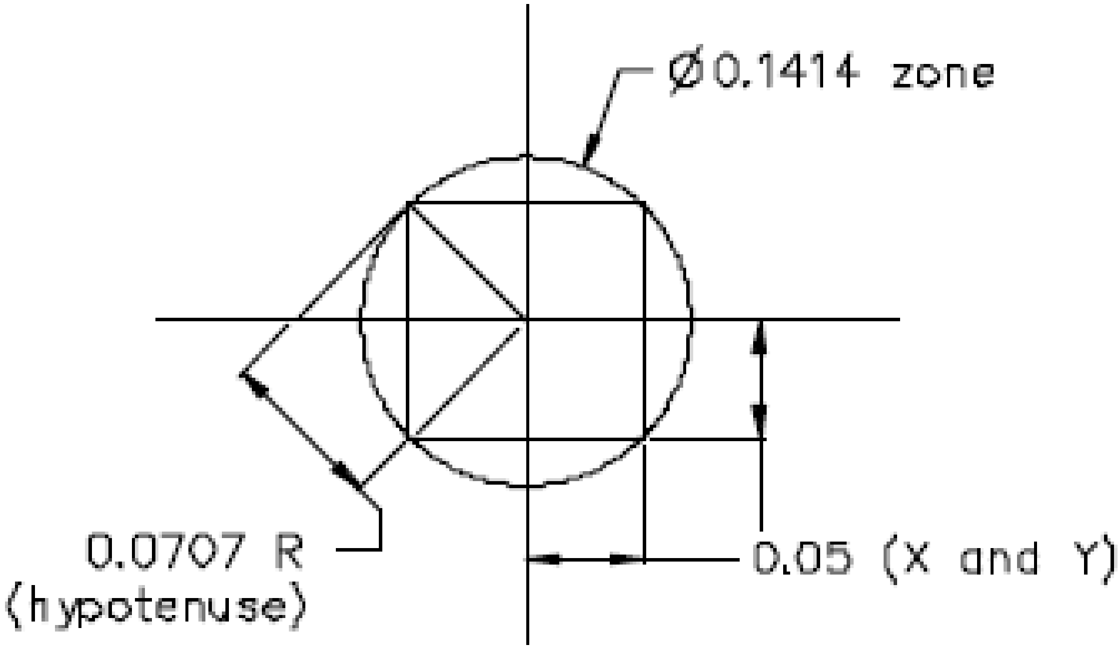
4 Pin Example



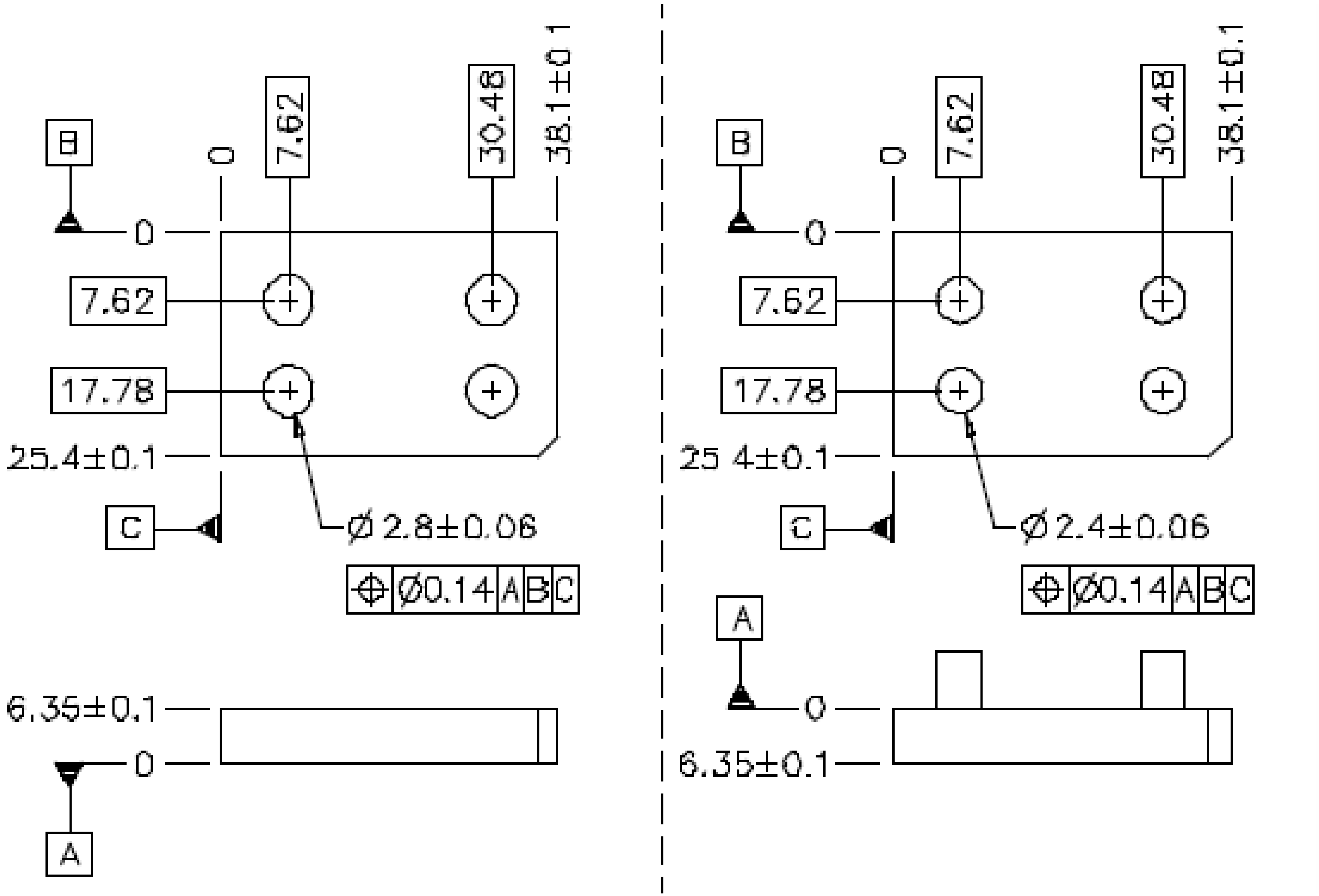
4 Pin Example



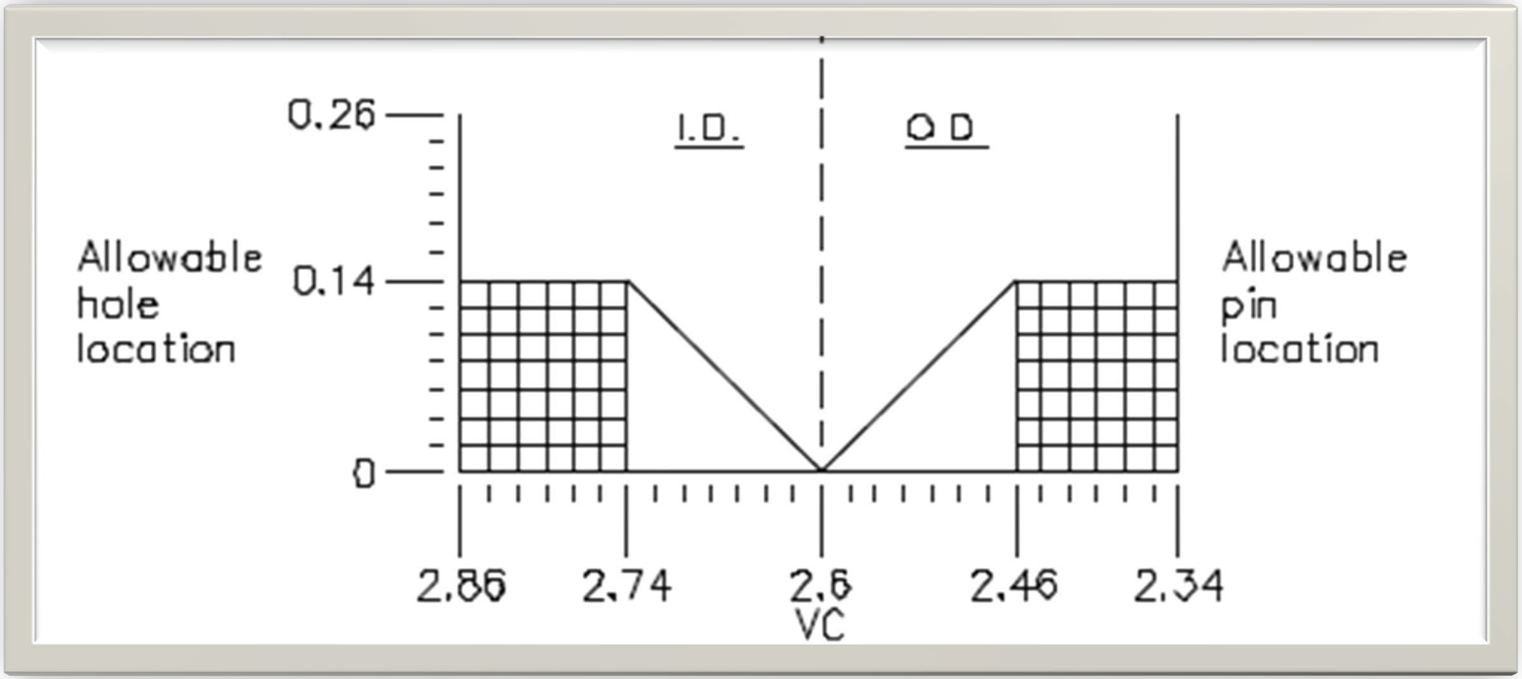
Tolerance Zone



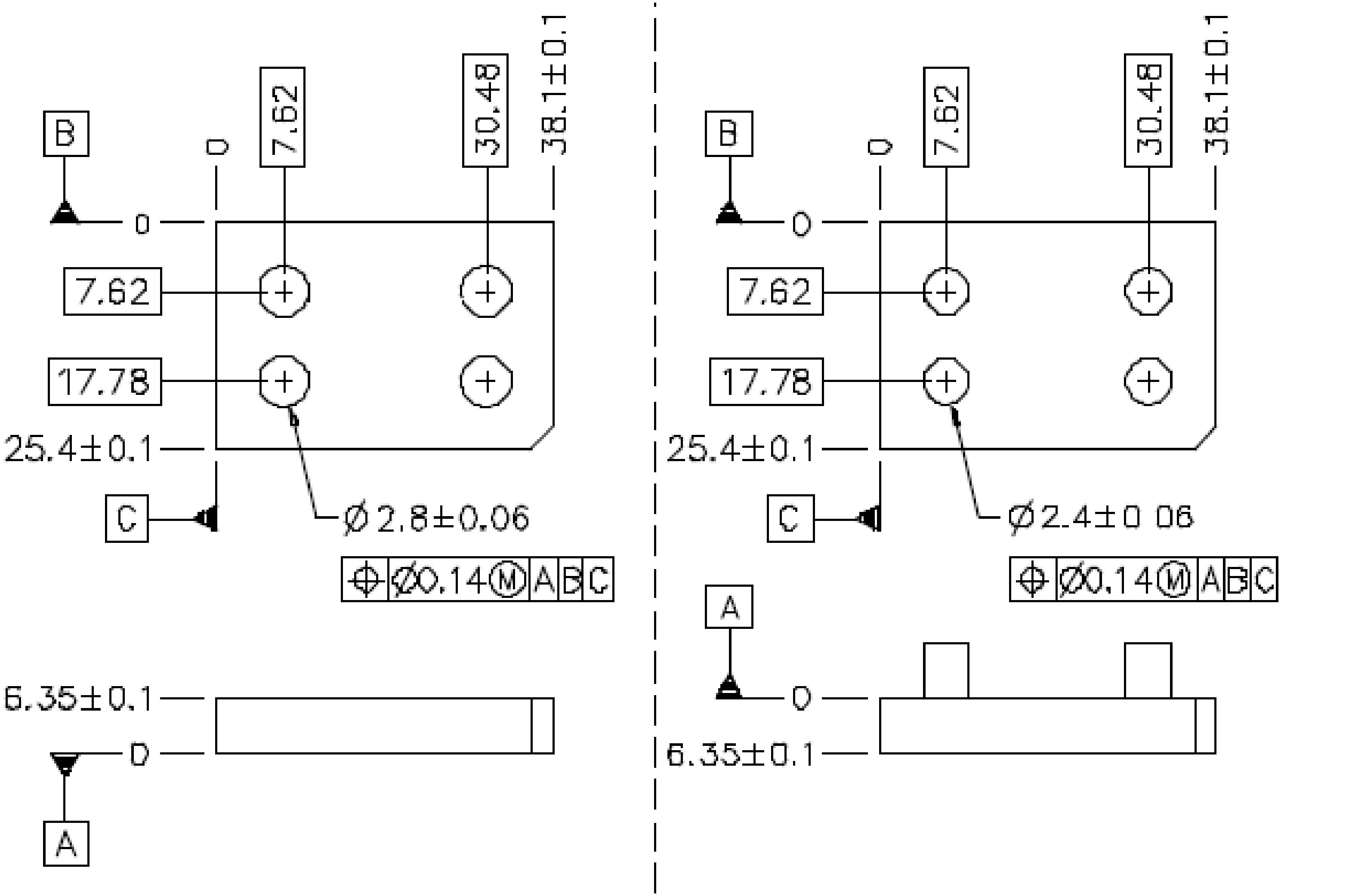
4 Pin Example



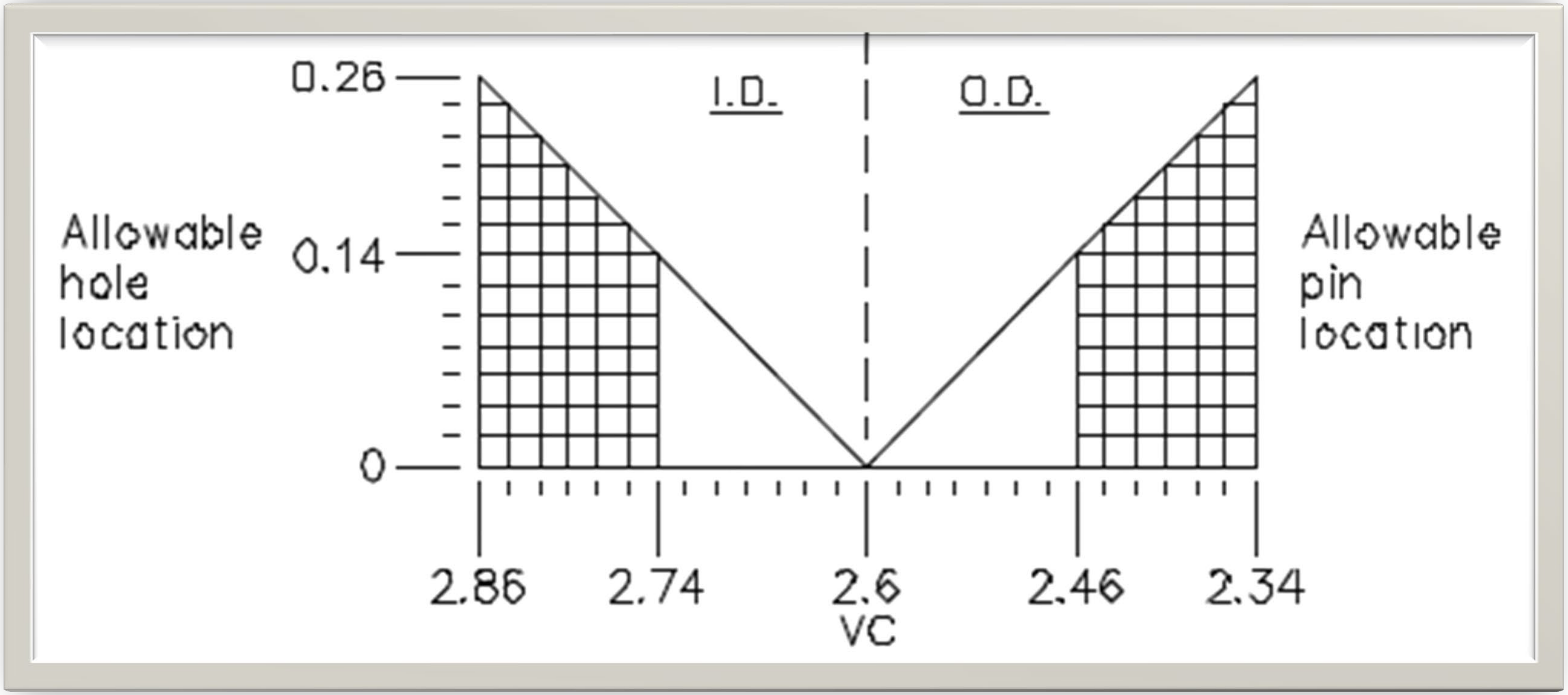
4 Pin Example



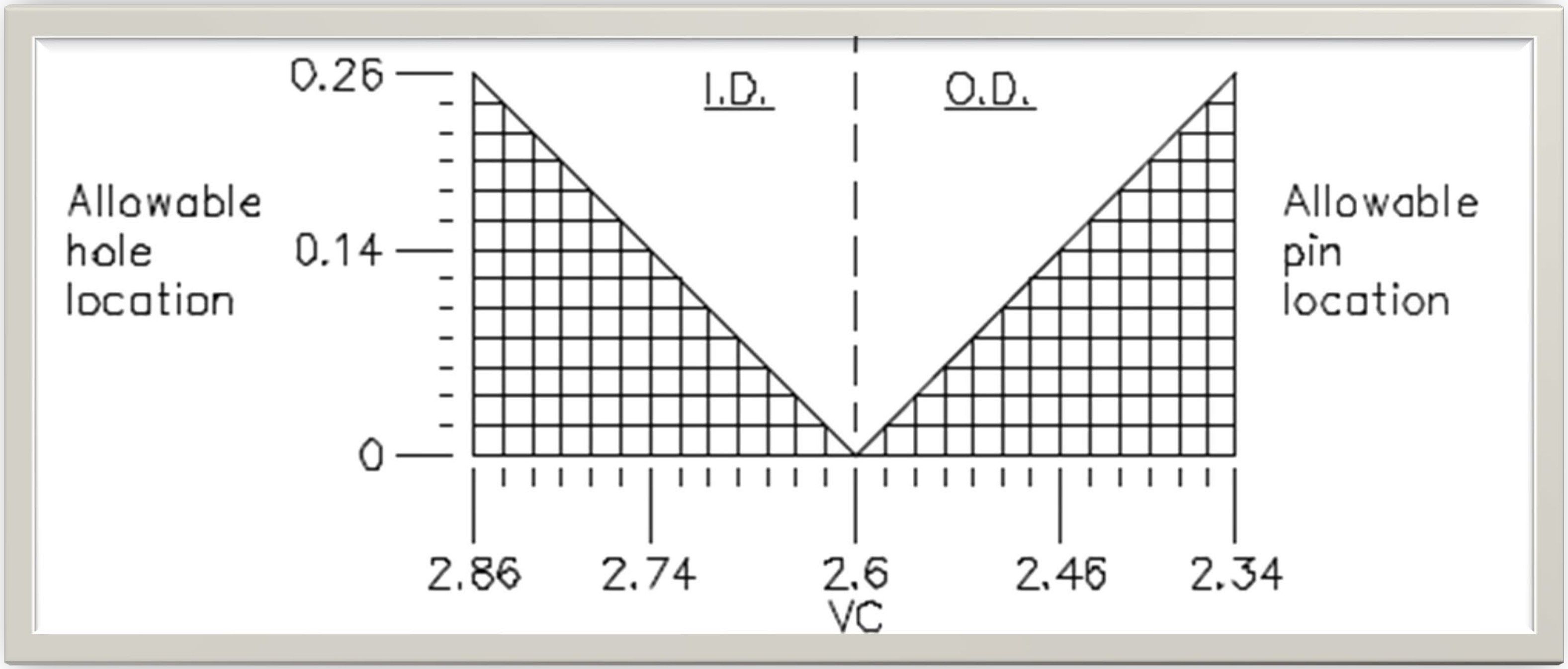
4 Pin Example



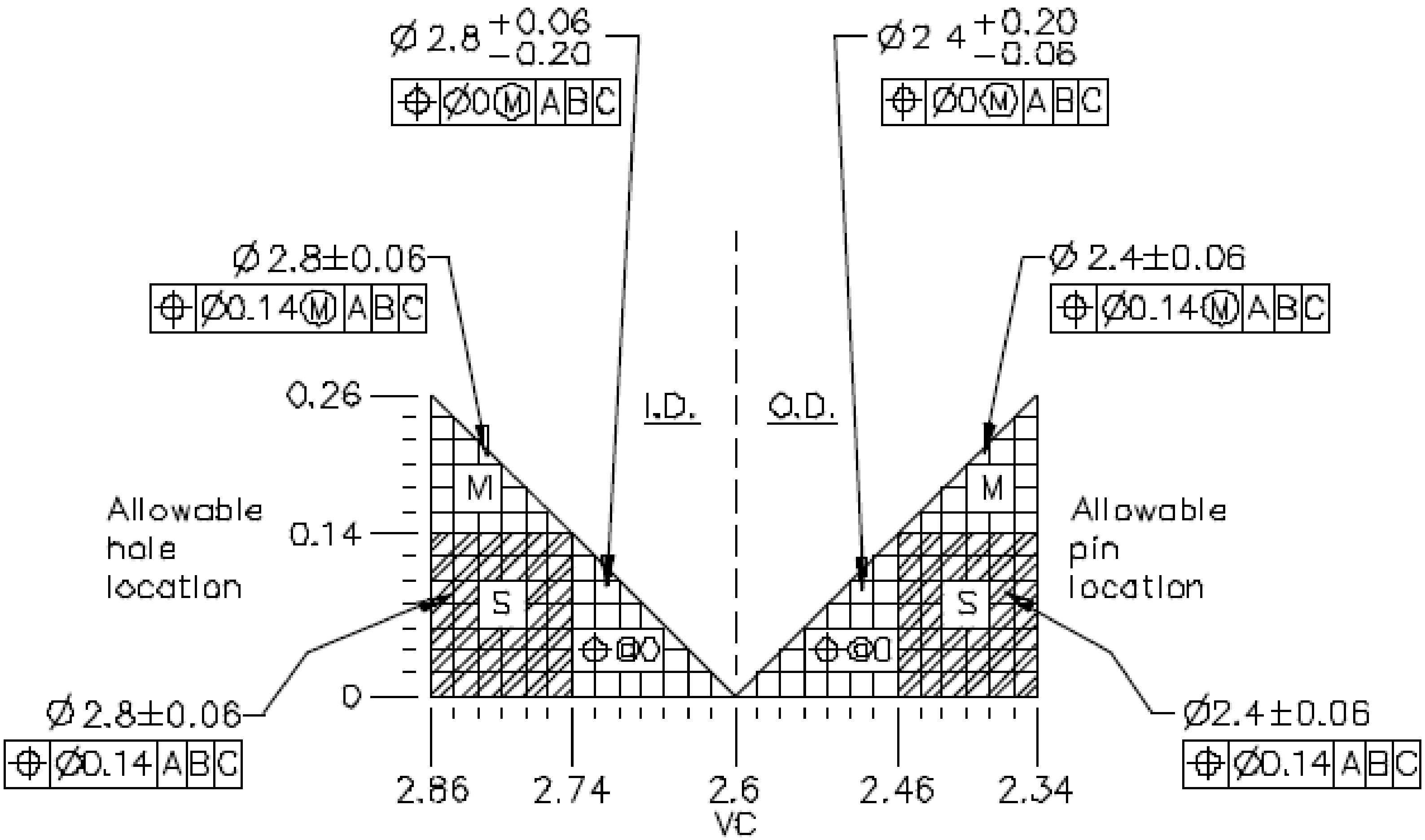
4 Pin Example



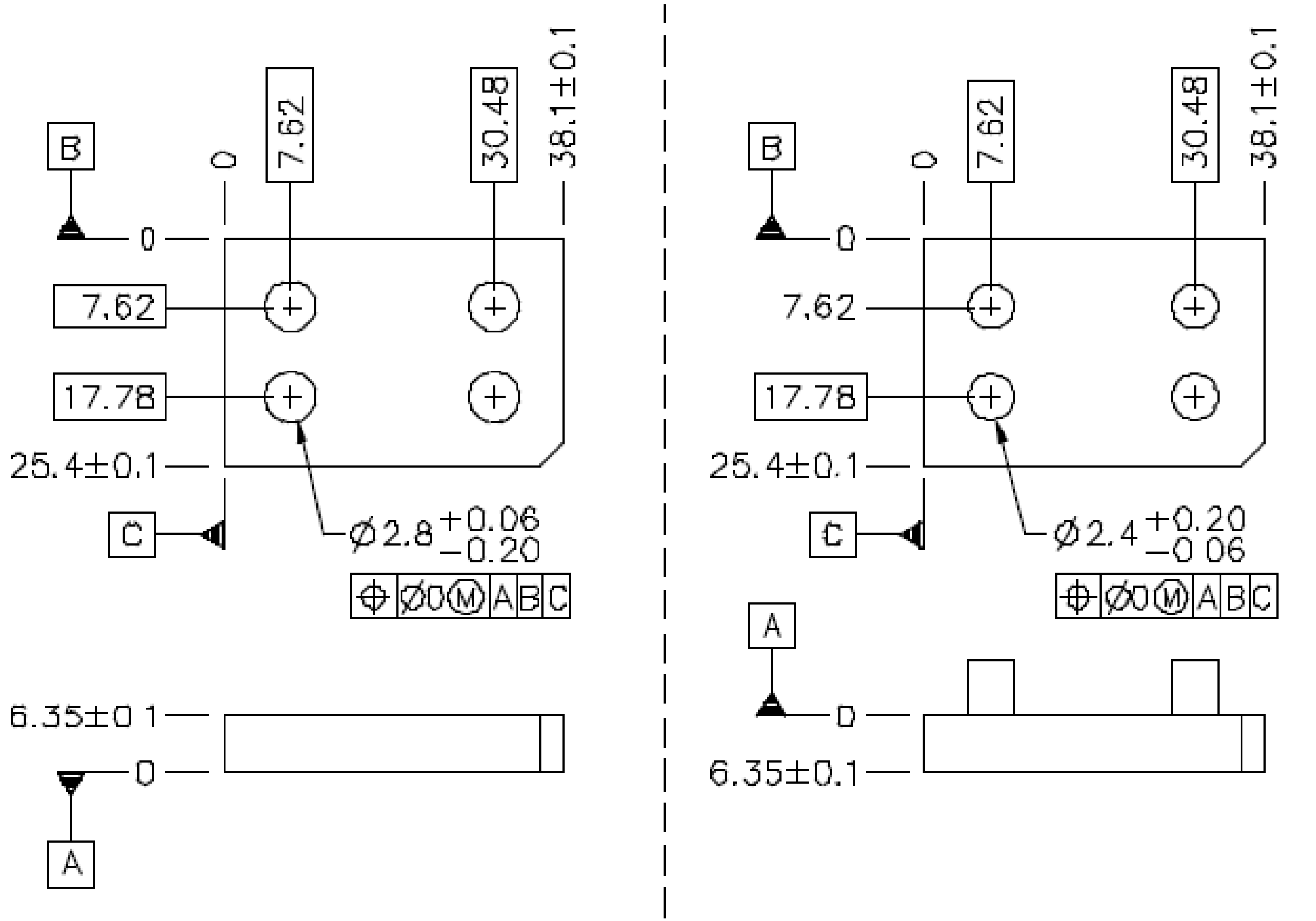
4 Pin Example



4 Pin Example Summary



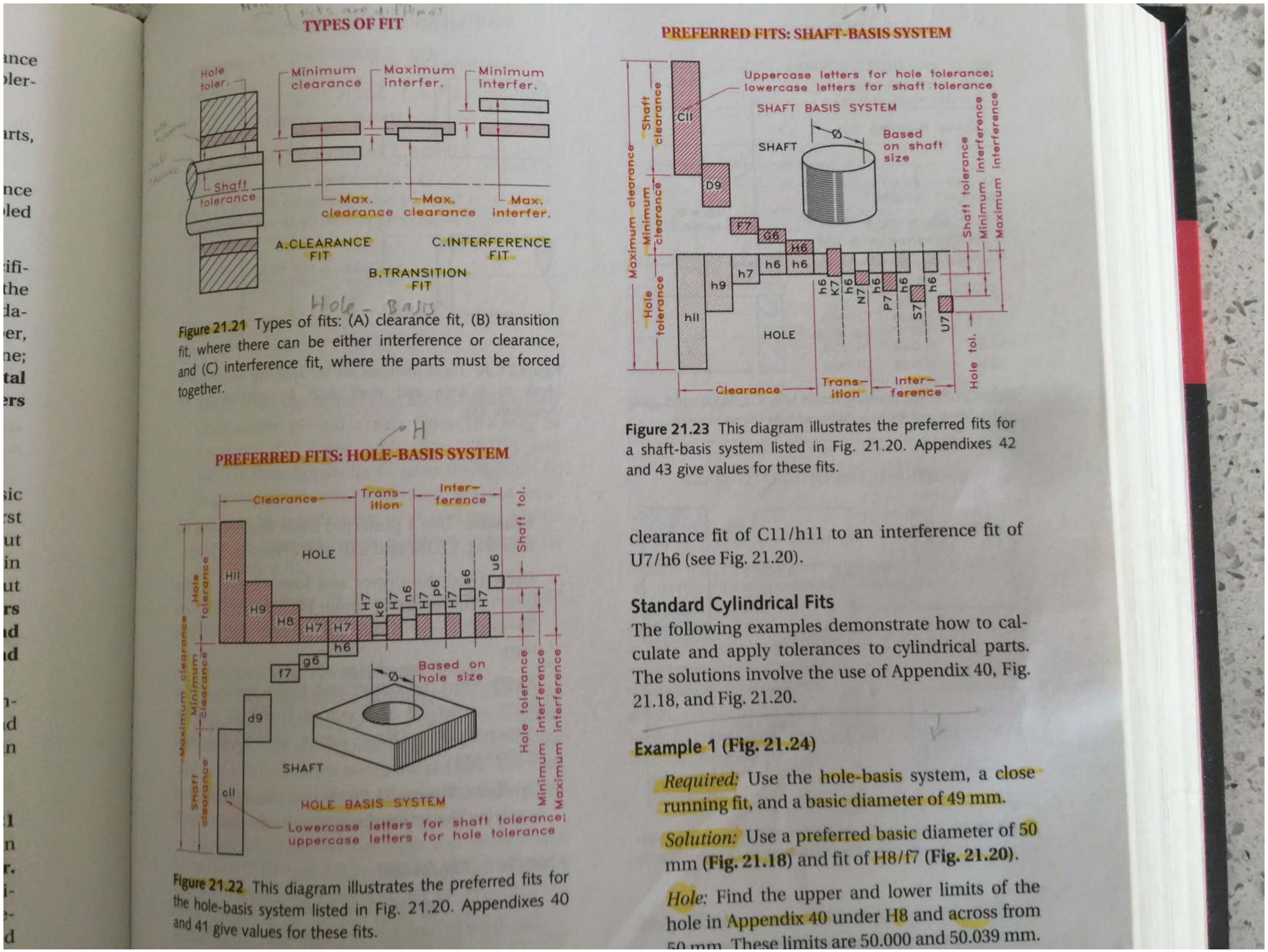
4 Pin Example



Summary & Recap of Tolerancing (GD&T)

- • **James H. Earle, "Engineering Design Graphics," 8th Edition, Addison Wesley, 2001**

Summary & Recap of Tolerancing



TYPES OF FIT

Figure 21.21 Types of fits: (A) clearance fit, (B) transition fit, where there can be either interference or clearance, and (C) interference fit, where the parts must be forced together.

PREFERRED FITS: HOLE-BASIS SYSTEM

Figure 21.22 This diagram illustrates the preferred fits for the hole-basis system listed in Fig. 21.20. Appendixes 40 and 41 give values for these fits.

PREFERRED FITS: SHAFT-BASIS SYSTEM

Figure 21.23 This diagram illustrates the preferred fits for a shaft-basis system listed in Fig. 21.20. Appendixes 42 and 43 give values for these fits.

clearance fit of C11/h11 to an interference fit of U7/h6 (see Fig. 21.20).

Standard Cylindrical Fits

The following examples demonstrate how to calculate and apply tolerances to cylindrical parts. The solutions involve the use of Appendix 40, Fig. 21.18, and Fig. 21.20.

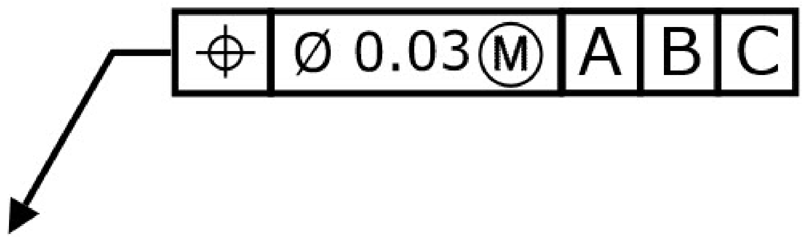
Example 1 (Fig. 21.24)

Required: Use the hole-basis system, a close running fit, and a basic diameter of 49 mm.

Solution: Use a preferred basic diameter of 50 mm (**Fig. 21.18**) and fit of H8/f7 (**Fig. 21.20**).

Hole: Find the upper and lower limits of the hole in Appendix 40 under H8 and across from 50 mm. These limits are 50.000 and 50.039 mm.

Summary & Recap of Tolerancing (GD&T)



Summary & Recap of Tolerancing

TABLE OF ANGULAR TOLERANCES

ANGULAR TOLERANCES				
LENGTH OF SHORTER LEG (mm)	UP TO 10	OVER 10 TO 50	OVER 50 TO 120	OVER 120 TO 400
TOLERANCE	±1°	±0°30'	±0°20'	±0°10'

Values in degrees and minutes taken from previous table

Figure 21.38 Extracted from Fig. 21.37, this table of values inserted on a drawing would indicate general tolerances for angles in degrees and minutes.

GEOMETRIC SYMBOLS

	TOLERANCE	CHARACTERISTIC	SYMBOL
INDIVIDUAL FEATURES	FORM	STRAIGHTNESS	—
		FLATNESS	▱
		CIRCULARITY	○
		CYLINDRICITY	⌭
INDIVIDUAL OR RELATED FEATURES	PROFILE	PROFILE OF A LINE	⌒
		PROFILE OF A SURFACE	⌓
RELATED FEATURES	ORIENTATION	ANGULARITY	∠
		PERPENDICULARITY	⊥
		PARALLELISM	//
	LOCATION	POSITION	⌖
		CONCENTRICITY	◎
	RUNOUT	CIRCULAR RUNOUT	↗
		TOTAL RUNOUT	⌰

Figure 21.39 These symbols specify the geometric characteristics of a part's features.

PROPORTIONS OF SYMBOLS

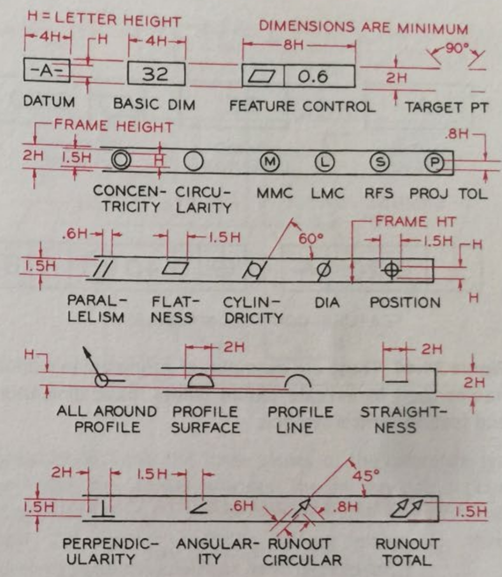
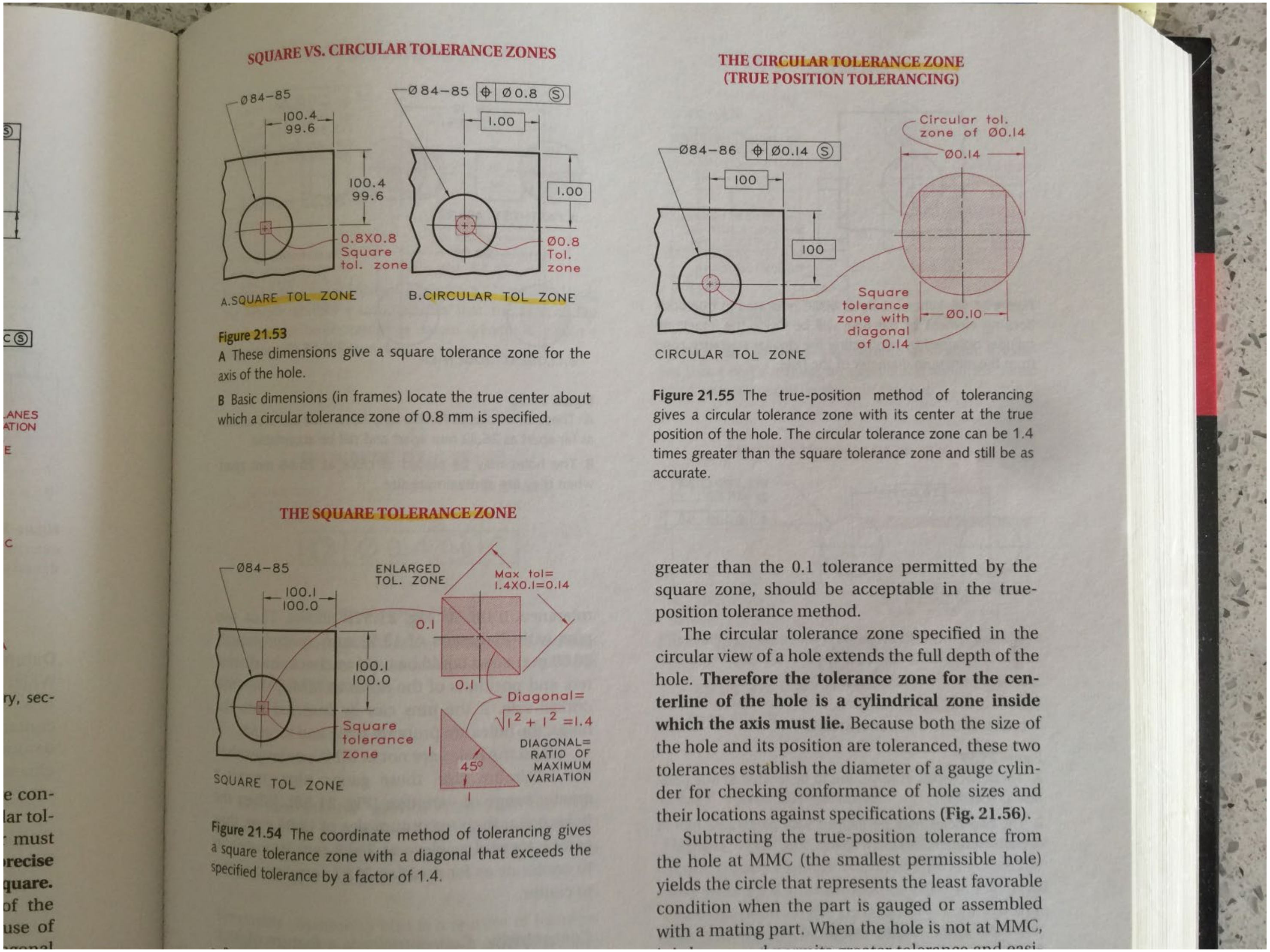


Figure 21.40 Use these general proportions (based on the letter height used) for drawing feature control symbols and frames.

21.13 Geometric Tolerances

Geometric tolerancing specifies tolerances that control location form, profile, orientation, and runout on a dimensioned part as covered by the ANSI *Y14.5M-1982 Standards* and the *Military Standards (Mil-Std)* of the U.S. Department of Defense. Before discussing those types of tolerancing, however, we need to introduce you to

Summary & Recap of Tolerancing



SQUARE VS. CIRCULAR TOLERANCE ZONES

Figure 21.53
A These dimensions give a square tolerance zone for the axis of the hole.
B Basic dimensions (in frames) locate the true center about which a circular tolerance zone of 0.8 mm is specified.

THE CIRCULAR TOLERANCE ZONE (TRUE POSITION TOLERANCING)

Figure 21.55 The true-position method of tolerancing gives a circular tolerance zone with its center at the true position of the hole. The circular tolerance zone can be 1.4 times greater than the square tolerance zone and still be as accurate.

THE SQUARE TOLERANCE ZONE

Figure 21.54 The coordinate method of tolerancing gives a square tolerance zone with a diagonal that exceeds the specified tolerance by a factor of 1.4.

greater than the 0.1 tolerance permitted by the square zone, should be acceptable in the true-position tolerance method.

The circular tolerance zone specified in the circular view of a hole extends the full depth of the hole. **Therefore the tolerance zone for the centerline of the hole is a cylindrical zone inside which the axis must lie.** Because both the size of the hole and its position are toleranced, these two tolerances establish the diameter of a gauge cylinder for checking conformance of hole sizes and their locations against specifications **(Fig. 21.56)**.

Subtracting the true-position tolerance from the hole at MMC (the smallest permissible hole) yields the circle that represents the least favorable condition when the part is gauged or assembled with a mating part. When the hole is not at MMC,

Summary & Recap of Tolerancing

CONCENTRICITY

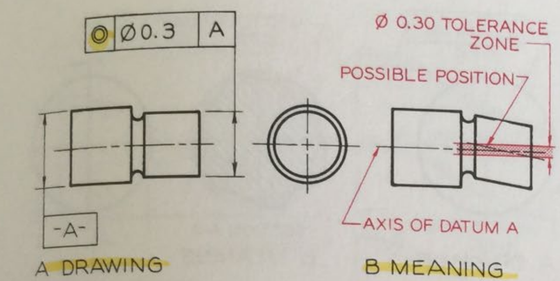


Figure 21.59 Concentricity is a tolerance of location. Here, the feature control frame specifies that the axis of the small cylinder be concentric to datum cylinder A, within a tolerance of 0.3 mm diameter.

SYMMETRY

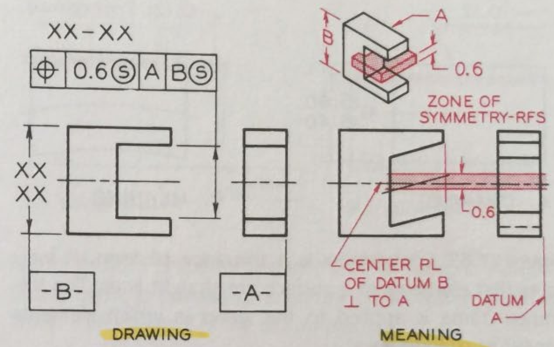


Figure 21.61 Symmetry is a tolerance of location. It specifies that a part's features be symmetrical about the center plane between parallel surfaces of the part.

FEATURE CONTROL FRAME

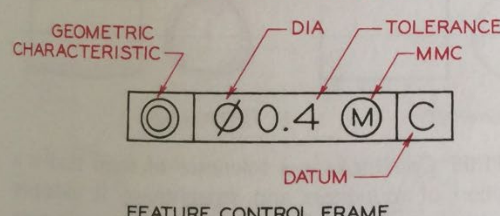


Figure 21.60 This typical feature control frame indicates that a surface is concentric to datum C within a cylindrical diameter of 0.4 mm at MMC.

FLATNESS

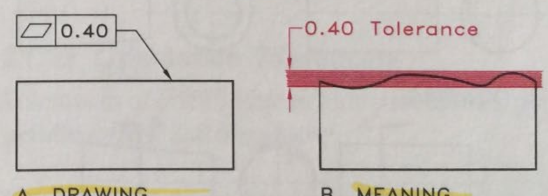


Figure 21.62 Flatness is a tolerance of form. It specifies a tolerance zone within which an object's surface must lie.

Summary & Recap of Tolerancing

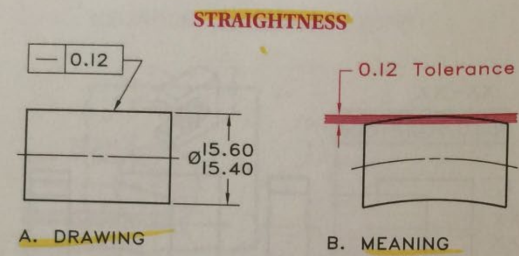


Figure 21.63 Straightness is a tolerance of form. It indicates that elements of a surface are straight lines. The tolerance frame is applied to the views in which elements appear as straight lines.

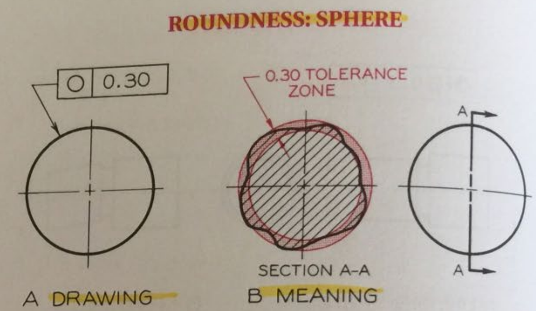


Figure 21.65 Roundness of a sphere means that any cross section through it is round within the specified tolerance.

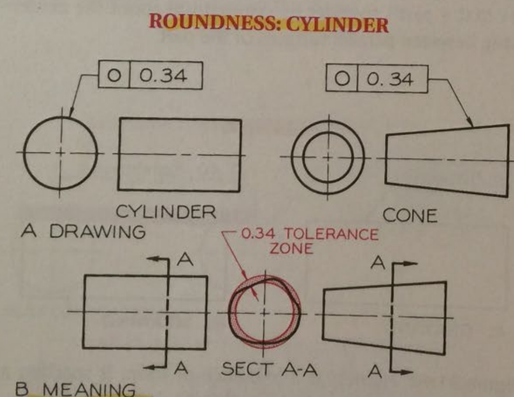


Figure 21.64 Roundness is a tolerance of form. It indicates that a cross section through a surface of revolution is round and lies within two concentric circles.

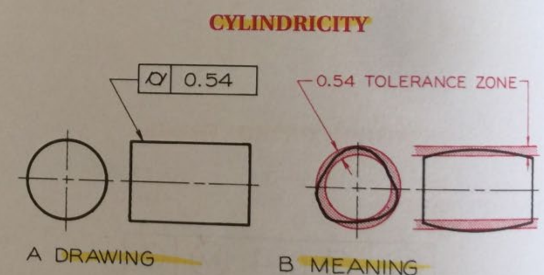


Figure 21.66 Cylindricity is a tolerance of form that is a combination of roundness and straightness. It indicates that the surface of a cylinder lies within a tolerance zone formed by two concentric cylinders.

mm on the radius. **Figure 21.65** specifies a 0.30 mm tolerance zone for the roundness of a sphere.

within 0.12 mm. On flat surfaces, straightness is

Cylindricity A surface of revolution is cylindrical

Summary & Recap of Tolerancing

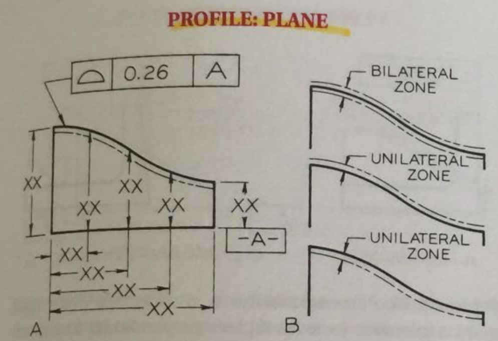


Figure 21.67 Profile is a tolerance of form for irregular curves of planes. (A) The curving plane is located by coordinates and is toleranced unidirectionally. (B) The tolerance may be applied by any of these methods.

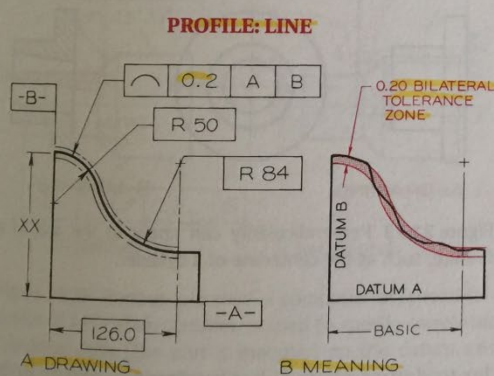


Figure 21.68 The profile of a line is a tolerance of form that specifies the variation allowed from the path of a line. Here, the line is formed by tangent arcs. The tolerance zone may be either bilateral or unilateral, as shown in Fig. 21.70.

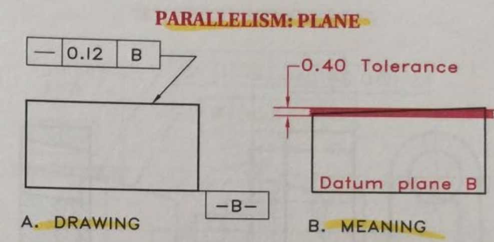


Figure 21.69 Parallelism is a tolerance of form. It indicates that a plane is parallel to a datum plane within specified limits. Here, plane B is the datum plane.

tangent arcs whose radii are given as basic dimensions. The radii are permitted to vary ±0.10 mm from the basic radii.

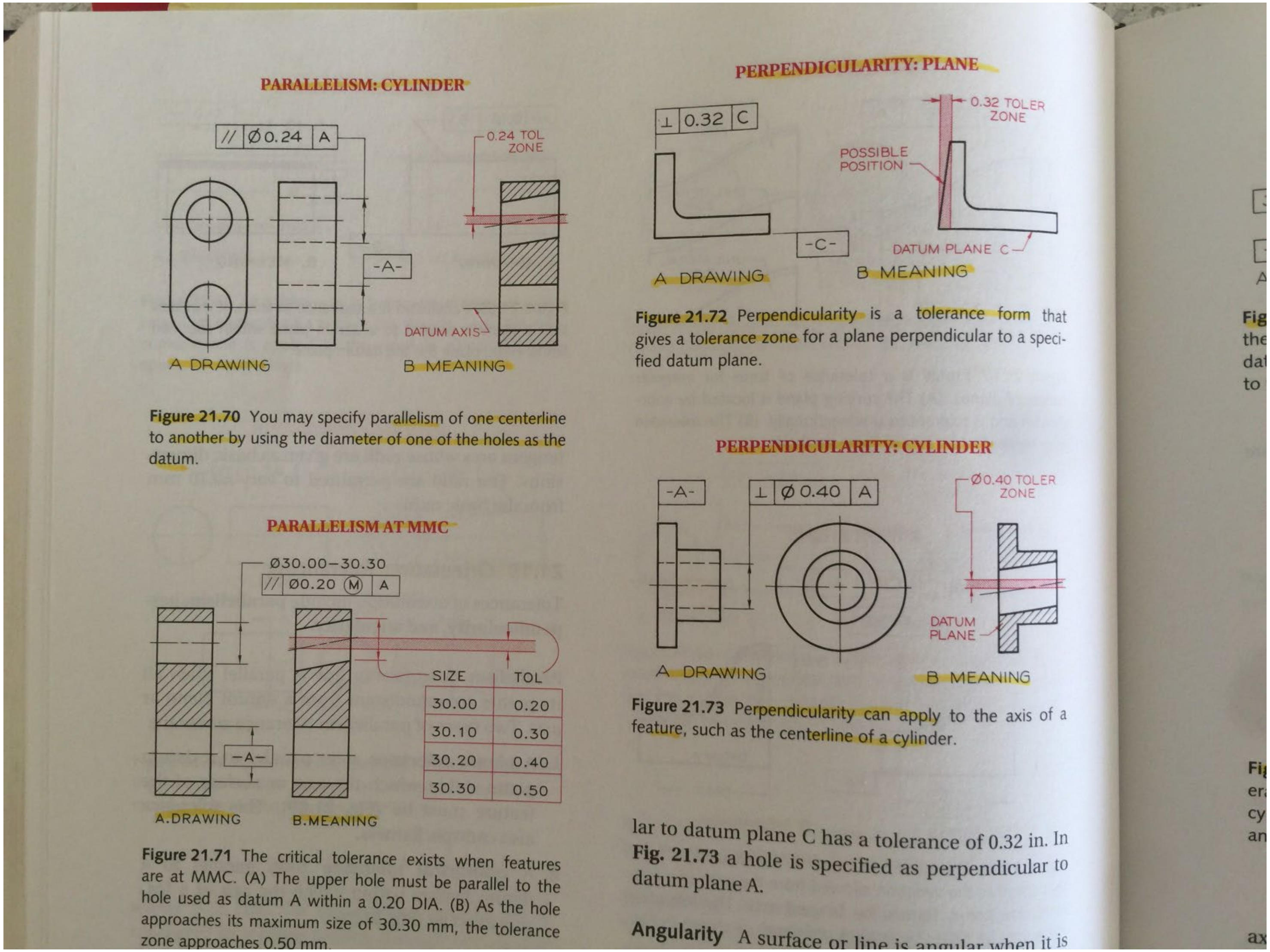
21.19 Orientation Tolerancing

Tolerances of orientation include **parallelism, perpendicularity, and angularity.**

Parallelism A surface or line is parallel when all its points are equidistant from a datum plane or axis. Two types of parallelism tolerance zones are:

1. A **planar tolerance** zone parallel to a datum plane within which the axis or surface of the feature must lie **(Fig. 21.69)**. This tolerance also controls flatness.
2. A **cylindrical tolerance** zone parallel to a datum feature within which the axis of a feature must lie **(Fig. 21.70)**.

Summary & Recap of Tolerancing



PARALLELISM: CYLINDER

Figure 21.70 You may specify parallelism of one centerline to another by using the diameter of one of the holes as the datum.

PARALLELISM AT MMC

SIZE	TOL
30.00	0.20
30.10	0.30
30.20	0.40
30.30	0.50

Figure 21.71 The critical tolerance exists when features are at MMC. (A) The upper hole must be parallel to the hole used as datum A within a 0.20 DIA. (B) As the hole approaches its maximum size of 30.30 mm, the tolerance zone approaches 0.50 mm.

PERPENDICULARITY: PLANE

Figure 21.72 Perpendicularity is a tolerance form that gives a tolerance zone for a plane perpendicular to a specified datum plane.

PERPENDICULARITY: CYLINDER

Figure 21.73 Perpendicularity can apply to the axis of a feature, such as the centerline of a cylinder.

lar to datum plane C has a tolerance of 0.32 in. In **Fig. 21.73** a hole is specified as perpendicular to datum plane A.

Angularity A surface or line is angular when it is

Summary & Recap of Tolerancing

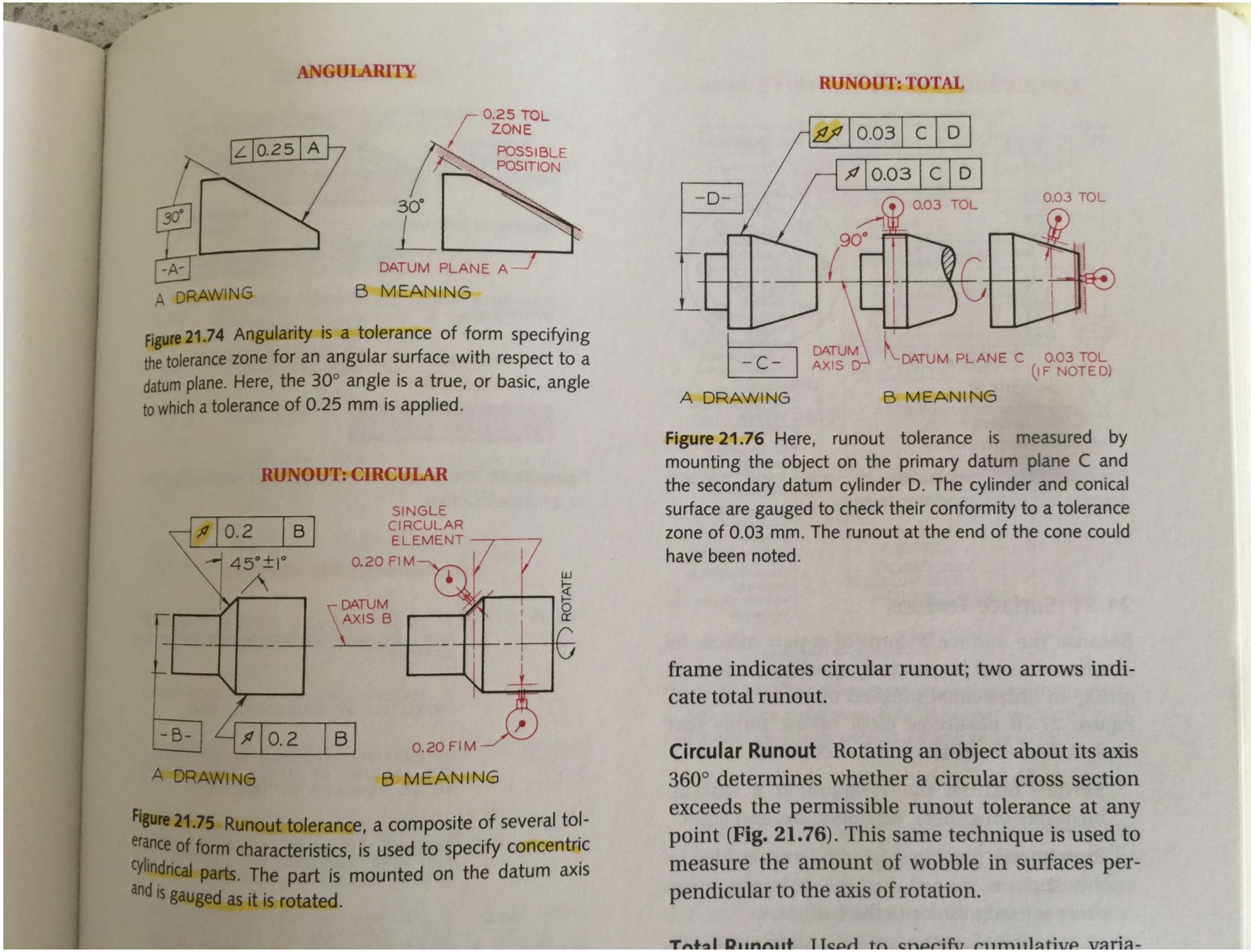


Figure 21.74 Angularity is a tolerance of form specifying the tolerance zone for an angular surface with respect to a datum plane. Here, the 30° angle is a true, or basic, angle to which a tolerance of 0.25 mm is applied.

Figure 21.75 Runout tolerance, a composite of several tolerance of form characteristics, is used to specify concentric cylindrical parts. The part is mounted on the datum axis and is gauged as it is rotated.

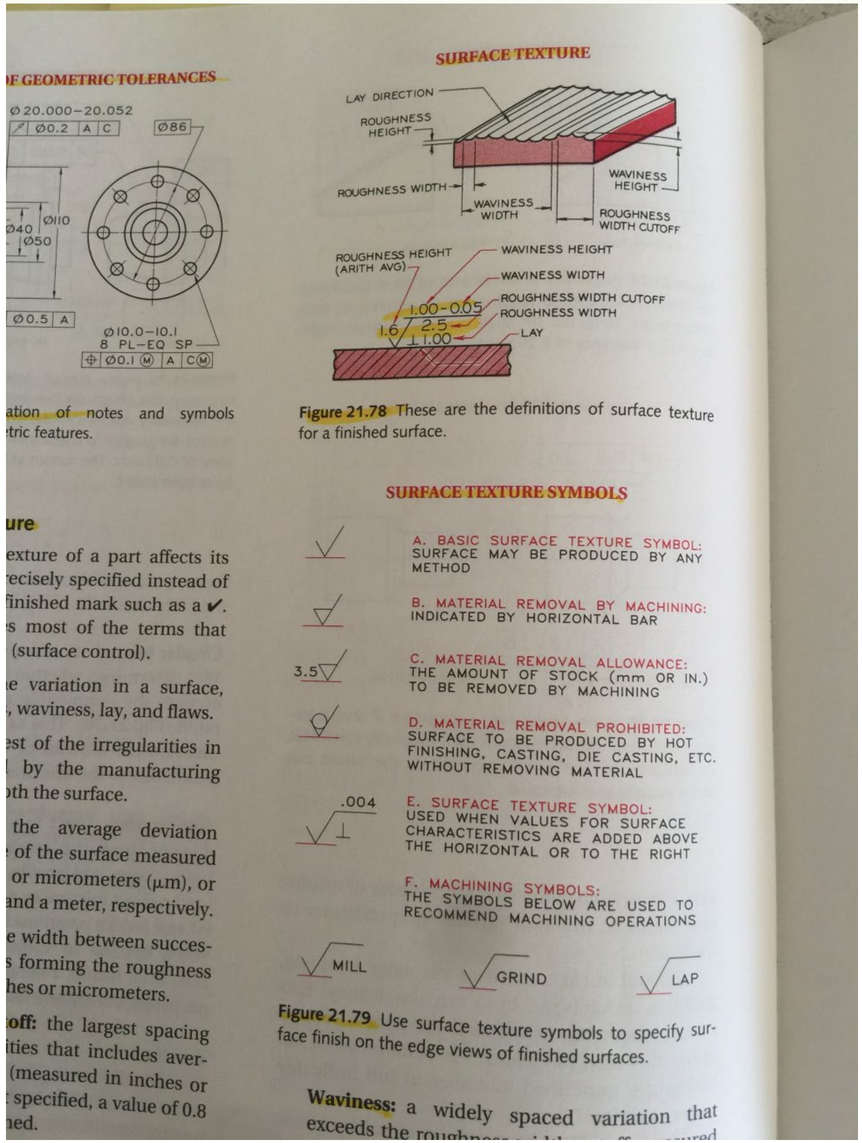
Figure 21.76 Here, runout tolerance is measured by mounting the object on the primary datum plane C and the secondary datum cylinder D. The cylinder and conical surface are gauged to check their conformity to a tolerance zone of 0.03 mm. The runout at the end of the cone could have been noted.

frame indicates circular runout; two arrows indicate total runout.

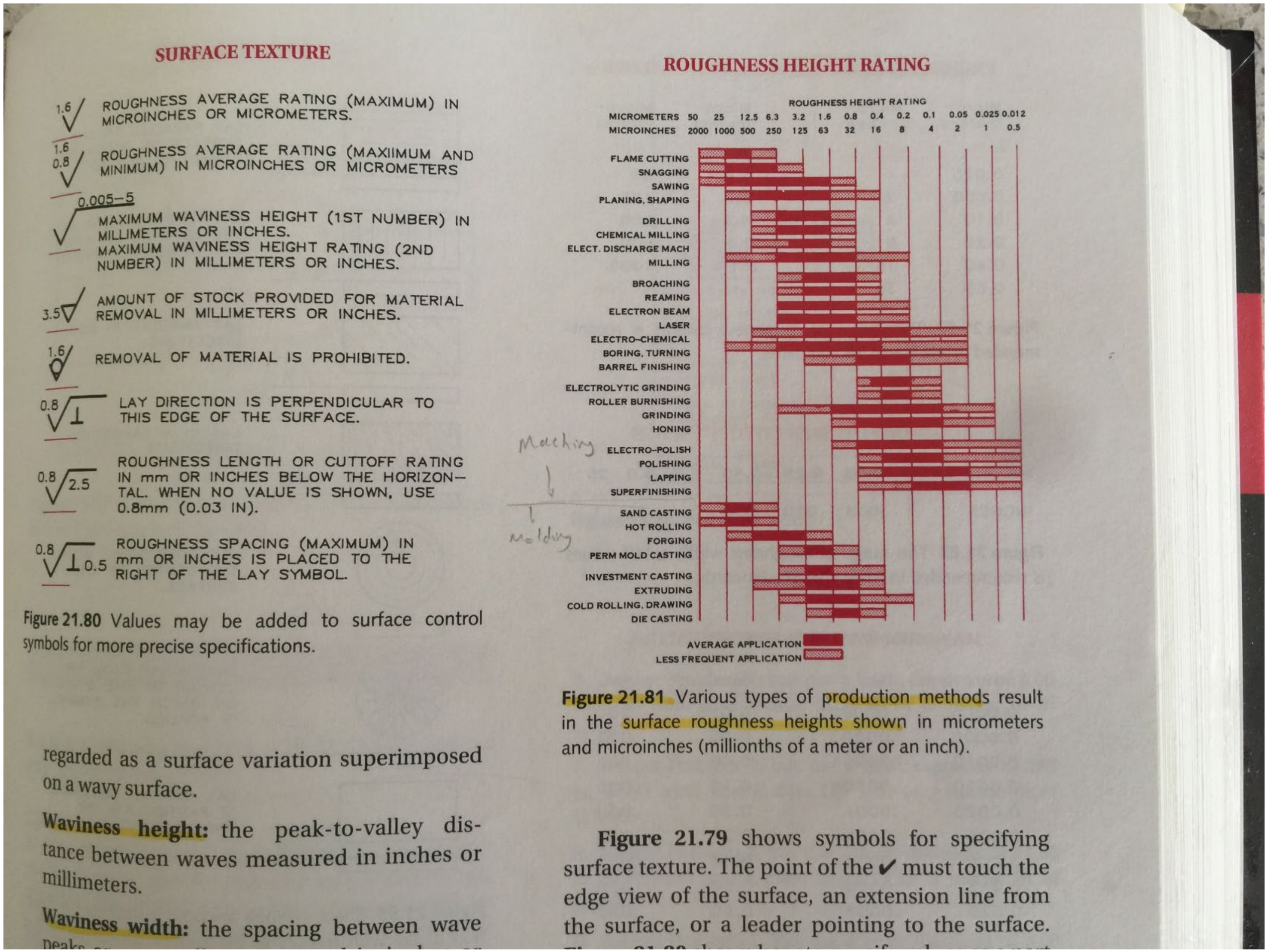
Circular Runout Rotating an object about its axis 360° determines whether a circular cross section exceeds the permissible runout tolerance at any point (**Fig. 21.76**). This same technique is used to measure the amount of wobble in surfaces perpendicular to the axis of rotation.

Total Runout Used to specify cumulative varia-

Summary & Recap of Tolerancing



Summary & Recap of Tolerancing



SURFACE TEXTURE

- 1.6 — ROUGHNESS AVERAGE RATING (MAXIMUM) IN MICROINCHES OR MICROMETERS.
- 1.6 / 0.8 — ROUGHNESS AVERAGE RATING (MAXIIMUM AND MINIMUM) IN MICROINCHES OR MICROMETERS
- 0.005–5 — MAXIMUM WAVINESS HEIGHT (1ST NUMBER) IN MILLIMETERS OR INCHES. MAXIMUM WAVINESS HEIGHT RATING (2ND NUMBER) IN MILLIMETERS OR INCHES.
- 3.5 — AMOUNT OF STOCK PROVIDED FOR MATERIAL REMOVAL IN MILLIMETERS OR INCHES.
- 1.6 — REMOVAL OF MATERIAL IS PROHIBITED.
- 0.8 ⊥ — LAY DIRECTION IS PERPENDICULAR TO THIS EDGE OF THE SURFACE.
- 0.8 / 2.5 — ROUGHNESS LENGTH OR CUTTOFF RATING IN mm OR INCHES BELOW THE HORIZONTAL. WHEN NO VALUE IS SHOWN, USE 0.8mm (0.03 IN).
- 0.8 ⊥ 0.5 — ROUGHNESS SPACING (MAXIMUM) IN mm OR INCHES IS PLACED TO THE RIGHT OF THE LAY SYMBOL.

Figure 21.80 Values may be added to surface control symbols for more precise specifications.

ROUGHNESS HEIGHT RATING

Figure 21.81 Various types of production methods result in the surface roughness heights shown in micrometers and microinches (millionths of a meter or an inch).

regarded as a surface variation superimposed on a wavy surface.

Waviness height: the peak-to-valley distance between waves measured in inches or millimeters.

Waviness width: the spacing between wave peaks ...

Figure 21.79 shows symbols for specifying surface texture. The point of the ✔ must touch the edge view of the surface, an extension line from the surface, or a leader pointing to the surface.