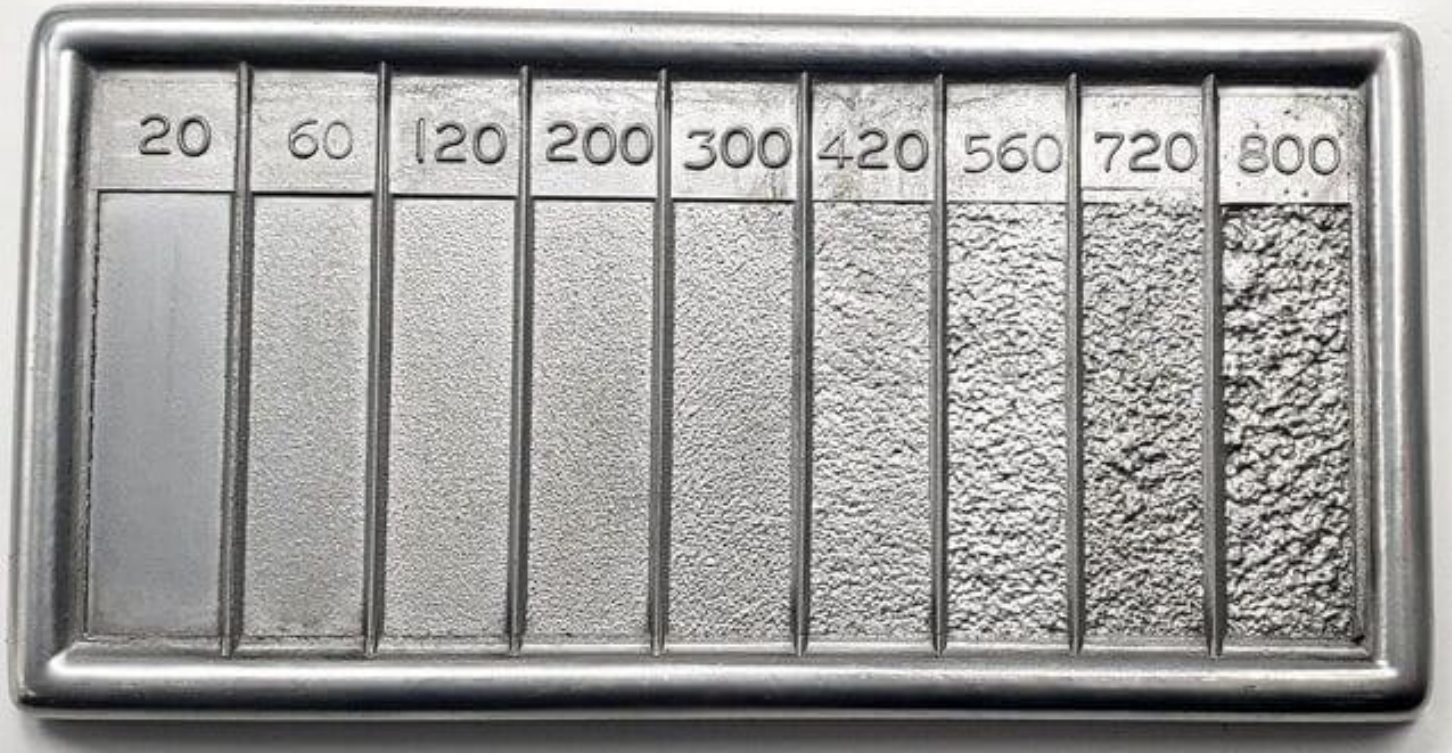
Computer-Aided Design

Machining Symbols and Tolerances (Geometric, Limits and Fits)

- Machining Symbols
- Tolerances: Geometric, Limits and Fits



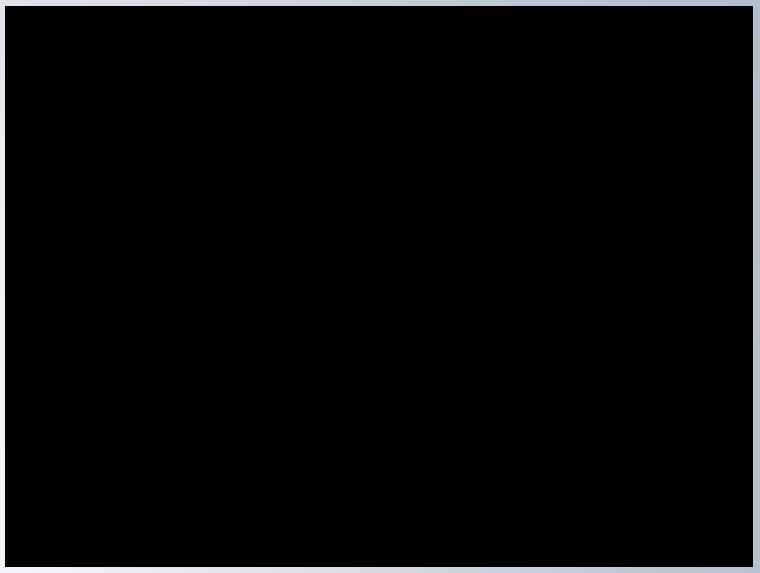
Machining Symbols

SURFACE ROUGHNESS

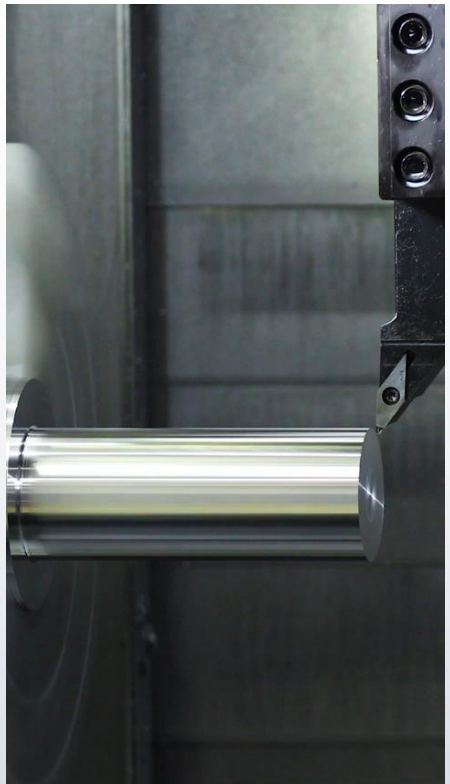
SHOP TIPS #293 Surface Roughness Finish 1 of 2 tubalcain (youtube.com)

- Roughness remains on the surface obtained by machining or non-chip manufacturing.
- It is measured by hand or precise control devices.

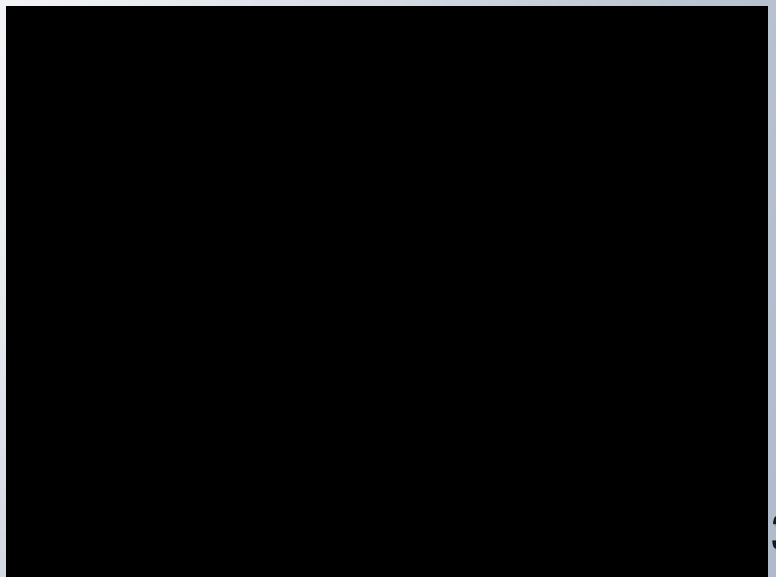
by hand



Machining



by profilometer



ROUGHNESS PARAMETERS

R_a: Arithmetic mean roughness height (micrometer, microinch)

R_z: Average distance between 5 highest protrusions and 5 deepest indentations of sample length

R_t: Sum of maximum height and maximum depth for the entire measuring length

average of all the symbols

general

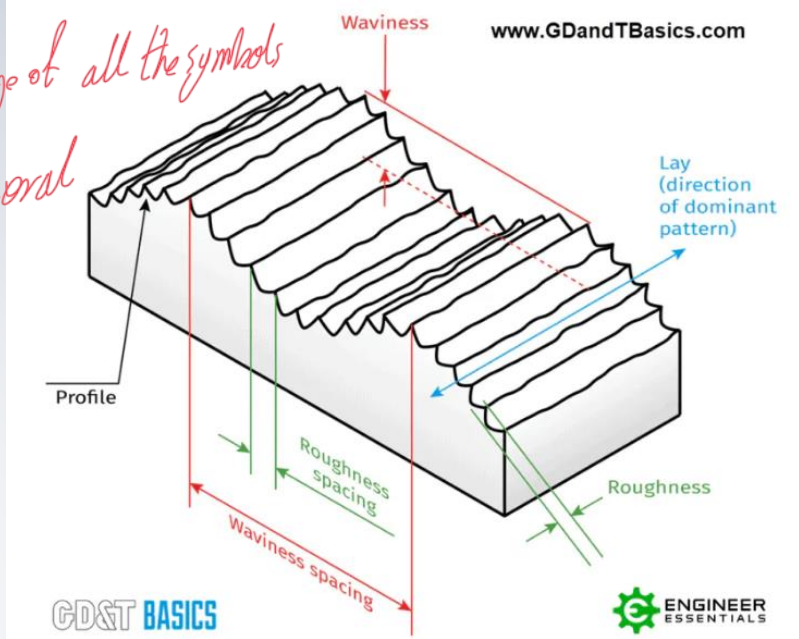
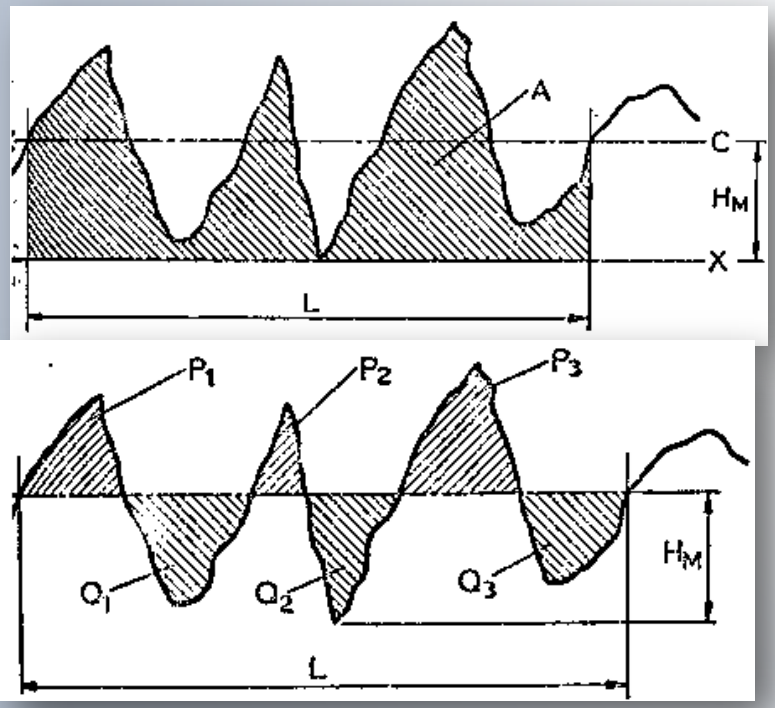


Figure 1 – Lay, Waviness, and Roughness

The Basics of Surface Finish | GD&T Basics (gdandtbasics.com)

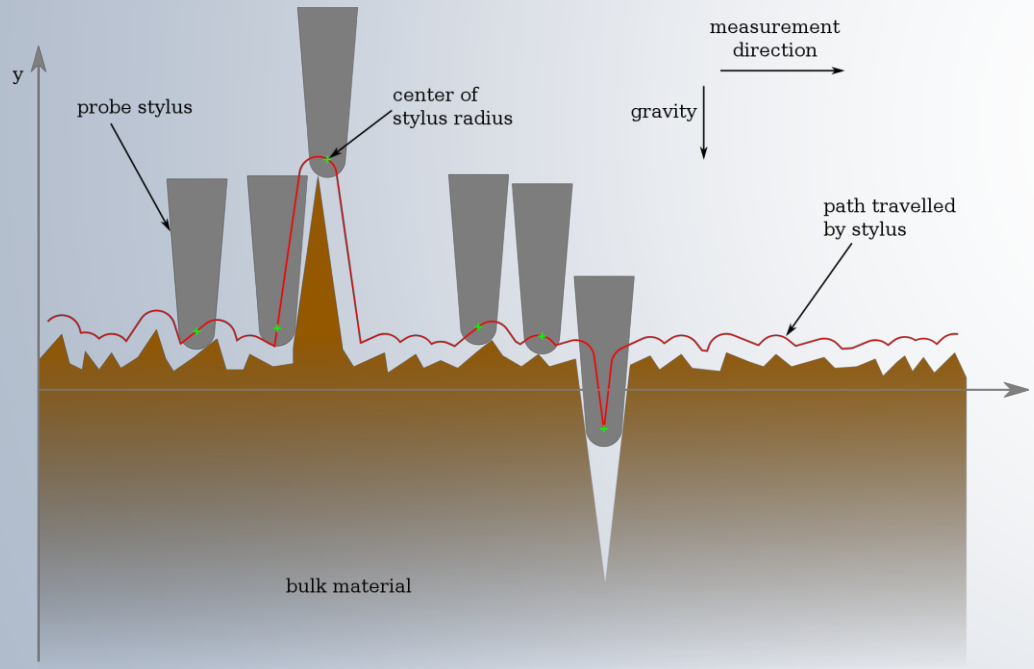


know R_a R_z R_t

$$R_z(\mu m) = \left(\frac{P_{area} + Q_{area}}{L}\right) \cdot \frac{1000}{V_q}$$

PROFILE HEIGHT PARAMETERS

How profilometer works?



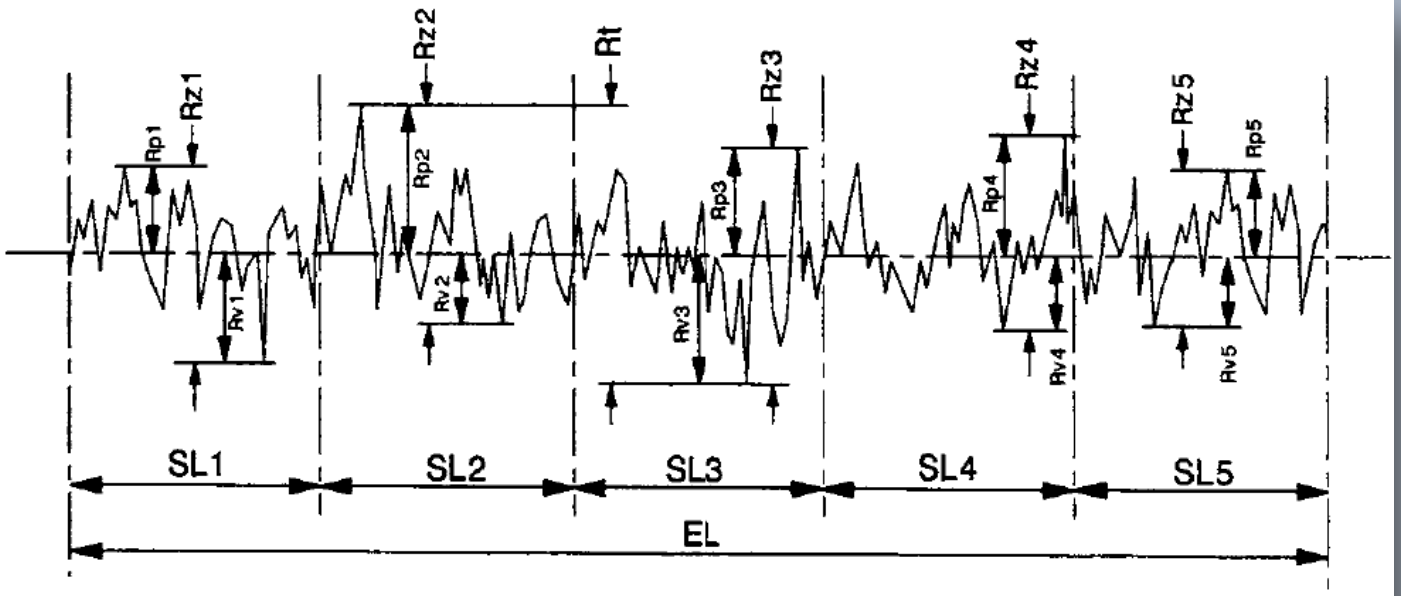
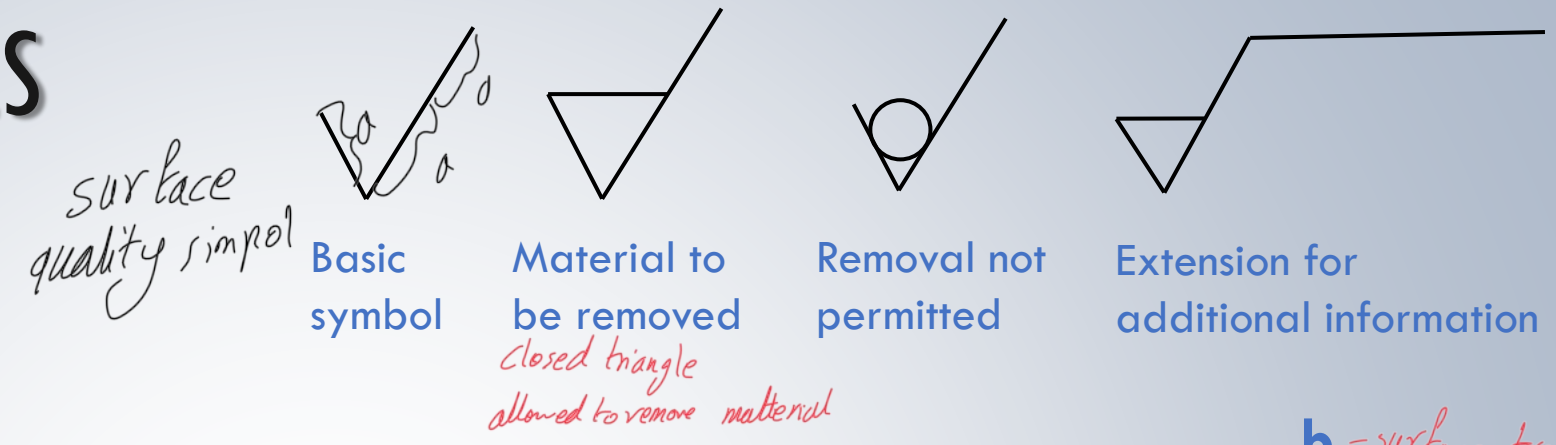


Figure 6.9 *A schematic profile and the parameters Rt, Rz, Rv, Rp*

Parameter		Description
Ra	Centre Line Average	$Ra = \frac{1}{n}\sum_{i=1}^{n}\lvert y_i\rvert = \frac{1}{L_r}\int_0^{L_r}\lvert y\rvert\,dx$
Rq	RMS Average	$Rq = \sqrt{\frac{1}{n}\sum_{i=1}^{n} y_i^2} = \sqrt{\frac{1}{L_r}\int_0^{L_r} y^2\,dx}$
Rt	EL peak to valley height	Peak to valley height within the EL
Rz	SL peak to valley height	Peak to valley height within a SL
Rp	Peak height	Highest peak to mean line height
Rv	Valley depth	Lowest valley to mean line depth

MACHINING SYMBOLS



a: R_a average roughness value number value in micrometers or roughness class number ($N1$ ... $N12$)

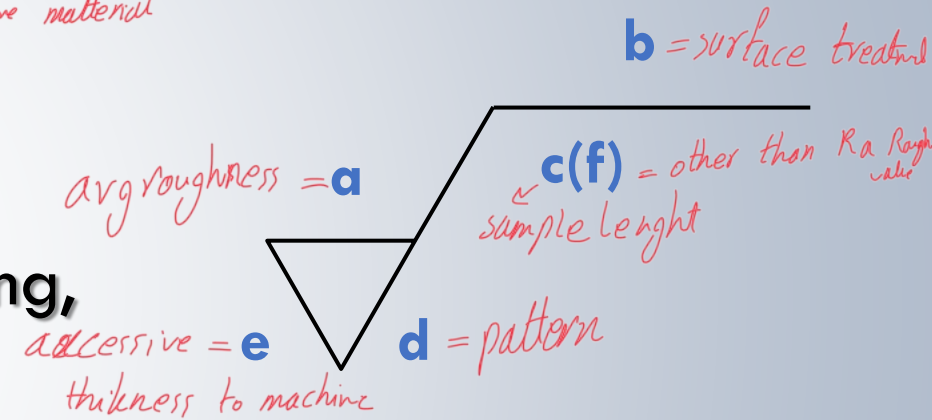
b: Surface treatment (cleaning, heat treatment, coating, painting etc.)

c: Sample length

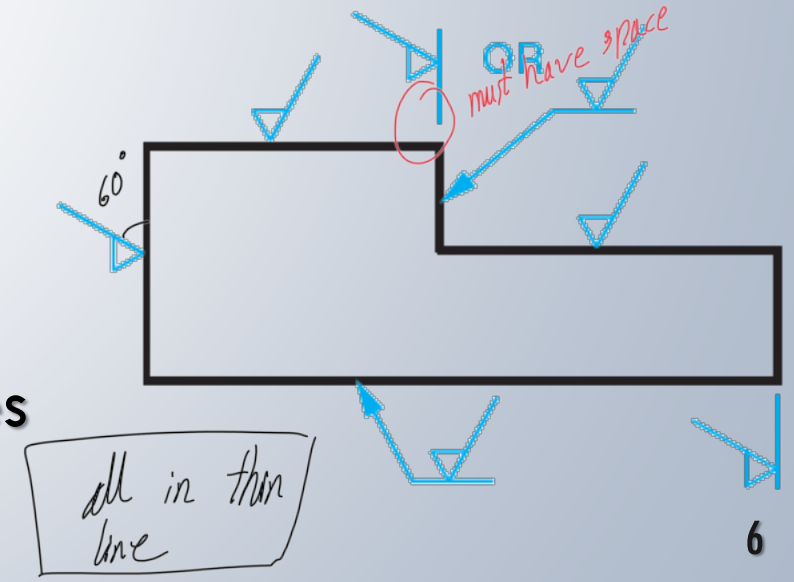
d: Lay pattern

e: Excessive thickness to be machined

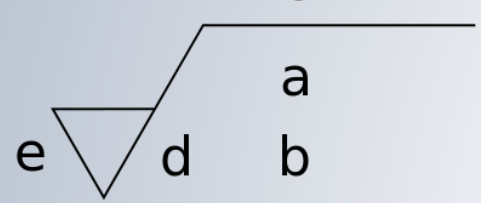
(f): Roughness value other than R_a written in parentheses



general micrometer



MACHINING SYMBOLS



b	Secondary surface parameter
c	Manufacturing method
e	Minimum material removal

Material removal not allowed

Material removal required

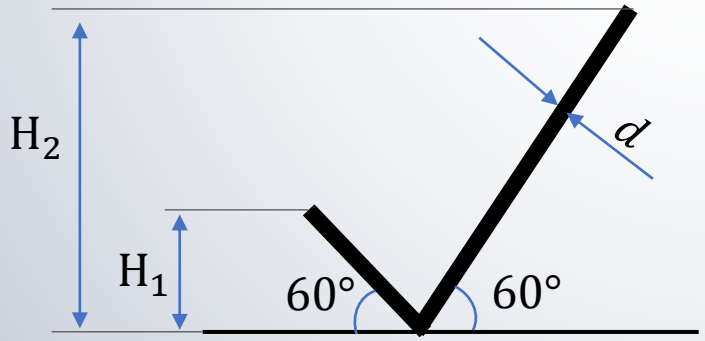
d	Lay
=	Parallel
⊥	Perpendicular
X	Cross-hatch
M	Multi-directional
C	Circular
R	Radial
P	Particulate

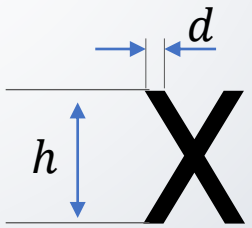
a	Surface parameter
	D F S-L / Rz N C V
D	Tolerance direction, upper (U) or lower (L)
F	Filter type, for example "2RC"
S	Short filter cutoff, for removing noise
L	Long filter cutoff, for removing waviness
R	Profile type, primary (P), waviness (W), or roughness (R)
z	Parameter type, for example "a" for Ra or "3z" for R3z
N	Assesment length; multiple of sampling length, usually 5
C	Comparison rule, "max" for 100%, "16%" for 116%
V	Specified value in micrometers

https://en.wikipedia.org/wiki/Surface_finish

MACHINING SYMBOL DIMENSIONS

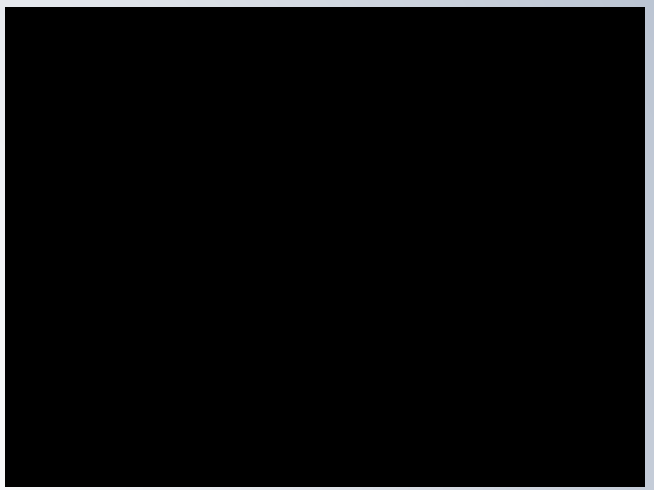
Line thickness	d	0.25	0.35	0.5	0.7	1.0	1.4	2.0
Number and text height	h	2.5	3.5	5	7	10	14	20
Machining symbol heights	$H_1 = h \cdot \sqrt{2}$	3.5	5	7	10	14	20	28
	$H_2 = h \cdot 2\sqrt{2}$	7	10	14	20	28	40	56

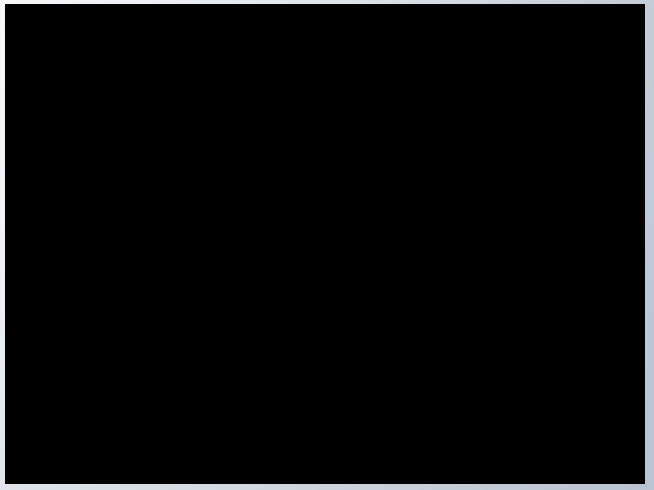




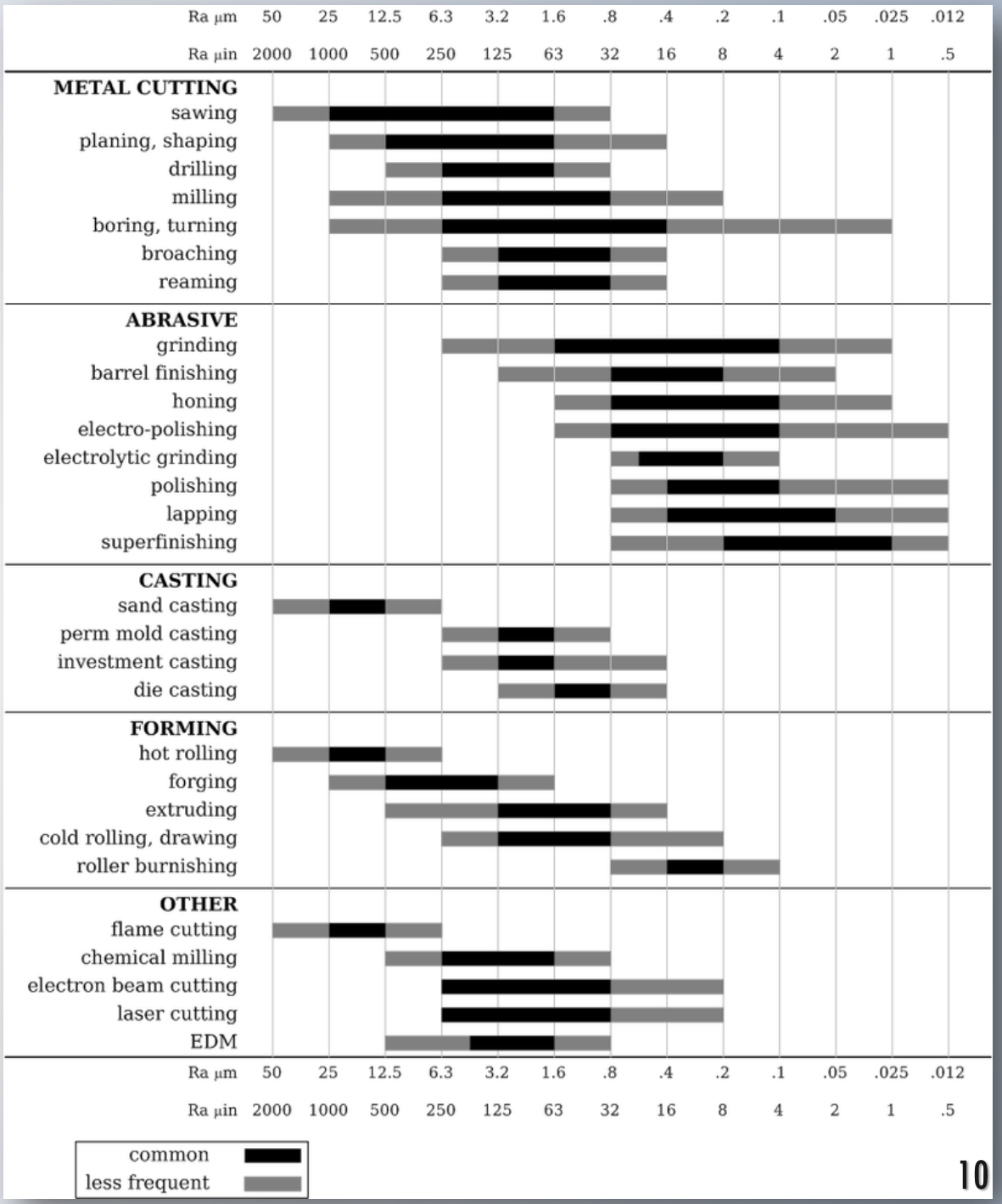
SURFACE QUALITY~ AVERAGE ROUGHNESS (R_a)

Surface Quality	Average Roughness (R_a) (µm)
N1	0.025
N2	0.05
N3	0.1
N4	0.2
N5	0.4
N6	0.8
N7	1.6
N8	3.2
N9	6.3
N10	12.5
N11	25
N12	50



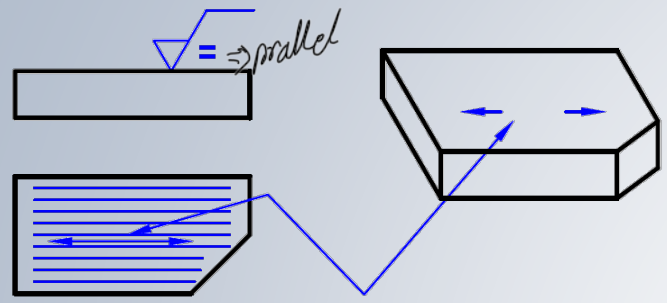


POSSIBLE SURFACE ROUGHNESS ACCORDING TO MANUFACTURING METHODS

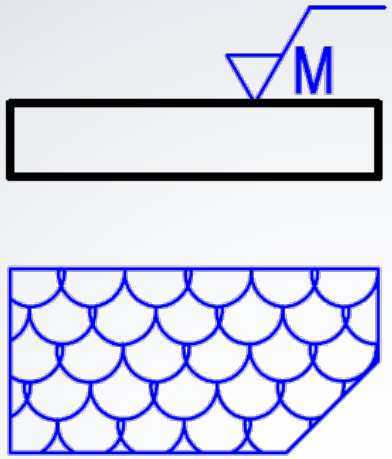


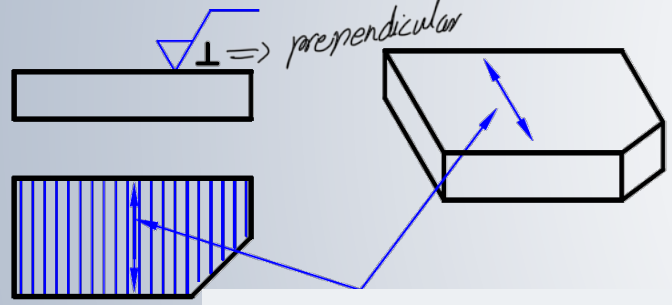
LAY PATTERNS

tells surface texture

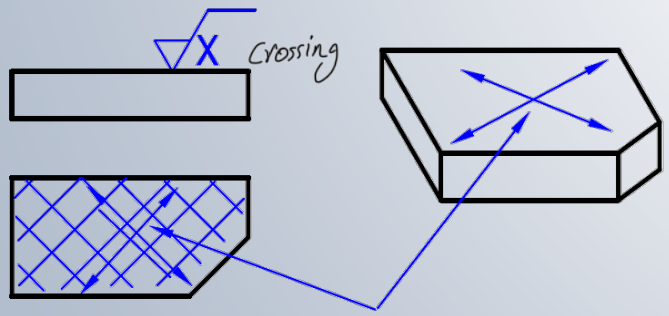


Direction of processing marks

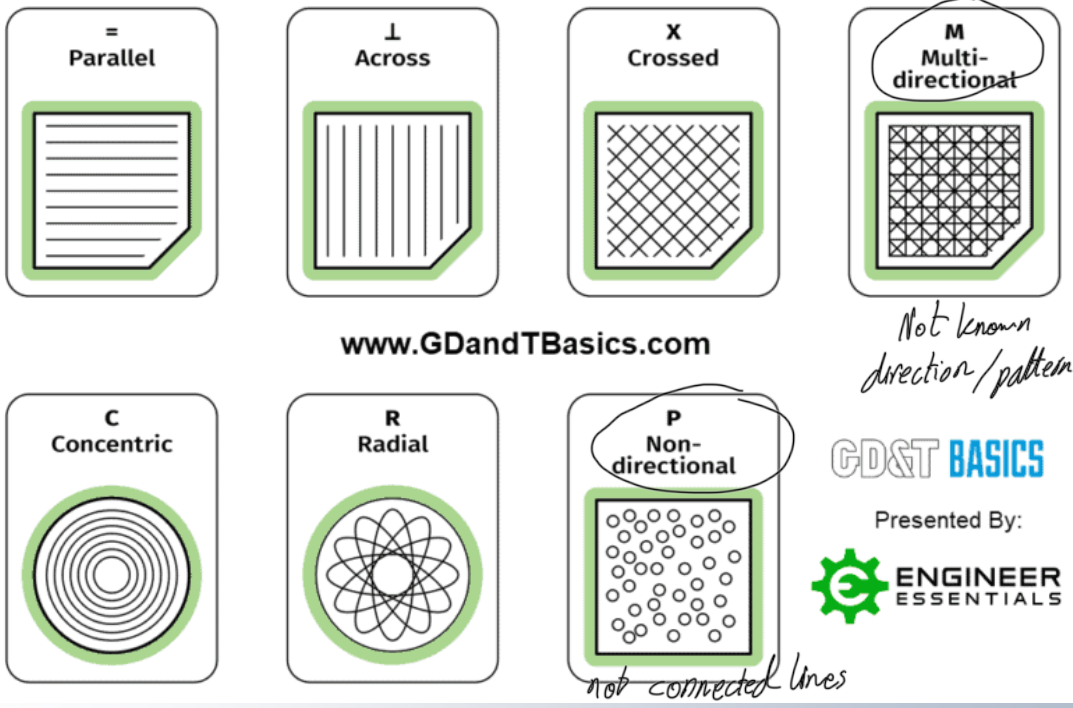


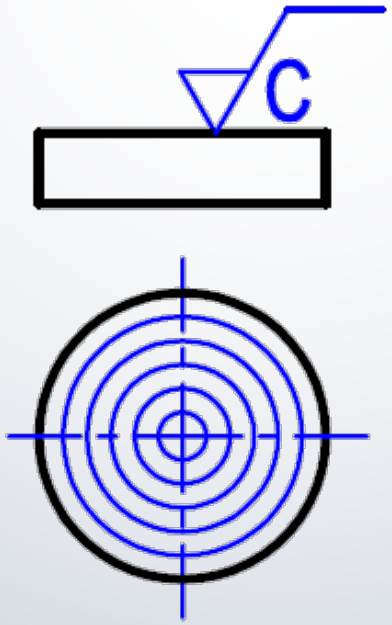


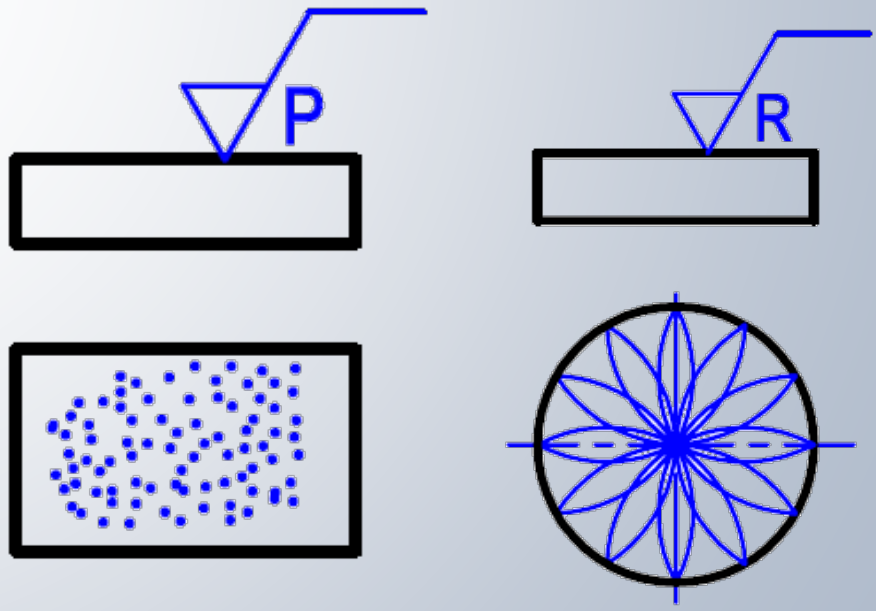
Direction of processing marks



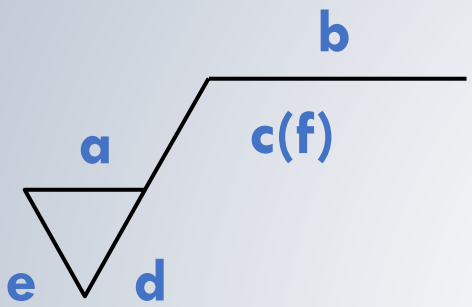
Direction of processing marks

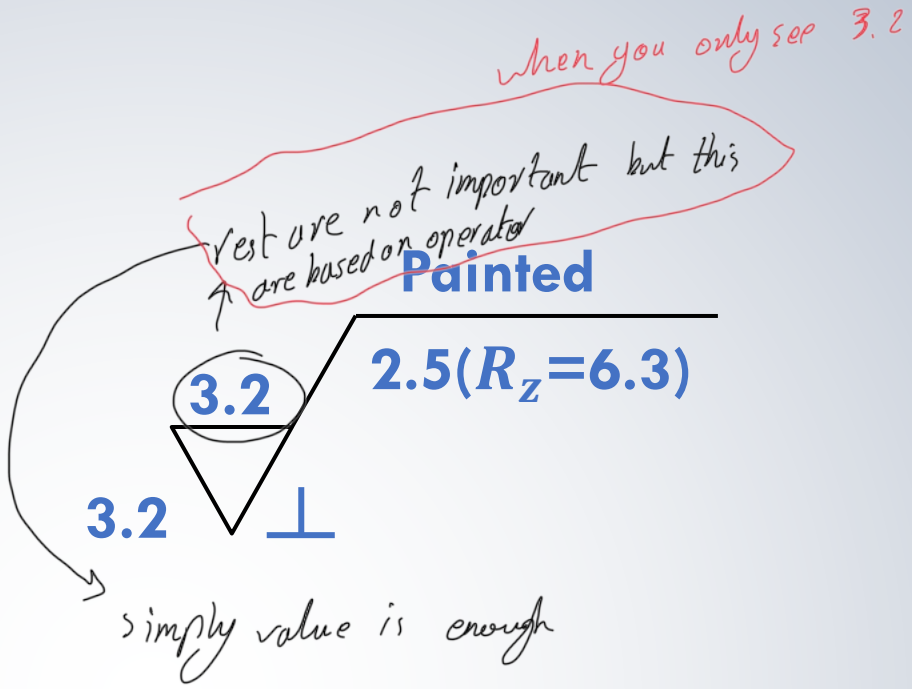




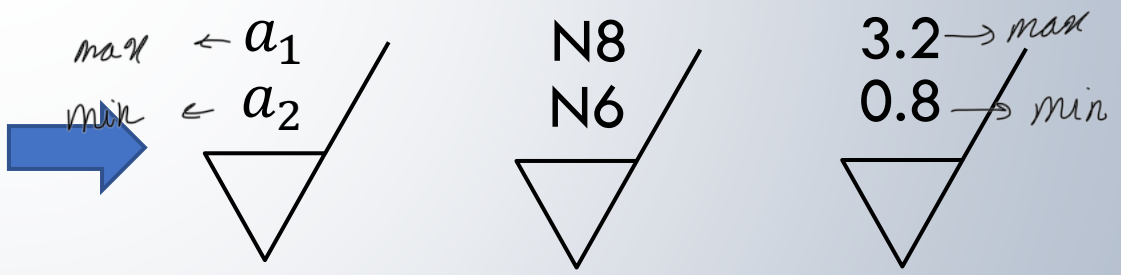


EXAMPLE

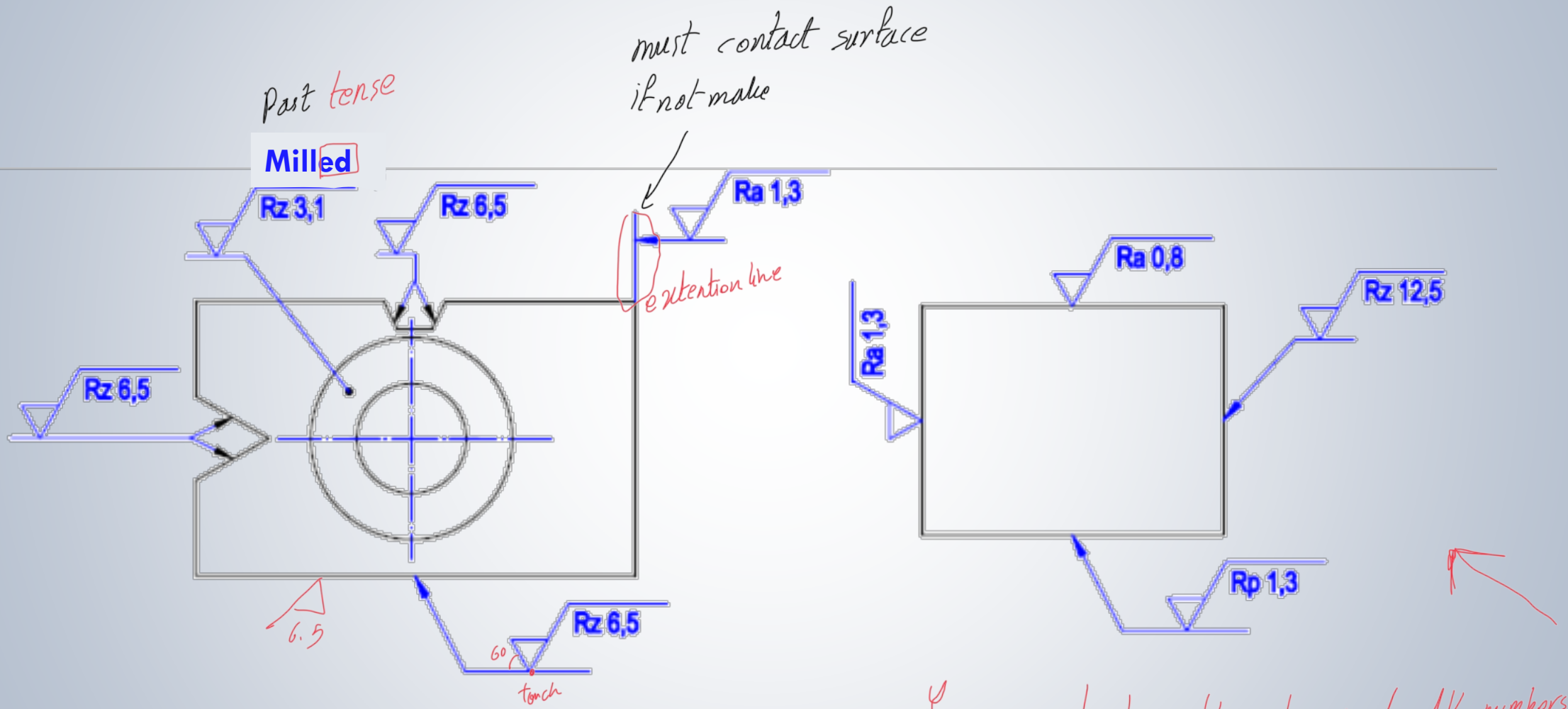




If upper and lower roughness limits are to be given



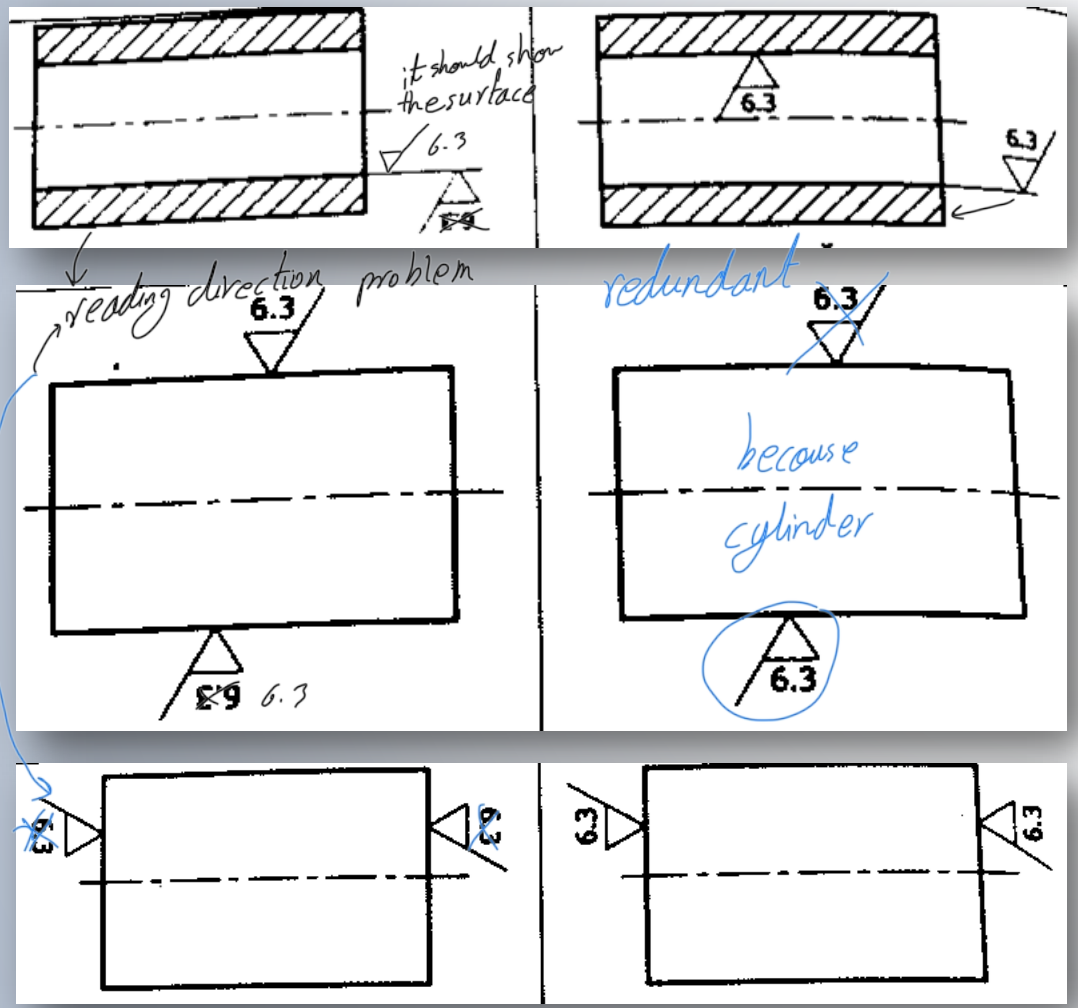
READING DIRECTION



USAGE OF SYMBOLS

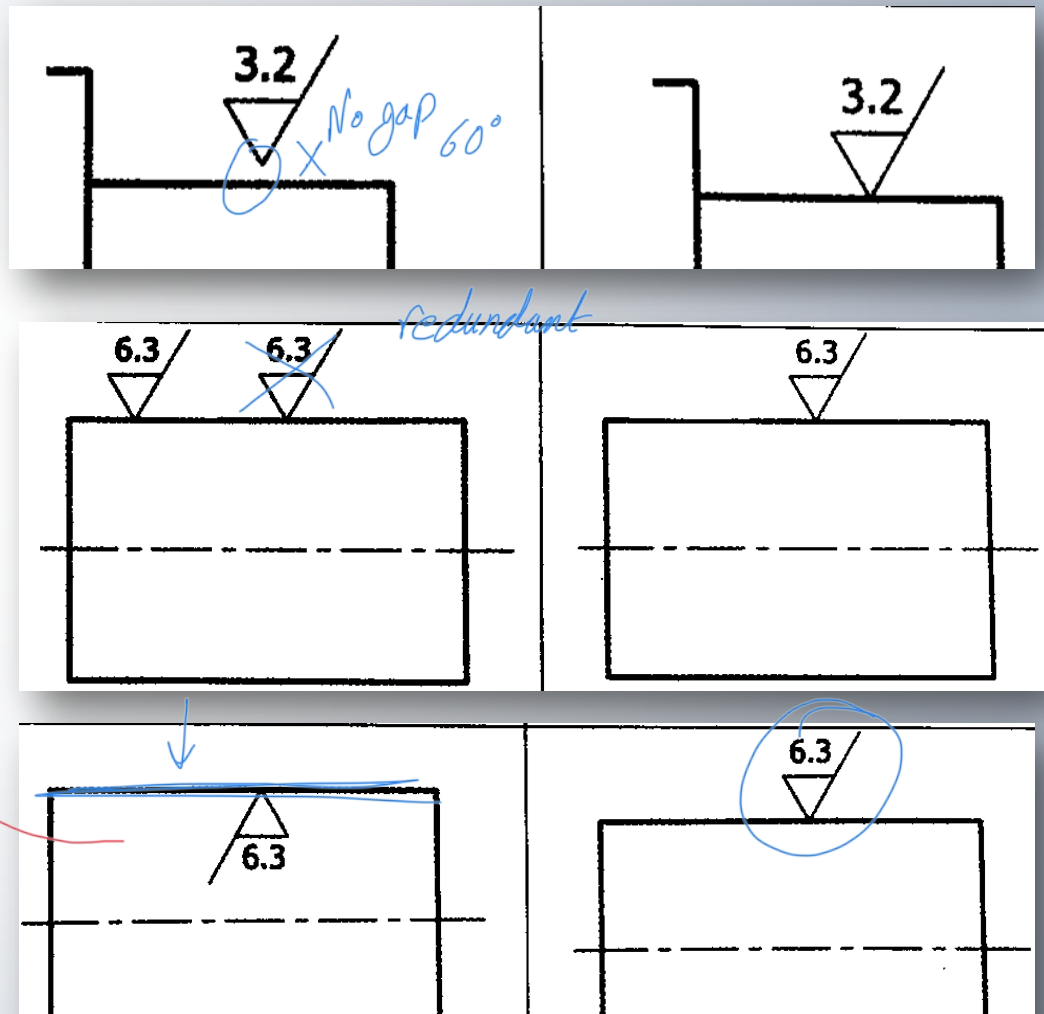
must point towards the surface

WRONG



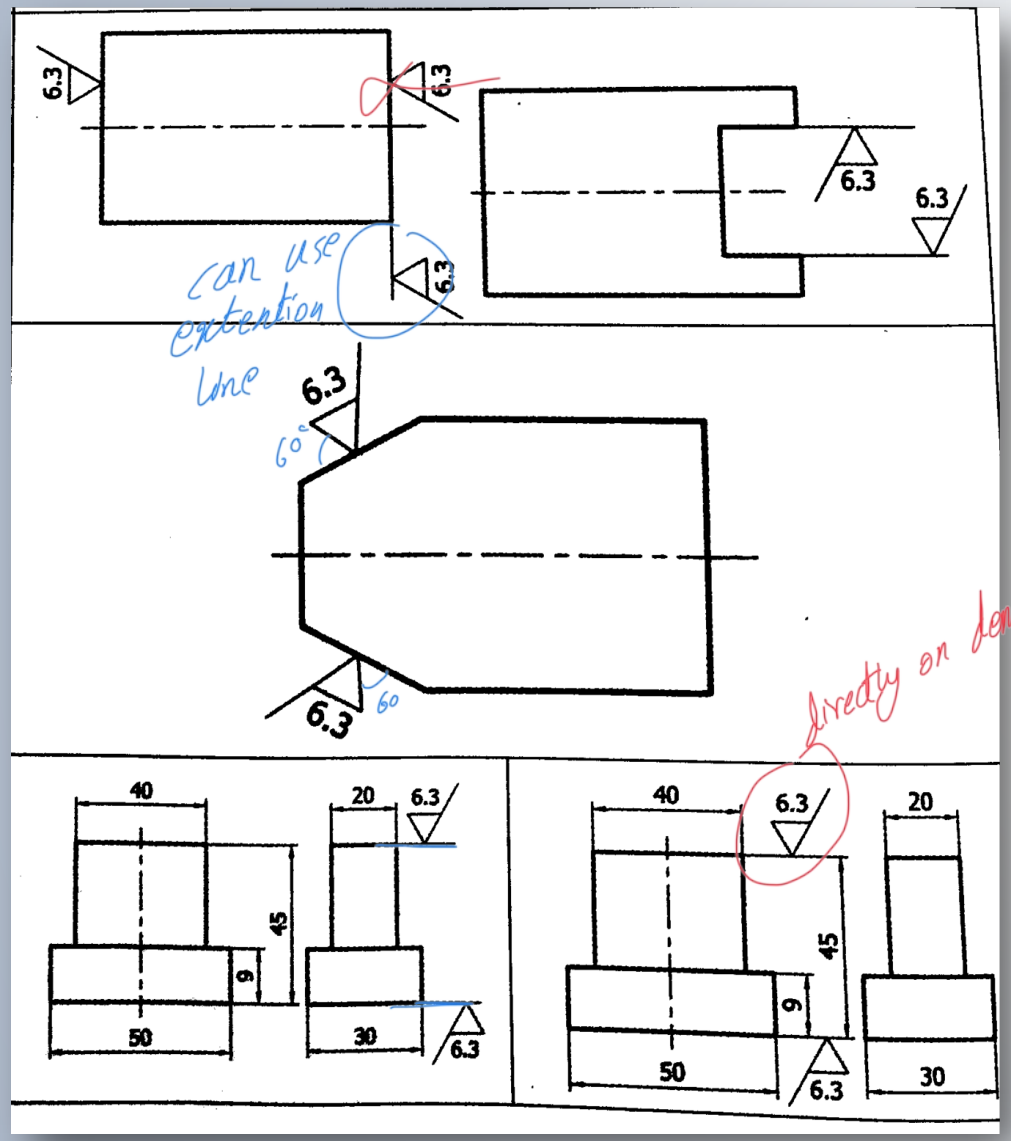
CORRECT

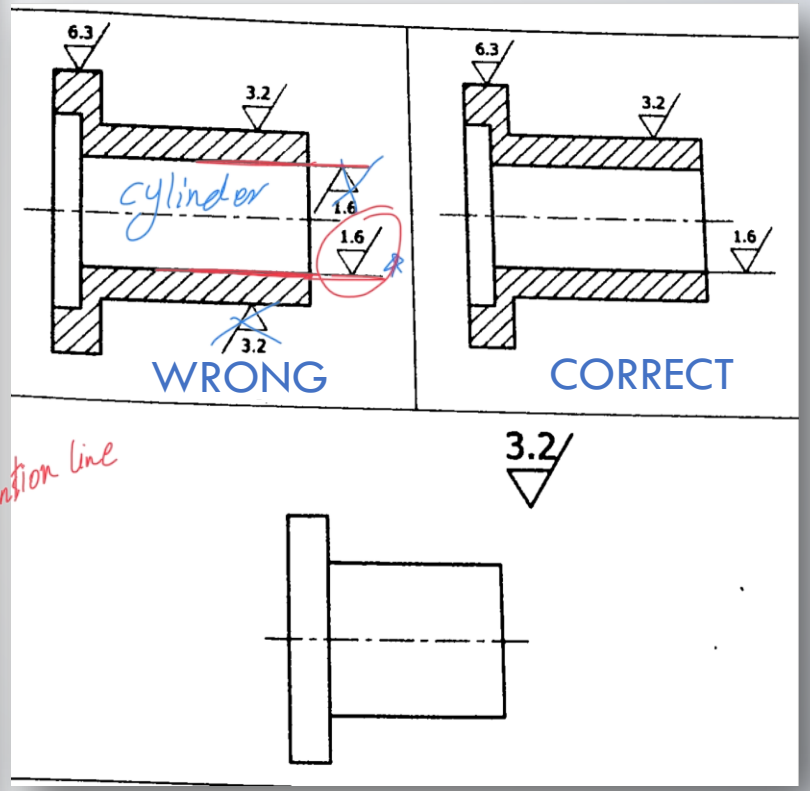
WRONG



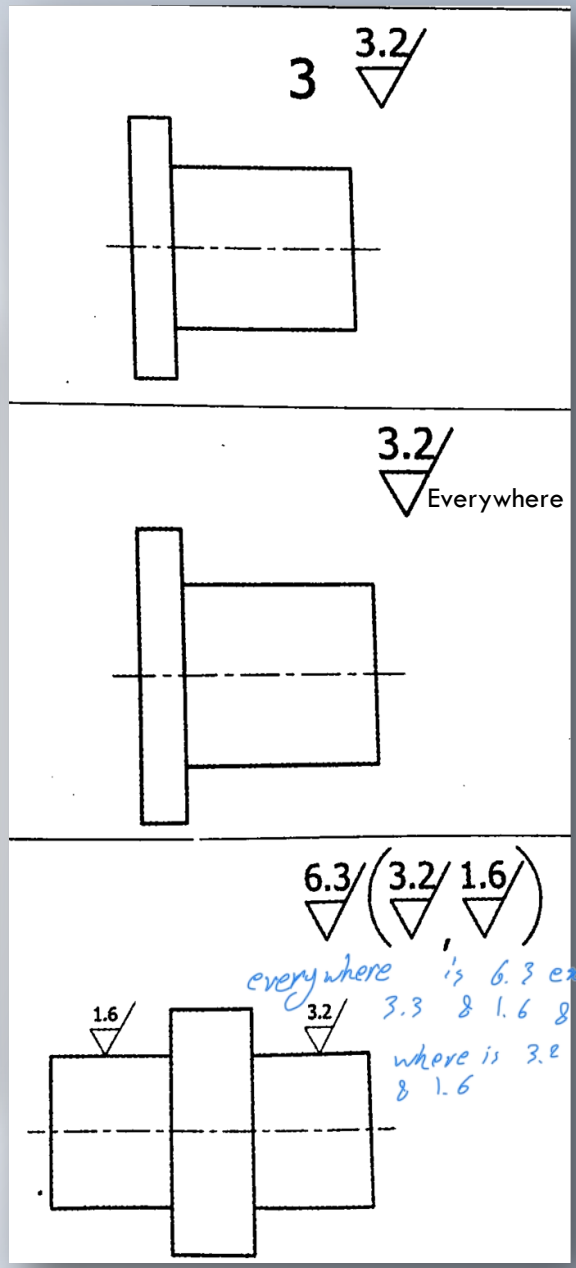
CORRECT

USAGE OF SYMBOLS

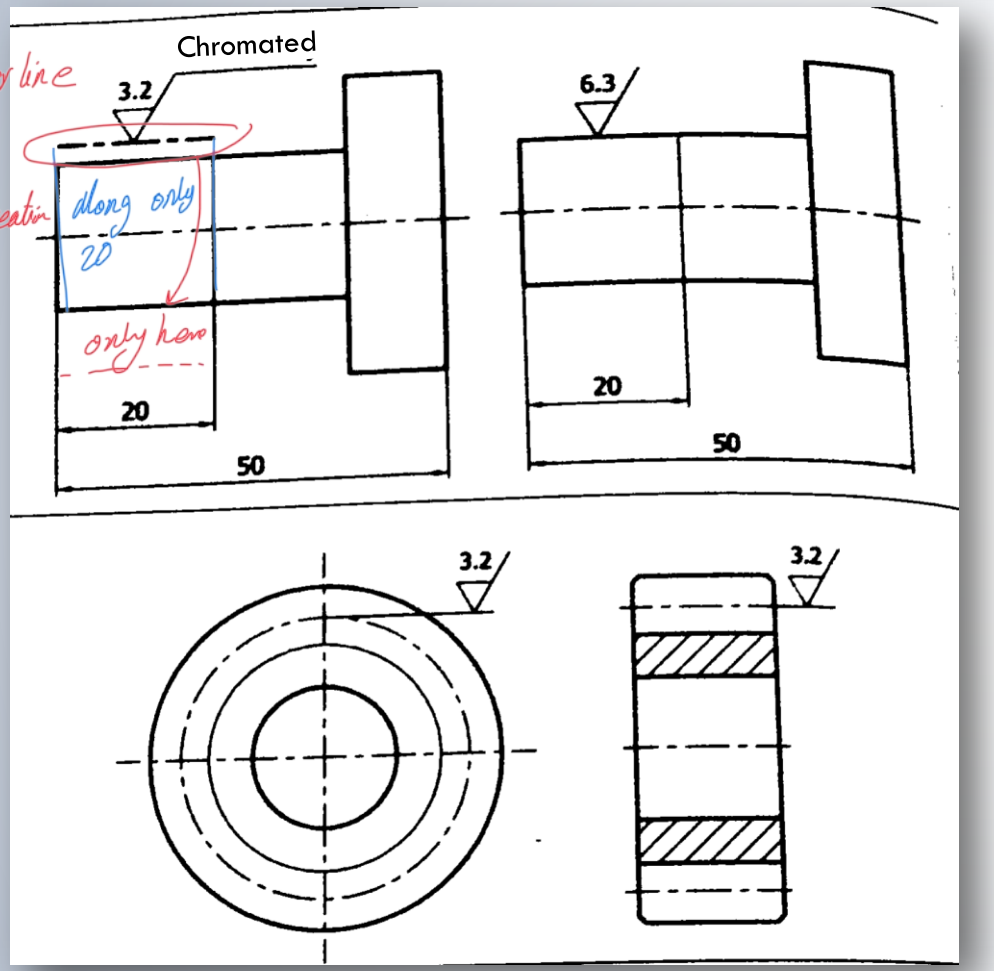


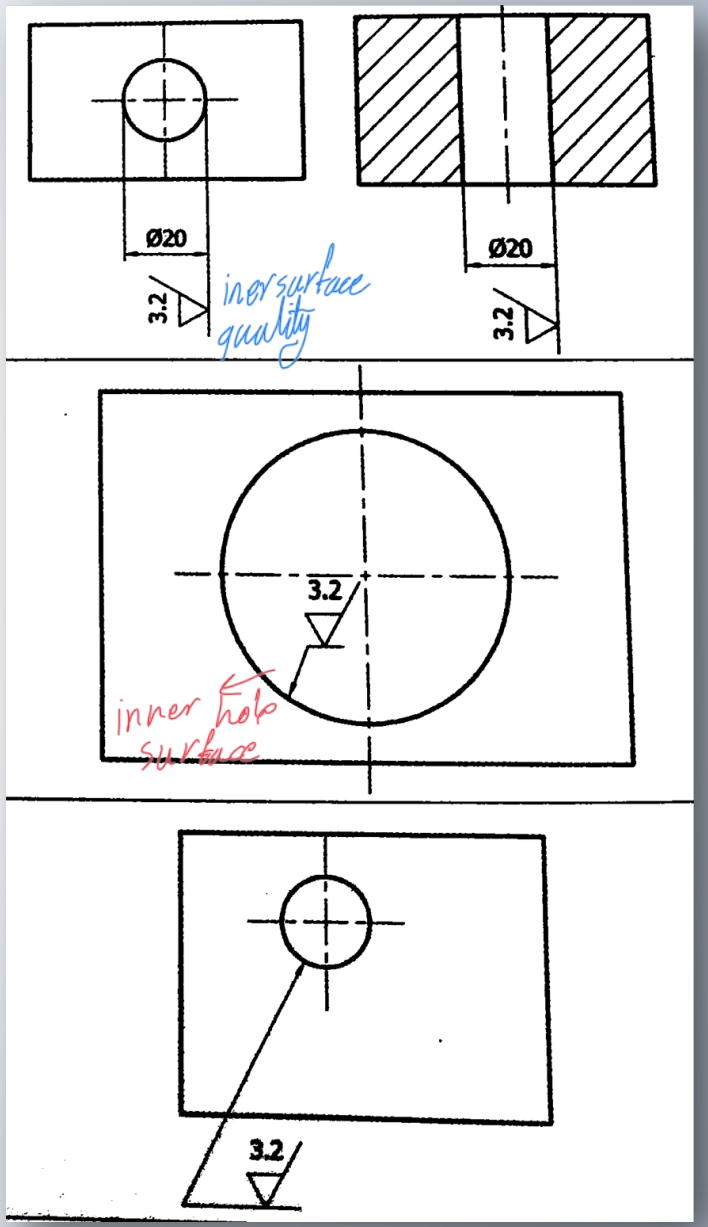


Common symbol is 1.5 times larger



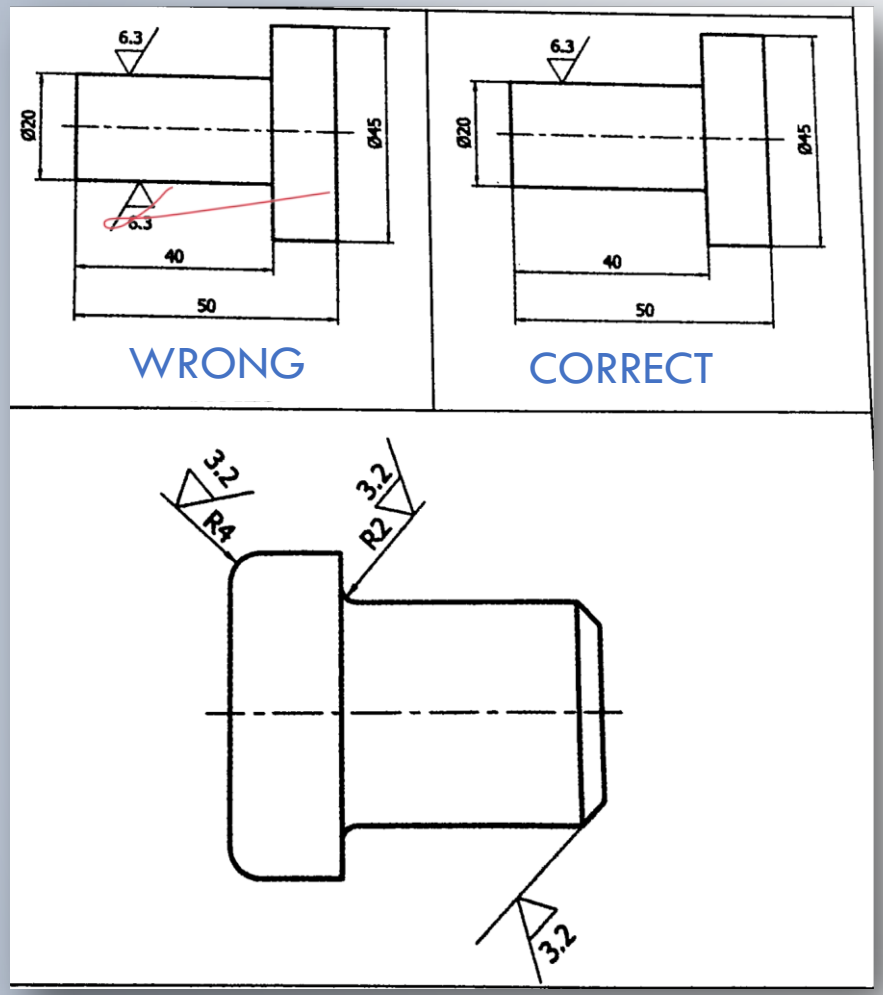
USAGE OF SYMBOLS

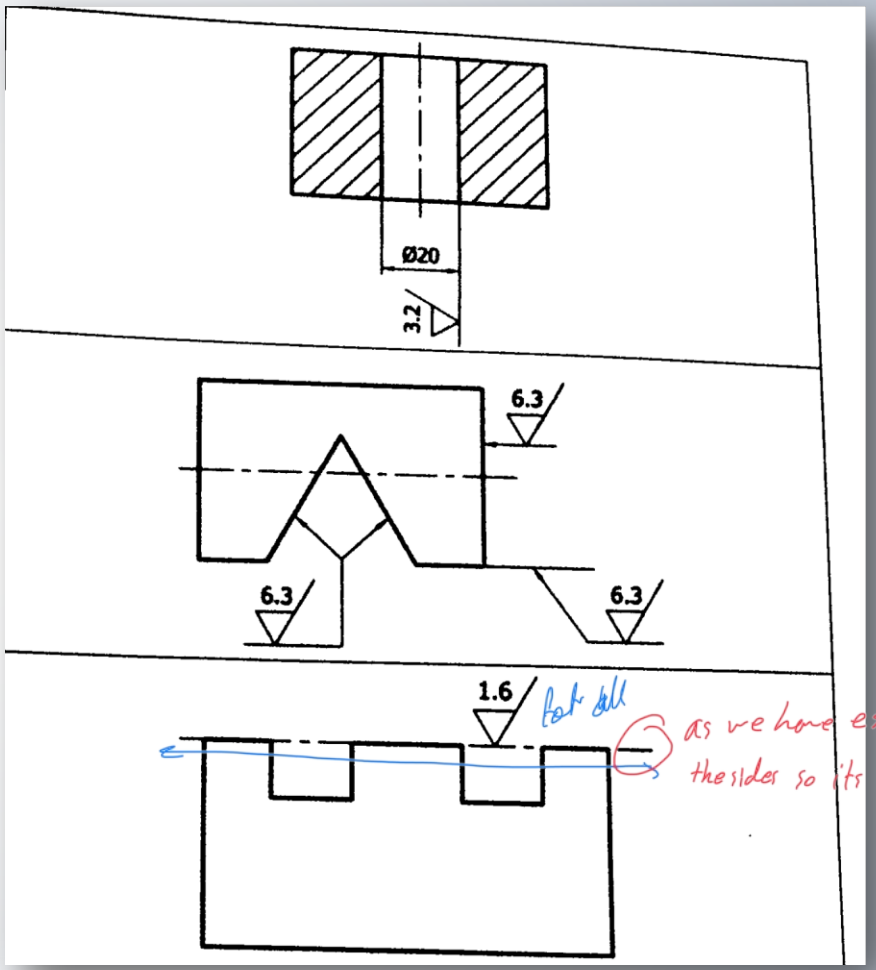




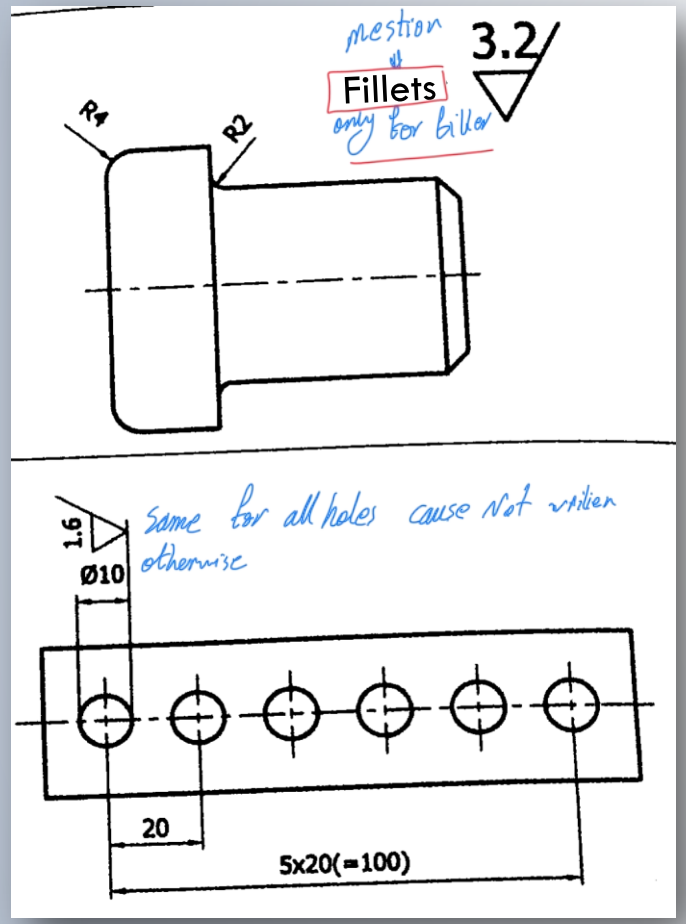
USAGE OF SYMBOLS

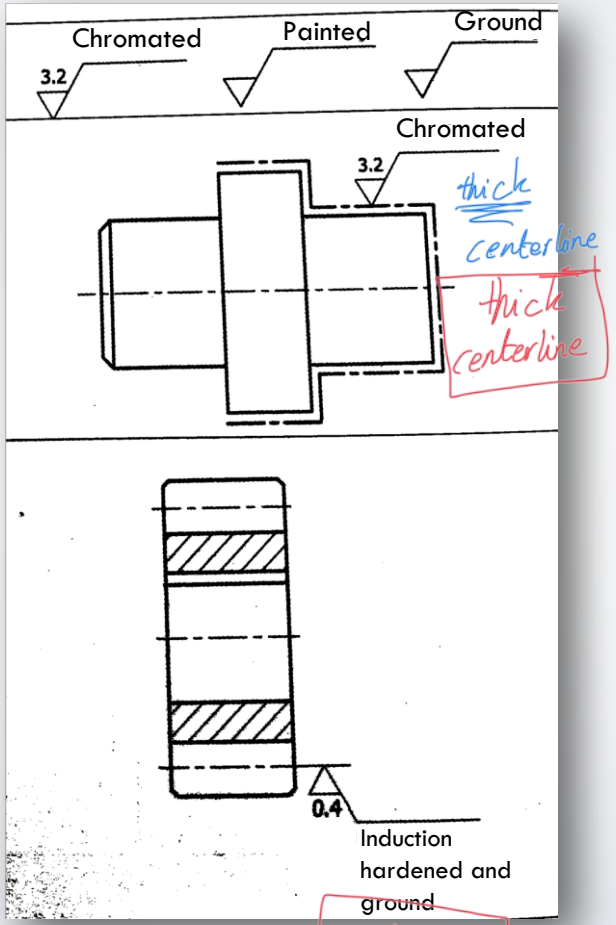
No diagonal or other views so cylindric

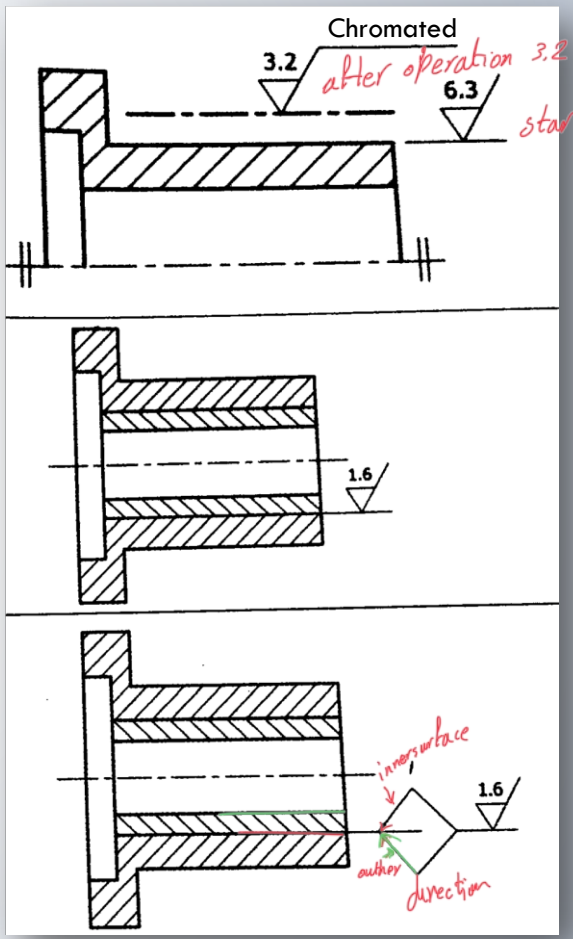




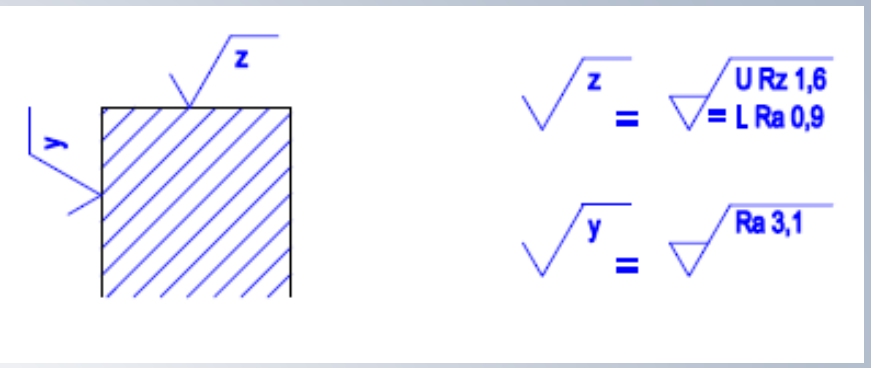
USAGE OF SYMBOLS

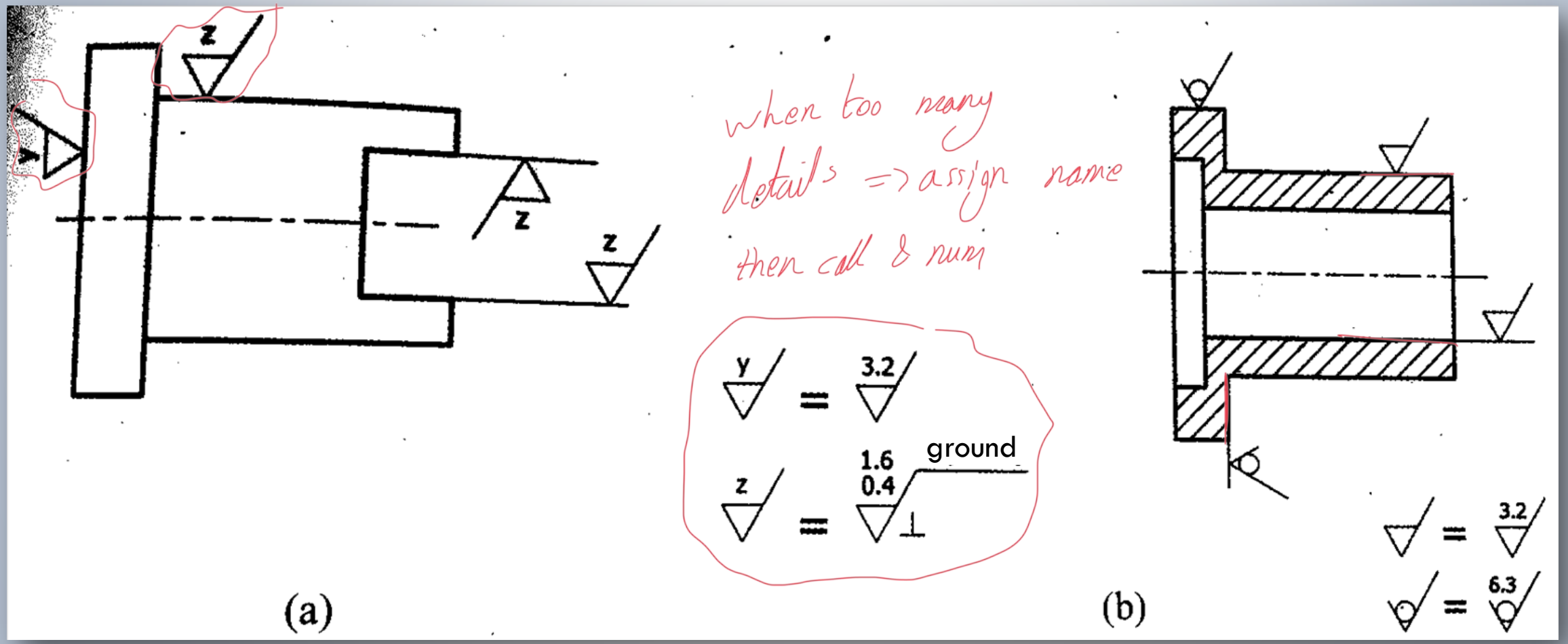




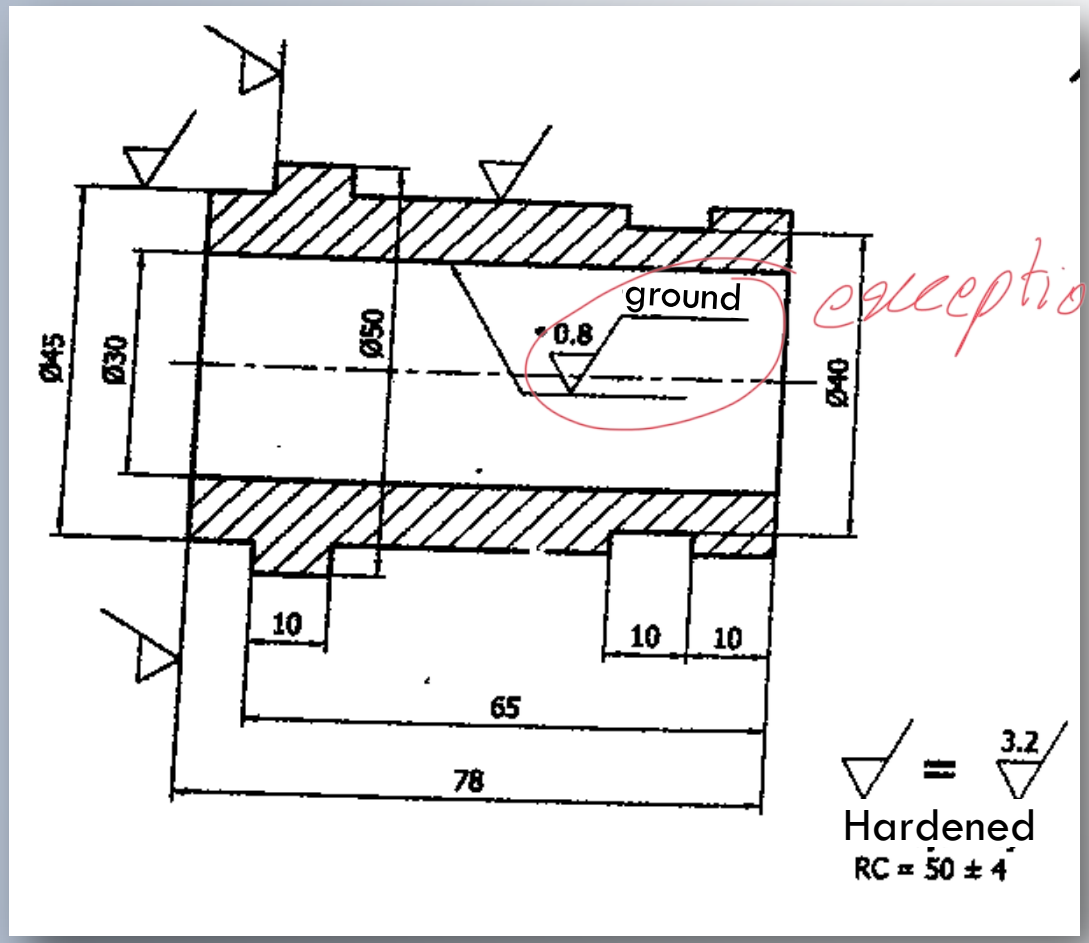


SIMPLIFICATION OF SYMBOLS

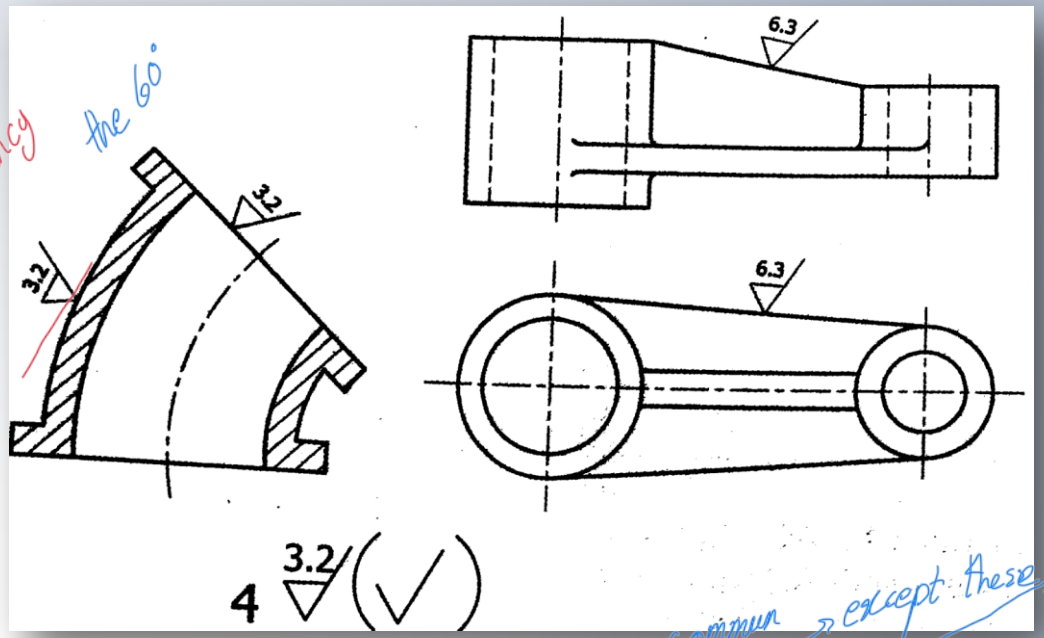




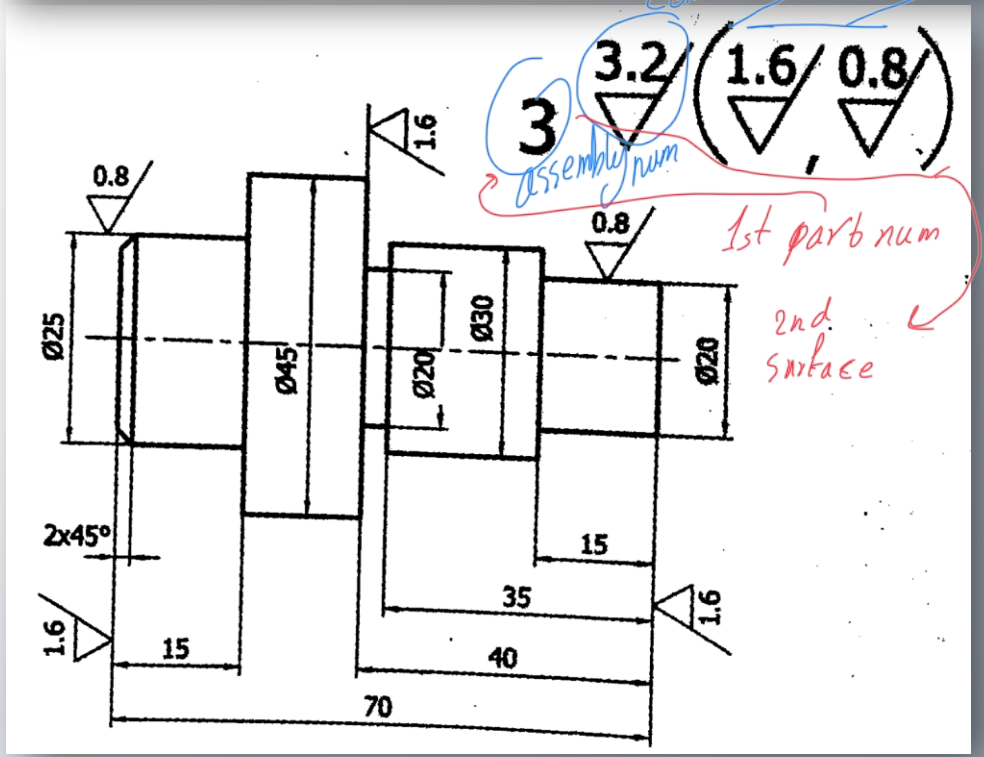
EXAMPLES



exception



make tangency the 60°



common → except these

assembly num

1st part num

2nd surface

REFERENCES

The content of this presentation is compiled from following sources:

1. S. Kurt, İ. Gerdemeli, C. E. İmrak, "Mühendislik Çizimin Esasları", Birsen Yayınevi, İstanbul, 2005.
2. Prof. C. Erdem Imrak, Ders Notları, MAK112E Computer Aided Technical Drawing, 2012.
3. Türkdemir, K. 2008. Teknik Resim I. Denizli: Boy Yayınları.
4. https://en.wikipedia.org/wiki/Surface_finish

Tolerances (Geometric, Limits and Fits)

Mostly from tolerance & machine

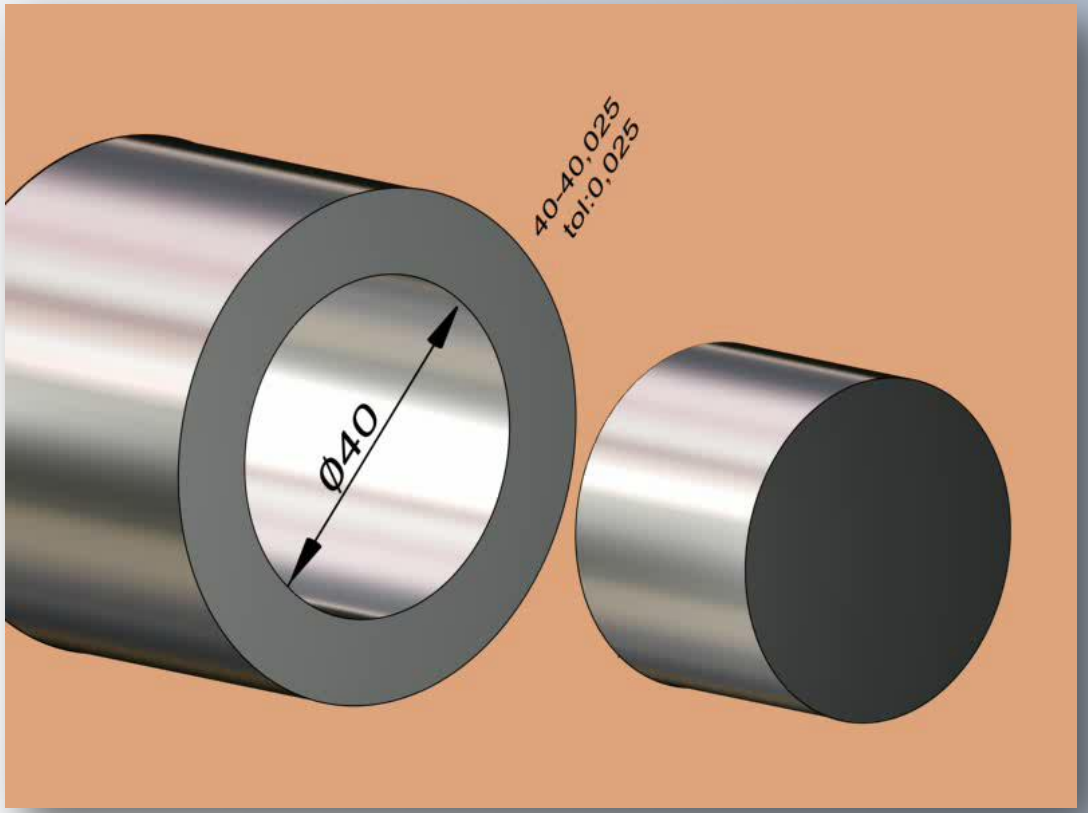
IMPORTANCE OF TOLERANCE

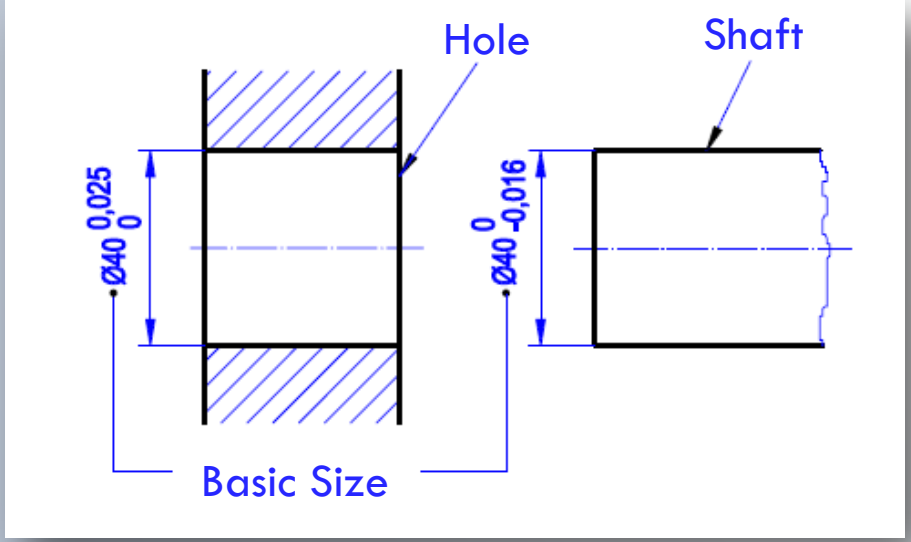
- Exact dimensions and shapes could not be attained in the shaping of physical objects.
- Part creation with low tolerances causes time and cost loss.
- Dimensional differences are allowed to the extent that they do not affect the interoperability of the interrelated parts.
- Dimension, shape and orientation tolerance values are given in the relevant drawings in order to ensure that the parts work together in the desired properties and easy assembly / disassembly.
- Tolerances are important for production of spare parts that will replace parts that deteriorate, wear and therefore need to be replaced.

DIMENSION TOLERANCE

higher tolerance → more accaptable
letting wider range of error

- **Basic size (BS):** The 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.

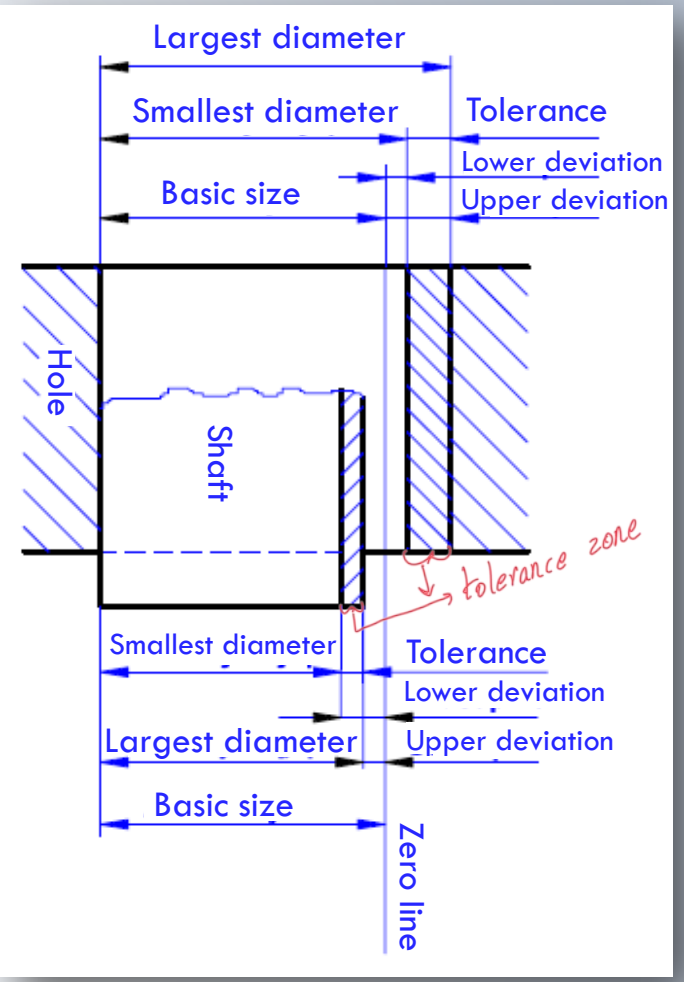




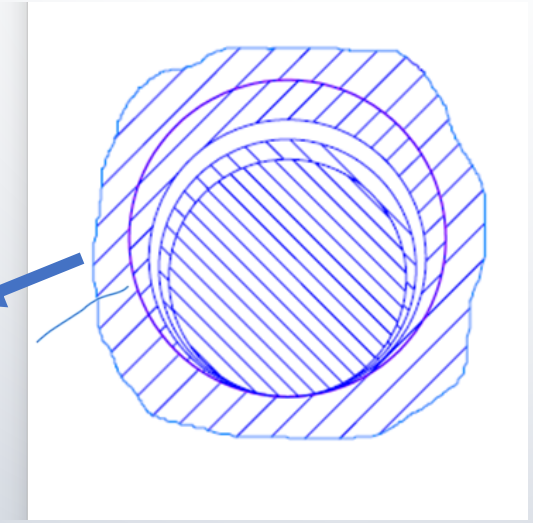
DEFINITIONS

max & min values

- **Limits of size:** The limits of size are the maximum and minimum sizes permitted for a specific dimension.
- **Tolerances:** The tolerance of a dimension is the total permissible variation in its size. The tolerance is the difference between the limits of size.



Both deviations of the hole are positive and negative for the shaft.



DEFINITIONS

intensional difference in the size

ex: if you want clearance between shaft & hole use

Allowance: An allowance is the intentional difference in size of mating parts. It is the minimum clearance (positive allowance) or maximum interference (negative allowance) between such parts.

empty space

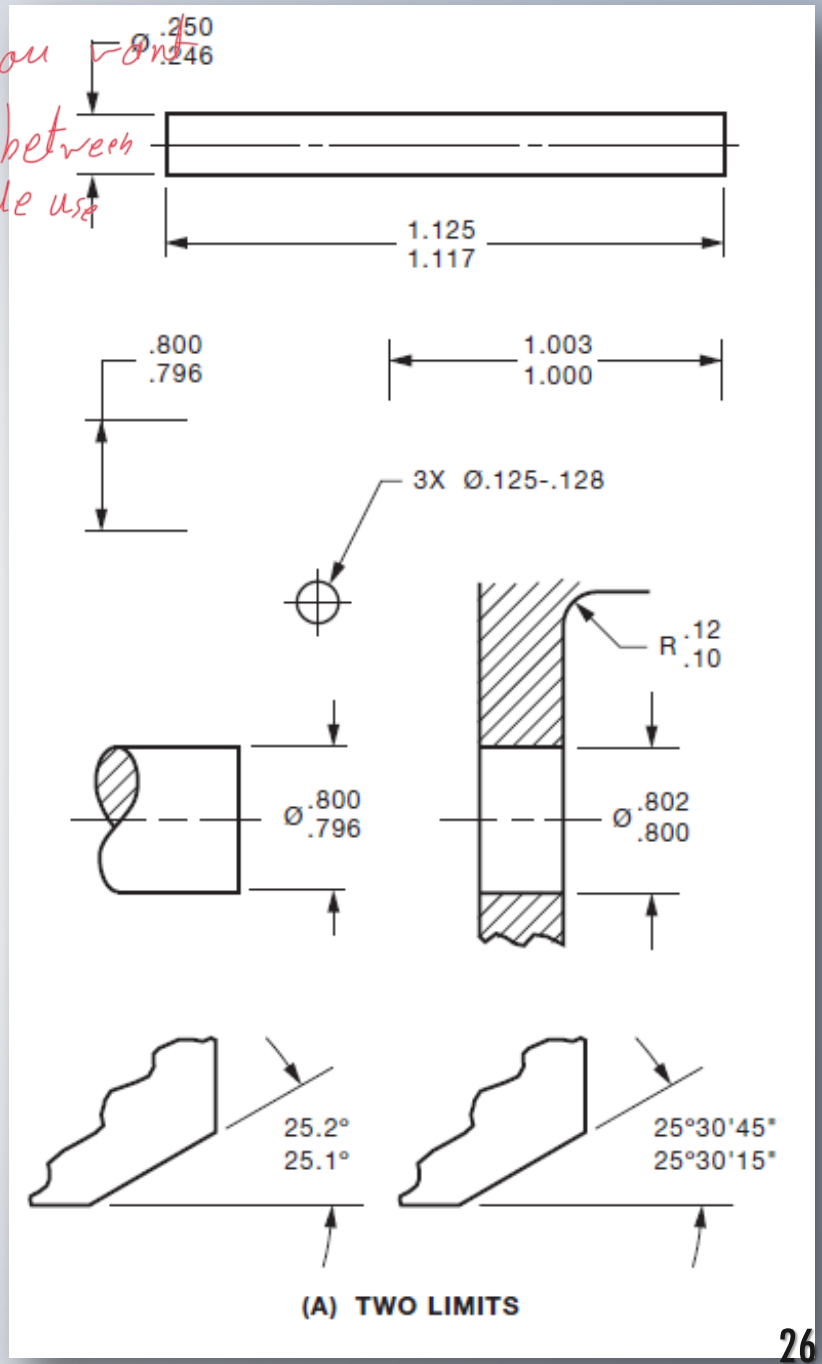
most تداول

- All dimensions on a drawing have tolerances.
- Some dimensions must be more exact than other dimensions and consequently have smaller tolerances.
- When dimensions require greater accuracy than the general note provides, individual tolerances or limits must be shown for that dimension.

show clearnce:

- positive allignce

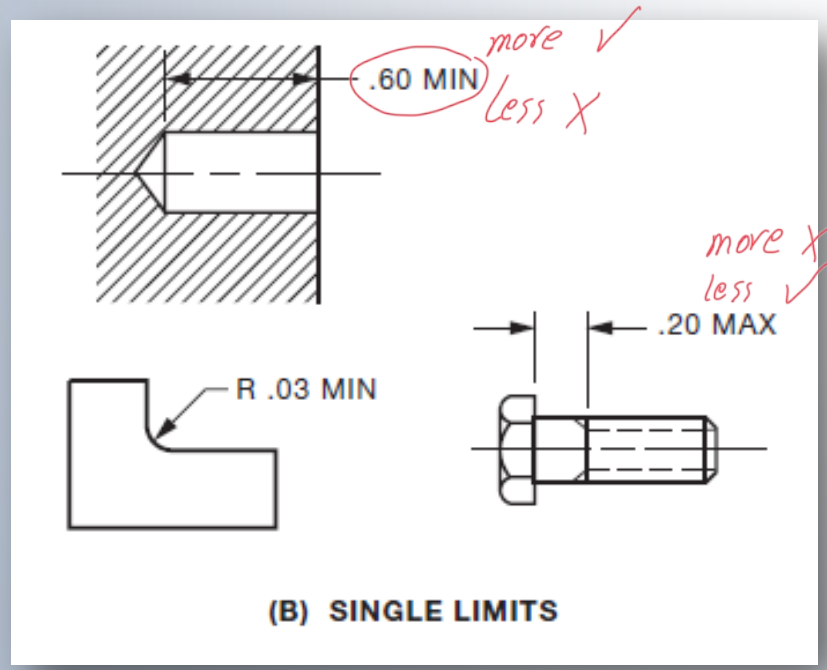
shaft larger than hole = interference



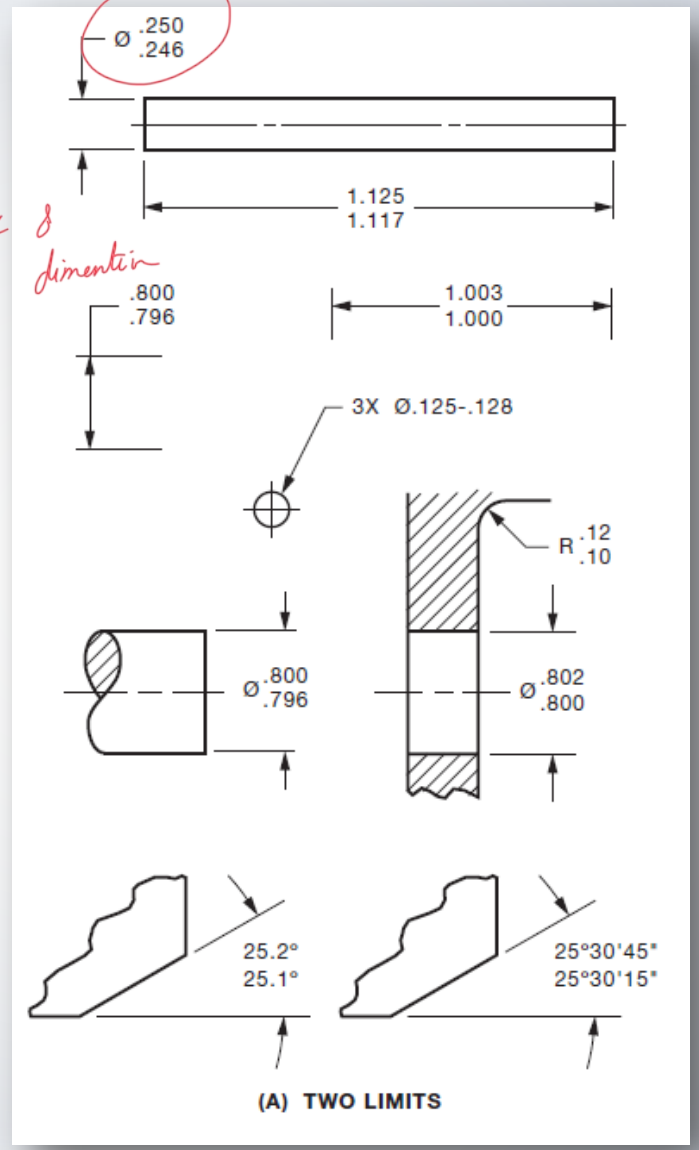
TOLERANCING METHODS

Tolerances are expressed in either of two ways: limit dimensioning or plus and minus tolerancing.

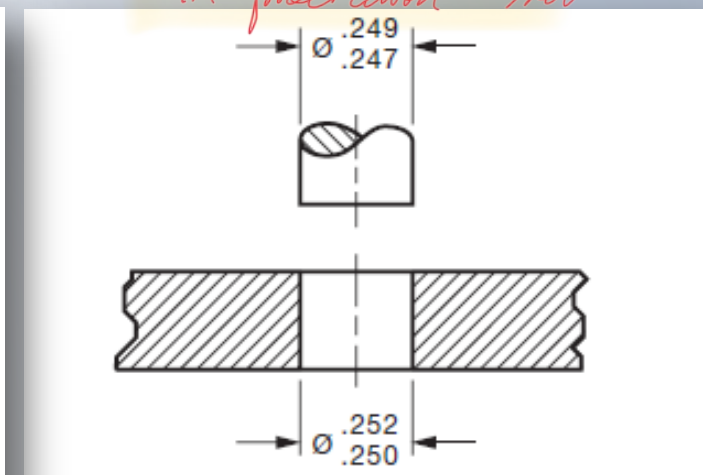
Limit dimensioning: In the limit dimensioning method, only the maximum and minimum dimensions are specified.



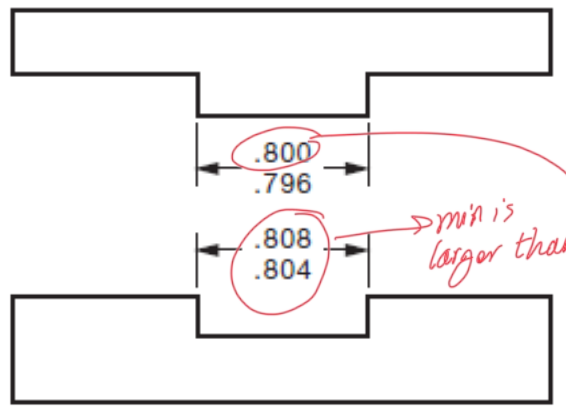
(B) SINGLE LIMITS



(A) TWO LIMITS



(A) CIRCULAR FEATURE



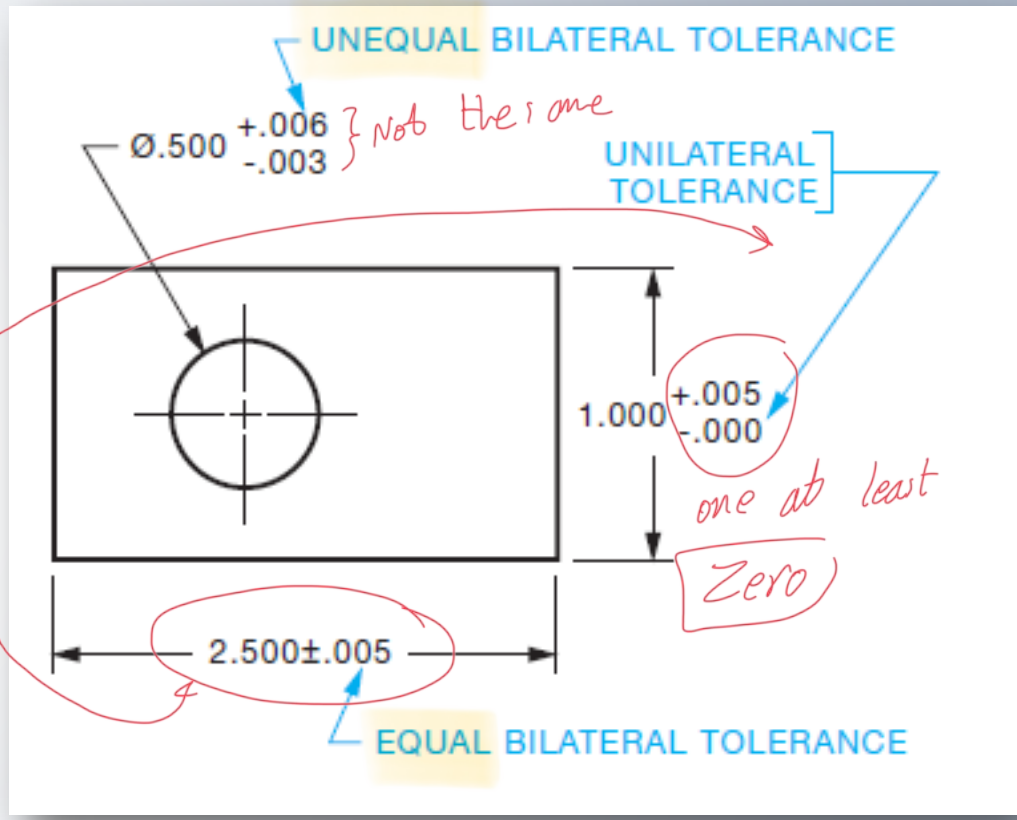
(B) FLAT FEATURE

TOLERANCING METHODS

Plus and Minus tolerancing: In this method the dimension of the specified size is given first and it is followed by a plus and minus tolerance expression. The tolerance can be bilateral or unilateral:

A **bilateral tolerance** is a tolerance that is expressed as plus and minus values. These values need not be the same size.

A **unilateral tolerance** is one that applies in one direction from the specified size, so the permissible variation in the other direction is zero.



TOLERANCING METHODS

General tolerance notes greatly simplify the drawing. The following examples illustrate the variety of application in this system.

if all tolerances the same or repeation

EXAMPLE 1

EXCEPT WHERE STATED OTHERWISE,
TOLERANCE ON DIMENSIONS ± .005

EXAMPLE 2

EXCEPT WHERE STATED OTHERWISE, TOLERANCES ON FINISHED DIMENSIONS TO BE AS FOLLOWS:

EXAMPLE 3

Dimension	Tolerance
Up to 3.00	.01
Over 3.00 to 12.00	.02
Over 12.00 to 24.00	.04
Over 24.00	.06

EXAMPLE 4

UNLESS OTHERWISE SPECIFIED
± .005 TOLERANCE ON MACHINED DIMENSIONS
± .04 TOLERANCE ON CAST DIMENSIONS
ANGULAR TOLERANCE ± 30′

DIMENSION ORIGIN SYMBOL

where does it starts from

- This symbol is used to indicate that a tolerance dimension between two features originates from one of these features.

origin sign shown non changing part

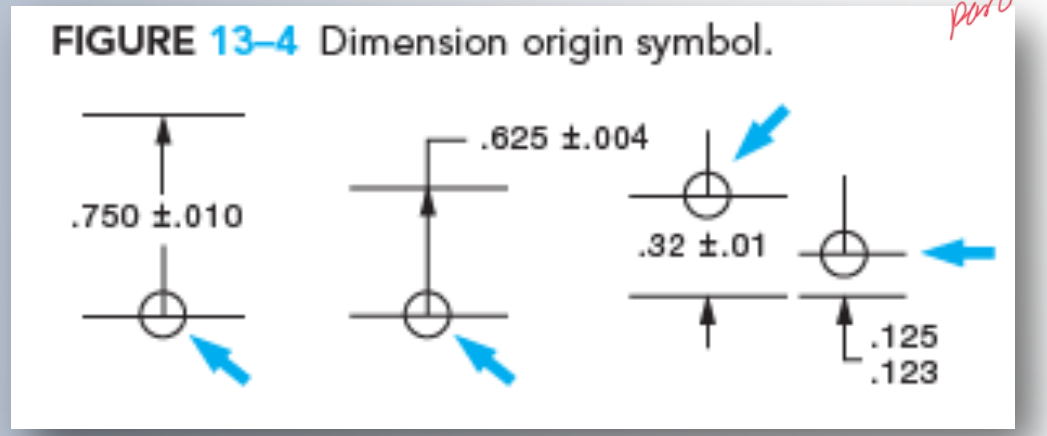


FIGURE 13–4 Dimension origin symbol.

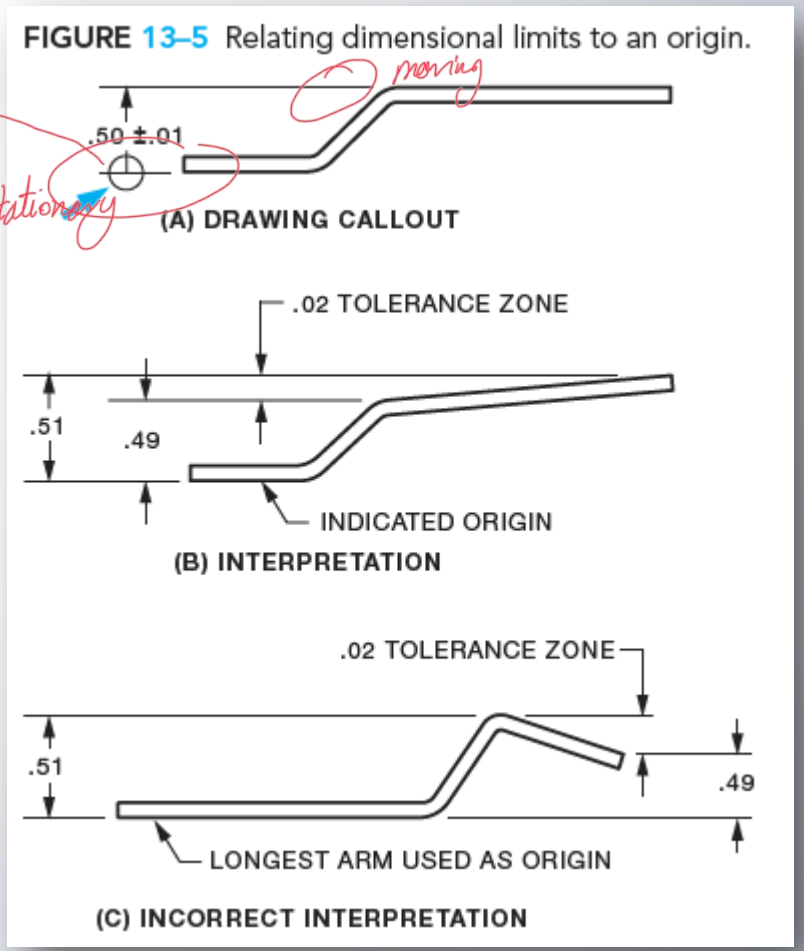
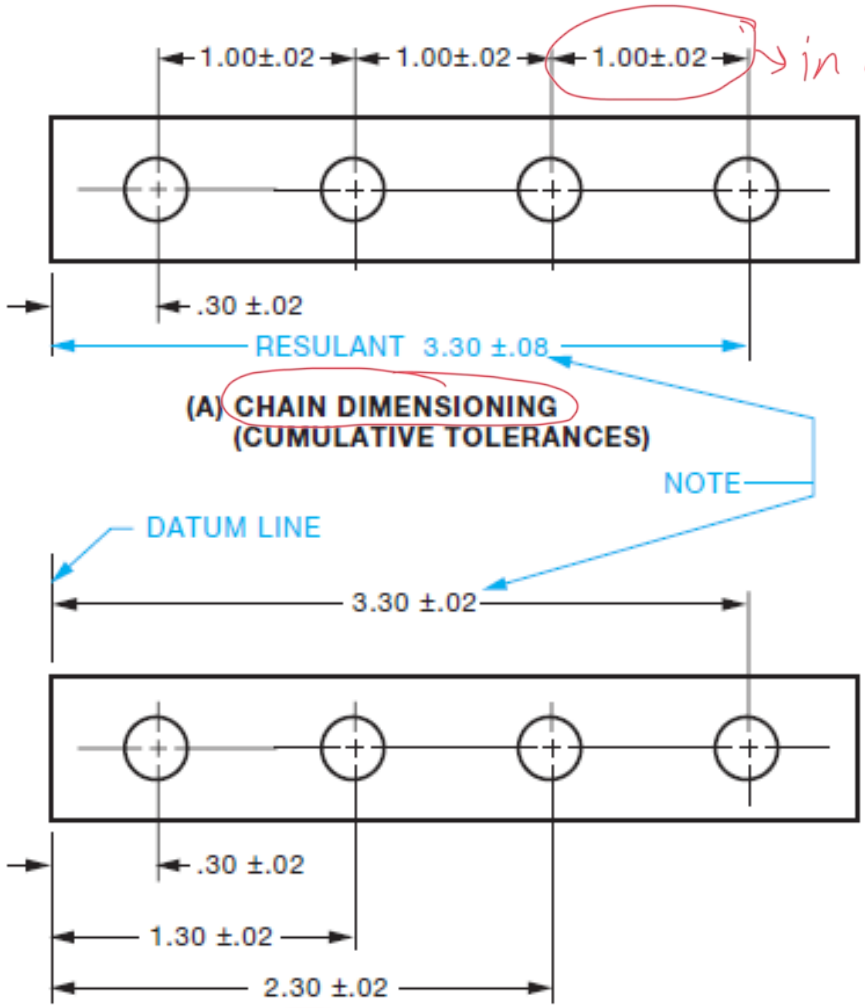


FIGURE 13–5 Relating dimensional limits to an origin.

TOLERANCES IN DIMENSIONING

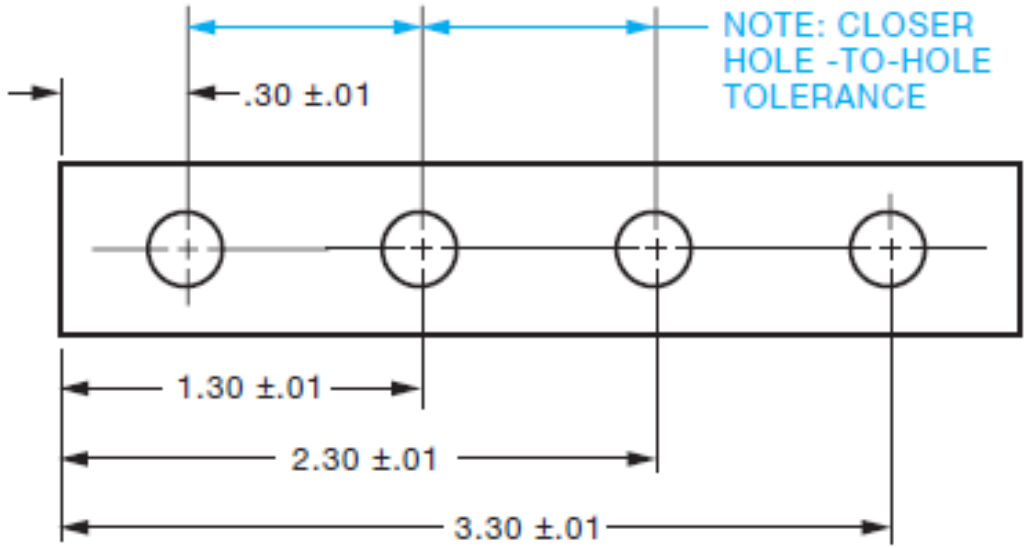
Comparison between chain dimensioning and base line dimensioning.



(A) CHAIN DIMENSIONING (CUMULATIVE TOLERANCES)

(B) BASE LINE DIMENSIONING (NON-CUMULATIVE TOLERANCES)

in chain the last circle position will be ...



(C) MAINTAINING SAME DIMENSION BETWEEN HOLES AS (A) OR (B) BUT WITH CLOSER TOLERANCE

METRIC FITS

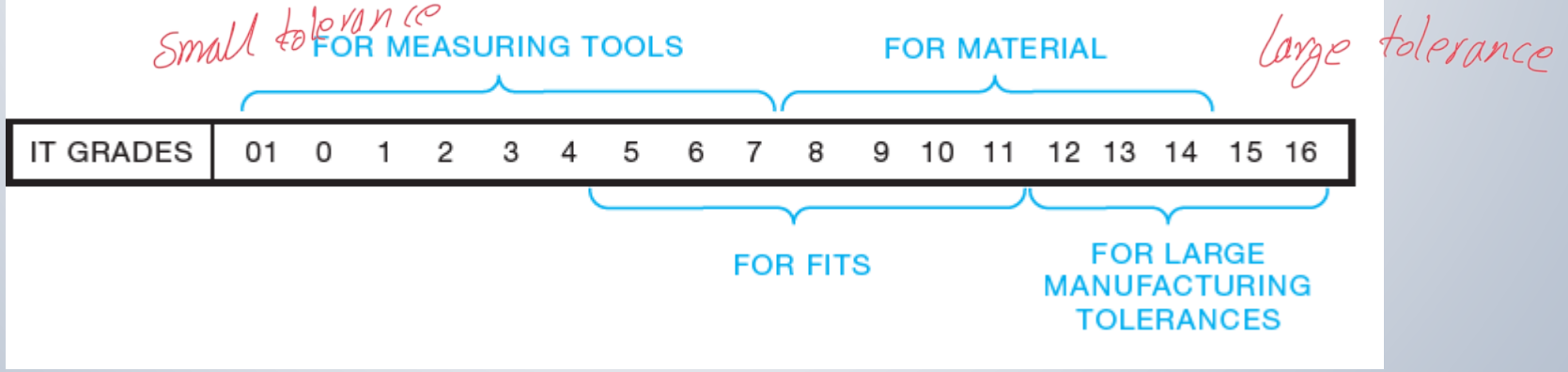
hole shaft system

- The general terms "hole" and "shaft" can also be taken as referring to the space containing or contained by two parallel faces of any part, such as the width of a slot, or the thickness of a key.

key = shaft slob = hole

- An "International Tolerance Grade" establishes the magnitude of the tolerance zone or the amount of part size variation allowed for internal and external dimensions alike.

Application of international tolerance (IT) grades.



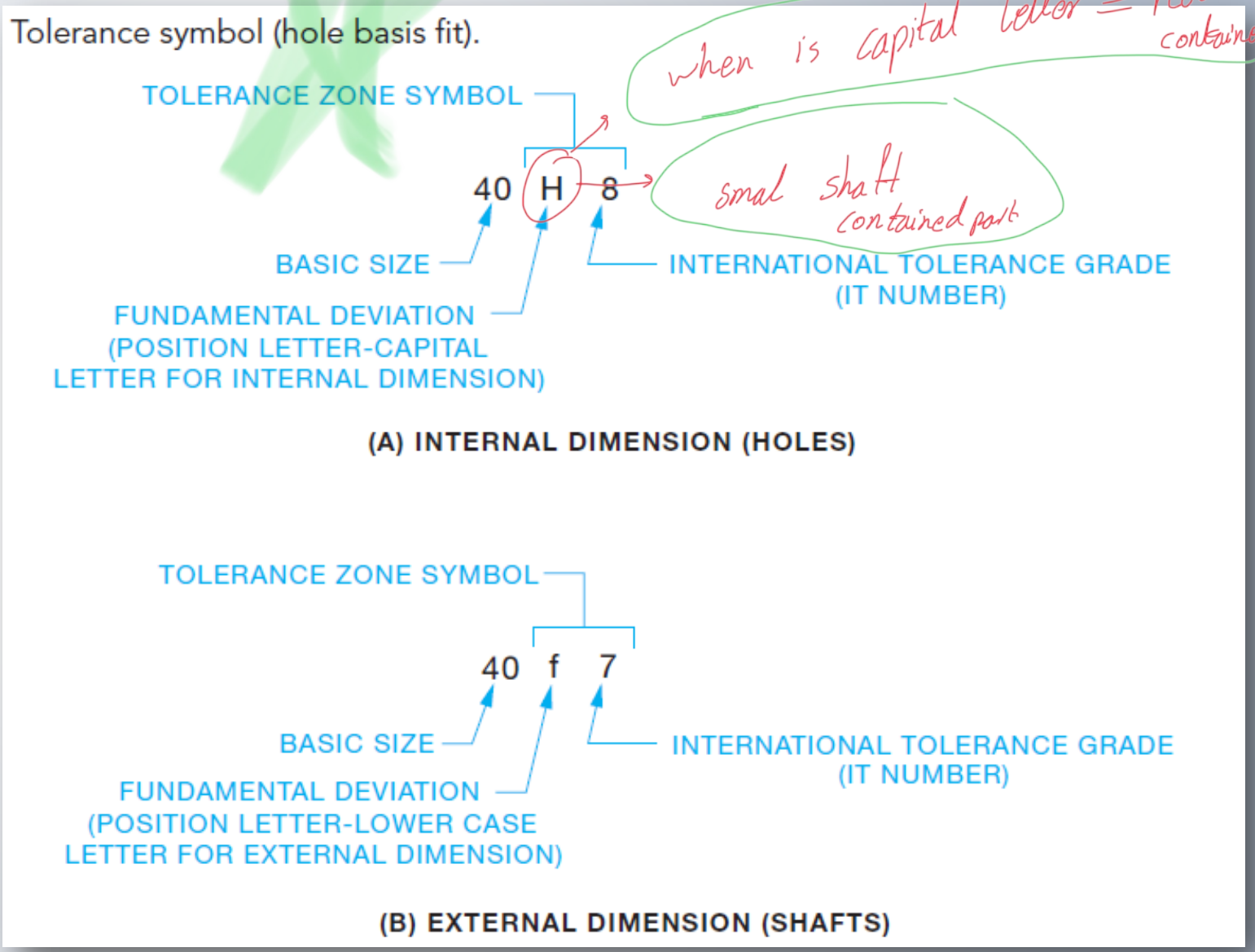
TOLERANCING GRADES FIR MACHINING PROCESSES

Tolerancing grades for machining processes.

MACHINING PROCESS	TOLERANCE GRADES									
	4	5	6	7	8	9	10	11	12	13
Lapping & Honing	■	■								
Cylindrical Grinding		■	■	■						
Surface Grinding		■	■	■	■					
Diamond Turning		■	■	■						
Diamond Boring		■	■	■						
Broaching		■	■	■	■					
Reaming		■	■	■	■	■	■			
Turning				■	■	■	■	■	■	■
Boring					■	■	■	■	■	■
Milling							■	■	■	■
Planing & Shaping							■	■	■	■
Drilling							■	■	■	■

METRIC TOLERANCE SYMBOL

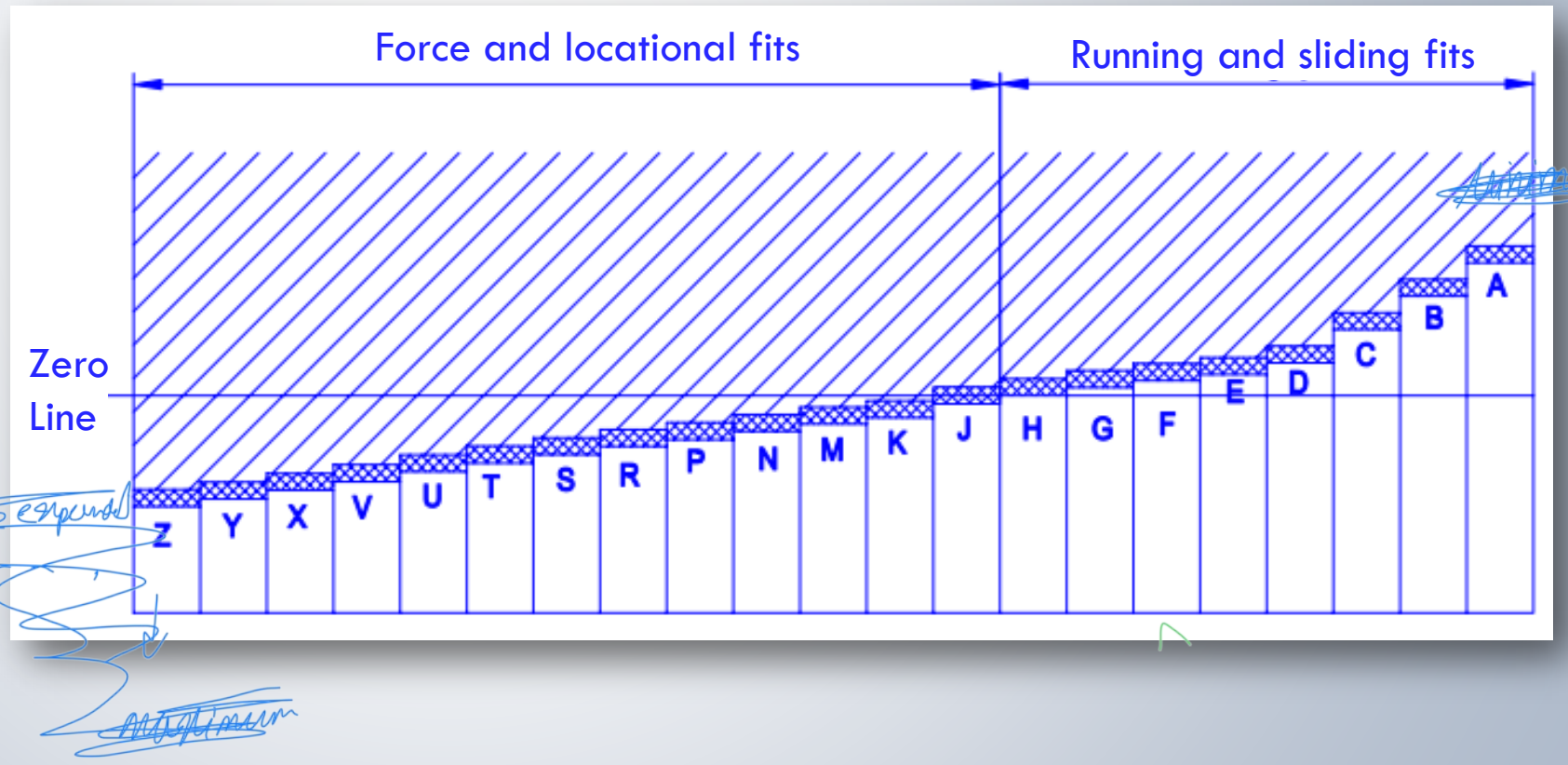
- Hole basis fits have a fundamental deviation of "H" on the hole, and shaft basis fits have a fundamental deviation of "h" on the shaft.
- Normally, the hole basis system is preferred.



TOLERANCE ZONES AND SYMBOLS

- Holes
  - A-H for lower deviation values
  - J-ZC for upper deviation values

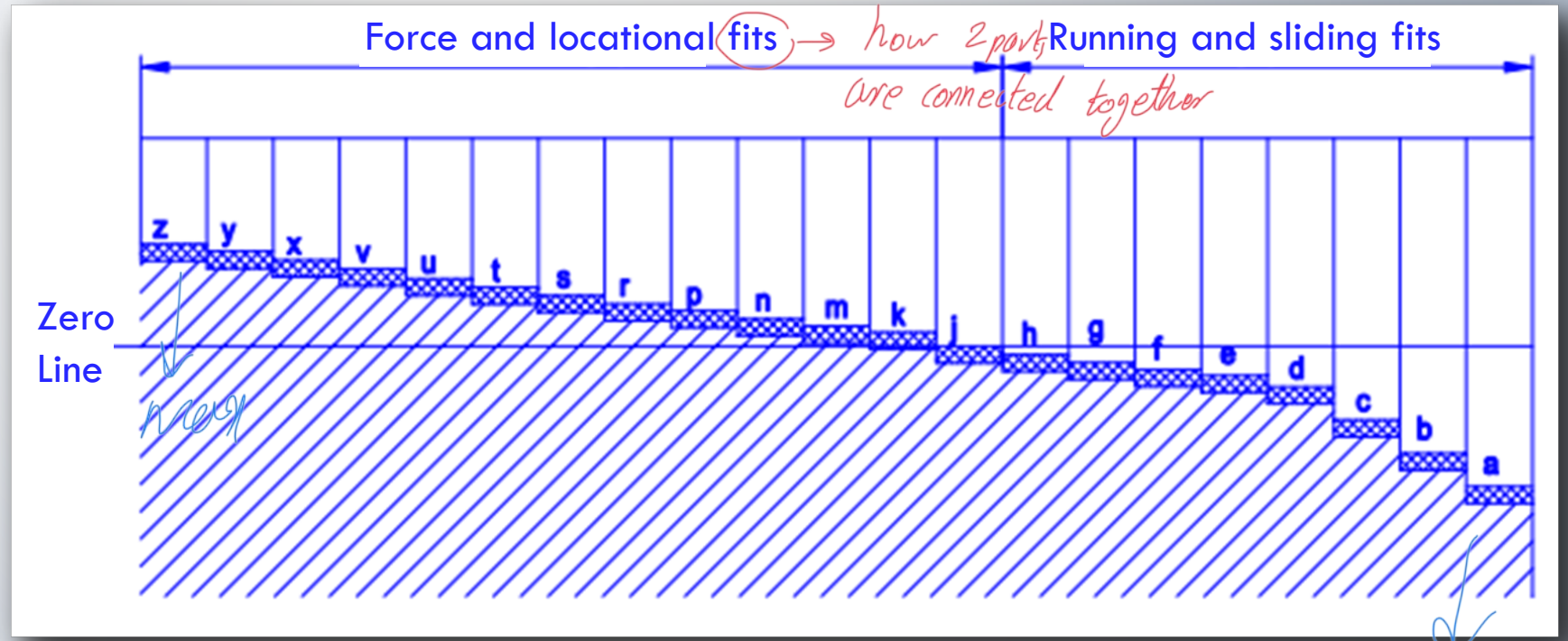
Tolerance areas for holes



TOLERANCE ZONES AND SYMBOLS

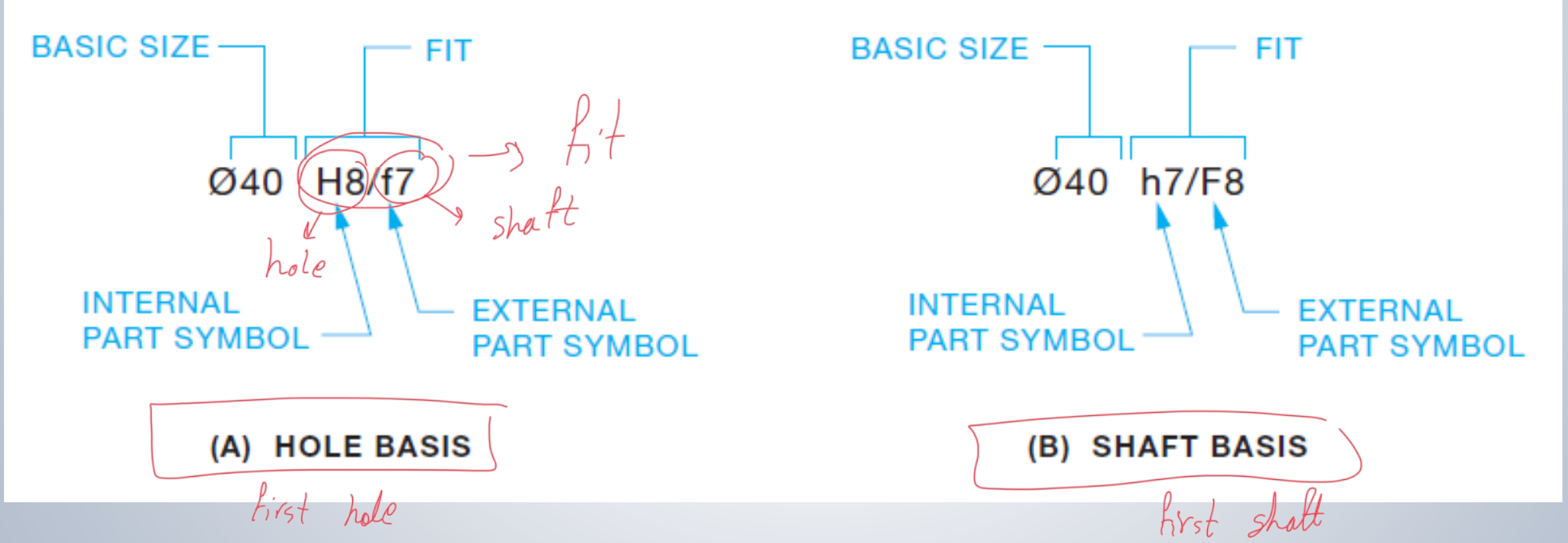
- Shafts
  - a-h for upper deviation values
  - j-zc for lower deviation values

Tolerance areas for shafts

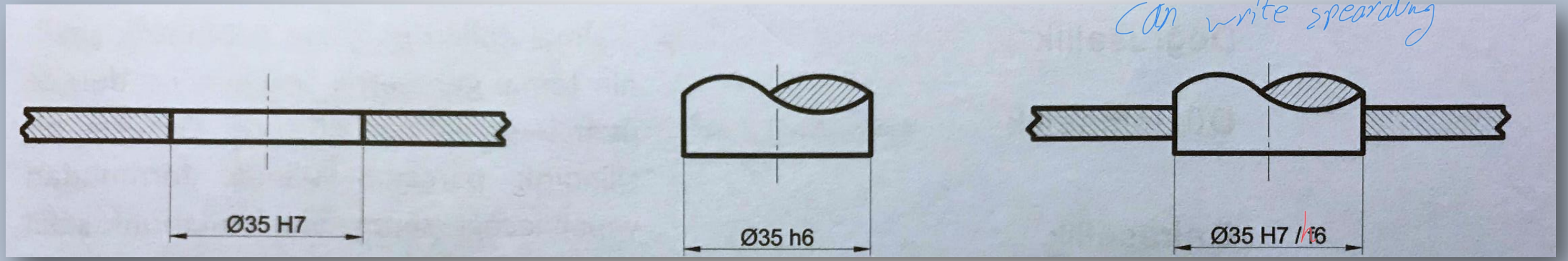


FIT SYMBOL

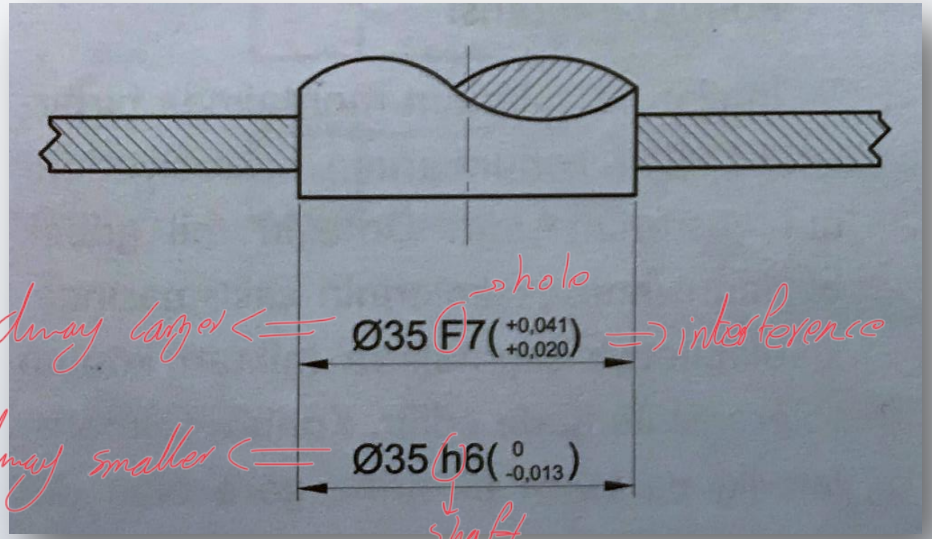
- A fit is specified by the basic size common to both components, followed by a symbol corresponding to each component, with the internal part symbol preceding the external part symbol.



DIMENSIONING OF SHAFTS AND HOLES



Can write spearating



or together

Sürmen, H. K., Erdem, S. S. 2020. Teknik Resim ve Uygulamaları. Ankara: Nobel Akademik Yayıncılık.

ISO TOLERANCE TABLE EXAMPLES

Ø 25 H8/f7 → hole

diameter rang

ISO Delik Toleransları (ISO 286-2)

Nominal delik ölçüsü (mm)

Çap Aralığı	3	6	10	18	30	40	50	65	80	100	120	140	160	180	200	225	250	280	315	355
	6	10	18	30	40	50	65	80	100	120	140	160	180	200	225	250	280	315	355	400
Mikrometre(µm)																				
E6	+28 +20	+34 +25	+43 +32	+53 +40	+66 +50	+79 +60	+94 +72	+110 +85	+129 +100	+142 +110	+161 +125									
E7	+32 +20	+40 +25	+50 +32	+61 +40	+75 +50	+90 +60	+107 +72	+125 +85	+146 +100	+162 +110	+185 +125									
E11	+95 +20	+115 +25	+142 +32	+170 +40	+210 +50	+250 +60	+292 +72	+335 +85	+390 +100	+430 +110	+485 +125									
E12	+140 +20	+175 +25	+212 +32	+250 +40	+300 +50	+360 +60	+422 +72	+485 +85	+560 +100	+630 +110	+695 +125									
E13	+200 +20	+245 +25	+302 +32	+370 +40	+440 +50	+520 +60	+612 +72	+715 +85	+820 +100	+920 +110	+1 015 +125									
F6	+18 +10	+22 +13	+27 +16	+33 +20	+41 +2	+49 +30	+58 +36	+68 43	+79 +50	+88 +56	+98 +62									
F7	+22 +10	+28 +13	+34 +16	+41 +20	+50 +25	+60 +30	+71 +36	+83 43	+96 +50	+108 +56	+119 +62									
F8	+28 +10	+35 +13	+43 +16	+53 +20	+64 +25	+76 +30	+90 +36	+106 43	+122 +50	+137 +56	+151 +62									
G6	+12 +4	+14 +5	+17 +6	+20 +7	+25 +9	+29 +10	+34 +12	+39 +14	+44 +15	+49 +17	+54 +18									
G7	+16 +4	+20 +5	+24 +6	+28 +7	+34 +9	+40 +10	+47 +12	+54 +14	+61 +15	+69 +17	+75 +18									
G8	+22 +4	+27 +5	+33 +6	+40 +7	+48 +9	+56 +10	+66 +12	+77 +14	+87 +15	+98 +17	+107 +18									
H6	+8 0	+9 0	+11 0	+13 0	+16 0	+19 0	+22 0	+25 0	+29 0	+32 0	+36 0									
H7	+12 0	+15 0	+18 0	+21 0	+25 0	+30 0	+35 0	+40 0	+46 0	+52 0	+57 0									
H8	+18 0	+22 0	+27 0	+33 0	+39 0	+46 0	+54 0	+63 0	+72 0	+81 0	+89 0									
H9	+30 0	+36 0	+43 0	+52 0	+62 0	+74 0	+87 0	+100 0	+115 0	+130 0	+140 0									

shaft

77

ISO Mil Toleransları (ISO 286-2)

Nominal Mil Ölçüsü (mm)

Çap Aralığı	3	6	10	18	30	40	50	65	80	100	120	140	160	180	200	225	250	280	315	355
	6	10	18	30	40	50	65	80	100	120	140	160	180	200	225	250	280	315	355	400
mikrometre(µm)																				
a12	-270 -390	-280 -430	-290 -470	-300 -510	-310 -560	-320 -570	-340 -640	-360 -660	-380 -730	-410 -760	-460 -860	-520 -920	-580 -980	-660 -1120	-740 -1200	-820 -1280	-920 -1440	-1050 -1570	-1200 -1770	-1350 -1920
d6	-30 -38	-40 -49	-50 -61	-65 -78	-80 -96	-100 -119	-120 -142	-145 -170	-170 -199	-190 -222	-210 -246									
e6	-20 -28	-25 -34	-32 -43	-40 -53	-50 -66	-60 -79	-72 -94	-85 -110	-100 -129	-110 -142	-125 -161									
e13	-20 -200	-25 -245	-32 -302	-40 -370	-50 -440	-60 -520	-72 -612	-85 -715	-100 -820	-110 -920	-125 -1015									
f5	-10 -15	-13 -19	-16 -24	-20 -29	-25 -36	-30 -43	-36 -51	-43 -61	-50 -70	-56 -79	-62 -87									
f6	-10 -18	-13 -22	-16 -27	-20 -33	-25 -41	-30 -49	-36 -58	-43 -68	-50 -79	-56 -88	-62 -98									
f7	-10 -22	-13 -28	-16 -34	-20 -41	-25 -50	-30 -60	-36 -71	-43 -83	-50 -96	-56 -108	-62 -119									
g5	-4 -9	-5 -11	-6 -14	-7 -16	-9 -20	-10 -23	-12 -27	-14 -32	-15 -35	-17 -40	-18 -43									
g6	-4 -12	-5 -14	-6 -17	-7 -20	-9 -25	-10 -29	-12 -34	-14 -39	-15 -44	-17 -49	-18 -54									
g7	-4 -16	-5 -20	-6 -24	-7 -28	-9 -34	-10 -40	-12 -47	-14 -54	-15 -61	-17 -69	-18 -75									
h4	-0 -4	-0 -4	-0 -5	-0 -6	-0 -7	-0 -8	-0 -10	-0 -12	-0 -14	-0 -16	-0 -18									
h5	-0 -5	-0 -6	-0 -8	-0 -9	-0 -11	-0 -13	-0 -15	-0 -18	-0 -20	-0 -23	-0 -25									
h6	-0 -8	-0 -9	-0 -11	-0 -13	-0 -16	-0 -19	-0 -22	-0 -25	-0 -29	-0 -32	-0 -36									
h7	-0 -12	-0 -15	-0 -18	-0 -21	-0 -25	-0 -30	-0 -35	-0 -40	-0 -46	-0 -52	-0 -57									
h8	-0 -18	-0 -22	-0 -27	-0 -33	-0 -39	-0 -46	-0 -54	-0 -63	-0 -72	-0 -81	-0 -89									
h9	-0 -30	-0 -36	-0 -43	-0 -52	-0 -62	-0 -74	-0 -87	-0 -100	-0 -115	-0 -130	-0 -140									
h10	-0 -48	-0 -58	-0 -70	-0 -84	-0 -100	-0 -120	-0 -140	-0 -160	-0 -185	-0 -210	-0 -230									
h11	-0 -75	-0 -90	-0 -110	-0 -130	-0 -160	-0 -190	-0 -220	-0 -250	-0 -290	-0 -320	-0 -360									
h12	-0 -120	-0 -150	-0 -180	-0 -210	-0 -250	-0 -300	-0 -350	-0 -400	-0 -460	-0 -520	-0 -570									
j5	+3 -2	+4 -2	+5 -3	+5 -4	+6 -5	+6 -7	+6 -9	+7 -11	+7 -13	+7 -16	+7 -18									
j6	+6 -2	+7 -2	+8 -3	+9 -4	+11 -5	+12 -7	+13 -9	+14 -11	+16 -13	+16 -16	+18 -18									
j7	+8 -4	+10 -5	+12 -6	+13 -8	+15 -10	+18 -12	+20 -15	+22 -18	+25 -21	+26 -26	+29 -28									

Sürmen, H. K., Erdem, S. S. 2020. Teknik Resim ve Uygulamaları. Ankara: Nobel Akademik Yayıncılık.

HOLE BASIS FITS SYSTEM

- Basic size will be the minimum size of the hole.
- For example, for a Ø25 H8/f7 fit, which is a Preferred Hole Basis Clearance Fit, the limits for the hole and shaft will be as follows:
  - Hole limits = Ø25.000 and Ø25.033
  - Shaft limits = Ø24.959 and Ø24.980
  - Minimum clearance = 0.020
  - Maximum clearance = 0.074

find in table

clearance => having free space between hole & shaft

less free space

min hole max shaft

max hole min shaft

more free space

difference between hole & shaft

HOLE BASIS FITS SYSTEM

Interference => shaft > larger than hole so they would interfere inorder to fit

- If a Ø25 H7/s6 Preferred Hole Basis Interference Fit is required, the limits for the hole and shaft will be as follows:
  - Hole limits = Ø25.000 and Ø25.021
  - Shaft limits = Ø25.035 and Ø25.048
  - Minimum interference = -0.014
  - Maximum interference = -0.048

generally
min shaft > max hole

always −

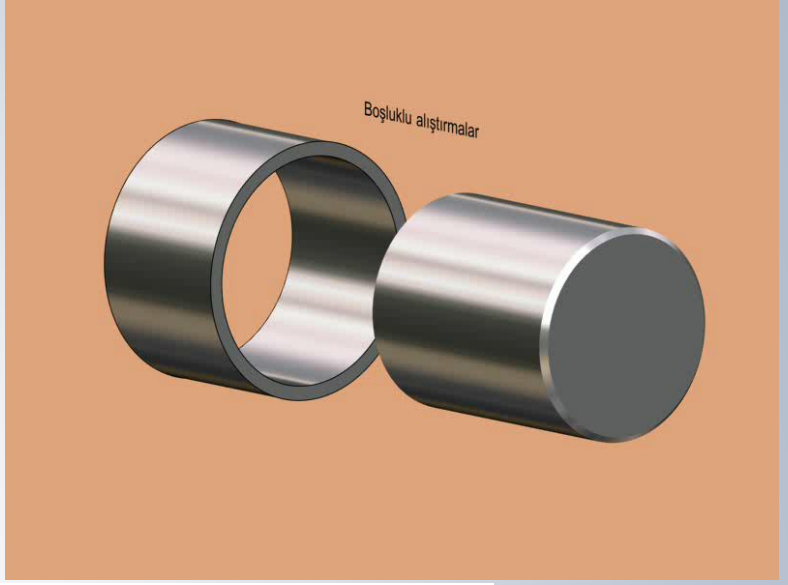
max hole min shaft

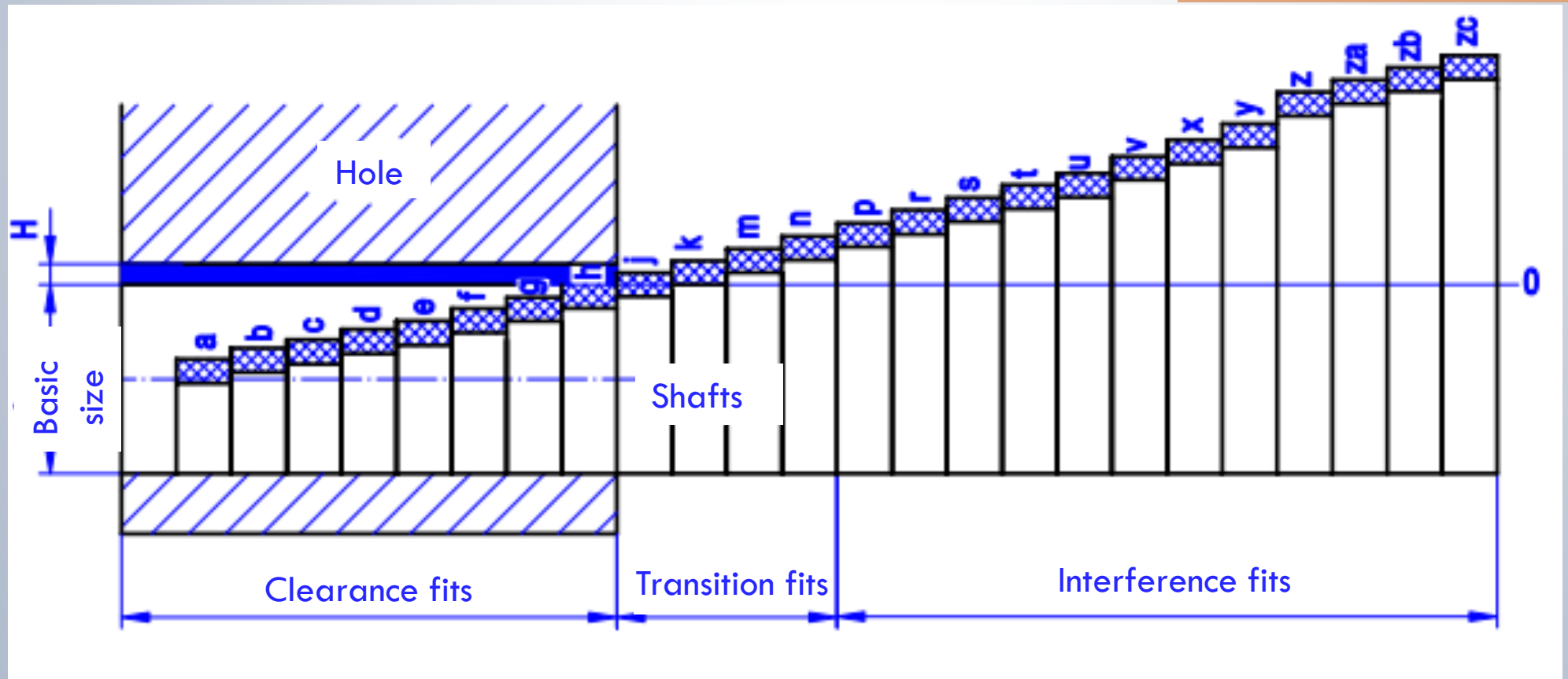
min hole max shaft

Hole - Shaft

Transition = can be both clearance / interference

HOLE BASIS FITS SYSTEM

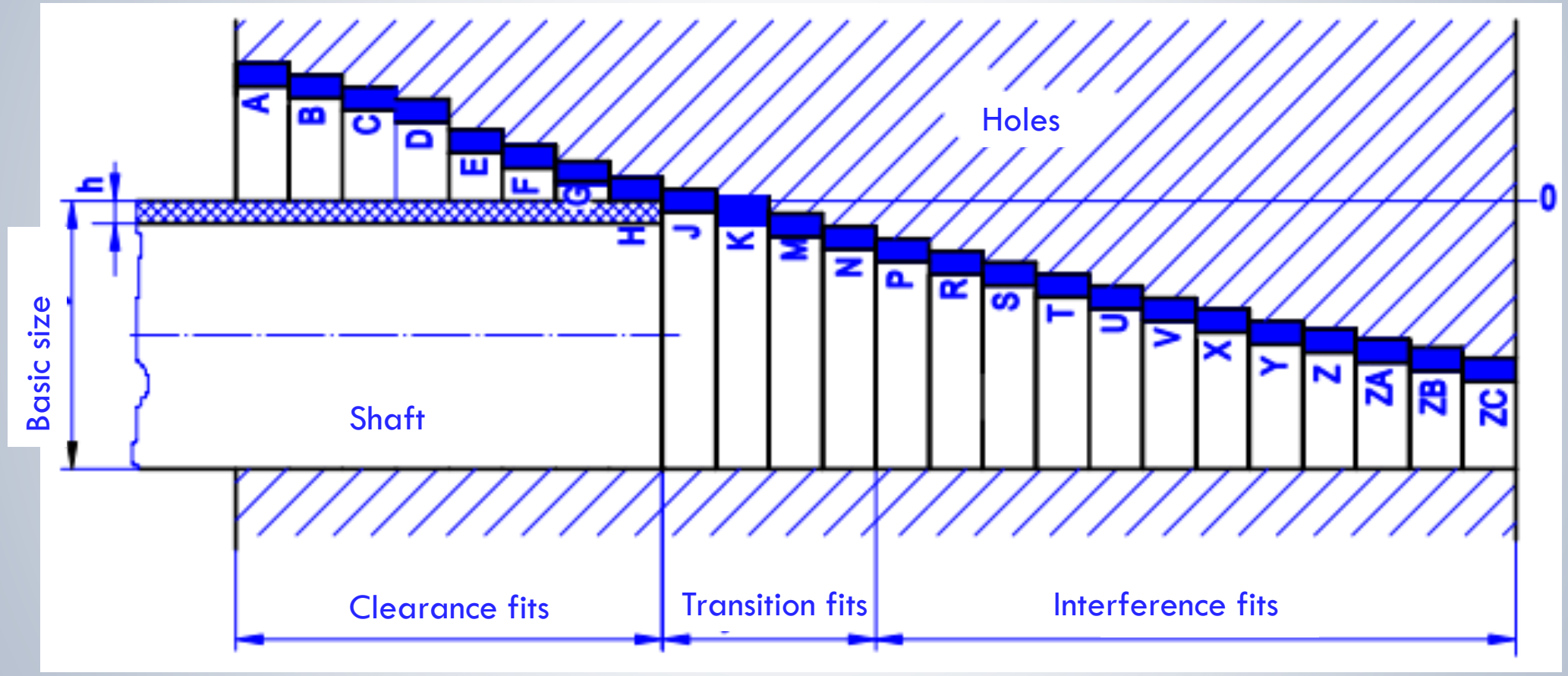




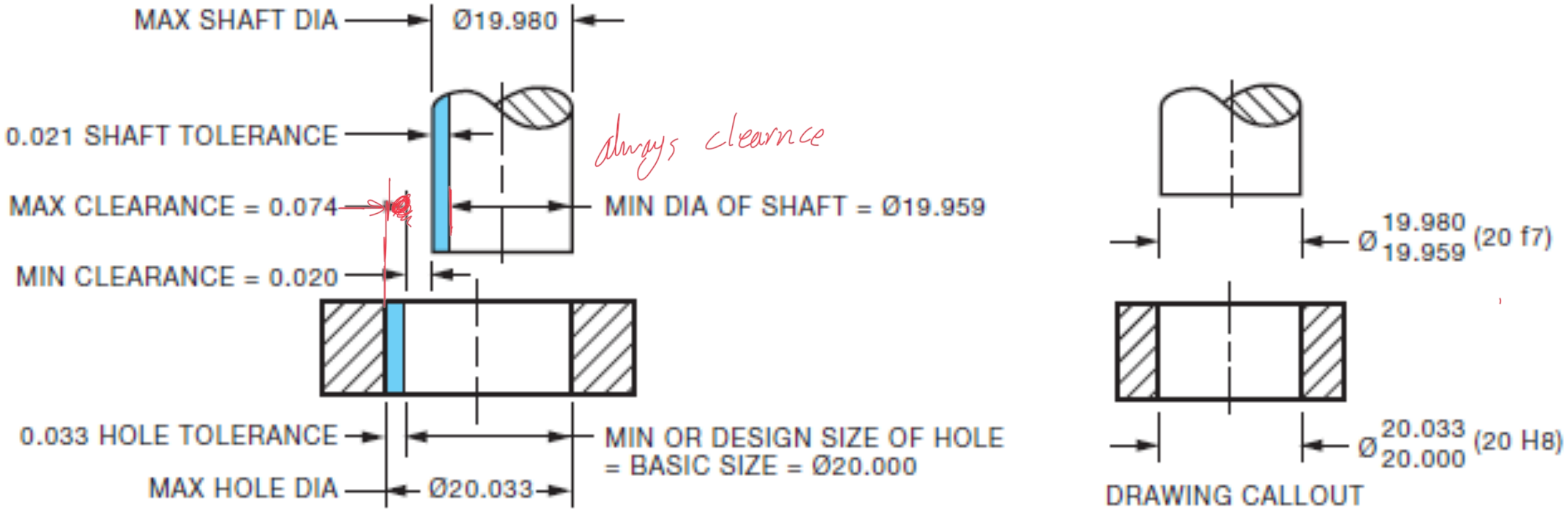
SHAFT BASIS FITS SYSTEM

- The basic size is the maximum shaft size.
- For example, for a Ø16 C11/h11 fit, which is a Preferred Shaft Basis Clearance Fit, the limits for the hole and shaft will be as follows:
  - Hole limits = Ø16.095 and Ø16.205
  - Shaft limits = Ø15.890 and Ø16.000
  - Minimum clearance = 0.095
  - Maximum clearance = 0.315

SHAFT BASIS FITS SYSTEM



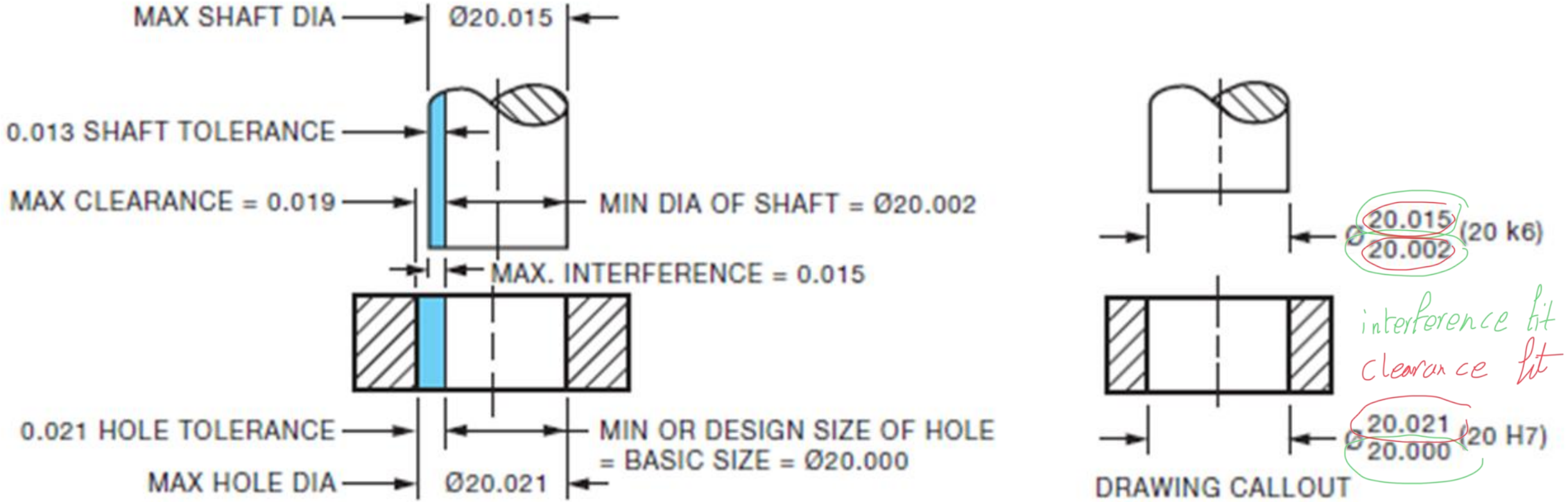
CLEARANCE FIT



EXAMPLE - H8/f7 PREFERRED HOLE BASIS FIT FOR A Ø20 HOLE (SEE APPENDIX, TABLE 24)

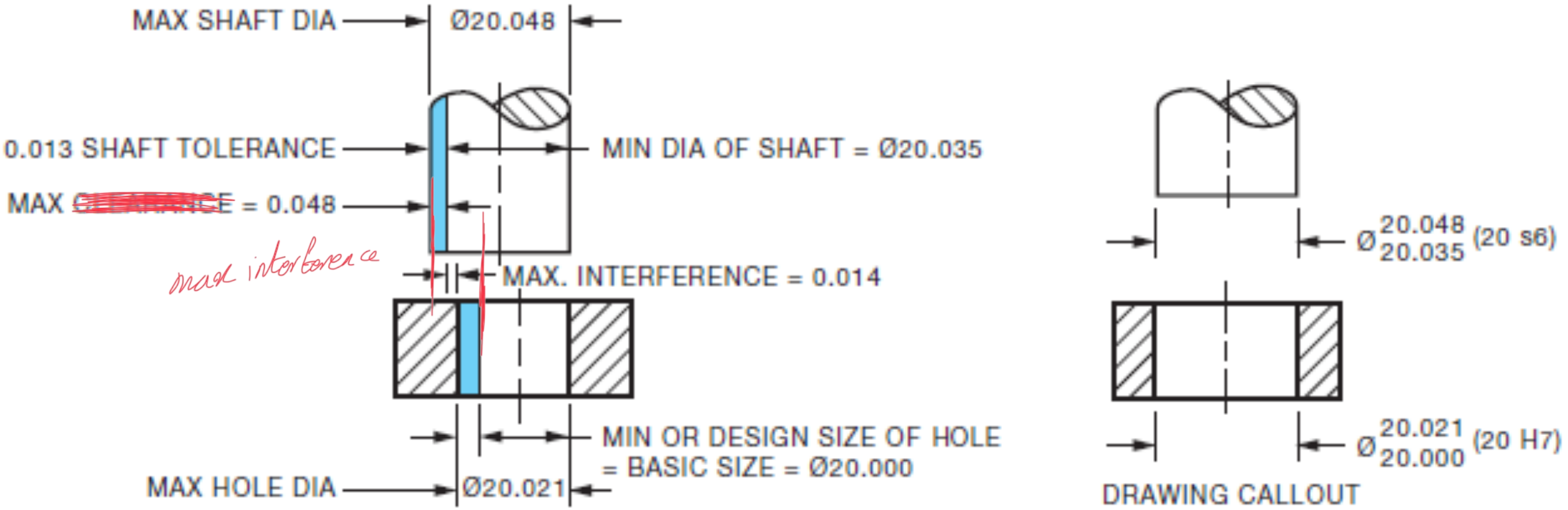
(A) CLEARANCE FIT

TRANSITION FIT



EXAMPLE - H7/k6 PREFERRED HOLE BASIS FIT FOR A Ø20 HOLE (SEE APPENDIX, TABLE 24)

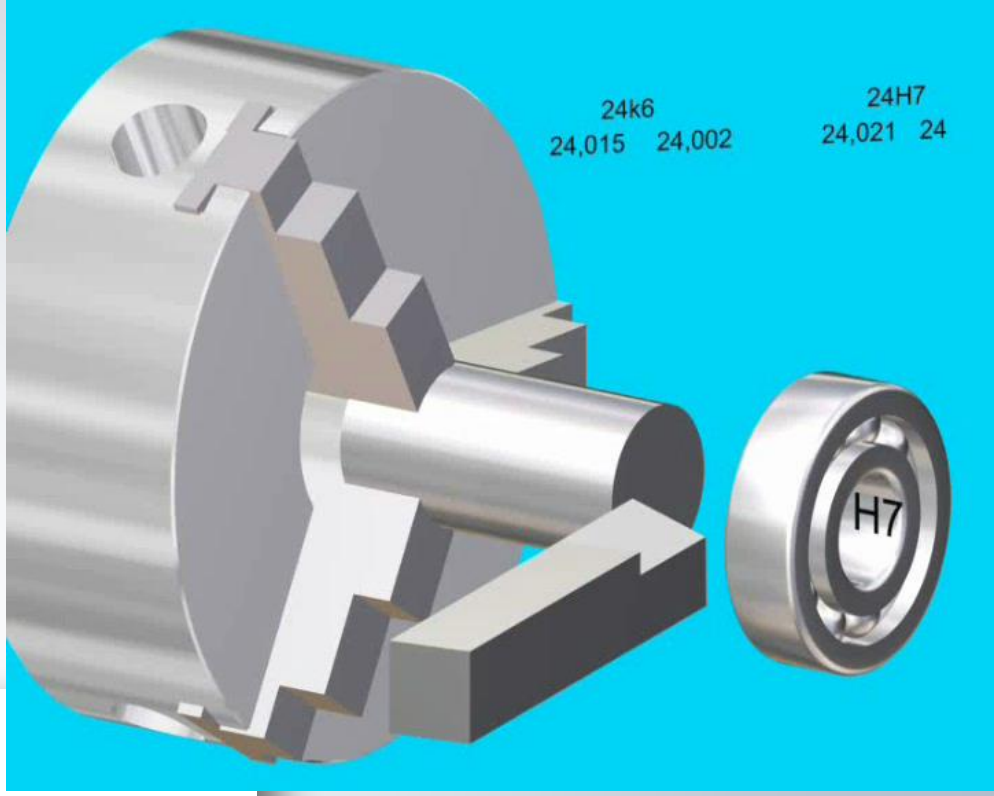
INTERFERENCE FIT

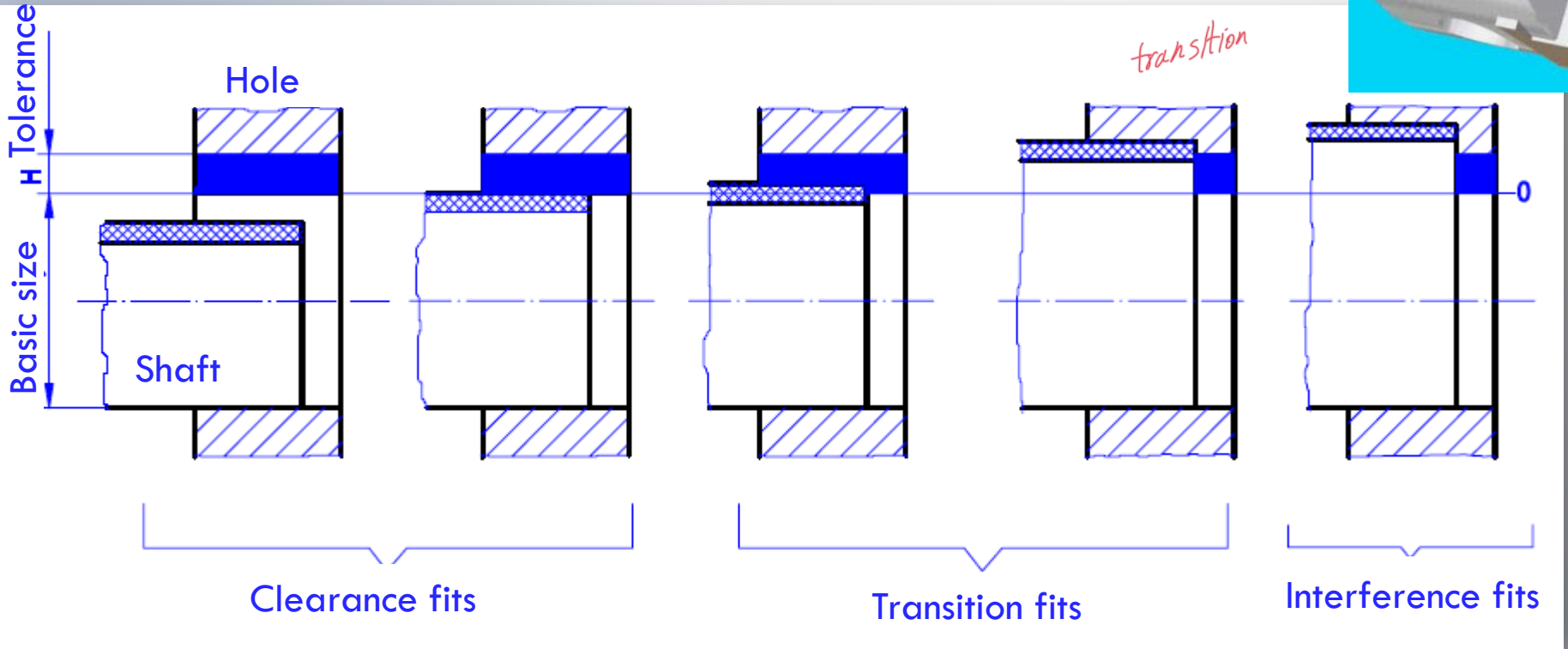


EXAMPLE - H7/s6 PREFERRED HOLE BASIS FIT FOR A Ø20 HOLE (SEE APPENDIX, TABLE 24)

FITS

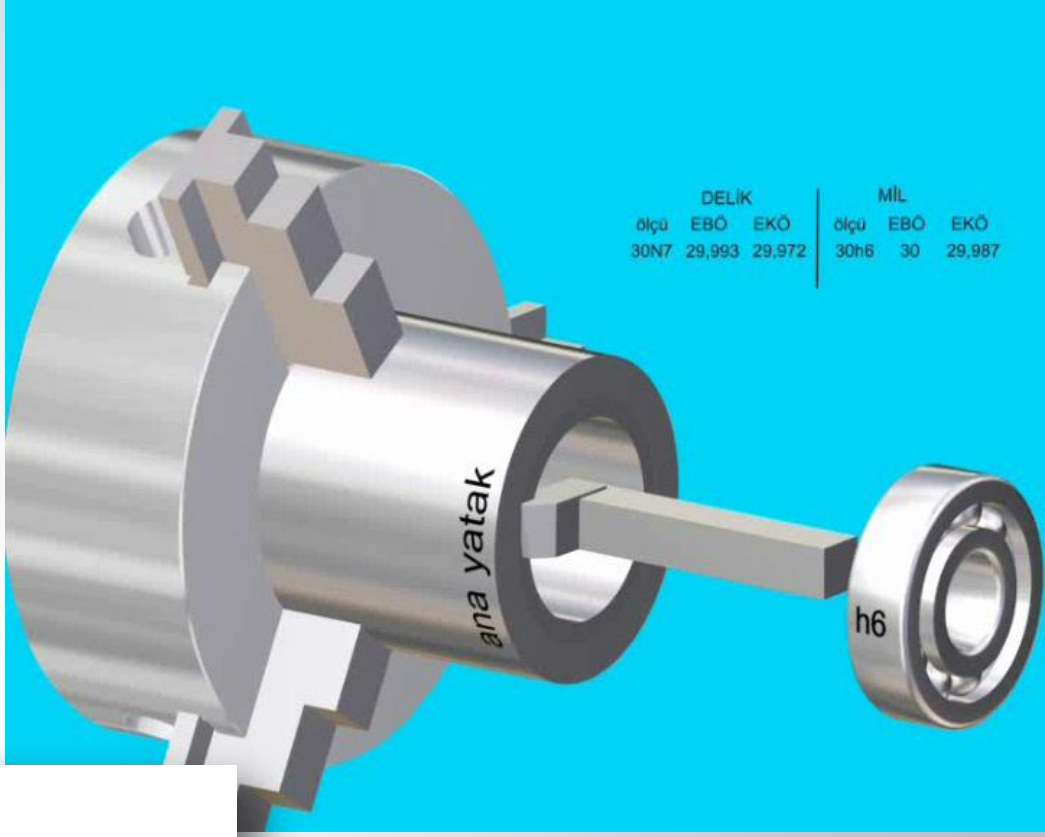
Hole bases fits with H tolerance

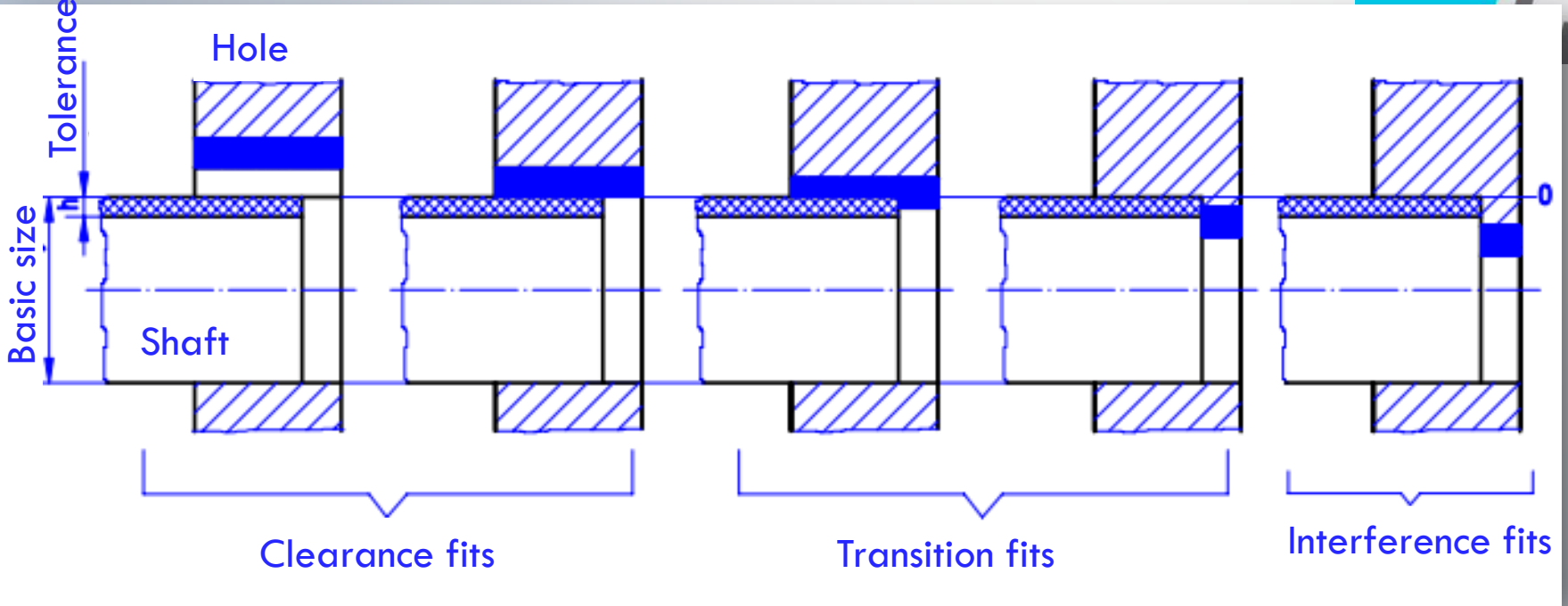




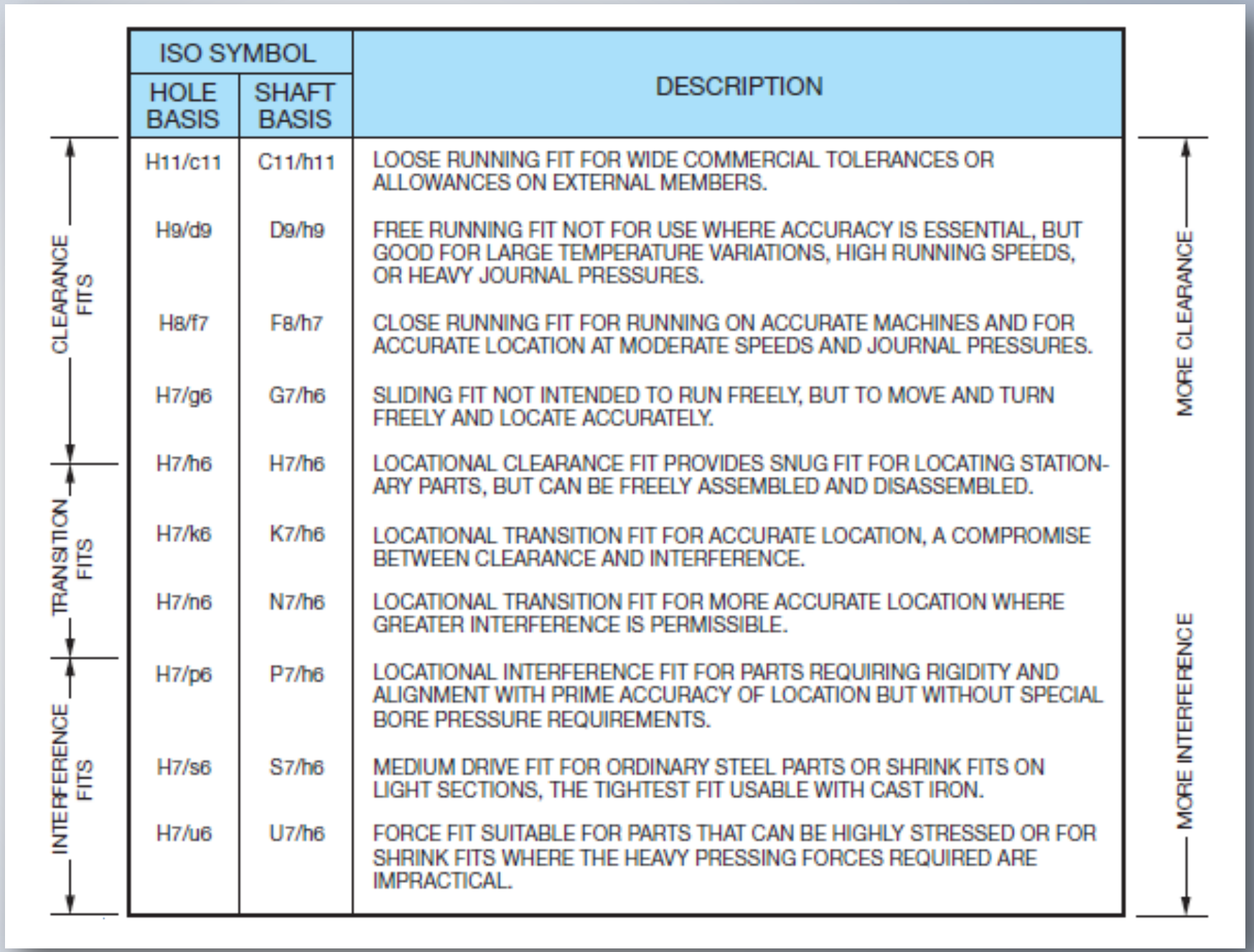
FITS

Shaft bases fits with h tolerance





PREFERRED METRIC 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 LOCATION AT 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.
TRANSITION FITS	H7/h6	H7/h6	LOCATIONAL CLEARANCE FIT PROVIDES SNUG FIT FOR LOCATING STATIONARY PARTS, BUT CAN BE FREELY ASSEMBLED AND DISASSEMBLED.
	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	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 THAT CAN BE HIGHLY STRESSED OR FOR SHRINK FITS WHERE THE HEAVY PRESSING FORCES REQUIRED ARE IMPRACTICAL.

MORE CLEARANCE

MORE INTERFERENCE

GEOMETRIC TOLERANCING

amount of allowed variation

FEATURES	TYPES OF TOLERANCE	CHARACTERISTICS	SYMBOLS	SEE UNITS
INDIVIDUAL FEATURES	FORM	STRAIGHTNESS	—	37 & 38
		FLATNESS	▱	39
		CIRCULARITY (ROUNDNESS)	○	39
		CYLINDRICITY	⌭	39
INDIVIDUAL OR RELATED FEATURES	PROFILE	PROFILE OF A LINE	⌒	44
		PROFILE OF A SURFACE	⌓	44
RELATED FEATURES	ORIENTATION	ANGULARITY	∠	41
		PERPENDICULARITY	⊥	
		PARALLELISM	//	
	LOCATION	POSITION	⌖	43
		CONCENTRICITY	◎	
		SYMMETRY	⌯	
	RUNOUT	CIRCULAR RUNOUT	↗	45
		TOTAL RUNOUT	⌰	
SUPPLEMENTARY SYMBOLS		MAXIMUM MATERIAL CONDITION	Ⓜ	38 & 43
		LEAST MATERIAL CONDITION	Ⓛ	
		PROJECTED TOLERANCE ZONE	Ⓟ	
		BASIC DIMENSION	XX	40
		DATUM FEATURE	A	40
		DATUM TARGET	Ø.50 / A2	42

* MAY BE FILLED IN

only lines

respect to where

center alignment - deviation

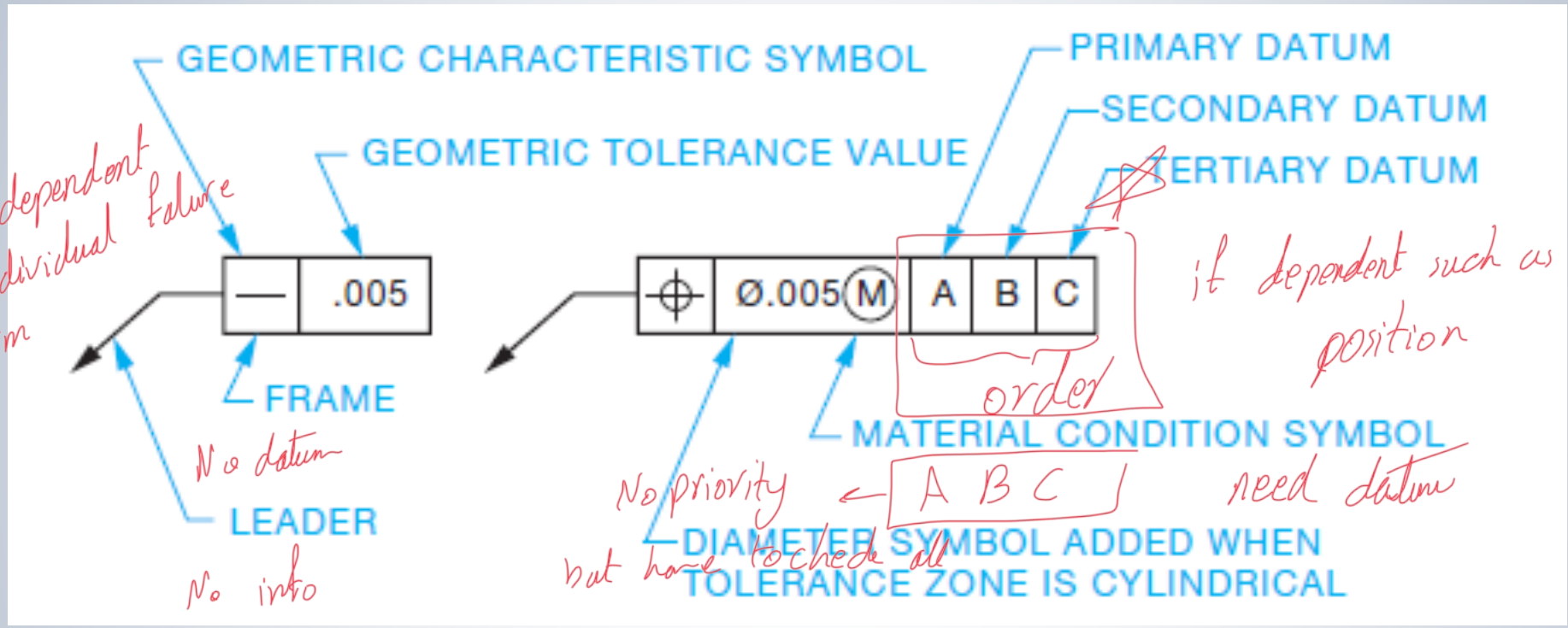
deviation of position between what must & what produced

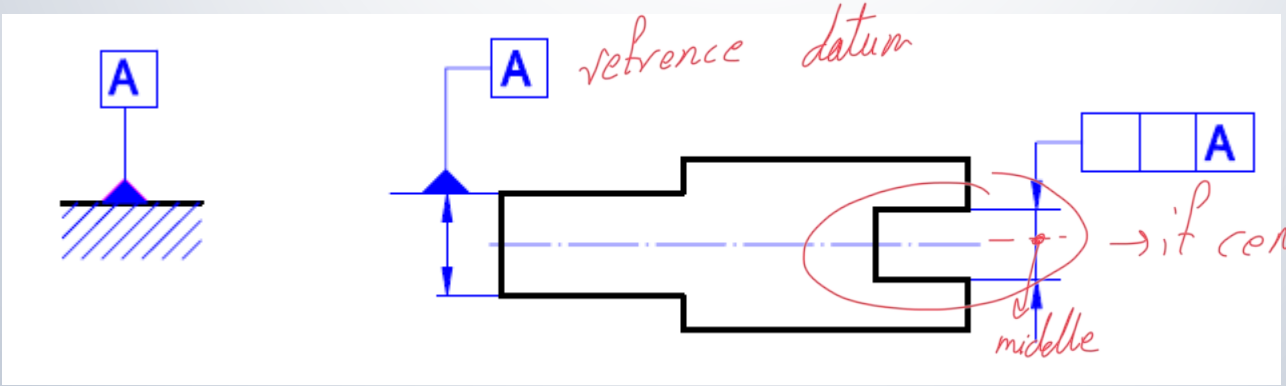
on only one side

full surface

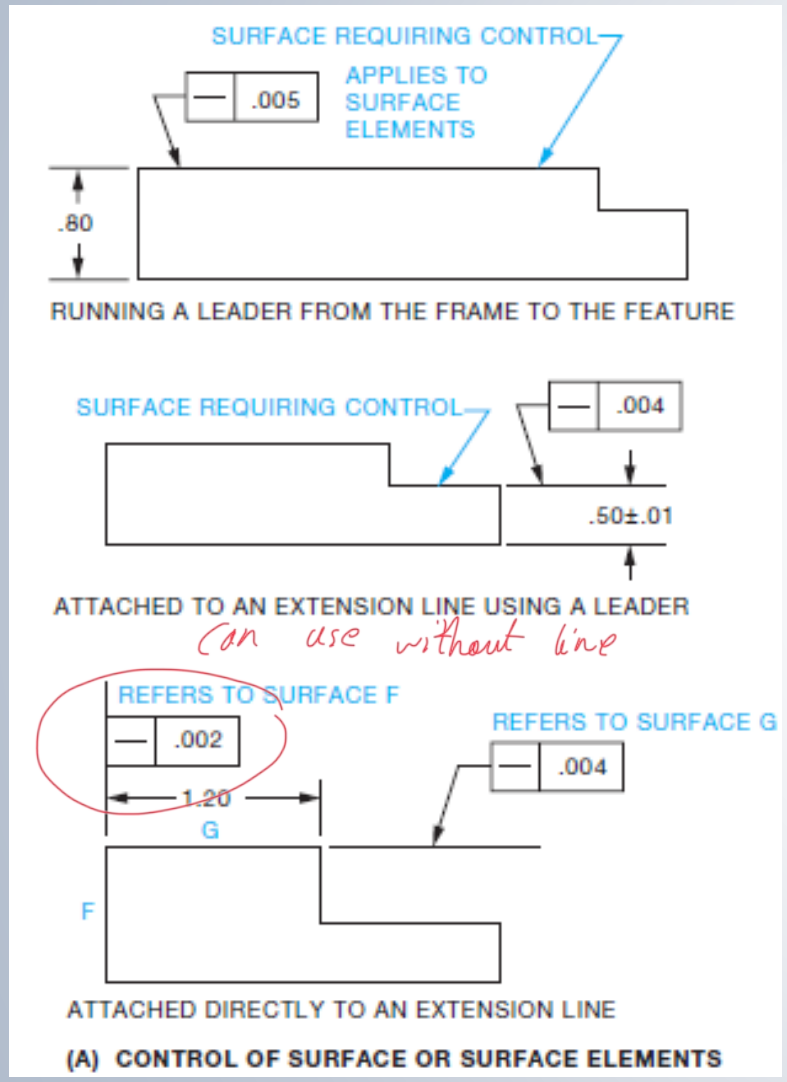
rotation around an axis

FEATURE CONTROL FRAME





FORM TOLERANCE



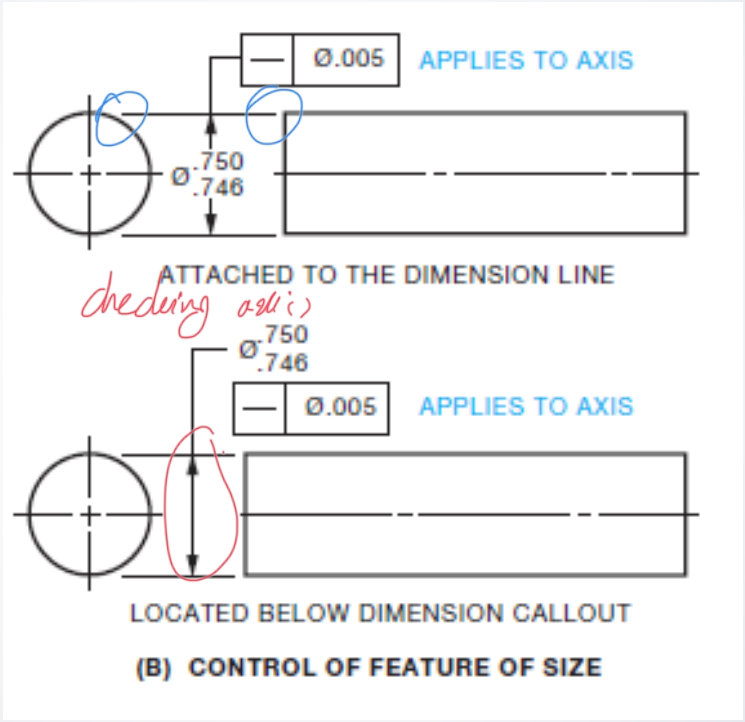


FIGURE 37–7 Preferred location of feature control frame when referring to a surface.

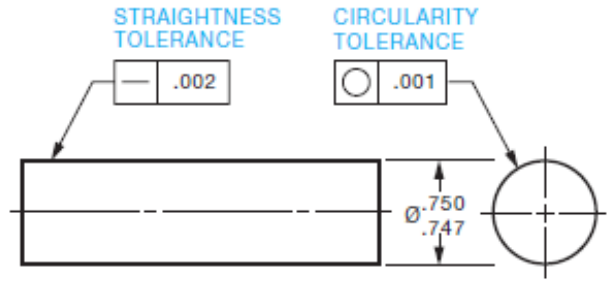
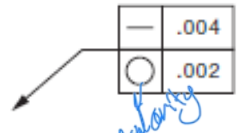
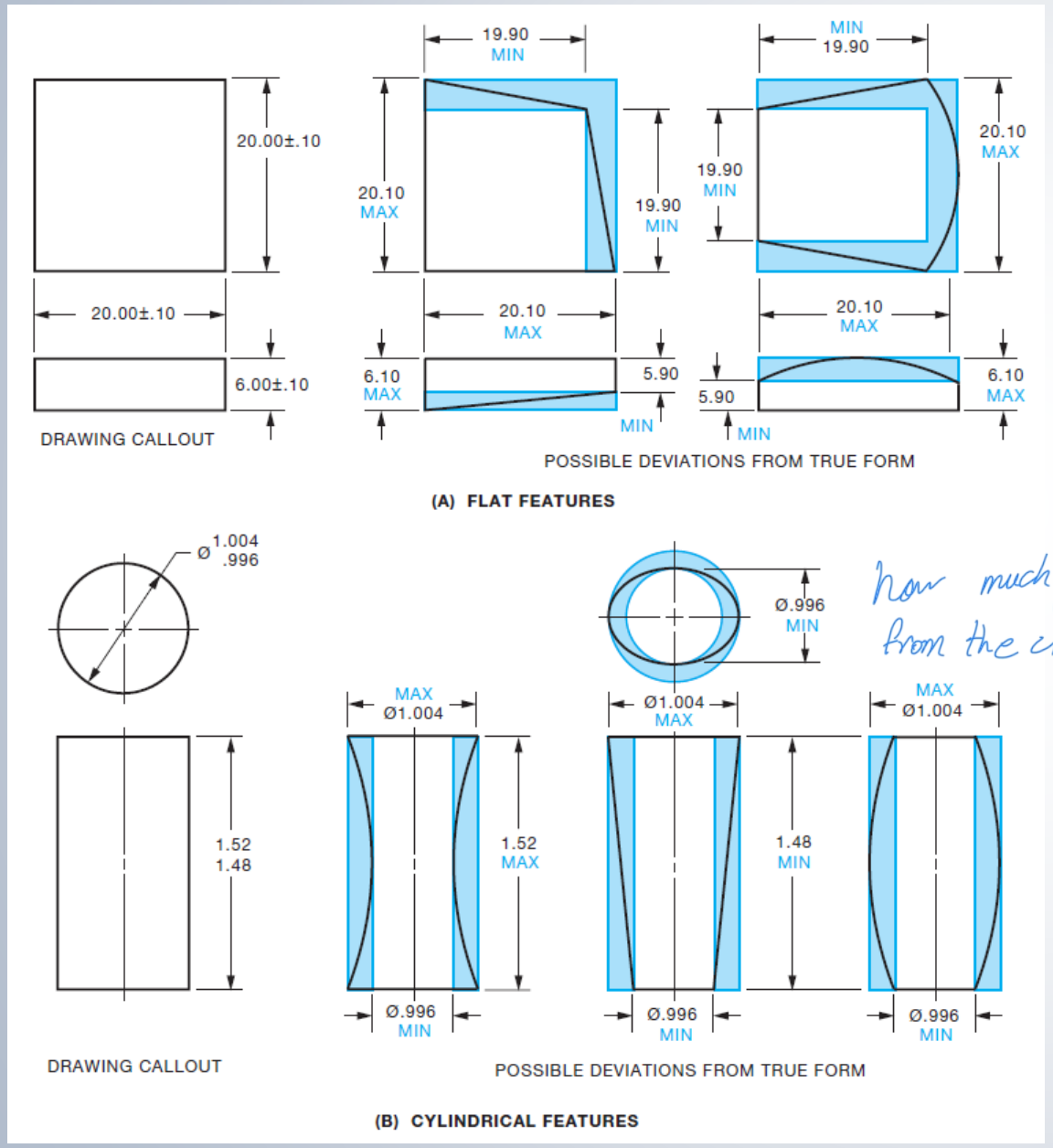


FIGURE 37–8 Combined feature control frames directed to one surface.

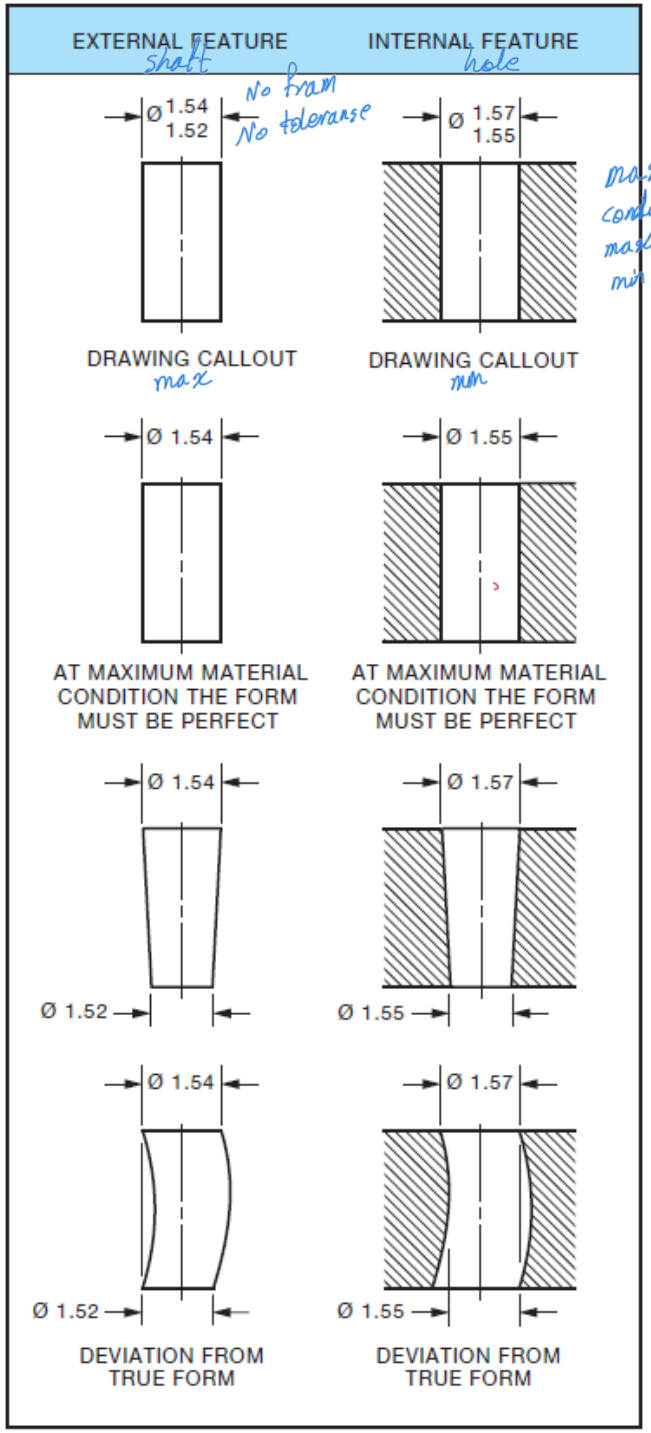


FORM TOLERANCE



max material condition
max shaft
min hole

how much you can deviate from the circle

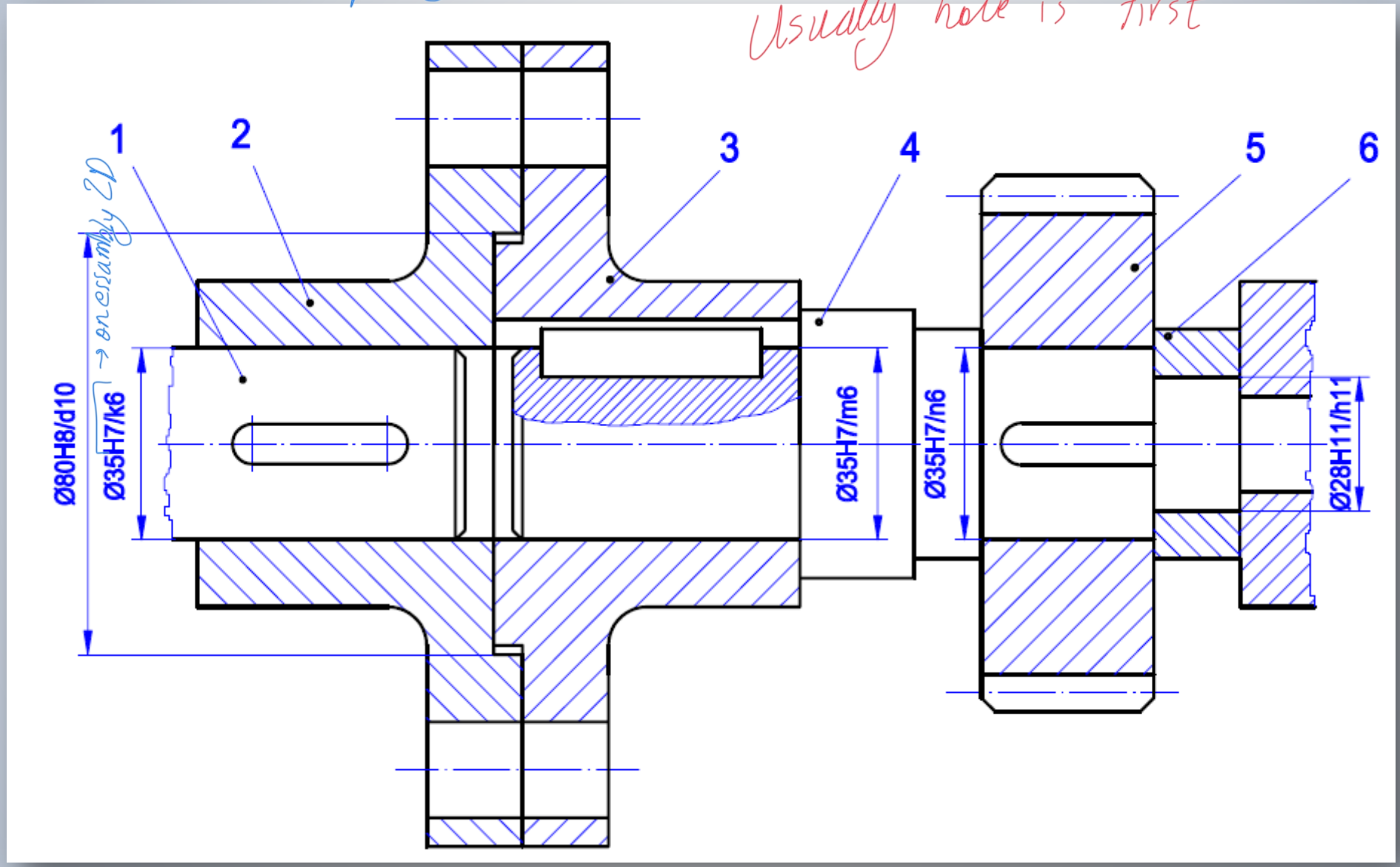


max material condition
max shaft
min hole

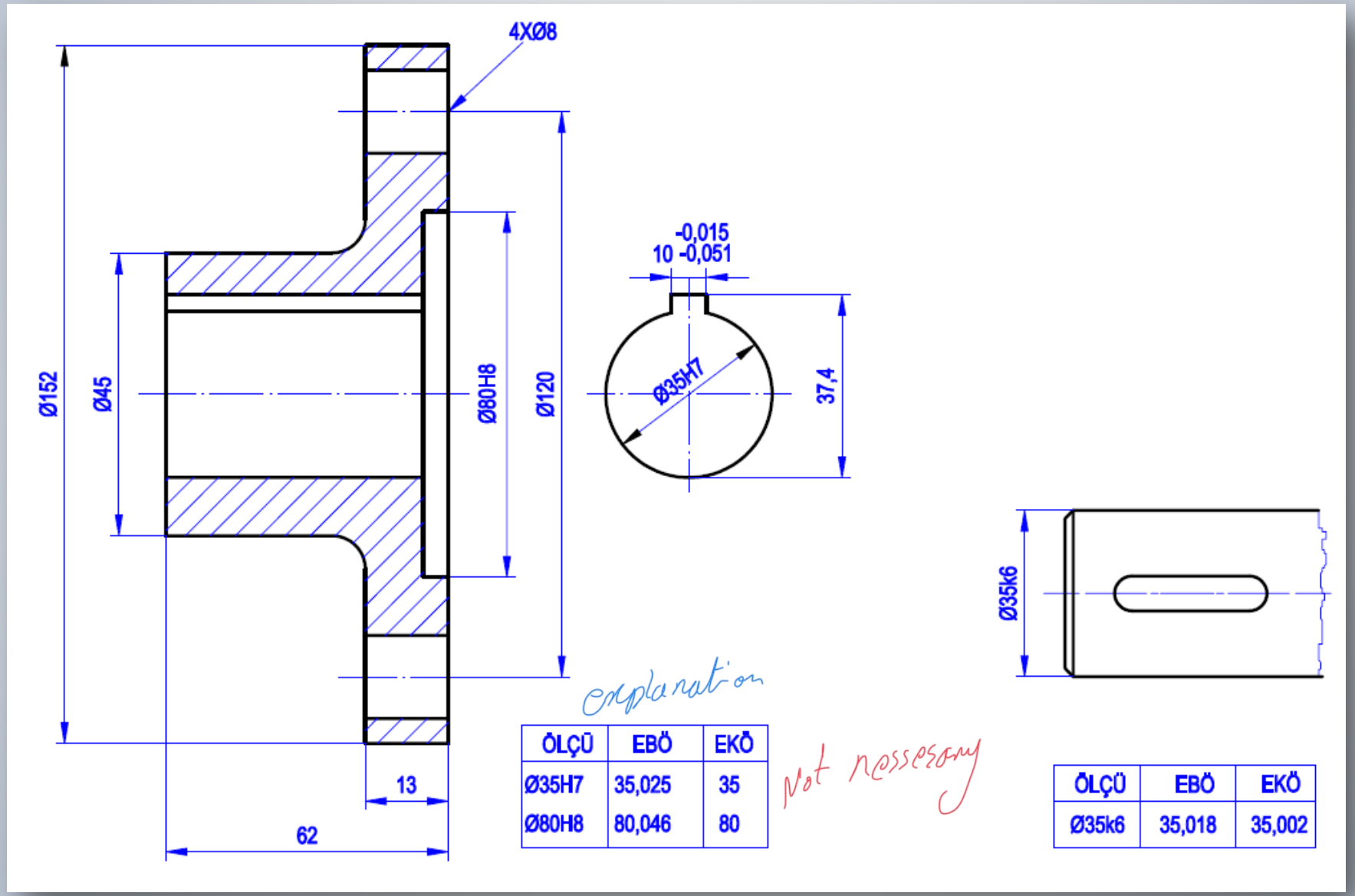
EXAMPLES

hole bases

Usually hole is first



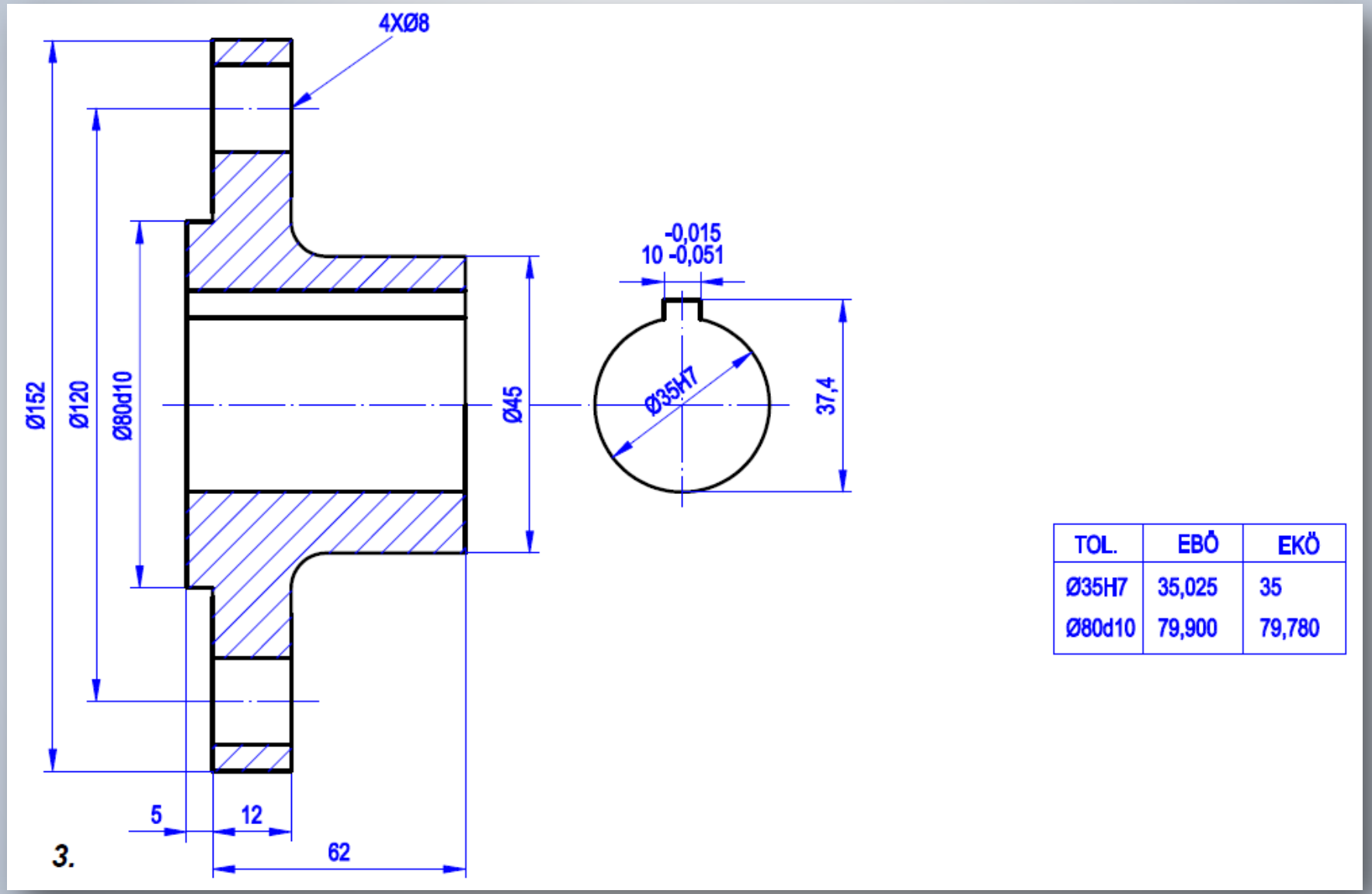
EXAMPLES



ÖLÇÜ	EBÖ	EKÖ
Ø35H7	35,025	35
Ø80H8	80,046	80

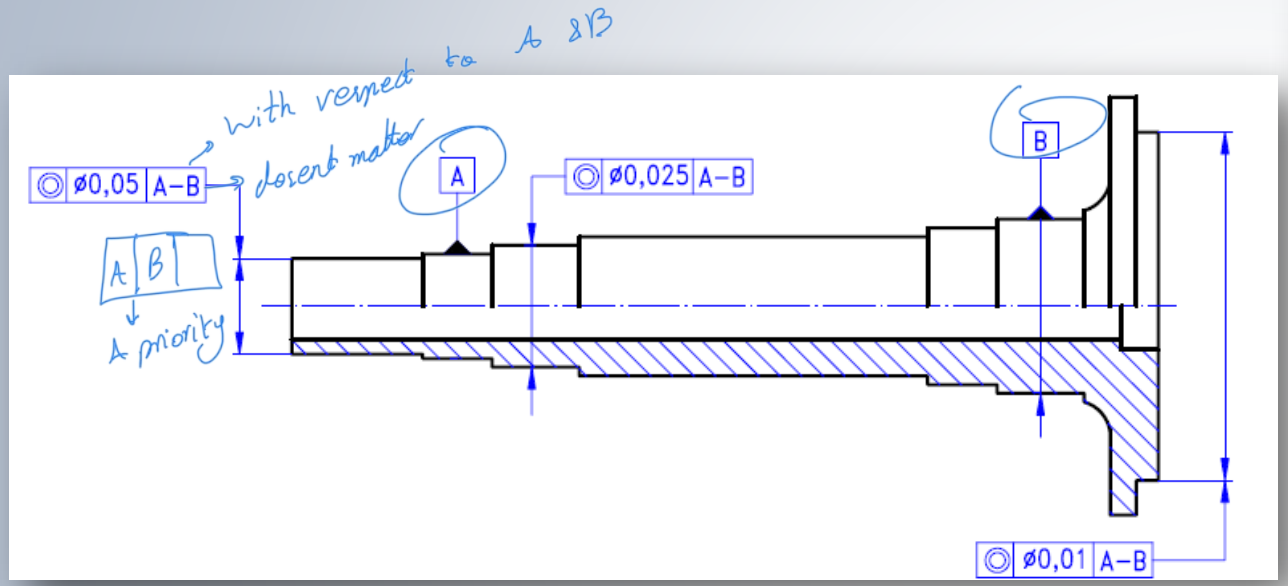
ÖLÇÜ	EBÖ	EKÖ
Ø35k6	35,018	35,002

EXAMPLES



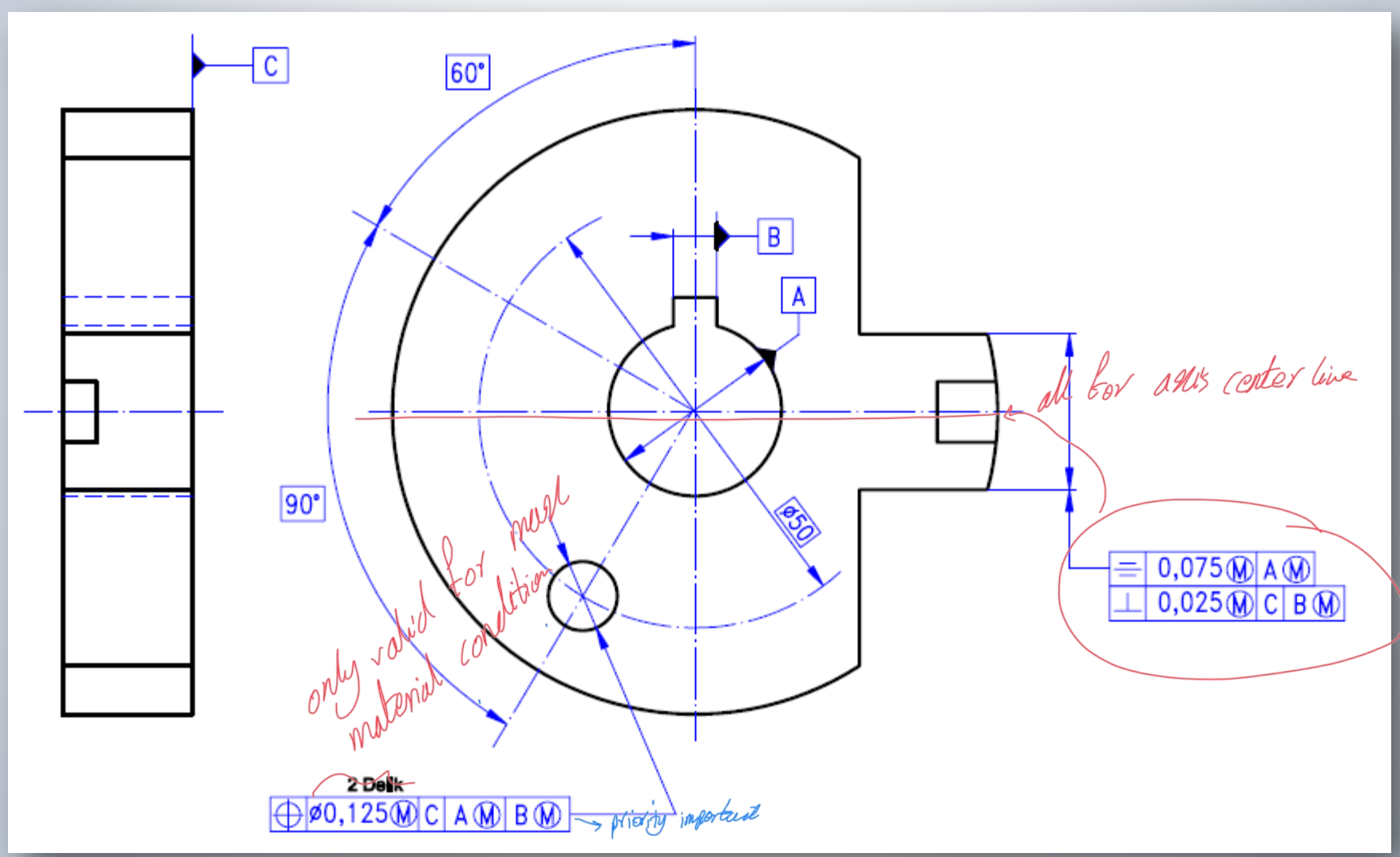
TOL.	EBÖ	EKÖ
Ø35H7	35,025	35
Ø80d10	79,900	79,780

EXAMPLES


with respect to A & B
⌀0,05 A–B
doesn't matter
A
⌀0,025 A–B
B
A B
A priority
⌀0,01 A–B

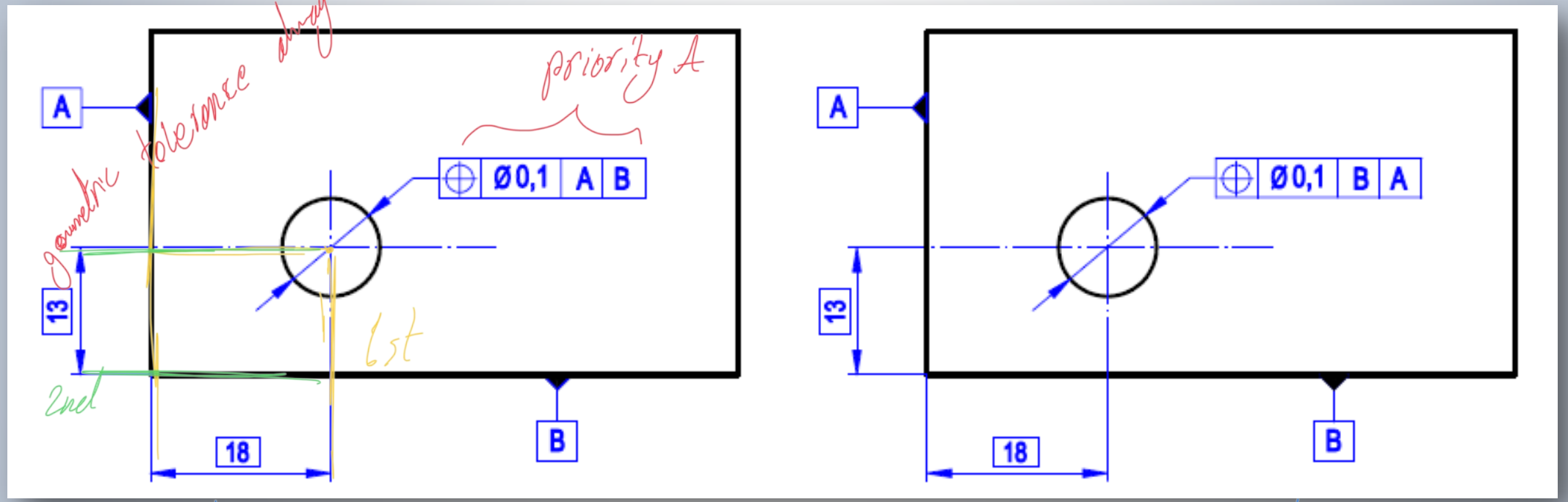
EXAMPLES

Material condition and datum priority



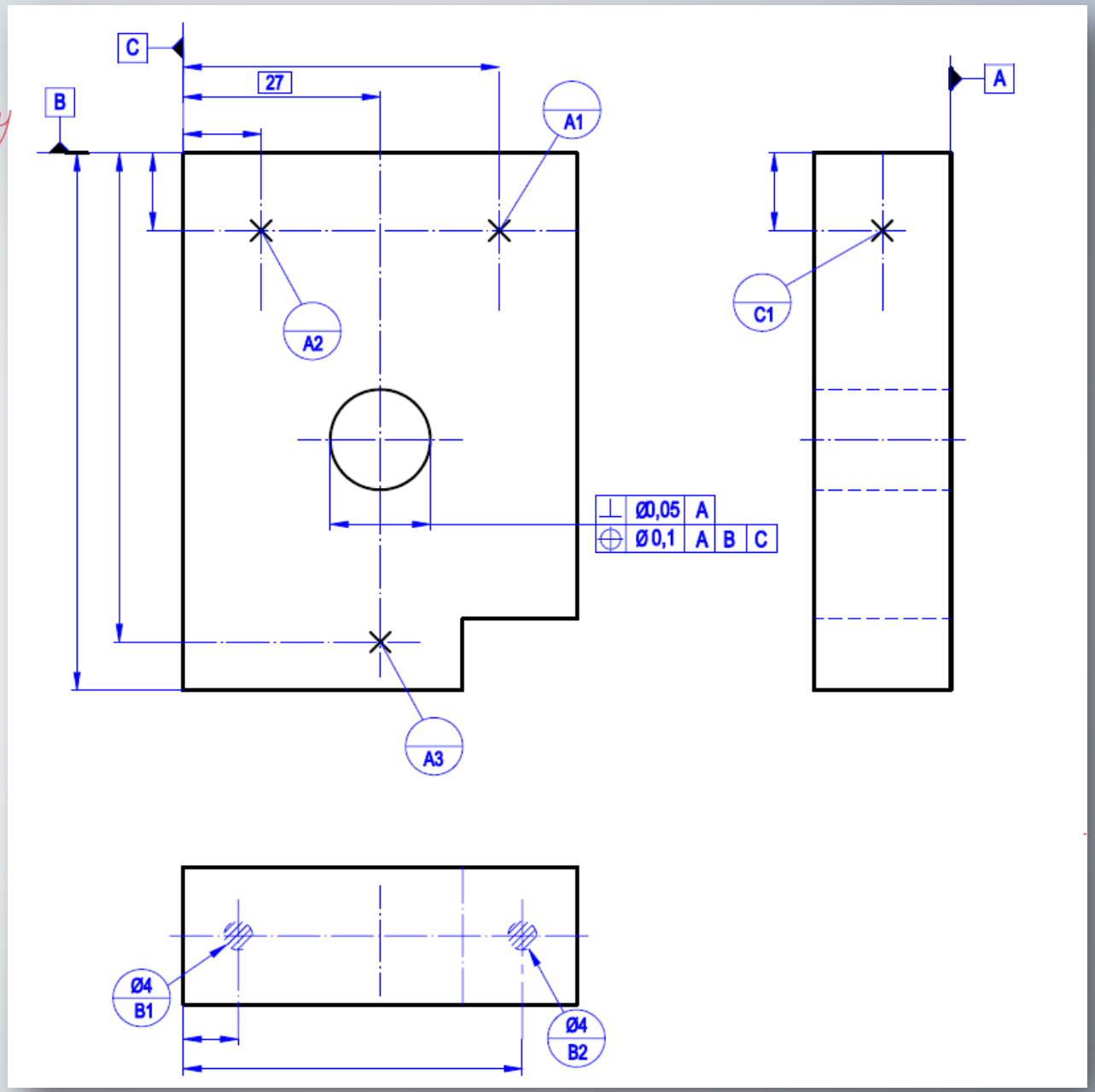
EXAMPLES

Datum priority

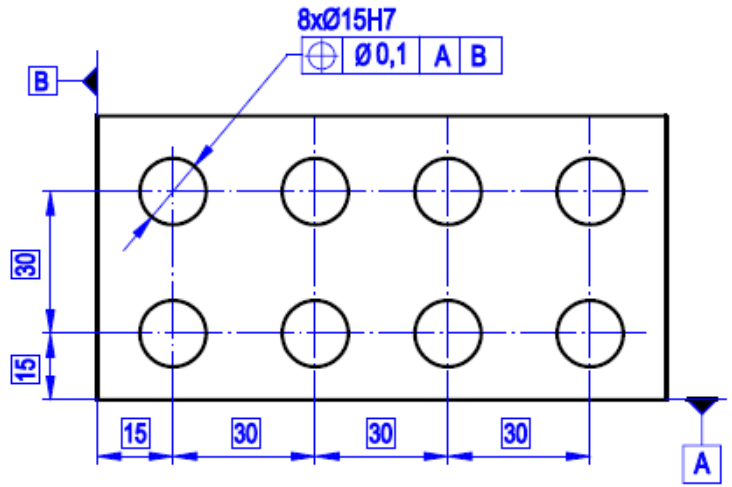


EXAMPLES

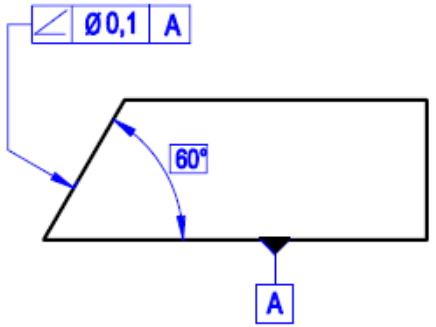
geometric tolerance = messured by Gauge



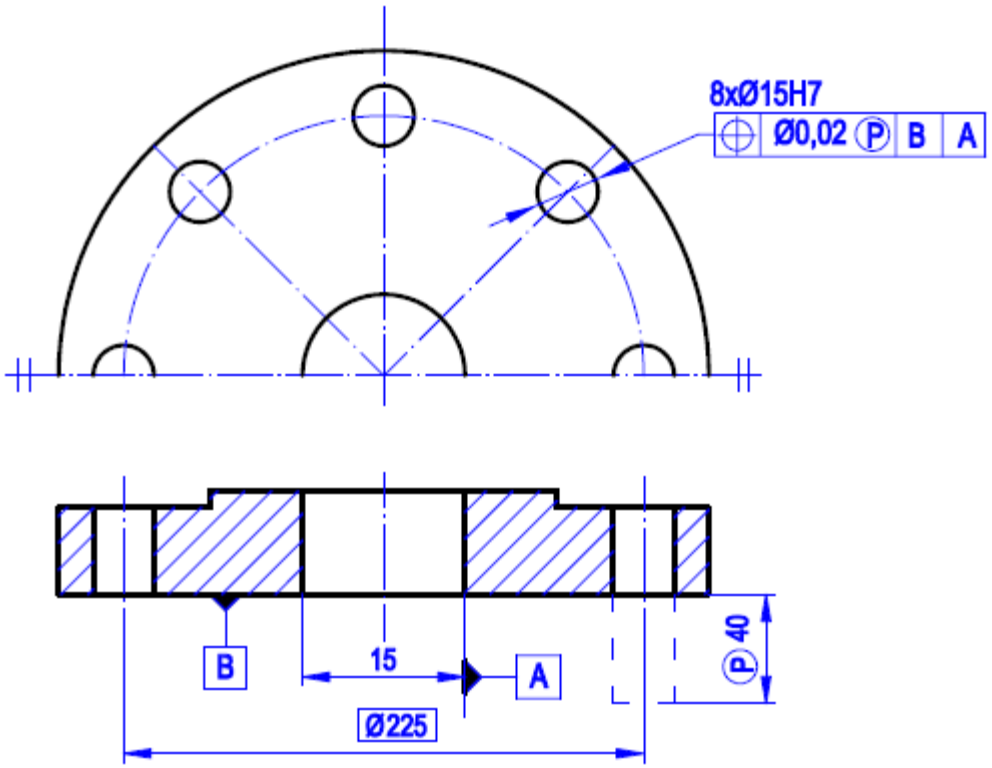
EXAMPLES



Position tolerance



Angularity



Projected tolerance

REFERENCES

The content of this presentation has been compiled from the following sources

1. Branoff, T. 2015. Interpreting engineering drawings. Cengage Learning.
2. Türkdemir, K. 2008. Teknik Resim I. Denizli: Boy Yayınları.