COMPUTER NUMERICAL CONTROL OF MACHINE TOOLS

Laboratory for Manufacturing Systems and Automation
Department of Mechanical Engineering and Aeronautics
University of Patras, Greece

Dr. Dimitris Mourtzis
Assistant Professor

Patras 2016

Chapter 7: Tolerancing

Table of Contents

Objectives

- Learn about **tolerancing** and how important this technique is, to mass production

- Learn various **tolerancing methods**

- Learn about **tolerancing standards** and the most common standards agencies

- And, ways of formatting **inch and metric tolerances**

Introduction

- After completing the construction of a piece, its **dimensions appear to deviate compared to its nominal values**

- These variations depend on the **accuracy of the machine-tools** used and the **available measuring devices**

- **Minor deviations**, depending on the intended use of the piece, **may be tolerated without destroying its functionality**

- **Tolerances** are defined as the **permissible deviations** from the prescribed shape, size or position of an element in one piece **with respect to the corresponding ones of the drawing**

(Φυλλάδιο εργαστηριακών ασκήσεων Μηχανουργείου,2012, Μούρτζης Δ. κ.α.)

Tolerances

- In most of the cases the craftsman who undertakes the construction of a piece is **unable to reproduce the exact dimensions** as outlined in the drawing

- For this reason **constructional drawings indicate the permissible deviations** from the nominal dimension

  - Example:

    A given dimension in a drawing is **1.50 ± .04 mm**, meaning that the piece **should have a particular dimension between 1.46** and **1.54 mm**, and that the **possible permissible tolerance** on this dimension is **0.08 mm**

(Φυλλάδιο εργαστηριακών ασκήσεων Μηχανουργείου,2012, Μούρτζης Δ. κ.α.)

Tolerances

Greater accuracy requires greater cost

- For this reason **not all the parts of a product are constructed with the same accuracy** (same tolerances)

- The selection of the desired **accuracy** of a product's part **depends on its final use** (assembly with other parts, e.g., shaft and hole, etc.).

- **Tolerances** are **heavily dependent on the machine tool** used for the manufacturing of the piece

- The following figure presents the typical achievable tolerances for various production material removal processes

Tolerances

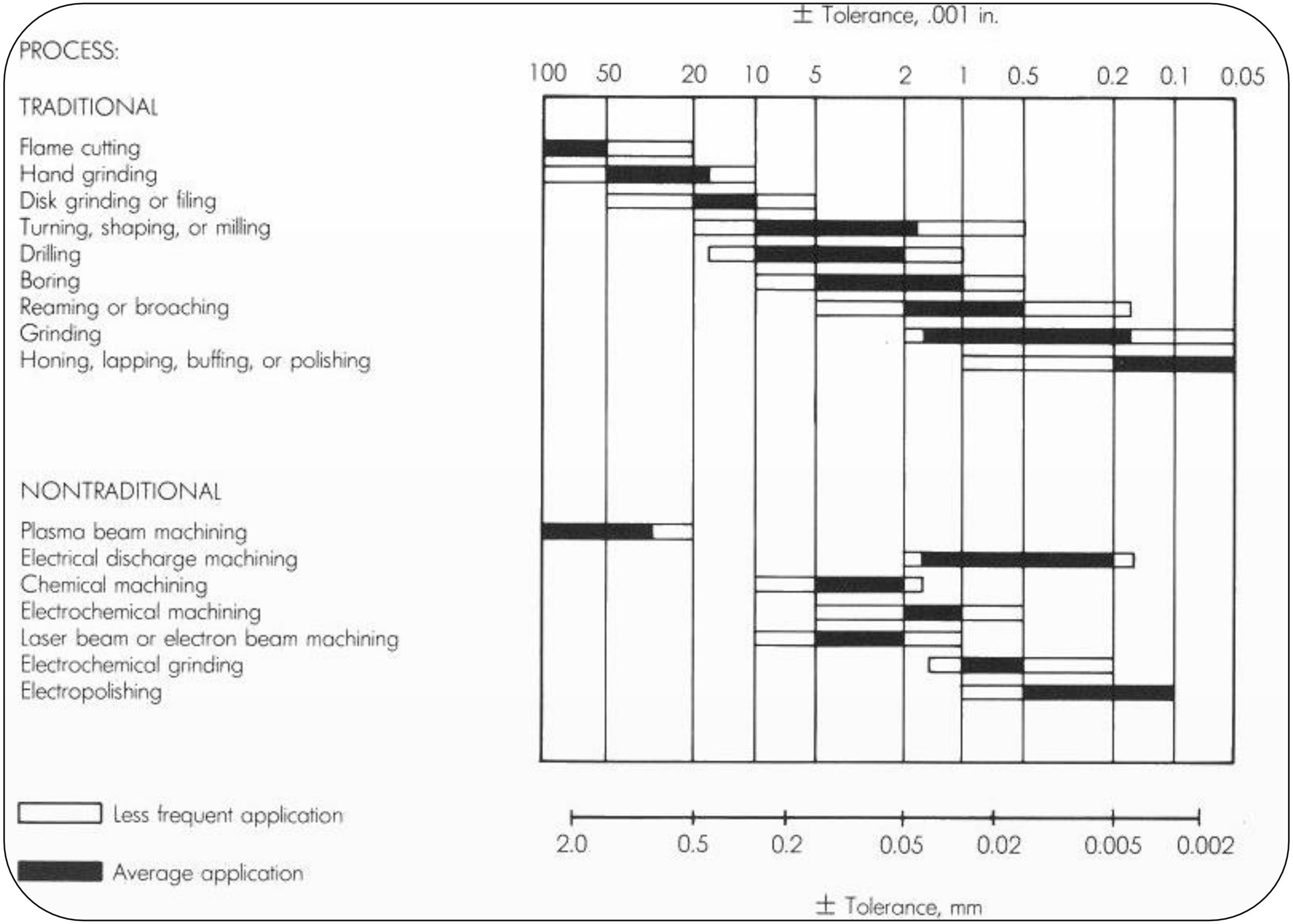


Figure 1: Typical tolerances for material removal processes *(Φυλλάδιο εργαστηριακών ασκήσεων Μηχανουργείου ,Μούρτζης Δ. κ.α. , ,Πανεπιστήμιο Πατρών)*

Tolerances

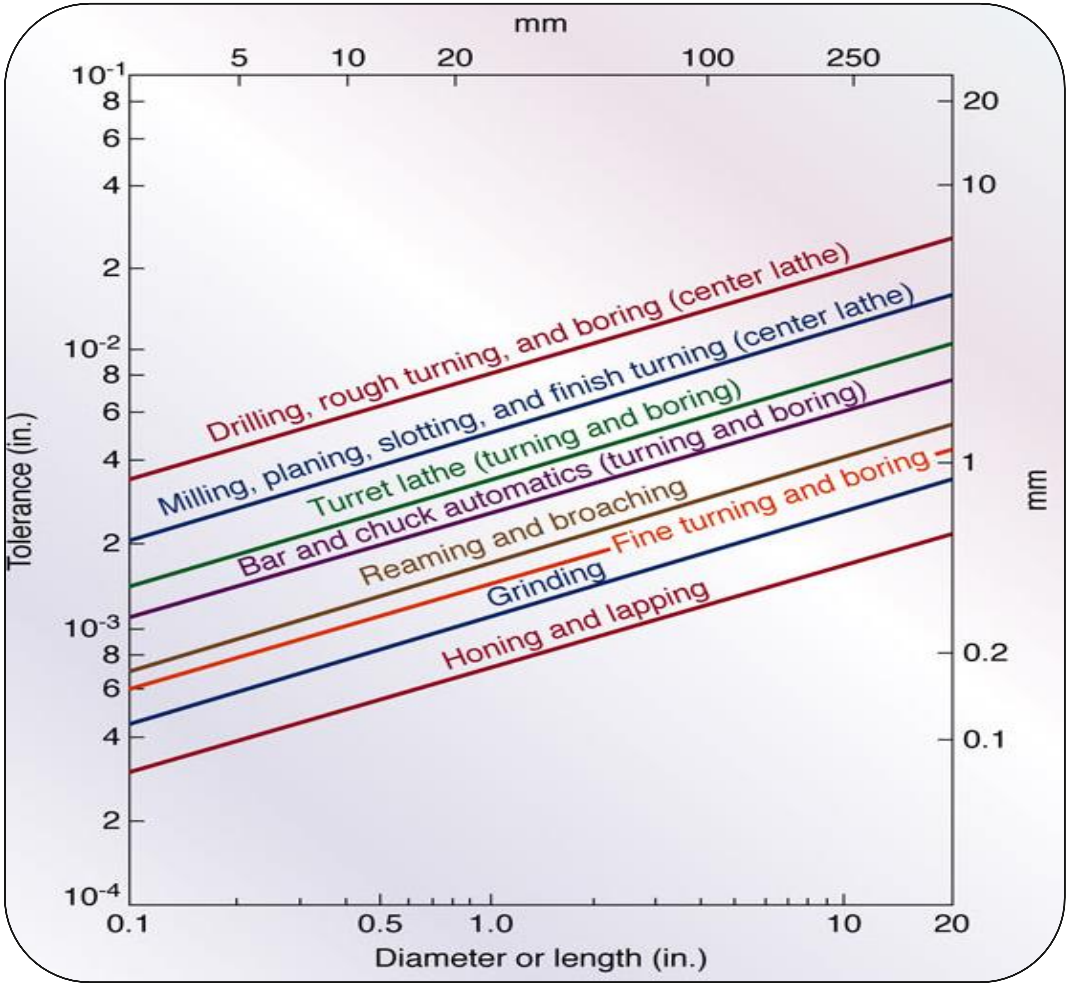


Figure 2: Range of Dimensional Tolerances in Machining as a Function of Workpiece Size

(source: Manufacturing, Engineering & Technology, Fifth Edition, by S. Kalpakjian and Steven R. Schmid)

I. Tolerancing for Interchangeability

Tolerancing / Interchangeability

- **Tolerancing is dimensioning for interchangeability**

  An interchangeable part is simply a mass produced part

  (a replacement part)

- **Tolerancing** allows a **range specification of accuracy** for every feature of a product, so the parts will fit together and function properly when assembled

- How is a feature on an interchangeable part dimensioned?
  - The feature **is not dimensioned using a single value**, but a **range** of values

$$1.00 \rightarrow \begin{matrix} 1.005 \\ .994 \end{matrix}$$

Tolerancing / Interchangeability

- **Tolerance** is the **total amount a specific dimension is permitted to vary**

- Usage of **generous tolerances** when possible is preferred because increased precision makes parts more **expensive to manufacture**

- A tolerance that specifies a large or small variation can be chosen

Understanding Tolerance

Size limits : 1.005
.994

Tolerance :1.005 - .994 = .011

Tolerancing / Interchangeability

- Why do we want a part's size to be controlled by two limits?

- It is necessary because it is **impossible to manufacture parts without some variation**

- The stated limits are a form of quality control

Tolerancing / Interchangeability

- Choosing the correct tolerance for a particular application depends on:

  - **the design intent** (end use) of the part
  - **cost**
  - **how it is manufactured**
  - **experience**

II. Tolerance Types

Tolerance Types

- The tolerancing methods presented are:

  1. **Limit dimensions**

  2. **Plus or minus tolerances**

  3. **Page or block tolerances**

1. Limit Dimensions

- **Limits** are the **maximum and minimum size** that a part can obtain and still pass inspection

  - For example, the diameter of a shaft might be specified as follows.

$$\varnothing^{1.001}_{.999} \quad \text{or} \quad \varnothing 1.001 - .999$$

1. Limit Dimension Order

- **External dimensions**:
  - The larger dimension is first or on top and the smaller dimension is last or on the bottom
- **Internal dimensions**:
  - The smaller dimension is first and the larger dimension is last

2. Plus or Minus Tolerances

- **Plus or minus tolerances** give a basic size and the variation that can occur around that basic size

$$10.0^{+0.1}_{-0.2}$$

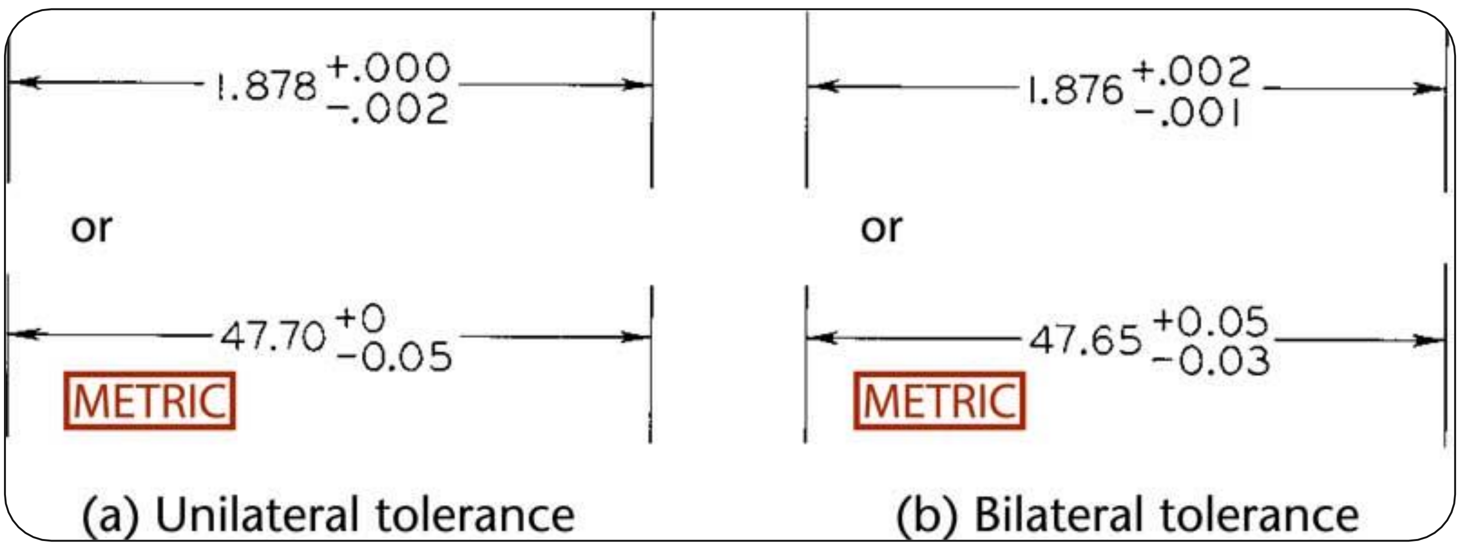


Figure 3: Example of plus or minus tolerances

3. Page or Block Tolerances

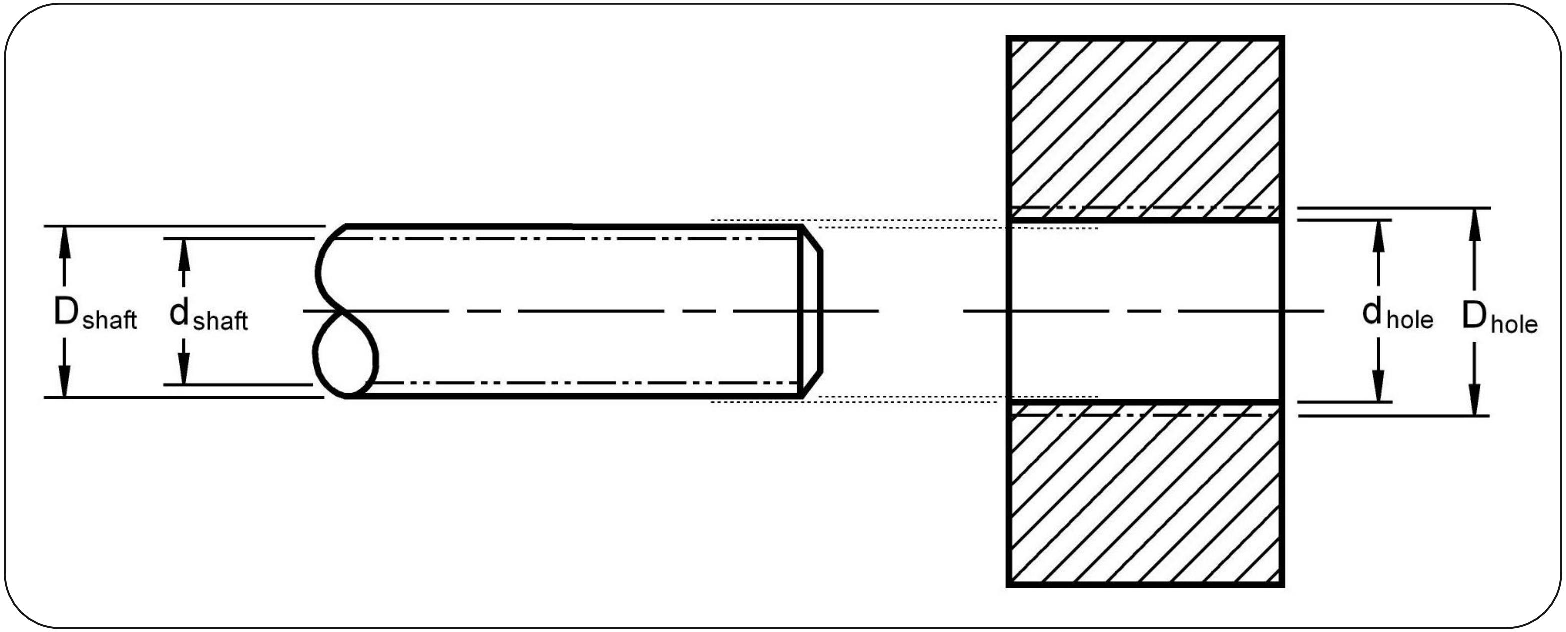
- A **page tolerance** is actually a general note that applies to all dimensions not covered by some other tolerancing type

UNLESS OTHERWISE SPECIFIED ALL:
.XX = ± .010 inch
.XXX = ± .005 inch
.XXXX = ± .002 inch

III. General Definitions

General Definitions

- What are the **limits**, **tolerance** and **allowance** for the following shaft/hole system? Are they the same or different?



General Definitions

- **Limits**

Are the **maximum** and **minimum diameters**

- **Tolerance**

Is the **difference between two limits**

- **Allowance** (Minimum Clearance)

Is the **difference** between the **largest shaft diameter** and the **smallest hole diameter**

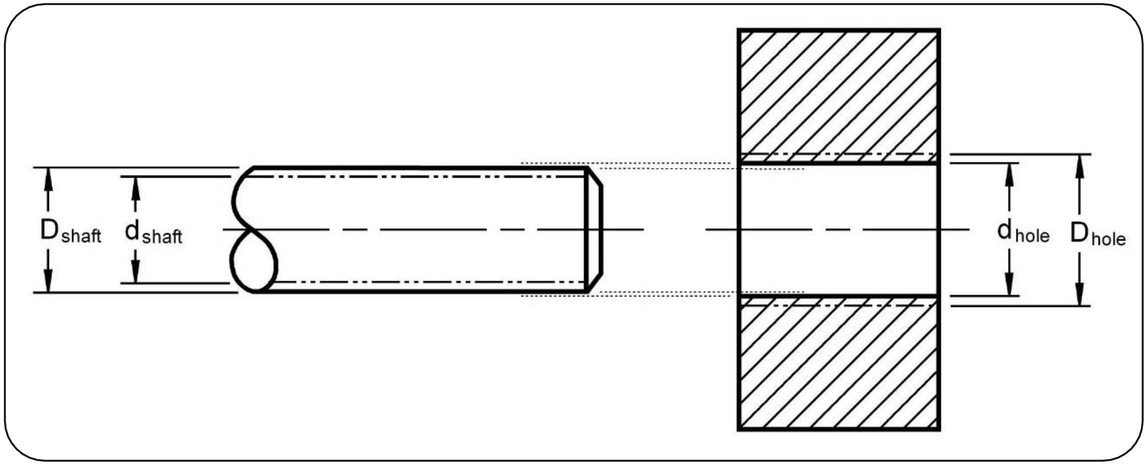
(K. Plantenberg , 2006,)

Example 1

- What are the **limits** of the shaft and the hole?

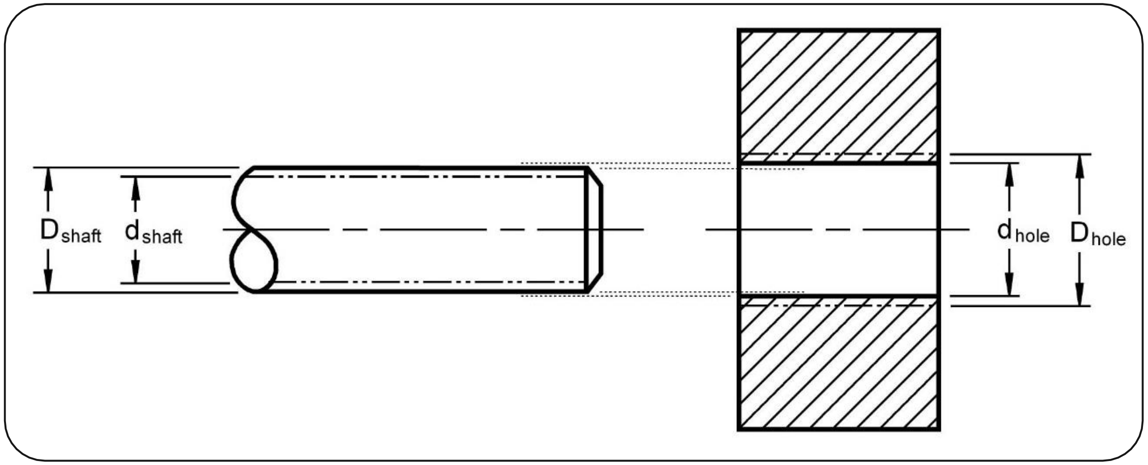
  - Shaft: D_{shaft} - d_{shaft}

  - Hole: d_{hole} - D_{hole}



Example 2

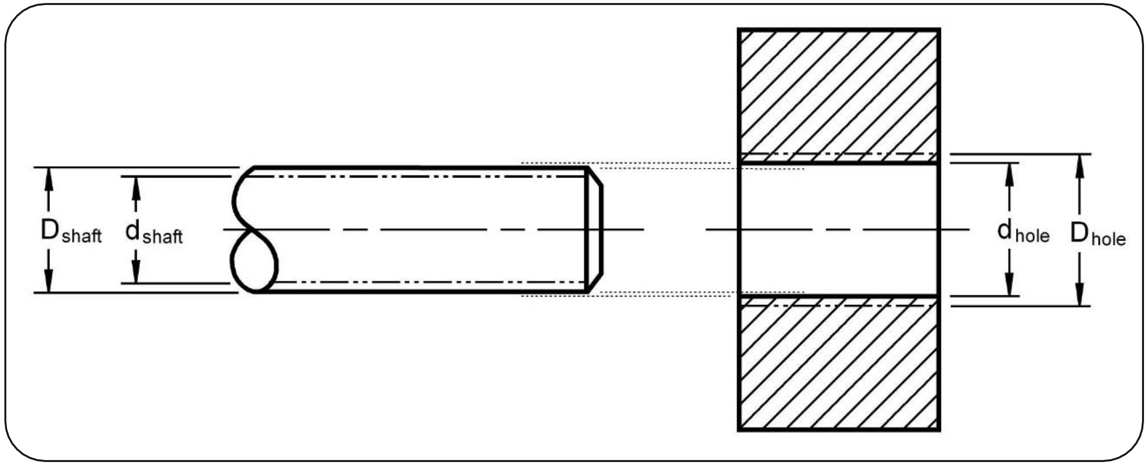
- What is the **tolerance** for the shaft and the hole?
  - Shaft: $D_{shaft} - d_{shaft} = ..$
  - Hole: $D_{hole} - d_{hole} = ..$



Example 3

- What is the **minimum clearance** (allowance)?

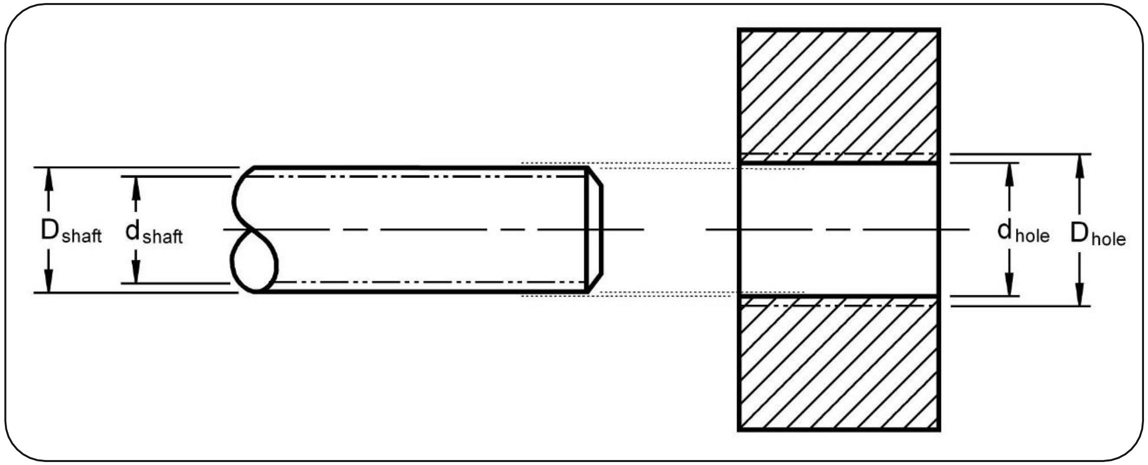
$$d_{hole} - D_{shaft} =$$



Example 4

- What is the **maximum clearance**?

$$D_{hole} - d_{shaft} =$$



IV. Tolerancing Standards

Tolerancing Standards

- **Standards** are needed to:

✓ Make it possible to manufacture parts at **different times** and in **different places** that still **assemble properly**

✓ Establish **dimensional limits for parts** that are to be **interchangeable**

Tolerancing Standards

- The two most common standards agencies are;

  - **A**merican **N**ational **S**tandards **I**nstitute (**ANSI**) / (**ASME**)

  - **I**nternational **S**tandards **O**rganization (**ISO**)

V. Inch Tolerances

Inch Tolerances Definitions

- **Limits**

The limits are the **maximum** and **minimum size** that the part is allowed to be

- **Basic Size**

The **basic size** is the size from which the **limits are calculated**

  - It is common for both the hole and the shaft and is usually the closest fraction

- **Tolerance**

The **tolerance** is the total amount a **specific dimension is permitted to vary**

(K. Plantenberg , 2006,)

Inch Tolerances Definitions

- **Maximum Material Condition (MMC):**

The MMC is the size of the part when it consists of the most material

- **Least Material Condition (LMC):**

The LMC is the size of the part when it consists of the least material

Inch Tolerances Definitions

- **Maximum Clearance:**

The maximum amount of space that can exist between the hole and the shaft

$$\text{Max. Clearance} = LMC_{hole} - LMC_{shaft}$$

- **Minimum Clearance (Allowance):**

The minimum amount of space that can exist between the hole and the shaft

$$\text{Min. Clearance} = MMC_{hole} - MMC_{shaft}$$

Types of Fits

- **Clearance Fit** :There is always a space, Min. Clearance > 0

- **Interference Fit** : There is never a space, Max. Clearance $\leq$ 0

- **Transition Fit** :Depending on the sizes of the shaft and hole there could be a space or no space, Max. Clearance > 0

  Min. Clearance < 0

- **Line Fit :**There is a space or a contact (hole diameter = shaft diameter), Max. Clearance > 0

  Min. Clearance = 0

(K. Plantenberg , 2006,)

Types of Fits

- From everyday life, list some examples of clearance and interference fits

Fit	Example
Clearance	Lock and Key Door and Door frame Coin and Coin slot
Interference	Pin in a bicycle chain Hinge pin Wooden peg and hammer toy

Example 5

- Determine the basic size and type of fit given the limits for the shaft and hole

Shaft Limits	Hole Limits		
1.500 – 1.498	1.503 – 1.505	1.5	Clearance
.755 - .751	.747 - .750	.75	Interference
.378 - .373	.371 - .375	.375	Transition
.250 - .247	.250 - .255	.25	Line

ANSI Standard Limits and Fits

The following fit types and classes are in accordance with the **ANSIB4.1-1967(R1994)standard**

- **RC: Running or Sliding Clearance fit**
  - Intended to provide running performance with suitable lubrication.
  - RC9 (loosest) – RC1 (tightest)

- **FN: Force Fits**
  - Force fits provide a constant bore pressure throughout the range of sizes.
  - FN1 – FN5 (tightest)

ANSI Standard Limits and Fits

- **Locational fits (LC, LT, LN)**

➢ **Locational fits** are intended to determine only the **location of the mating parts**

LC = Locational clearance fits

LT = Locational transition fits

LN = Locational interference fits

Example 6

- Given a basic size of .50 inches and a fit of RC8, calculate the limits for both the hole and the shaft

  - Use the **ANSI limits and fit tables** given in the following Appendix

Appendix

Basic size = .5

Fit = RC8

Nominal Size Range Inches		Class RC5		Class RC6		Class RC7		Class RC8		Class RC9	
		Standard Limits		Standard Limits		Standard Limits		Standard Limits		Standard Limits	
Over	To	Hole	Shaft	Hole	Shaft	Hole	Shaft	Hole	Shaft	Hole	Shaft
0	– 0.12	+0.6 0	-0.6 -1.0	+1.0 [illegible]	-0.6 [illegible]	+1.0 [illegible]	-1.0 -1.6	+1.6 0	-2.5 -3.5	+2.5 0	-4.0 -5.6
0.12	– 0.24	+0.7 0	-0.8 -1.3	[illegible]	[illegible]	[illegible]	-1.2 -1.9	+1.8 0	-2.8 -4.0	+3.0 0	-4.5 -6.0
0.24	– 0.40	+0.9 0	-1.0 -1.6	[illegible]	[illegible]	[illegible]	-1.6 -2.5	+2.2 0	-3.0 -4.4	+3.5 0	-5.0 -7.2
0.40	– 0.71	+1.0 0	-1.2 -1.9	[illegible] 0	[illegible] -2.2	[illegible] 0	-2.0 -3.0	+2.8 0	-3.5 -5.1	+4.0 0	-6.0 -8.8
0.71	– 1.19	+1.2 0	-1.6 -2.4	+2.0 0	-1.6 -2.8	+2.0 0	-2.5 -3.7	+3.5 0	-4.5 -6.5	+5.0 0	-7.0 -10.5
1.19	– 1.97	+1.6 0	-2.0 -3.0	+2.5 0	-2.0 -3.6	+2.5 0	-3.0 -4.6	+4.0 0	-5.0 -7.5	+6.0 0	-8.0 12.0
1.97	– 3.15	+1.8 0	-2.5 -3.7	+3.0 0	-2.5 -4.3	+3.0 0	-4.0 -5.8	+4.5 0	-6.0 -9.0	+7.0 0	-9.0 -13.5
3.15	– 4.73	+2.2 0	-3.0 -4.4	+3.5 0	-3.0 -5.2	+3.5 0	-5.0 -7.2	+5.0 0	-7.0 -10.5	+9.0 0	-10.0 -15.0
4.73	– 7.09	+2.5 0	-3.5 -5.1	+4.0 0	-3.5 -6.0	+4.0 0	-6.0 -8.5	+6.0 0	-8.0 -12.0	+10.0 0	-12.0 -18.0
7.09	– 9.85	+2.8 0	-4.0 -5.8	+4.5 0	-4.0 -6.8	+4.5 0	-7.0 -9.8	+7.0 0	-10.0 -14.5	+12.0 0	-15.0 -22.0
9.85	– 12.41	+3.0 0	-5.0 -7.0	+5.0 0	-5.0 -8.0	+5.0 0	-8.0 -11.0	+8.0 0	-12.0 -17.0	+12.0 0	-18.0 -26.0
12.41	– 15.75	+3.5 0	-6.0 -8.2	+6.0 0	-6.0 -9.5	+6.0 0	-10.0 -13.5	+9.0 0	-14.0 -20.0	+14.0 0	-22.0 -31.0
15.75	– 19.69	+4.0 0	-8.0 -10.5	+6.0 0	-8.0 -12.0	+6.0 0	-12.0 -16.0	+10.0 0	-16.0 -22.0	+16.0 0	-25.0 -35.0

Example 7

- Given a basic size of .50 inches and a fit of RC8, calculate the limits for both the hole and the shaft.

  - Standard Limits Hole = +2.8   0
  - Standard Limits Shaft = -3.5   -5.1

- These are the values that we add/subtract from the basic size to obtain the limits

Example 8

- Given a basic size of .50 inches and a fit of RC8, calculate the limits for both the hole and the shaft.

  - Hole Limits = .50 - 0 = .5000

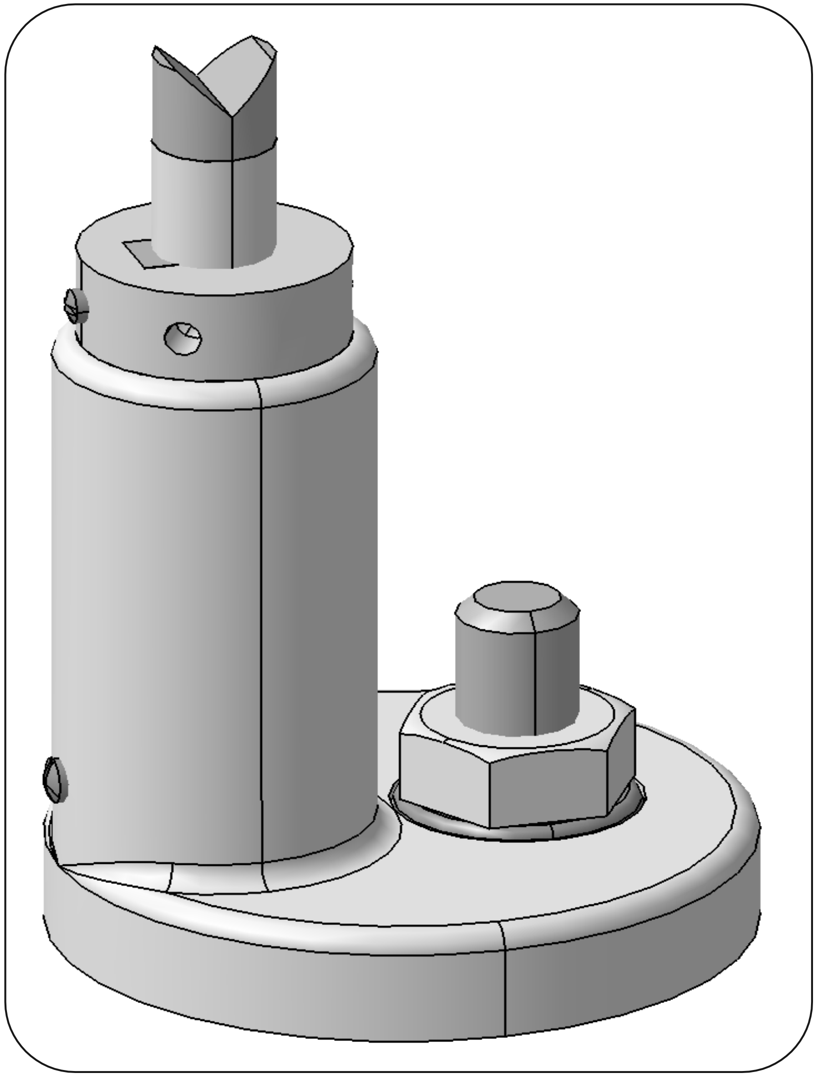
    .50 + .0028 = .5028

  - Shaft Limits = .50 - .0035 = .4965

    .50 - .0051 = .4949

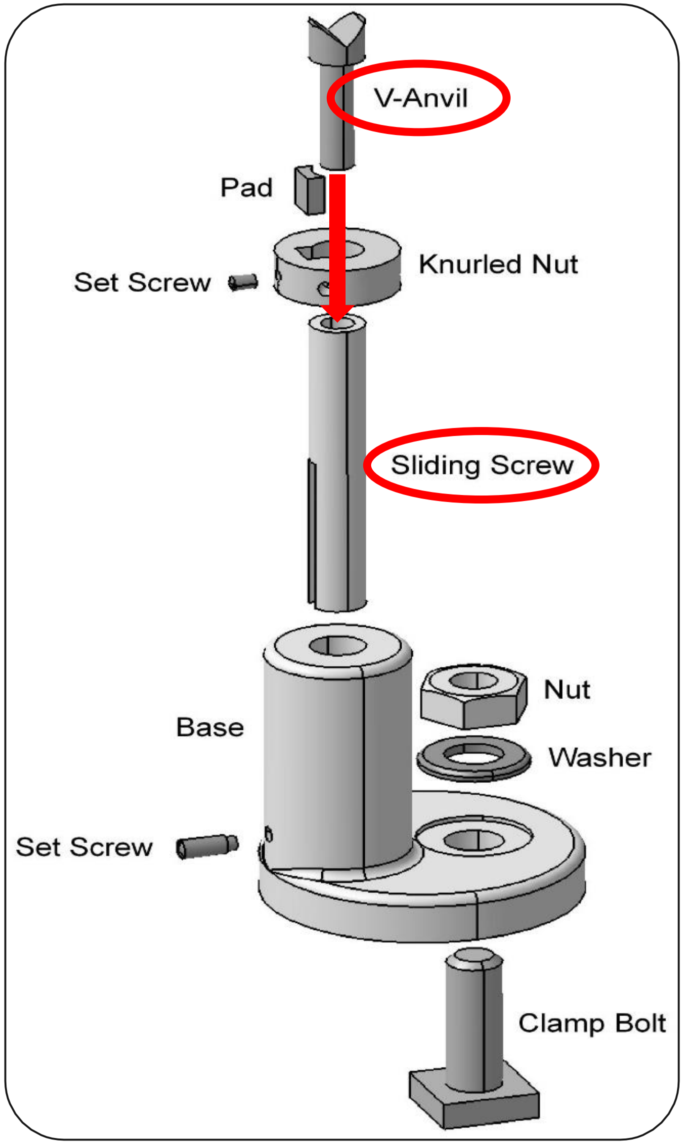
Example 9

- Consider the **Milling Jack assembly** shown
  - Notice that there are **many parts that fit into or around other parts**
  - Each of these parts is **toleranced** to ensure **proper fit** and **function**



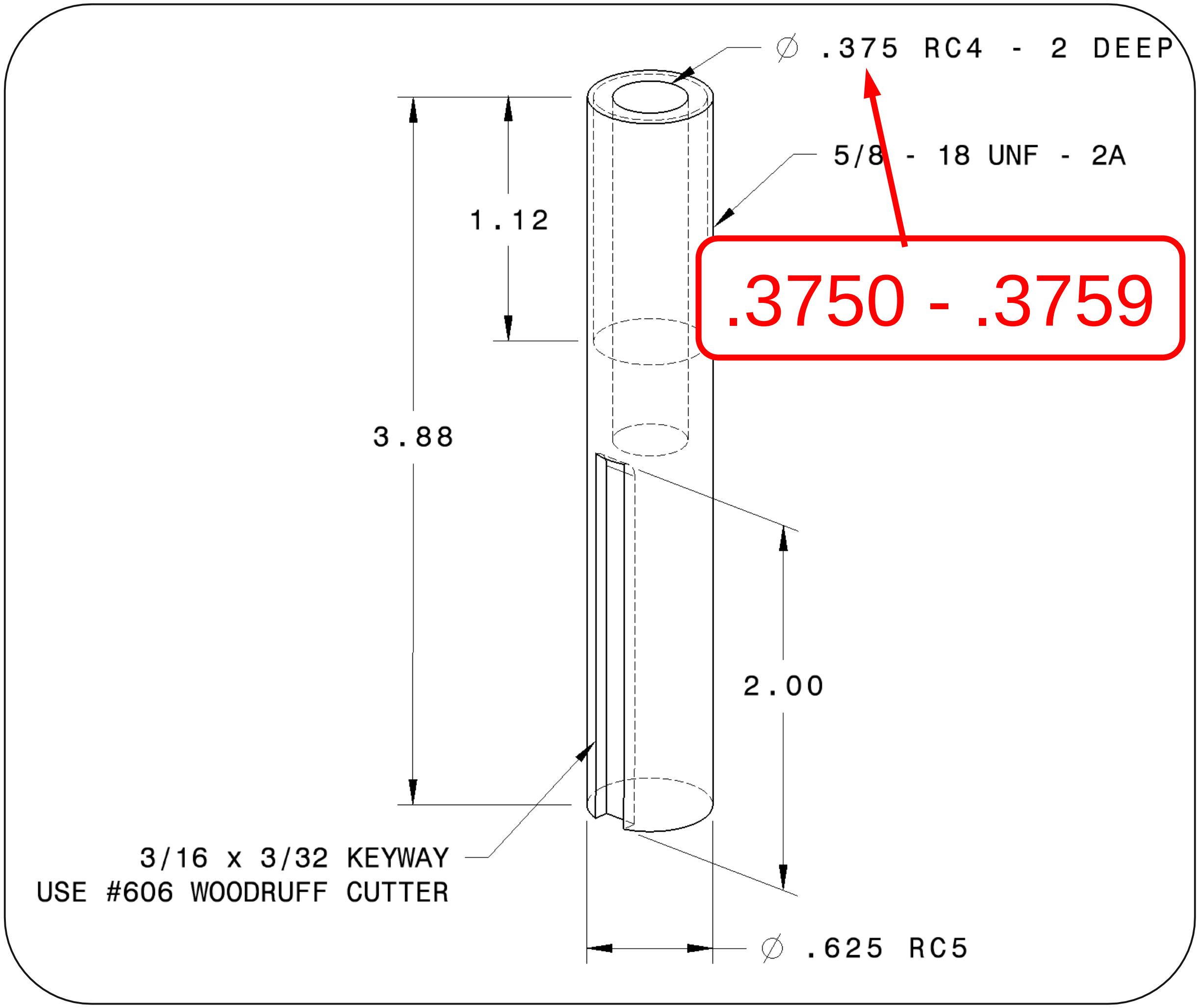
(K. Plantenberg , 2006,)

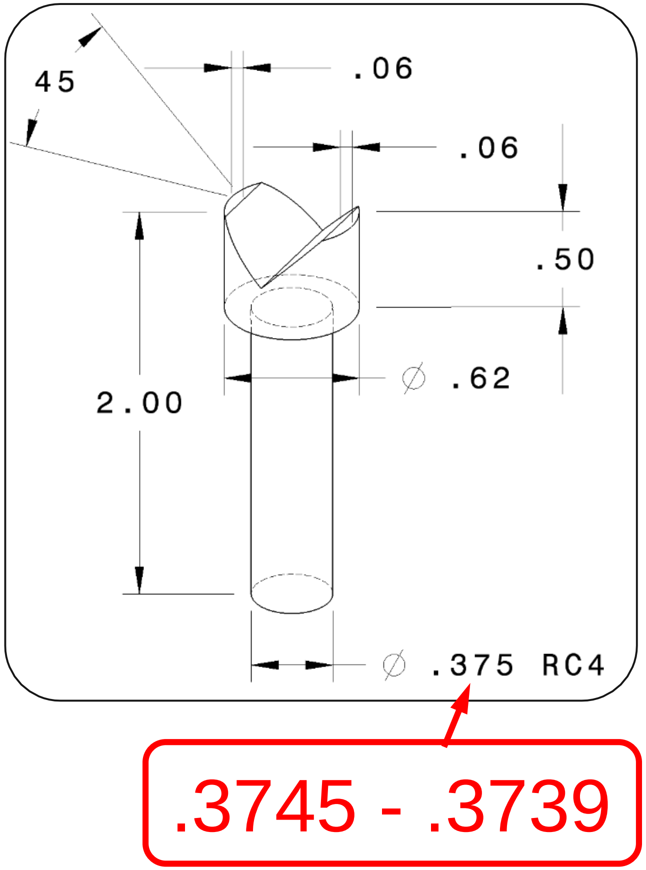
Example 9



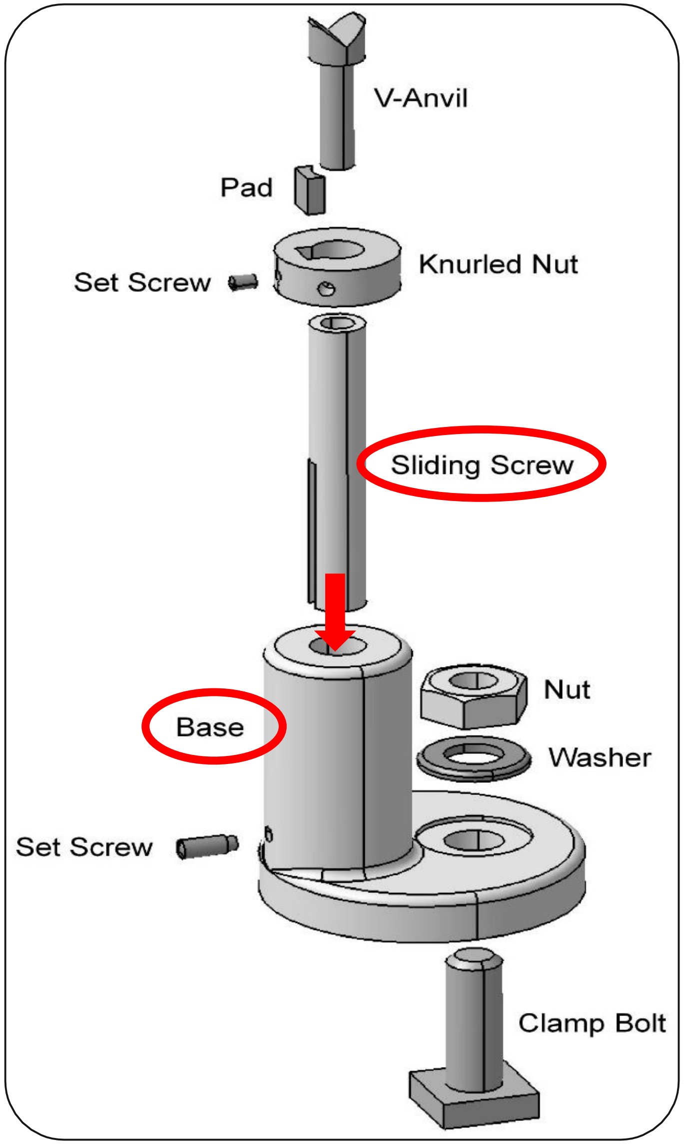
- The V-Anvil fits into the Sliding Screw with a RC4 fit
- The basic size is .375 (3/8). Determine the limits for both parts

Example 9


∅ .375 RC4 - 2 DEEP
5/8 - 18 UNF - 2A
1.12
3.88
2.00
3/16 x 3/32 KEYWAY
USE #606 WOODRUFF CUTTER
∅ .625 RC5
.3750 - .3759

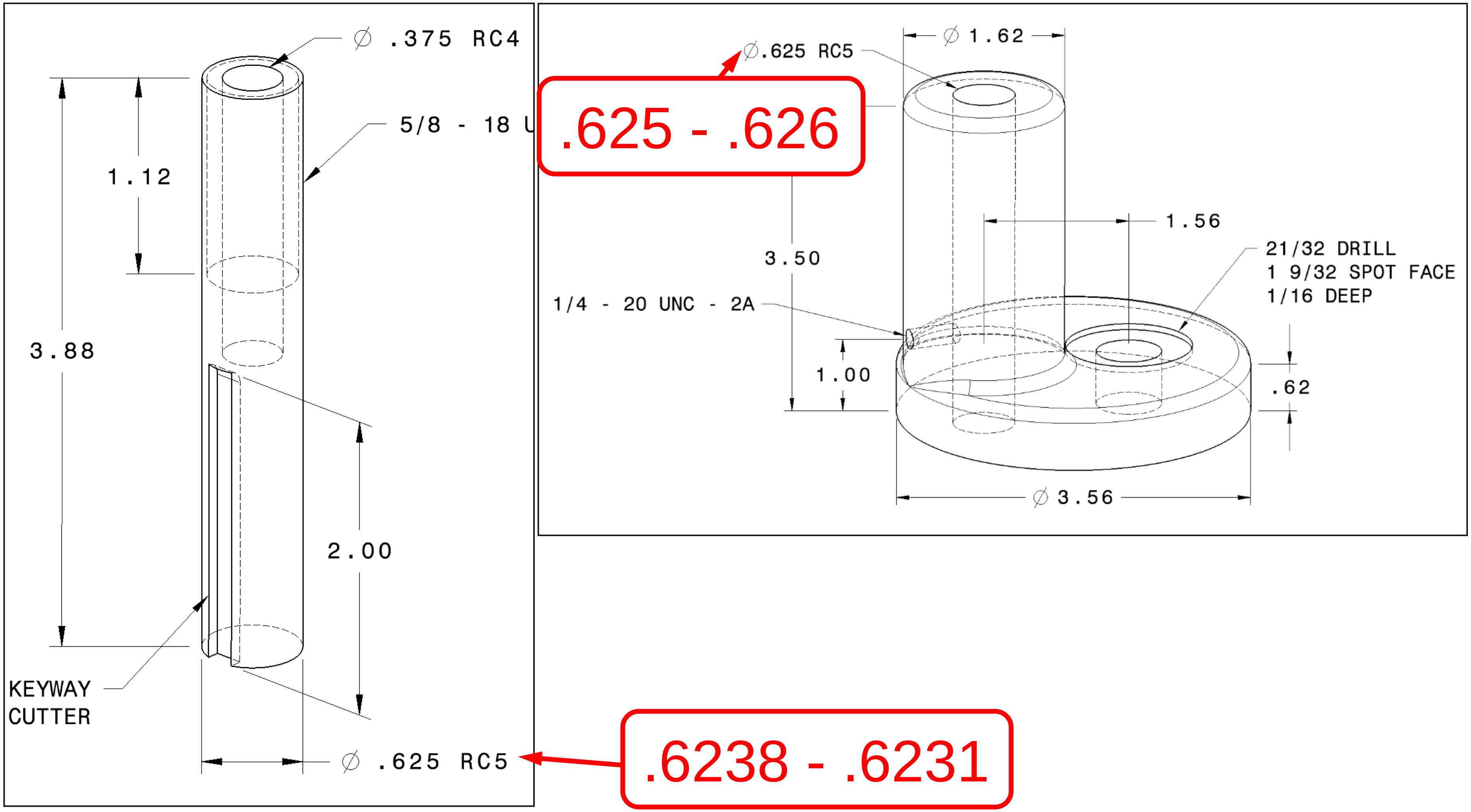

45
.06
.06
.50
2.00
∅ .62
∅ .375 RC4
.3745 - .3739

Example 10



- The Sliding Screw fits into the Base with a RC5 fit
- The basic size is .625 (5/8). Determine the limits for both parts

Example 10



∅ .375 RC4
5/8 - 18 U
1.12
3.88
2.00
KEYWAY CUTTER
∅ .625 RC5
∅.625 RC5
∅ 1.62
.625 - .626
1.56
21/32 DRILL
1 9/32 SPOT FACE
1/16 DEEP
3.50
1/4 - 20 UNC - 2A
1.00
.62
∅ 3.56
.6238 - .6231

VI. Metric Tolerances

Metric Tolerances Definitions

- Limits, Basic Size, Tolerance, MMC and LMC have the same definition as in the inch tolerance section

- **Upper deviation**

The upper deviation is the difference between the basic size and the permitted maximum size of the part

$$UD = |\text{basic size} - D_{max}|$$

- **Lower deviation**

The lower deviation is the difference between the basic size and the minimum permitted size of the part

$$LD = |\text{basic size} - D_{min}|$$

Metric Tolerances Definitions

- **Fundamental deviation**

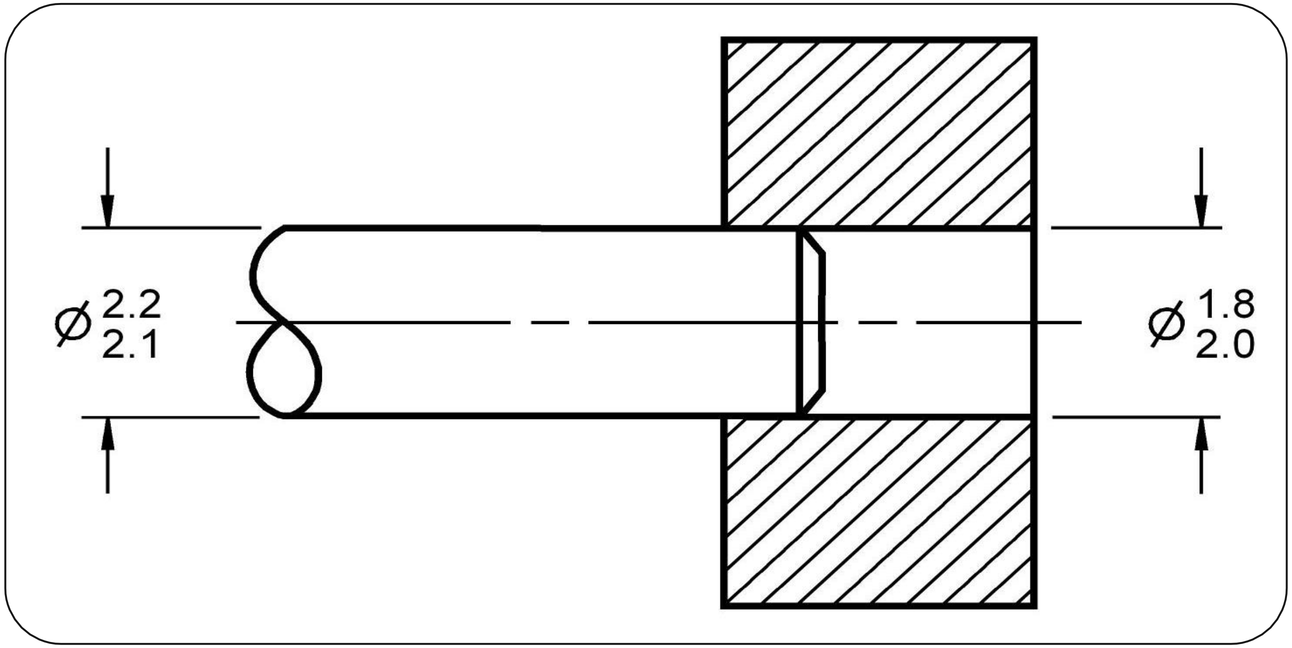
The fundamental deviation is the closest deviation to the basic size

- ➢ The fundamental deviation is the smaller of the UD and the LD
- ➢ A letter in the fit specification represents the fundamental deviation

Example: Metric Fit = H11/c11

Exercise 1

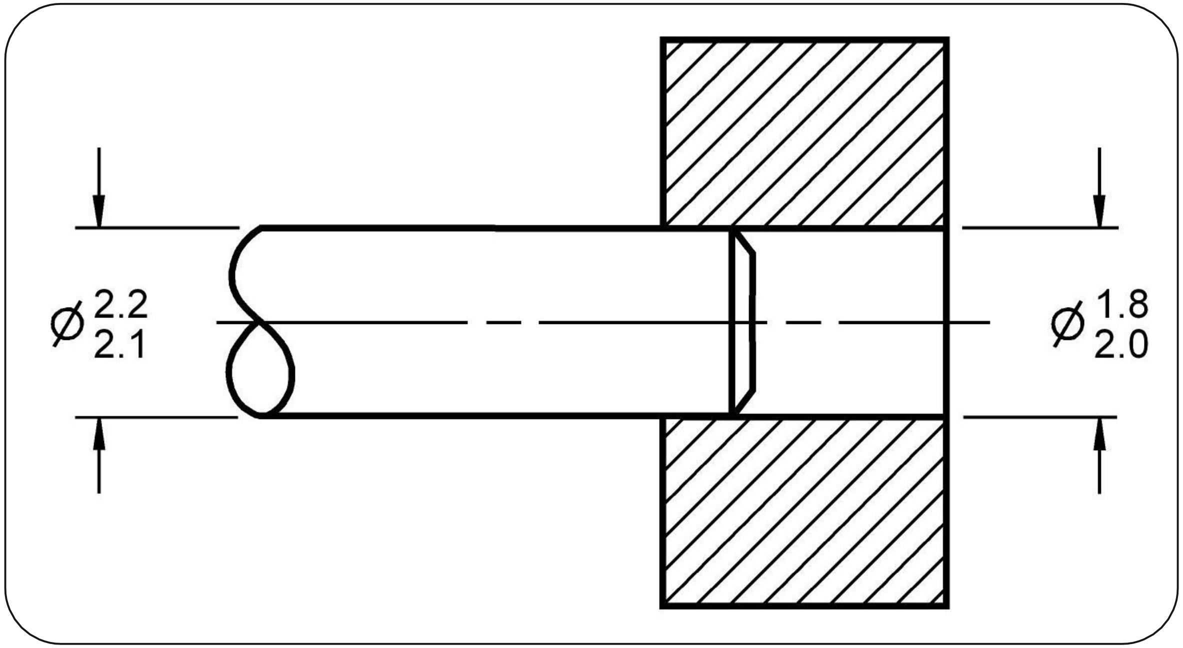
- Fill in the following table



	Shaft	Hole
UD	0.2	0
LD	0.1	0.2
FD	0.1	0

Exercise 2

- Fill in the following table



Type of fit	Interference

Metric Tolerances Definitions

- **International tolerance grade number (IT#)**

The IT#'s are a set of tolerances that vary according to the basic size and provide the same relative level of accuracy within a given grade

- The number in the fit specification represents the IT#
- A smaller number provides a smaller tolerance

Metric Tolerances Definitions

- **Tolerance zone**

The fundamental deviation in combination with the IT# defines the tolerance zone

- The IT# establishes the magnitude of the tolerance zone or the amount that the dimension can vary

- The fundamental deviation establishes the position of the tolerance zone with respect to the basic size

(K. Plantenberg , 2006,)

Available Metric Fits

Hole Basis	Shaft Basis	Fit
H11/c11	C11/h11	Loose running
H9/d9	D9/h9	Free running
H8/f7	F8/h7	Close running
H7/g6	G7/h6	Sliding
H7/h6	H7/h6	Locational clearance
H7/k6 or H7/n6	K7/h6 or N7/h6	Locational transition
H7/p6	P7/h6	Locational interference
H7/s6	S7/h6	Medium drive
H7/u6	U7/h6	Force

Tolerance Designation

- A Metric fit is specified by stating the fundamental deviation and the IT#
- IT# = the amount that the dimension can vary (tolerance zone size)
- Fundamental deviation (letter) = establishes the position of the tolerance zone with respect to the basic size
- Hole = upper case
- Shaft = lower case

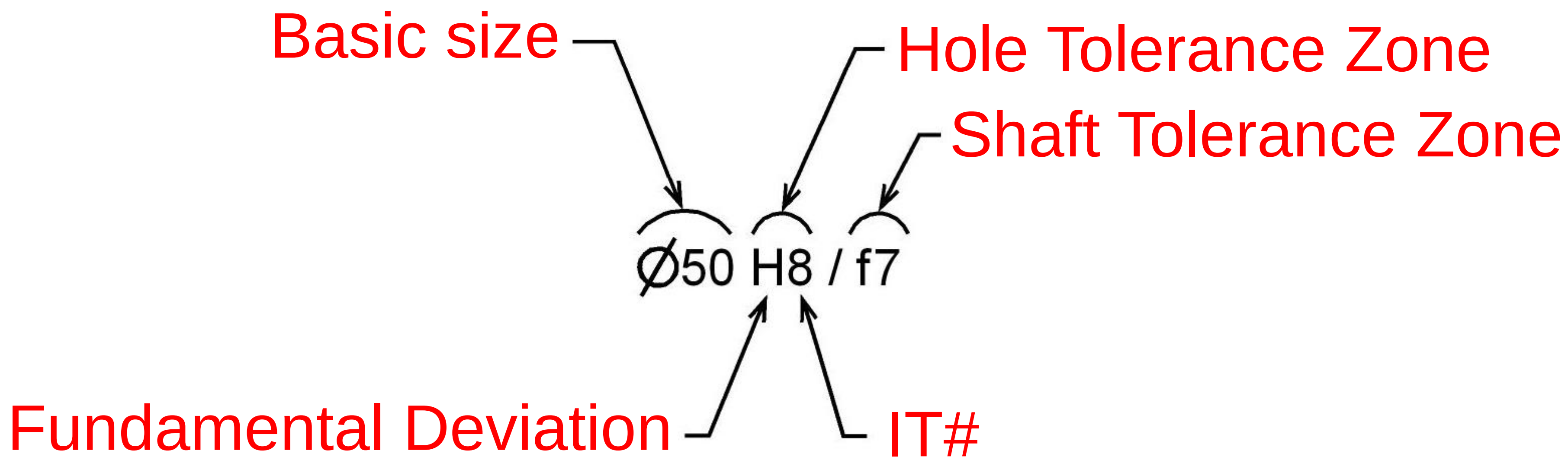
Tolerance Designation

- Fits are specified by using the:
  - fundamental deviation (letter)
  - IT# (International Tolerance Grade #)

- When specifying the fit:
  - The hole = upper case letter
  - The shaft = lower case letter

Example 11

- Fill in the appropriate name for the fit component



Basic Hole / Basic Shaft Systems

- Metric limits and fits are divided into two different systems; the **basic hole system** and the **basic shaft system**
- **Basic hole system**: The basic hole system is used when you want the basic size to be attached to the hole dimension.
  - For example, if you want to tolerance a shaft based on a hole produced by a standard drill, reamer, broach, or another standard tool
- **Basic shaft system**: The basic shaft system is used when you want the basic size to be attached to the shaft dimension
  - For example, if you want to tolerance a hole based on the size of a purchased a standard drill rod

Example 12

- Identify the type of fit and the system used to determine the limits of the following shaft and hole pairs

Shaft	Hole	Type of Fit	System
9.987 – 9.972	10.000 – 10.022	Clearance	Hole
60.021 – 60.002	60.000 – 60.030	Transition	Hole
40.000 – 39.984	39.924 – 39.949	Interference	Shaft

(K. Plantenberg , 2006,)

Example 13

- Find the limits, tolerance, type of fit, and type of system for a n30 H11/c11 fit
  - Use the tolerance tables given below

Basic Size	Loose Running		Free Running		Close Running		Sliding		Locational Clearance	
	Hole H11	Shaft c11	Hole H9	Shaft d9	Hole H8	Shaft f7	Hole H7	Shaft g6	Hole H7	Shaft h6
1 max	1.060	0.940	1.025	0.980	1.014	0.994	1.010	0.998	1.010	1.000
min	1.000	0.880	1.000	0.955	1.000	0.984	1.000	0.992	1.000	0.994
1.2 max	1.260	1.140	1.225	1.180	1.214	1.194	1.210	1.198	1.210	1.200
min	1.200	1.080	1.200	1.155	1.200	1.184	1.200	1.192	1.200	1.194
1.6 max	1.660	1.540	1.625	1.580	1.614	1.594	1.610	1.598	1.610	1.600
min	1.600	1.480	1.600	1.555	1.600	1.584	1.600	1.592	1.600	1.594
2 max	2.060	1.940	2.025	1.980	2.014	1.994	2.010	1.998	2.010	2.000
min	2.000	1.880	2.000	1.955	2.000	1.984	2.000	1.992	2.000	1.994
16 max	16.110	15.905	16.043	15.950	16.027	15.984	16.018	15.994	16.018	16.000
min	16.000	15.795	16.000	15.907	16.000	15.966	16.000	15.983	16.000	15.989
20 max	20.130	19.890	20.052	19.935	20.033	19.980	20.021	19.993	20.021	20.000
min	20.000	19.760	20.000	19.883	20.000	19.959	20.000	19.980	20.000	19.987
25 max	25.130	24.890	25.052	24.935	25.033	24.980	25.021	24.993	25.021	25.000
min	25.000	24.760	25.000	24.883	25.000	24.959	25.000	24.980	25.000	24.987
30 max	30.130	29.890	30.052	29.935	30.033	29.980	30.021	29.993	30.021	30.000
min	30.000	29.760	30.000	29.883	30.000	29.959	30.000	29.980	30.000	29.987

Example 14

- Find the limits, tolerance, type of fit, and type of system for a n30 H11/c11 fit.

	Shaft	Hole
Limits	29.890 – 29.760	30.000 – 30.130
Tolerance	0.13	0.13
System	Hole	
Fit	Clearance – Loose Running	

Example 15

- Find the limits, tolerance, type of fit, and type of system for a n30 P7/h6 fit
  - ➢ Use the tolerance tables given below

Basic Size	Locational Transition		Locational Transition		Locational Interference		Medium Drive		Force	
	Hole K7	Shaft h6	Hole N7	Shaft h6	Hole P7	Shaft h6	Hole S7	Shaft h6	Hole U7	Shaft h6
1 max	1.000	1.000	0.996	1.000	0.994	1.000	0.986	1.000	0.982	1.000
min	0.990	0.994	0.986	0.994	0.984	0.994	0.976	0.994	0.972	0.994
1.2 max	1.200	1.200	1.196	1.200	1.194	1.200	1.186	1.200	1.182	1.200
min	1.190	1.194	1.186	1.194	1.184	1.194	1.176	1.194	1.172	1.194
1.6 max	1.600	1.600	1.596	1.600	1.594	1.600	1.586	1.600	1.582	1.600
min	1.590	1.594	1.586	1.594	1.584	1.594	1.576	1.594	1.572	1.594
2 max	2.000	2.000	1.996	2.000	1.994	2.000	1.986	2.000	1.982	2.000
min	1.990	1.994	1.986	1.994	1.984	1.994	1.976	1.994	1.972	1.994
16 max	16.006	16.000	15.995	16.000	15.989	16.000	15.979	16.000	15.974	16.000
min	15.988	15.989	15.977	15.989	15.971	15.989	15.961	15.989	15.956	15.989
20 max	20.006	20.000	19.993	20.000	19.986	20.000	19.973	20.000	19.967	20.000
min	19.985	19.987	19.972	19.987	19.965	19.987	19.952	19.987	19.946	19.987
25 max	25.006	25.000	24.993	25.000	24.986	25.000	24.973	25.000	24.960	25.000
min	24.985	24.987	24.972	24.987	24.965	24.987	24.952	24.987	24.939	24.987
30 max	30.006	30.000	29.993	30.000	29.986	30.000	29.973	30.000	29.960	30.000
min	29.985	29.987	29.972	29.987	29.965	29.987	29.952	29.987	29.939	29.987

Example 16

- Find the limits, tolerance, type of fit, and type of system for a n30 P7/h6 fit

	Shaft	Hole
Limits	30.000 – 29.987	29.965 – 29.986
Tolerance	0.013	0.021
System	Shaft	
Fit	Locational Interference	

VII. Selecting Tolerances

Selecting Tolerances

- **Tolerances** will govern the method of manufacturing

  - When the **tolerances are reduced**, the **cost** of manufacturing **rises very rapidly**

  - Specify **as generous a tolerance as possible** without interfering with the function of the part

Selecting Tolerances

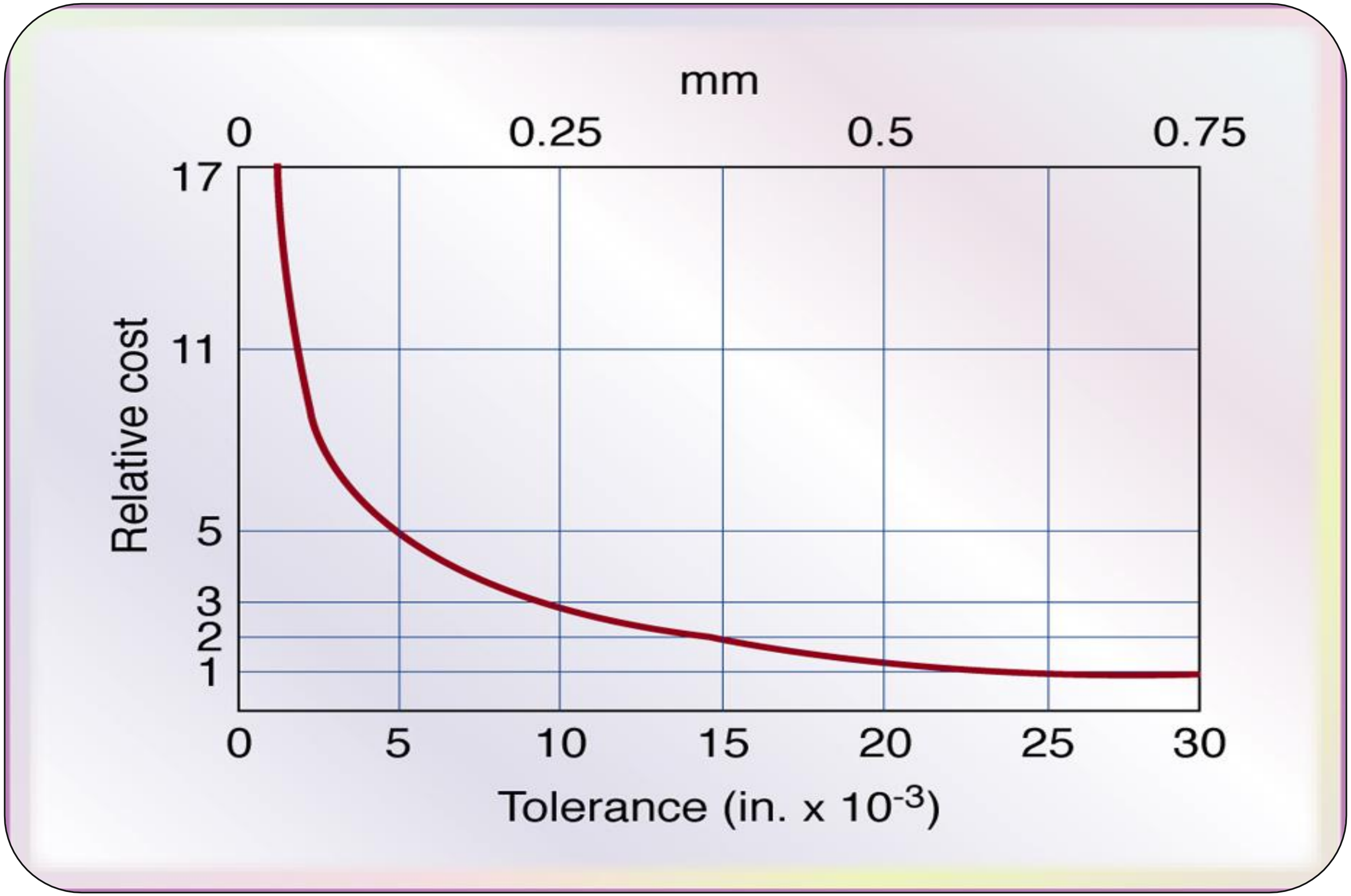


Figure 4: Dependence of manufacturing cost on dimensional tolerances

(source: Manufacturing, Engineering & Technology, Fifth Edition, by S. Kalpakjian and Steven R. Schmid)

Selecting Tolerances

- Choosing the most appropriate tolerance depends on many factors such as:
  - **Length of engagement**
  - **Bearing load**
  - **Speed**
  - **Lubrication**
  - **Temperature**
  - **Humidity**
  - **Material**
- **Experience** also plays a significant role

Machining and IT Grades

	IT Grades							
Machining Operation	4	5	6	7	8	9	10	11
Lapping & Honing	■	■						
Cylindrical Grinding		■	■	■				
Surface Grinding		■	■	■	■			
Diamond Turning		■	■	■				
Diamond Boring		■	■	■				
Broaching		■	■	■	■			
Reaming			■	■	■	■	■	
Turning				■	■	■	■	■
Boring					■	■	■	■
Milling							■	■
Planing & Shaping							■	■
Drilling							■	■
Punching							■	■
Die Casting								■

VIII.Tolerance Accumulation

Tolerance Accumulation

- Figures 5-7 compare the tolerance values resulting from the following **three methods of dimensioning**

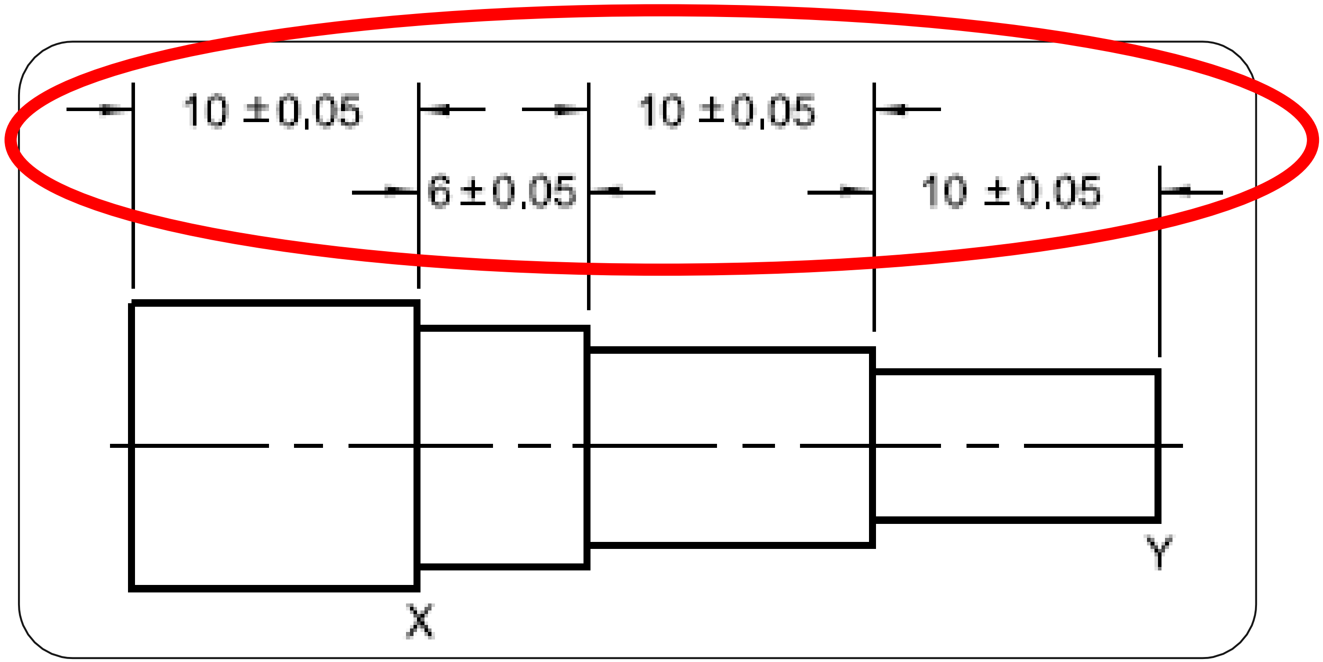


Figure 5: Chain Dimensioning-Greatest tolerance accumulation between X and Y

(a) Chain Dimensioning: The maximum variation between two features is equal to the sum of the tolerances on the intermediate distances; this results in the greatest tolerance accumulation. In this figure the tolerance accumulation between surfaces X and Y is ± 0.15

(K. Plantenberg , 2006,)

Tolerance Accumulation

(b) Base Line Dimensioning: The maximum variation between two features is equal to the sum of the tolerances on the two dimensions from their origin to the features; this results in a reduction of the tolerance accumulation. In this figure, the tolerance accumulation between surfaces X and Y is ±0.1

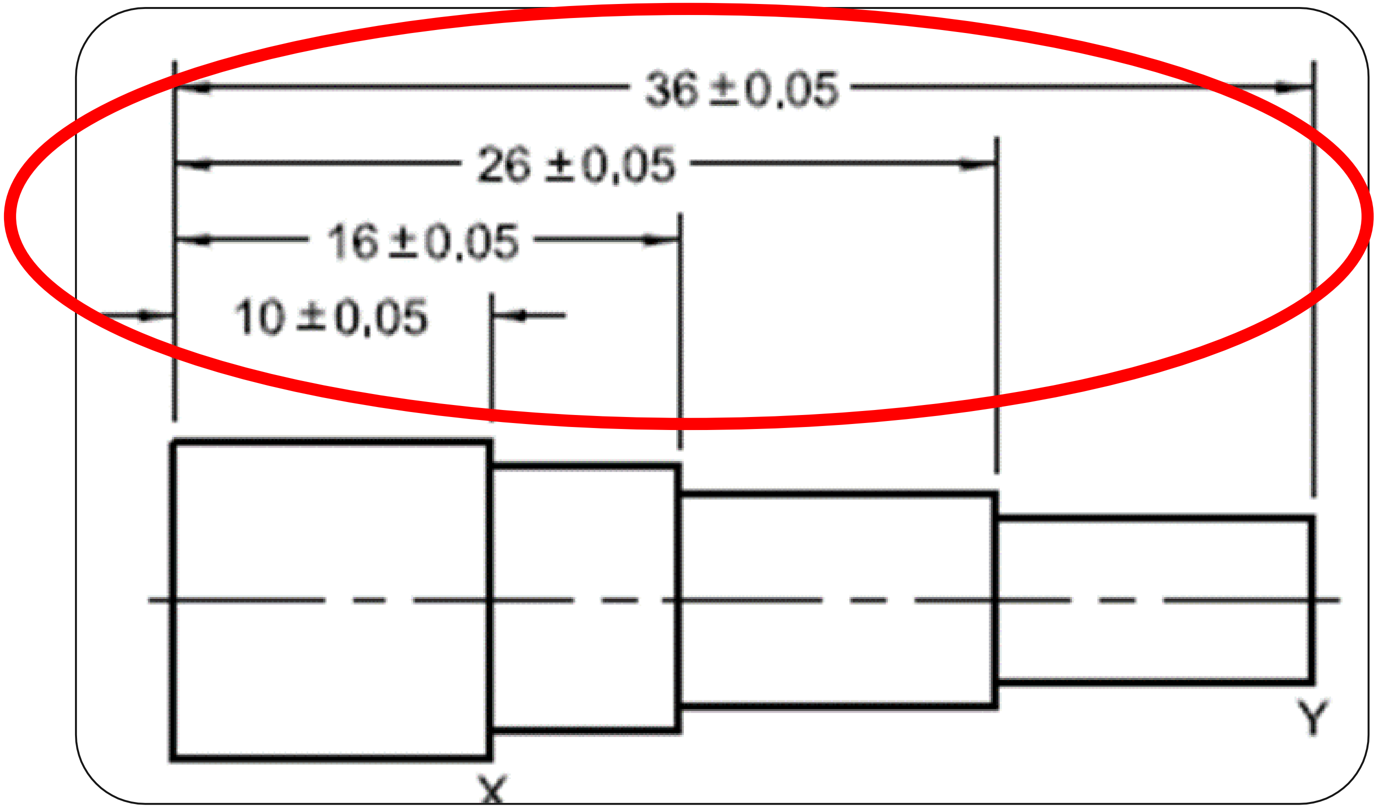


Figure 6: Base Line Dimensioning-Lesser tolerance accumulation between X and Y

Tolerance Accumulation

(c) Direct Dimensioning: The maximum variation between two features is controlled by the tolerance on the dimension between the features; this results in the least tolerance. In this figure, the tolerance between surfaces X and Y is ±0.05

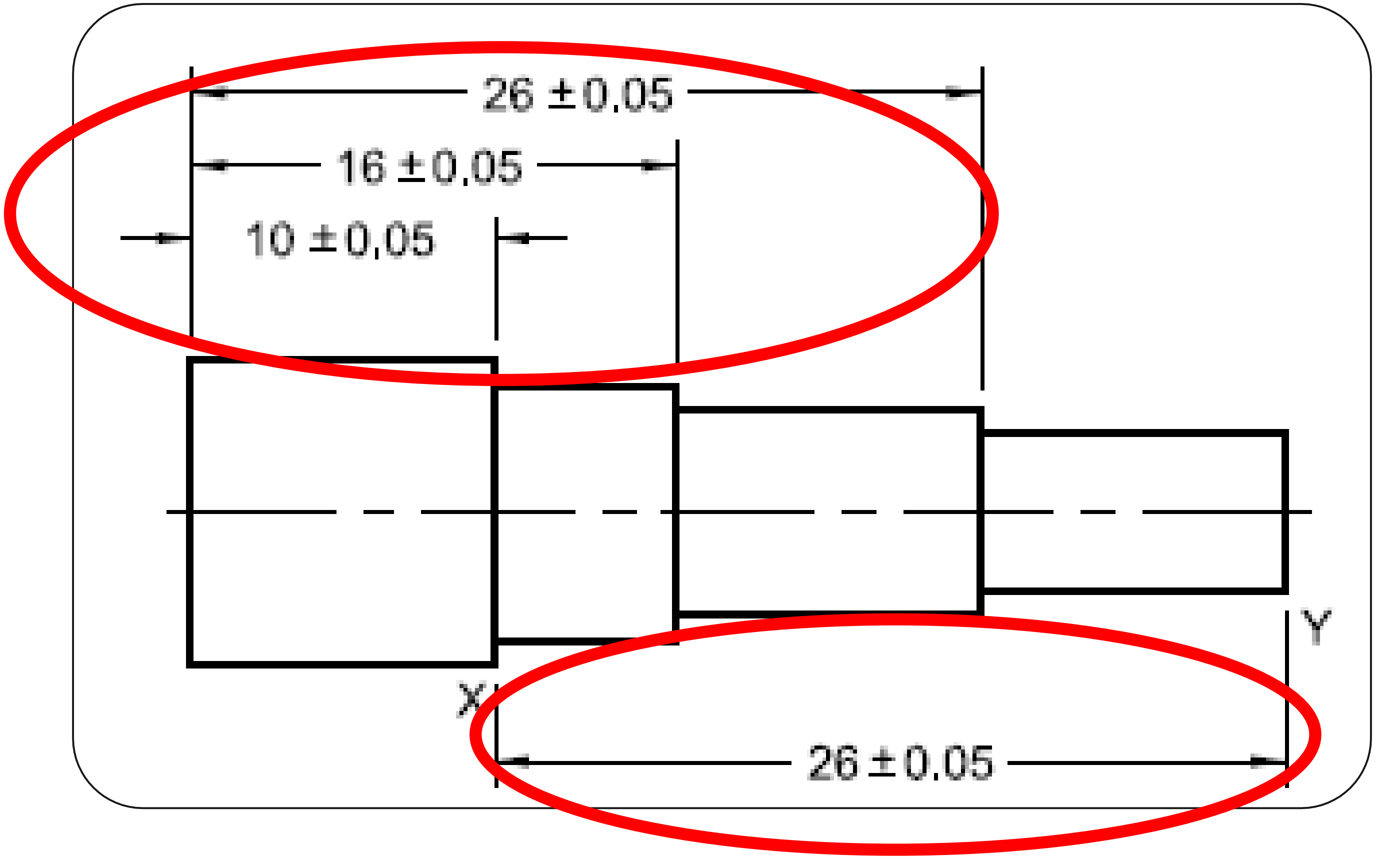
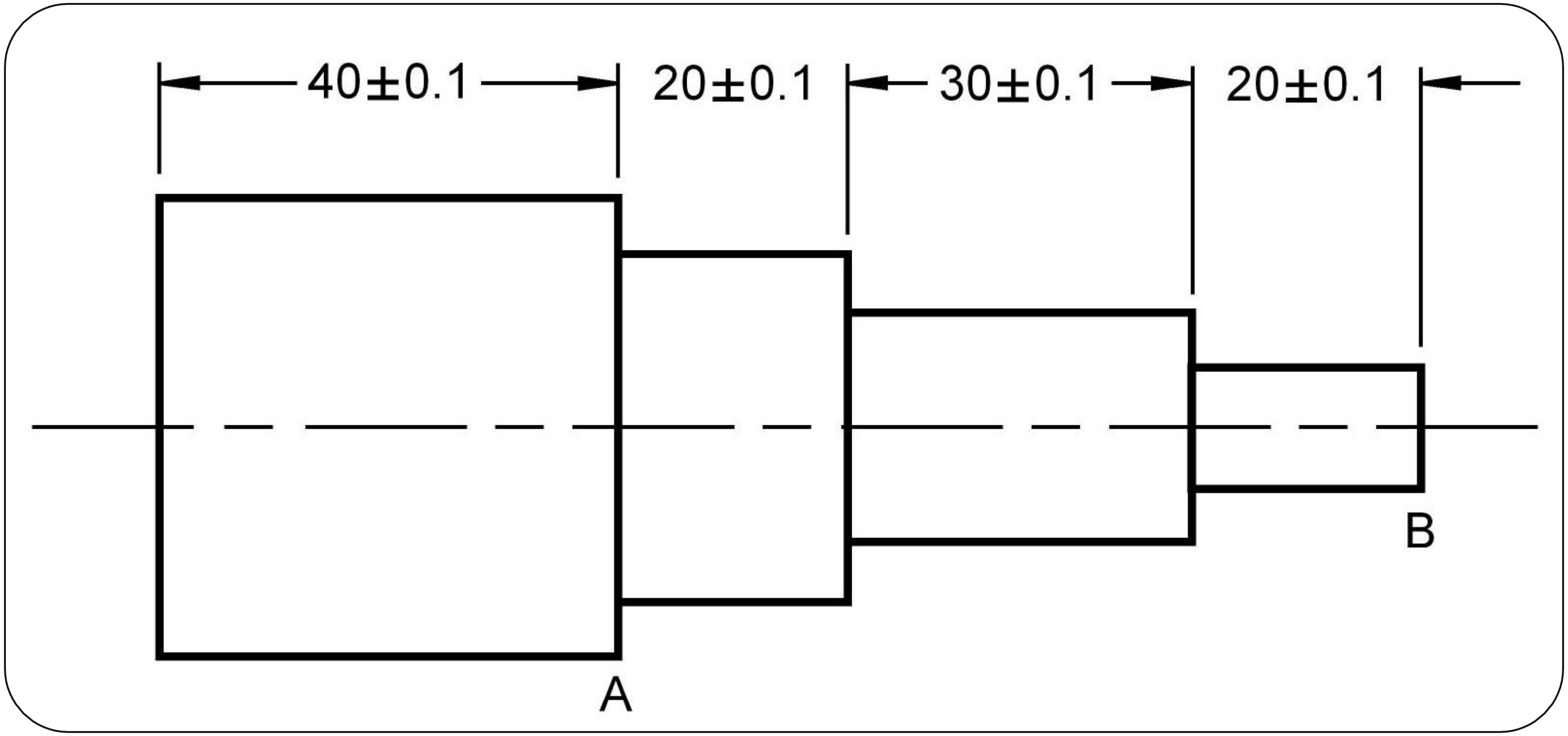


Figure 7 :Direct Dimensioning-Least tolerance between X and Y

(K. Plantenberg , 2006,)

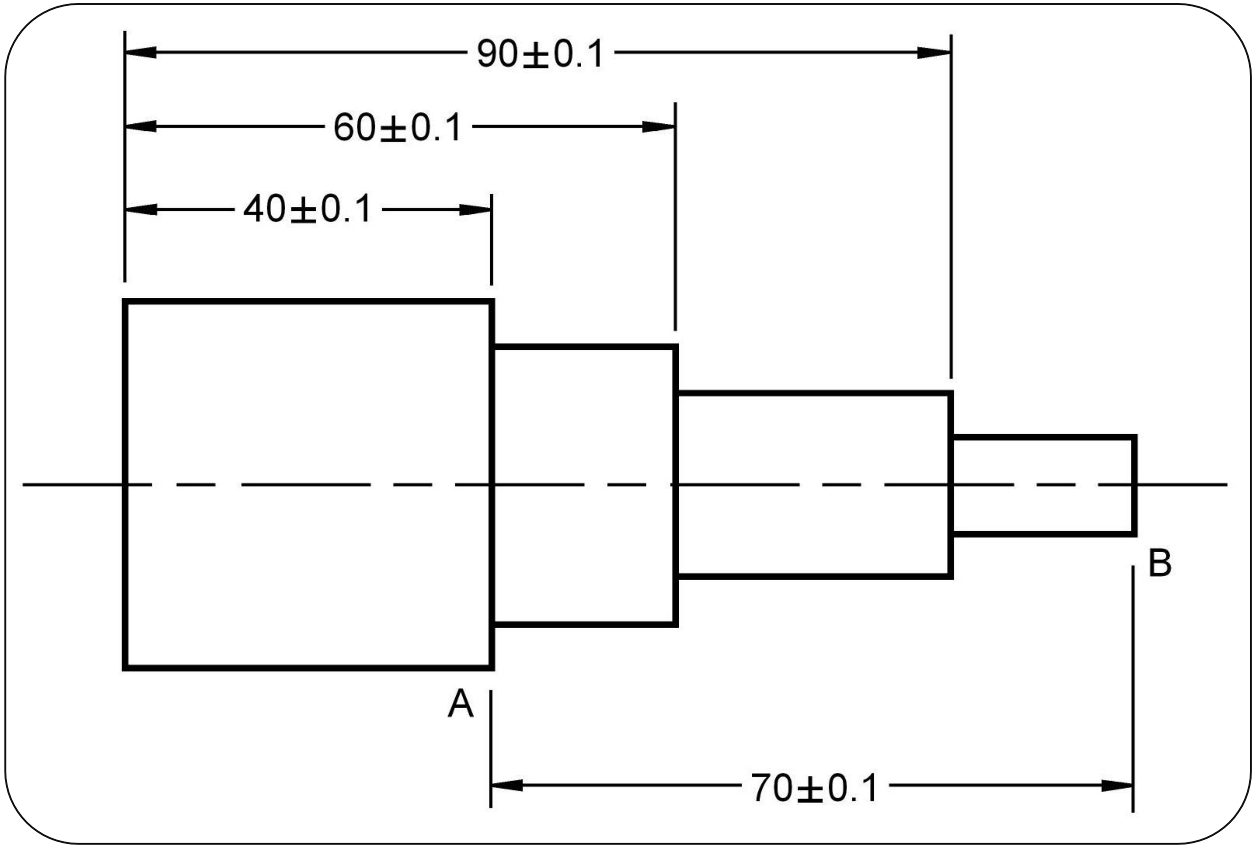
Tolerance Accumulation

- The **tolerance** between two features of a part **depends on the number of controlling dimensions**



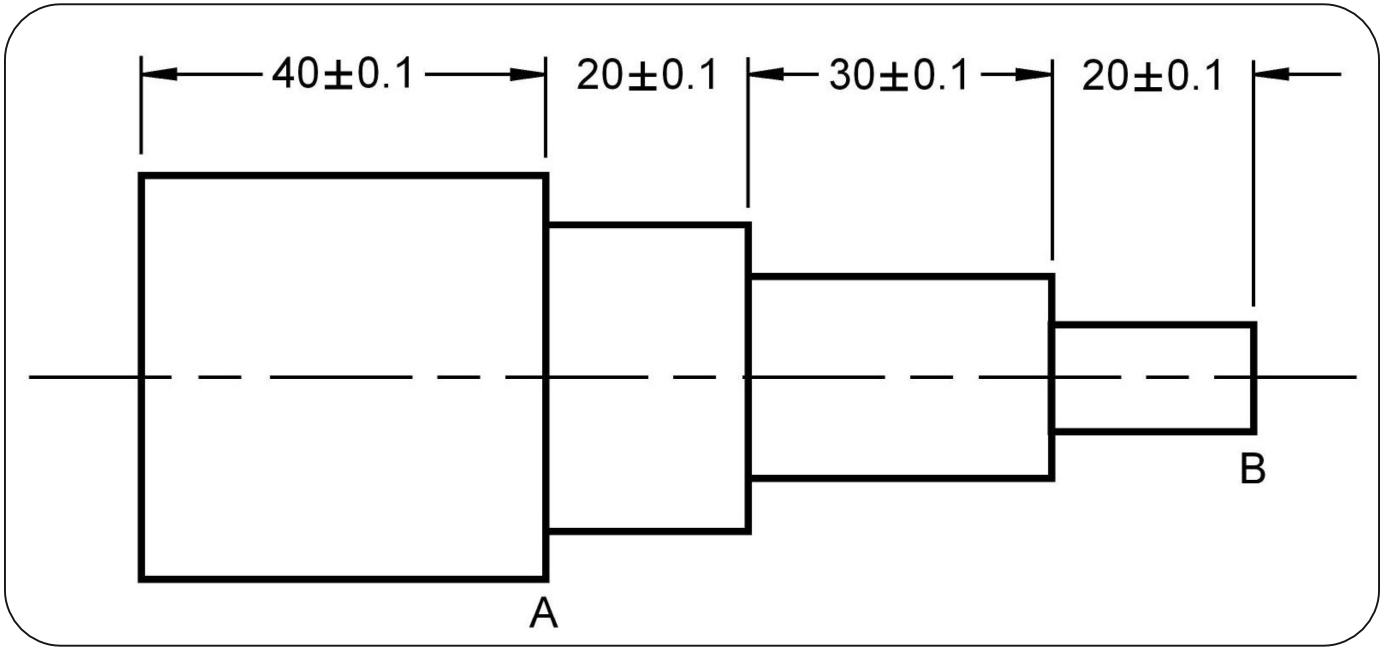
Tolerance Accumulation

- The **distance** could be controlled by a **single dimension** or **multiple dimensions**



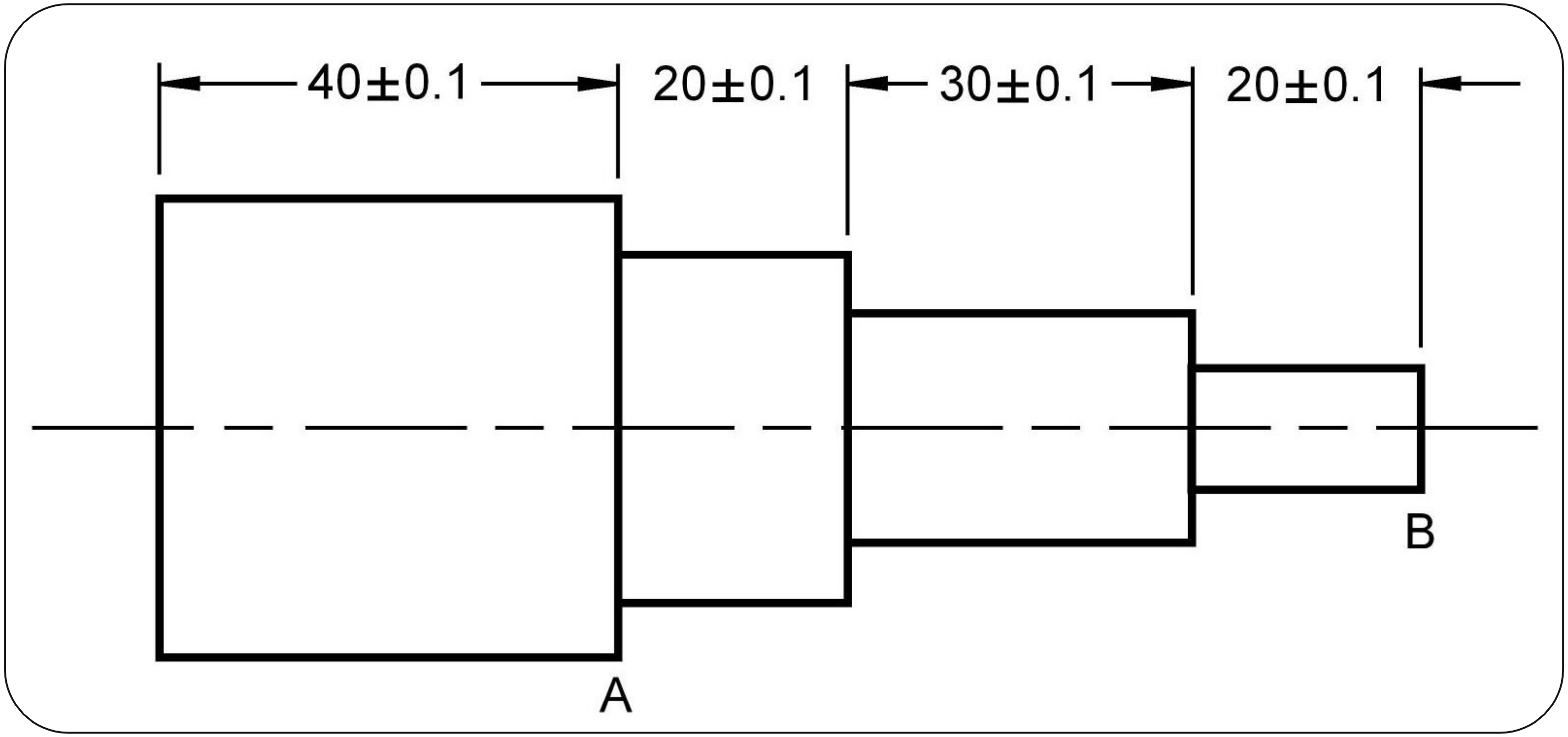
Tolerance Accumulation

- The **maximum variation** between two features is equal to the **sum of the tolerances placed on the controlling dimensions**



Tolerance Accumulation

- As the **number of controlling dimensions** increases, the **tolerance accumulation** increases



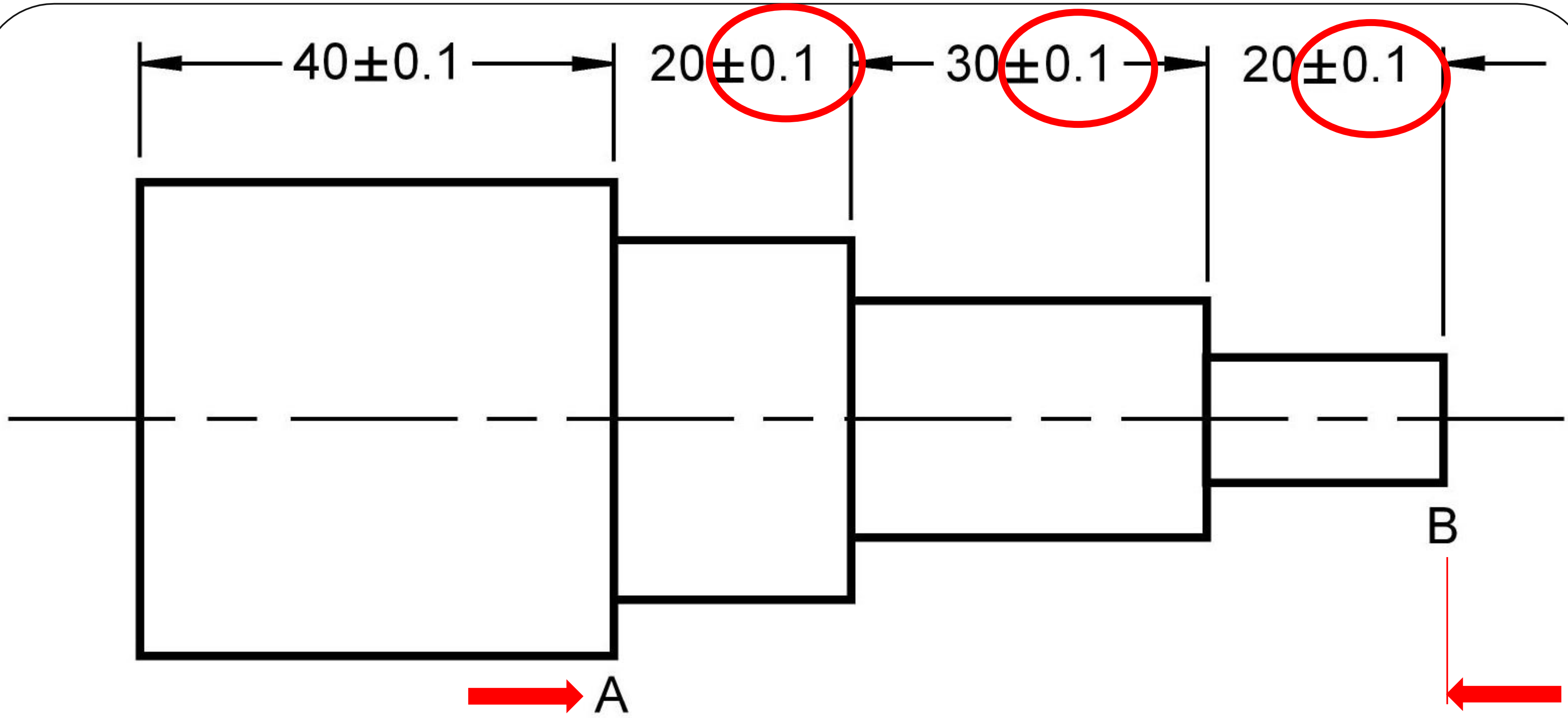
Tolerance Accumulation

- **Remember**: **even if the dimension does not have a stated tolerance, it has an implied tolerance**

- **Example:**

  What is the tolerance accumulation for the distance between surface A and B for the following three dimensioning methods?

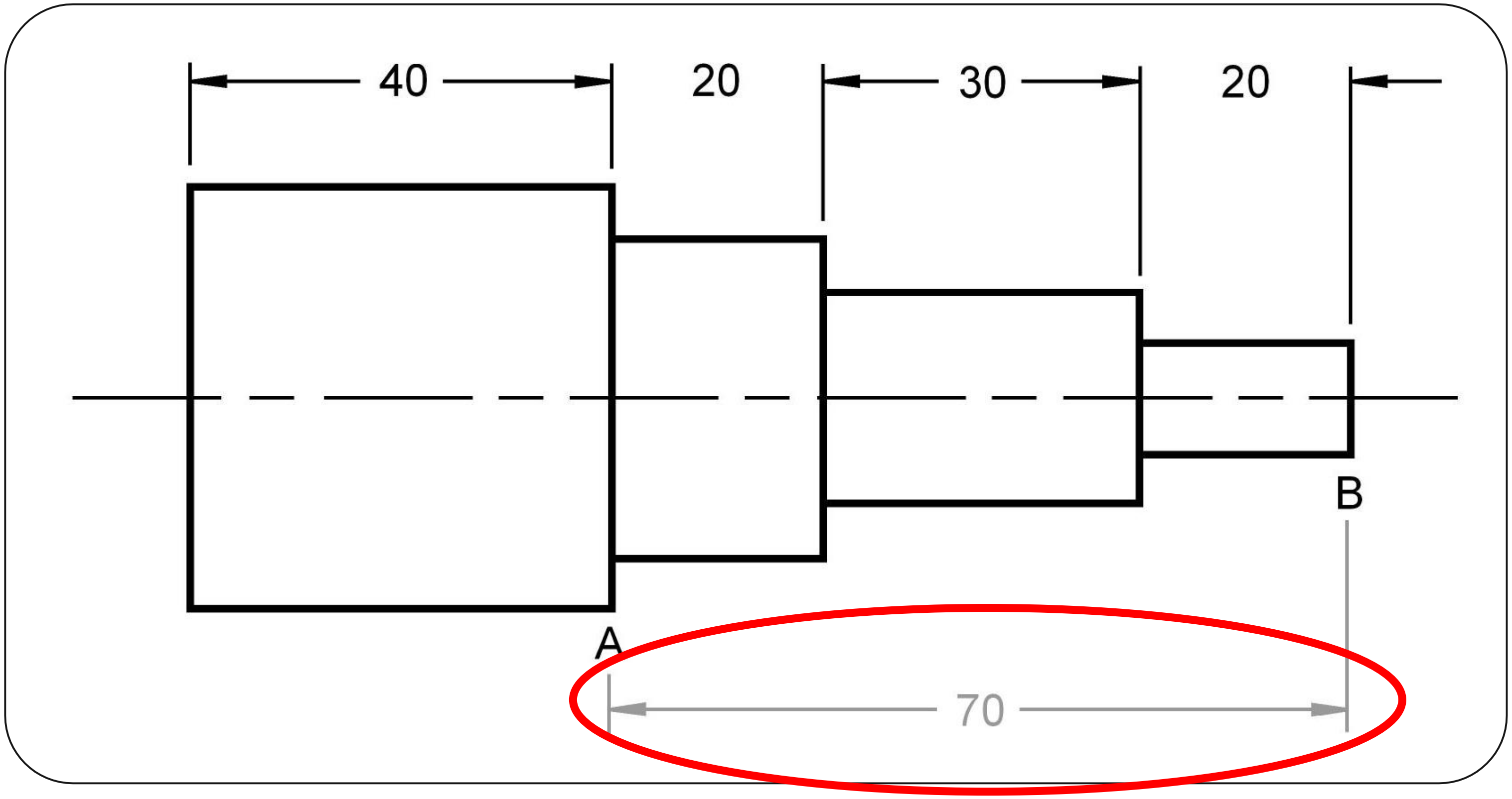
Tolerance Accumulation



Tolerance accumulation between surface A and B = 0.3

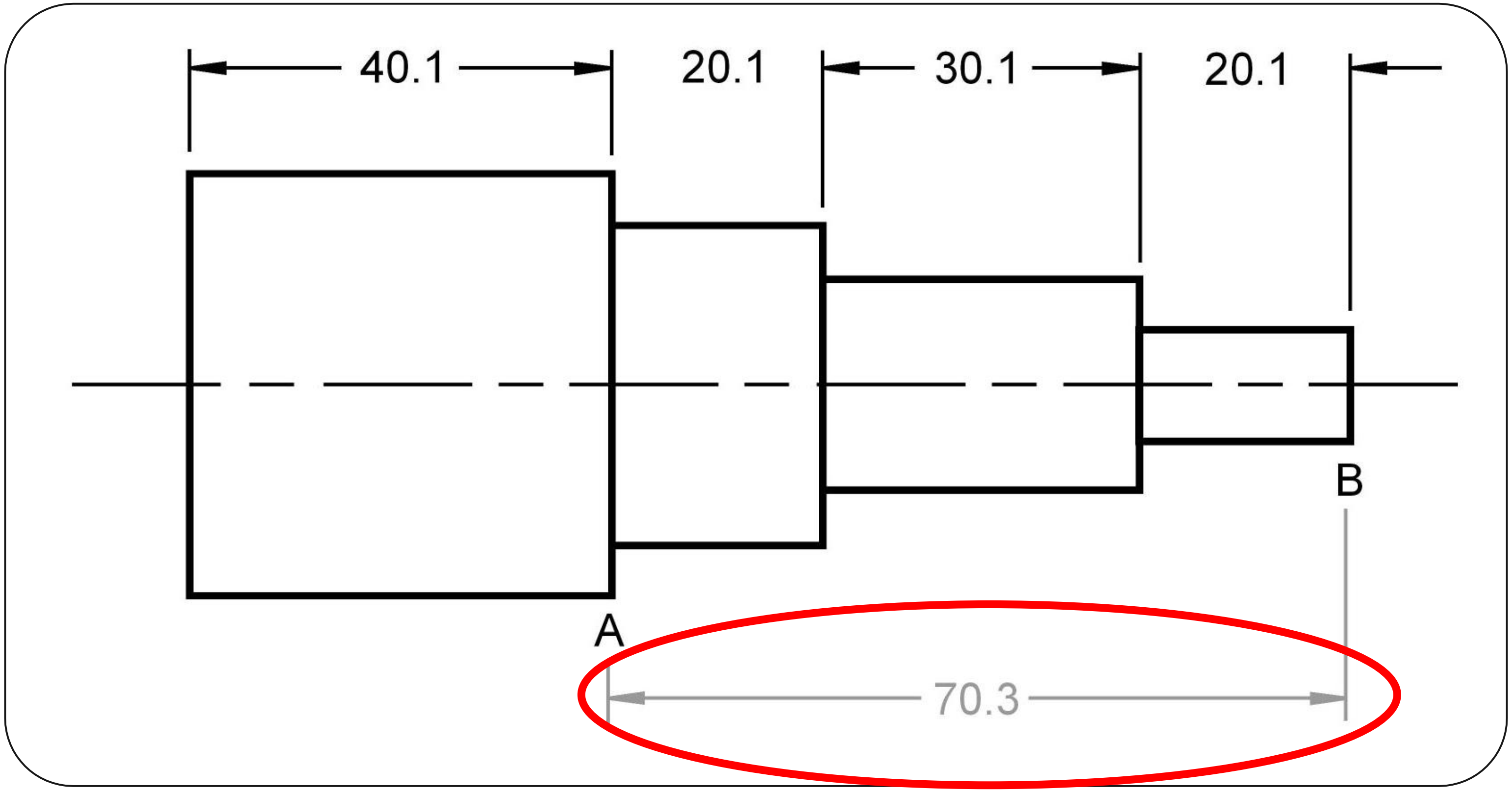
(K. Plantenberg , 2006,)

Tolerance Accumulation



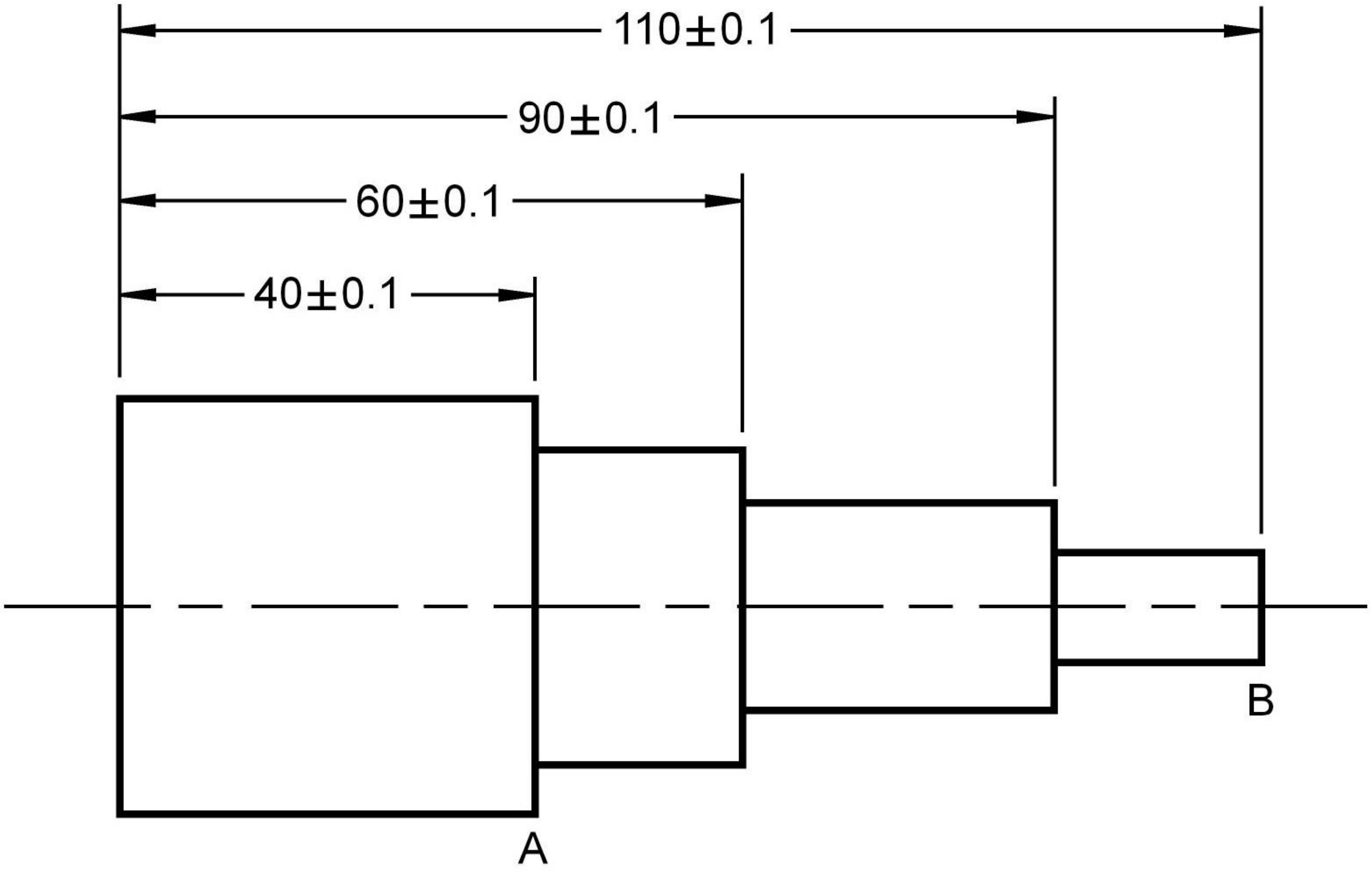
Perfect

Tolerance Accumulation



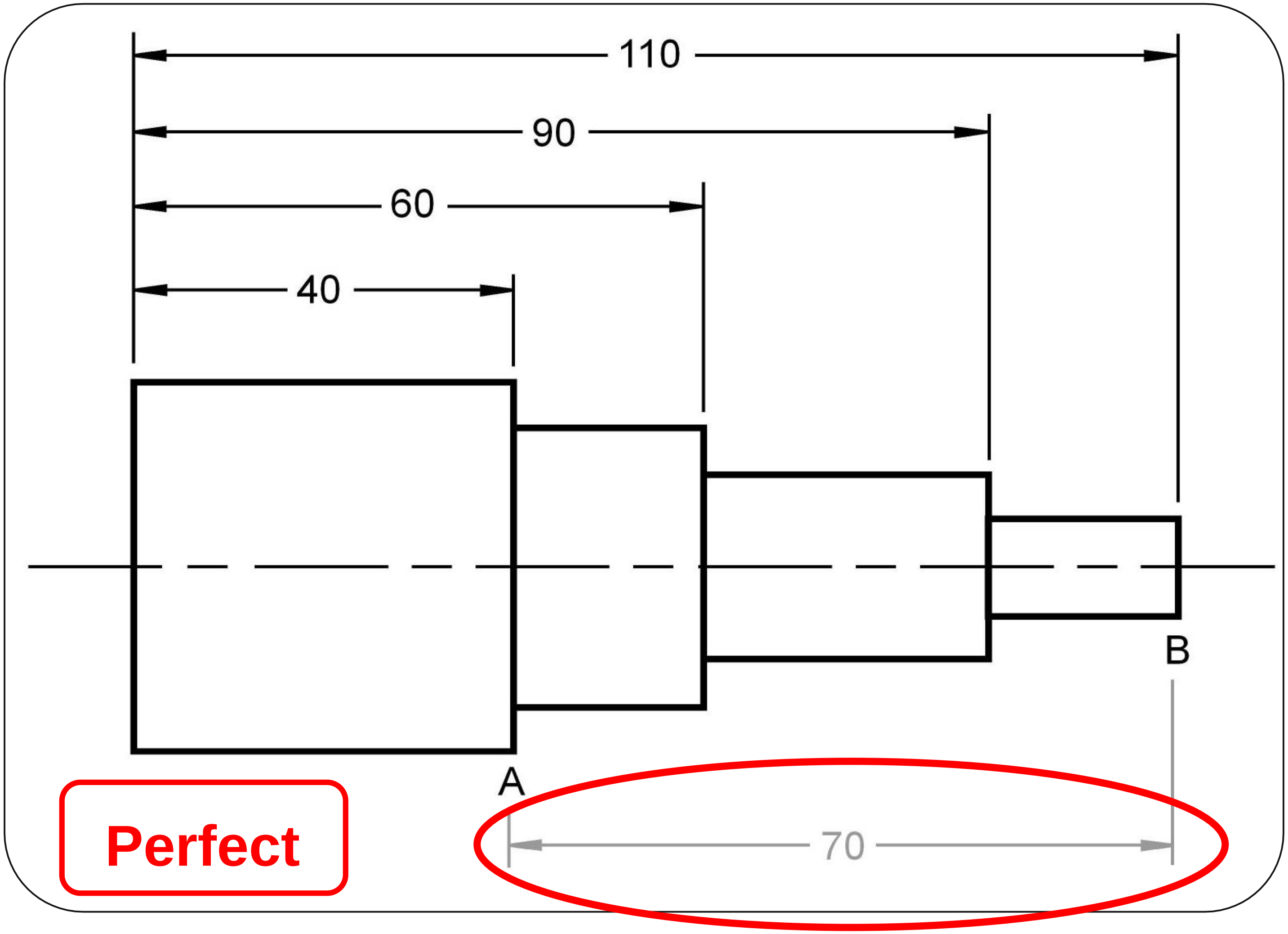
Worst Case

Tolerance Accumulation

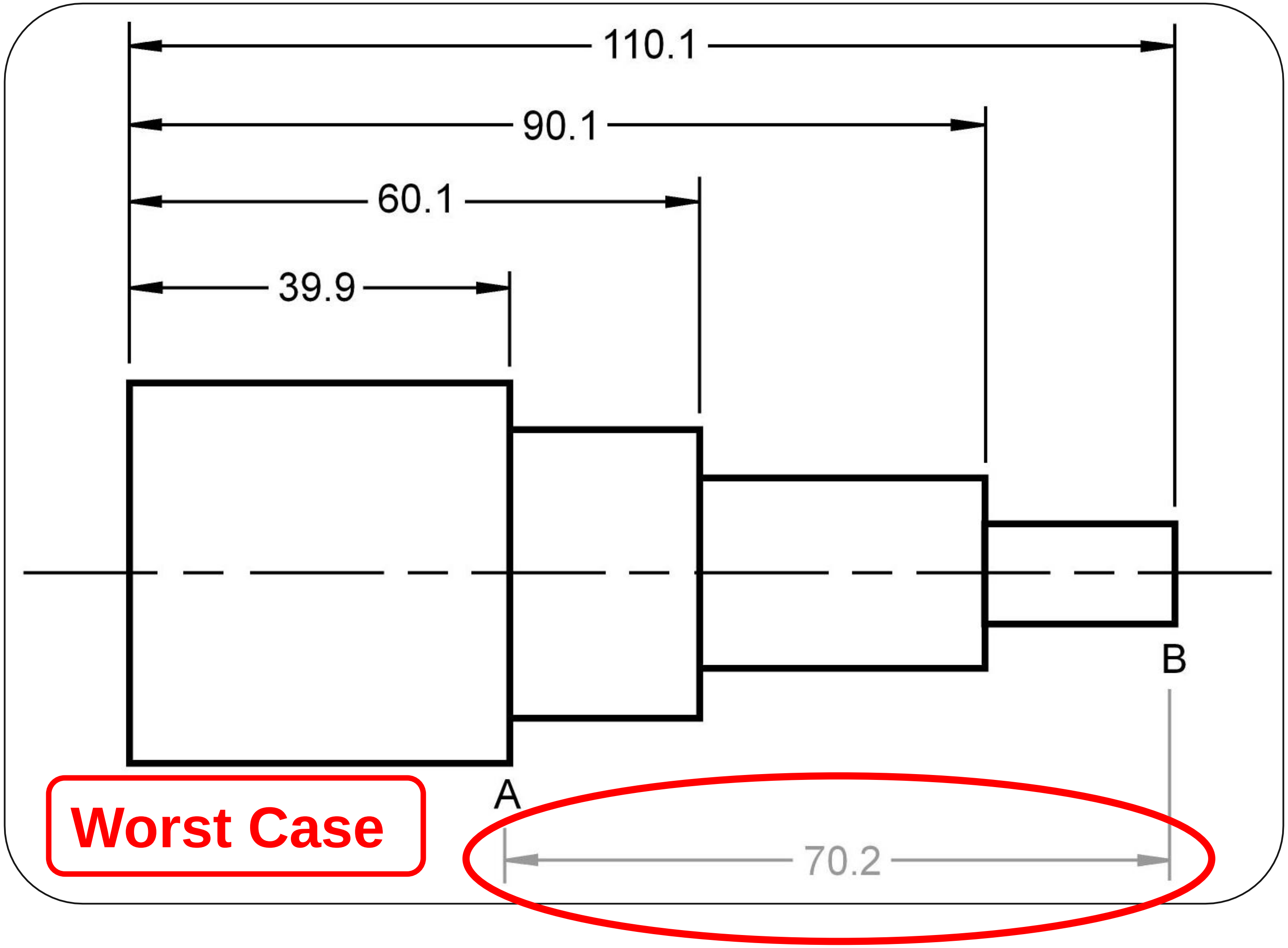


Tolerance accumulation between surface A and B = 0.2

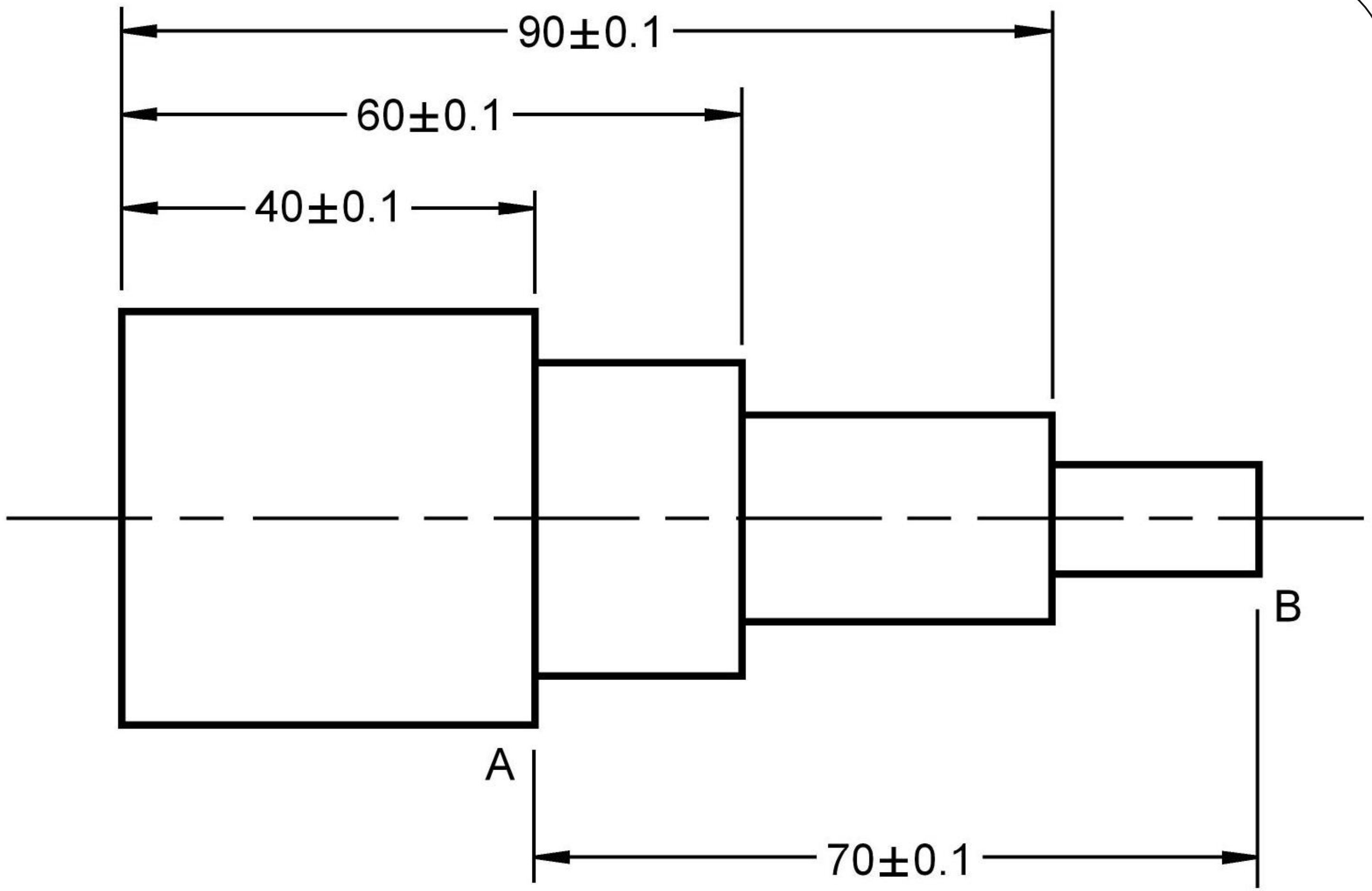
Tolerance Accumulation



Tolerance Accumulation



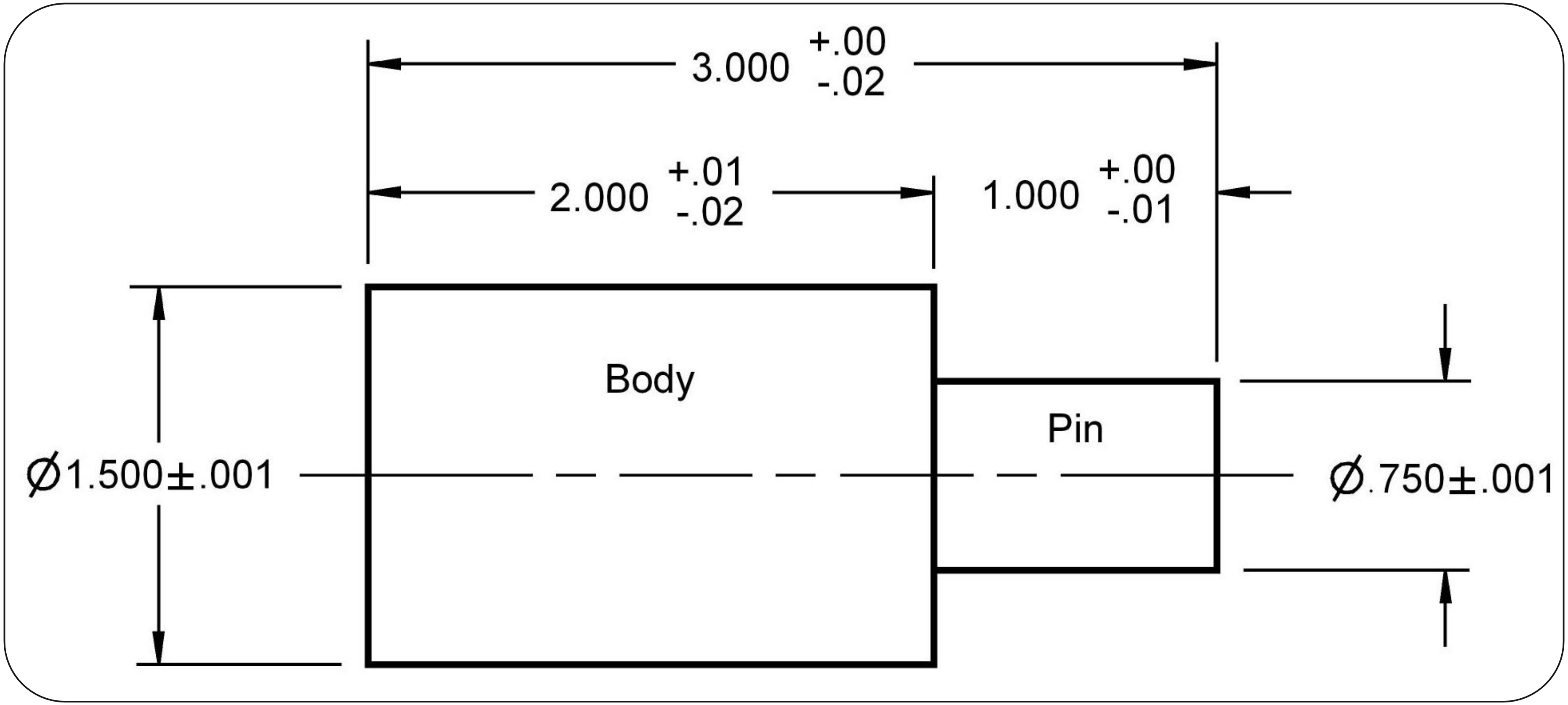
Tolerance Accumulation



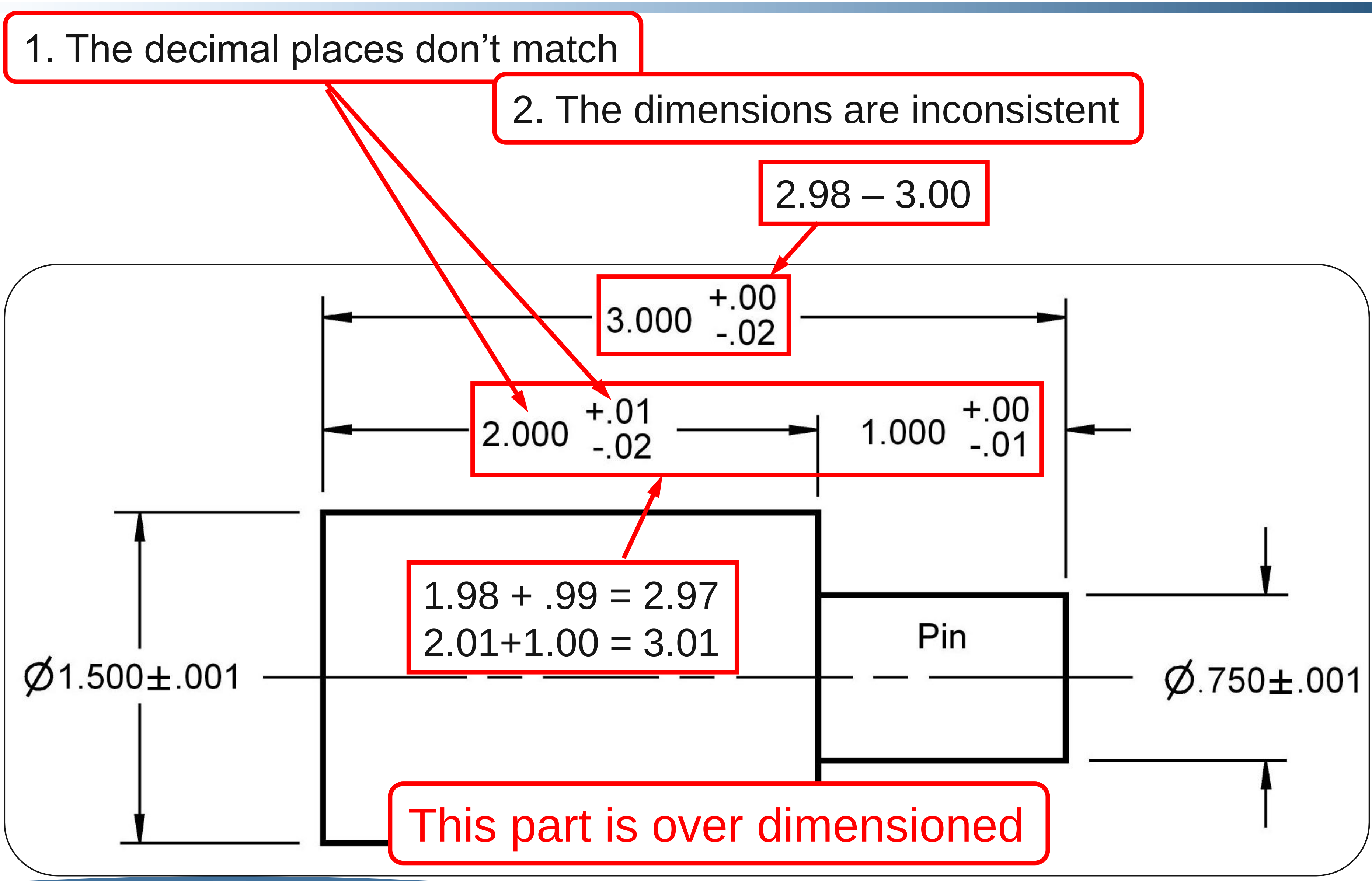
Tolerance accumulation between surface A and B = 0.1

Example 17

- Assuming that the diameter **dimensions are correct**, explain why this object is **dimensioned incorrectly**



Example 17


1. The decimal places don't match
2. The dimensions are inconsistent
2.98 – 3.00
3.000 +.00 -.02
2.000 +.01 -.02
1.000 +.00 -.01
1.98 + .99 = 2.97
2.01+1.00 = 3.01
Pin
Ø1.500±.001
Ø.750±.001
This part is over dimensioned

IX. Formatting Tolerances

Formatting Metric Tolerances

- Tolerances from standardized fit tables are listed on drawings as:

$\varnothing\,^{20.240}_{20.110}$ ($\varnothing$20 C11)

$\varnothing$20 C11 $\left(\varnothing\,^{20.240}_{20.110}\right)$

$\varnothing$20 C11

The person reading the print must have access to the standard fit tables

(K. Plantenberg , 2006,)

Formatting Metric Tolerances

- **Unilateral tolerances**

$40^{\;0}_{-0.02}$ or $40^{+0.02}_{\;0}$

➢ A single zero without a plus or minus sign

- **Bilateral tolerances**

$10^{+0.25}_{-0.10}$ not ~~$10^{+0.25}_{-0.1}$~~

➢ Both the plus and minus values have the same number of decimal places

Formatting Metric Tolerances

- **Limit dimensions**

15.45 15.00	not	~~15.45 15~~

➢ Both values should have the same number of decimal places

- **Using Basic dimensions with the tolerance**

45 ± 0.15	not	~~45.00 ± 0.15~~

➢ The number of decimal places in the basic dimension does not have to match the number of decimal places in the tolerance

Formatting Inch Tolerances

- **Unilateral and Bilateral tolerances**

$.500^{+.000}_{-.002}$	not	$.500^{0}_{-.002}$
$.500^{+.001}_{-.002}$	not	$.50^{+.001}_{-.002}$

➢ The basic dimension and the plus and minus values should have the same number of decimal places

Formatting Inch Tolerances

- **Limit dimensions**

.250 .252	not	~~.25 .252~~

  - ➢ Both values should have the same number of decimal places

- **Using Basic dimensions with the tolerance**

2.000±0.015	not	~~2.0±0.015~~

  - ➢ The number of decimal places in the basic dimension should match the number of decimal places in the tolerance

Formatting Angular Tolerances

- **Angular tolerances**

$30.0^{o} \pm .2^{o}$ not ~~$30^{o} \pm .2^{o}$~~

➢ Both the angle and the plus and minus values have the same number of decimal places

(K. Plantenberg , 2006,)

Summary 1/2

- **Tolerance** is the difference between two limits
- If a feature’s size is toleranced , it is allowed to **vary within a range of values or limits**
- **Tolerancing** enables an engineer to **design interchangeable** or **replacement parts**
- **Allowance** is the **difference between the largest shaft diameter** and the **smallest hole diameter**
- The two most common standards agencies are; **A**merican **N**ational **S**tandards **I**nstitute (**ANSI**) / (**ASME**) and **I**nternational **S**tandards **O**rganization (**ISO**)

Summary 2/2

- **The International Tolerance Grade number** (**IT#**) is **set of tolerances** that vary according to the basic size and **provide the same relative level of accuracy** within a given grade 00

References

1. Dimensioning and Tolerancing, Engineering Drawing and Related Documentation Practices ,ASME Y14.5-2009

2. Engineering Graphics Essentials With AutoCAD 2007, K. Plantenberg

3. Manufacturing, Engineering & Technology, Fifth Edition, by S. Kalpakjian and S. R. Schmid

4. Tolerancing Topics - Exercises presentation ,2006, K.Plantenberg

5. Φυλλάδιο εργαστηριακών ασκήσεων Μηχανουργείου,2012.Δημητρακόπουλος Γ., Μούρτζης Δ., Πανδρεμένος Ι., Παπακώστας Ν., Σταυρόπουλος Π., Φυσικόπουλος Α.