School of Mechanical and Manufacturing Engineering

MMAN2130 Design and Manufacturing

Term 3 – 2019

Week 4

Limits, Fits and Tolerances

Limits, fits and tolerances

Today's Lecture

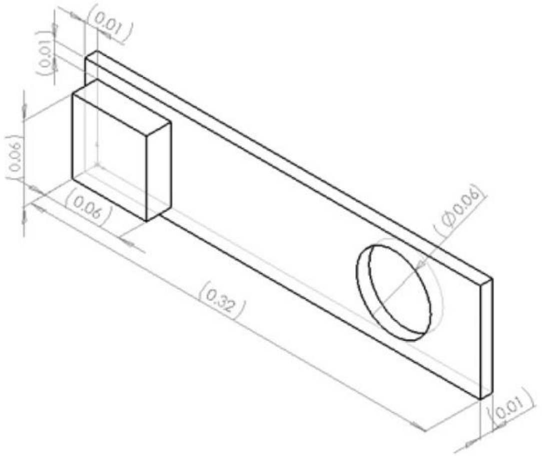
- Errors in manufacturing
- Why tolerancing?
- Limits and fits
- Symbols and drawings

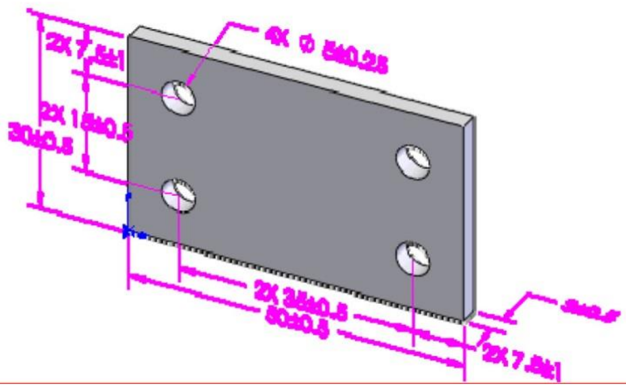
Definitions

Dimension: Numerical value that defines the size, location, orientation, form or other geometric location.

Tolerance: The total amount a specific dimension is permitted to vary. The tolerance is the difference between the max and min limits.

Limits: The maximum and minimum value the specific dimension can be.





Dimensions

- CLEAR COMMUNICATION is the primary purpose of dimensions.
- Every dimension must have an associated tolerance.
- Unless otherwise specified, angles shown on drawings are assumed to be 90 degrees.
- Avoid over dimensioning a part!
- Dimensioning to hidden/construction lines should be avoided whenever possible. Hidden lines are less clear than visible lines.
- The depth of blind, counterbored, or countersunk holes can be specified in a note along with the diameter.
- Dimensions should be placed in a view that most clearly describes the feature being dimensioned.

Why Tolerances?

Errors in manufacturing

- Stock material has errors
- Manufacturing will introduce errors
- Errors can be size/magnitude, geometry or finish.

In your engineering drawing;

- Include allowable errors as tolerances.
- Show the preferred manufacturing process.
- Show the finish required.

Why Tolerances?

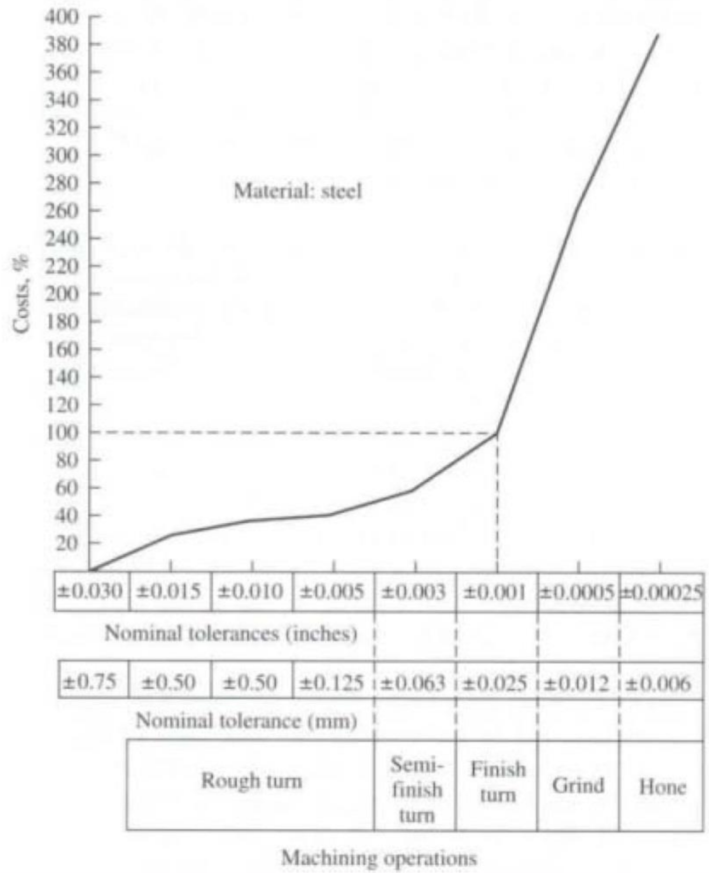
How can you determine the likely error in the manufacturing process?

- Get advice from workshop staff, TAFE tutors etc.
- Use established formulae and standards to calculate the error.

Golden Rule:

"Always specify as large tolerance as possible without sacrificing the quality."

Tight Tolerances Increase Cost



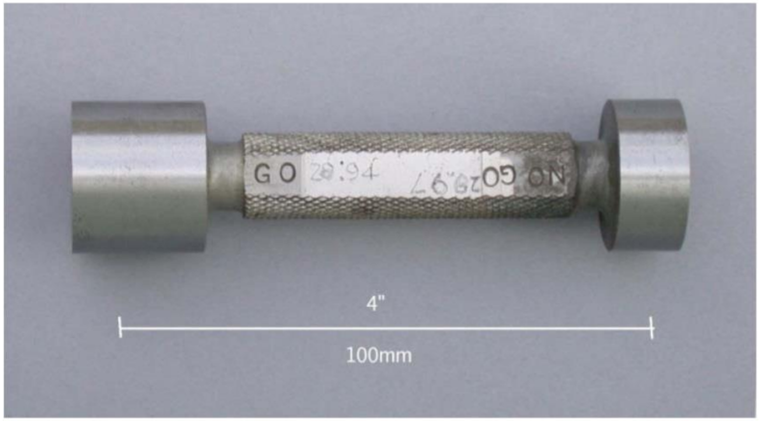
Why Tolerances?

- Tolerancing saves money
- Ensures interchangeability of mating parts
- Maximises quality
- Uniform interpretation of specification
  - No guess work for the manufacturer
  - Clear who is at fault if there is an issue

Limit Gauges

- Gauging is the process of determining if a part is within the required tolerance
- Limit gauges measure a single dimension qualitatively
- Purpose built or generic
- Types:
  - Go/NoGo
  - Fixed limit gauge
  - Indicating limit gauge (Micrometer, Vernier Caliper)

Go/NoGo: Plug Gauge

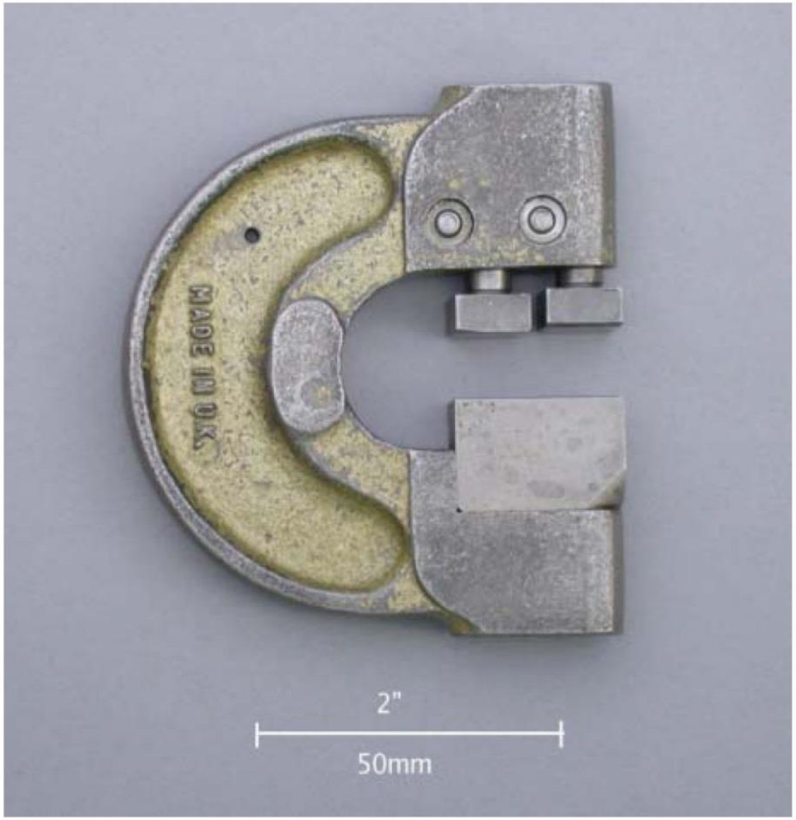


Does not return a size, but returns a "state"

Name derived from two tests

1) Must **pass** one test (GO)
2) Must **fail** another test (NO GO)

Go/NoGo: Snap Gauge



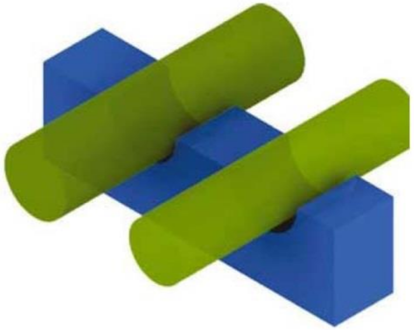
First snap guage invented in 1943 (WW2) to speed the production of parts.

Indicating Limit Gauge

Three Classes of Tolerances

1. Size Tolerances
   - Concerned with the sizing of dimensions
2. Geometric Tolerances
   - Concerned with the geometrical shape or form
3. Surface Tolerances
   - Concerned with the surface finish
   - A generic value will suffice for your project
   - Check with TAFE for what is achievable

General Definitions



- **Shaft**
  - A member fits into another member
  - May be stationary or rotating
  - A member which has to fit into a space between two restrictions
- **Hole**
  - A member which houses or fits the shaft
  - May be stationary or rotating
  - Space between two restriction into which a member has to fit

General Definitions

- Nominal Size
  - A size an item is designated as a matter of convenience
  - M20 screws tread, 75X10 flat bar stock
  - Always in mm and expressed to the nearest whole

- Basic Size
  - A size from which the limits of size are derived by applying upper and lower deviations
  - Same for shaft and hole
  - *Usually same as nominal size*

General Definitions

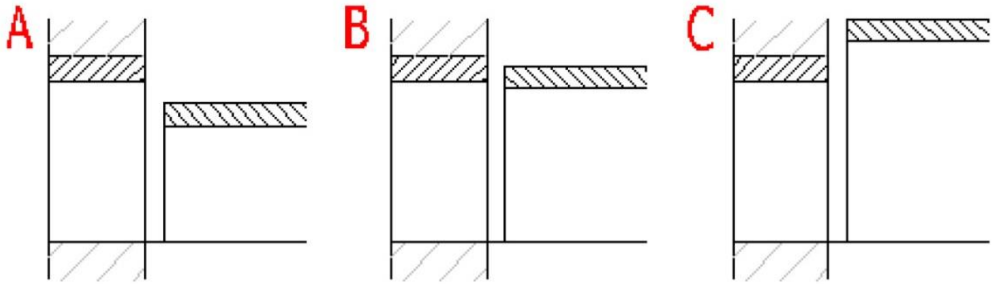
- Limits of Size
  - Extreme of sizes allowed for a toleranced dimension
  - Maximum Material Limit (MML)
    - The maximum (upper) limit of size for an external feature (shaft)
    - The minimum (lower) limit of size for an internal feature (hole)
  - Least Material Limit (LML)
    - The minimum (lower) limit of size for an external feature (shaft)
    - The maximum (upper) limit of size for an internal feature (hole)

General Definitions

- Deviations
  - The difference between the basic size and the actual size
  - The extreme deviations referred to as *upper and lower deviations*
- Tolerance
  - The difference between the maximum and minimum limits of size
  - The difference between the upper and lower deviation

General Definitions

A relative motion between a shaft and a hole resulting from the final sizes after manufacture.



A) Clearance fit: Always fits under all conditions.

- When you want something to rotate or slide freely! Axle running on bearings, bullet in chamber.

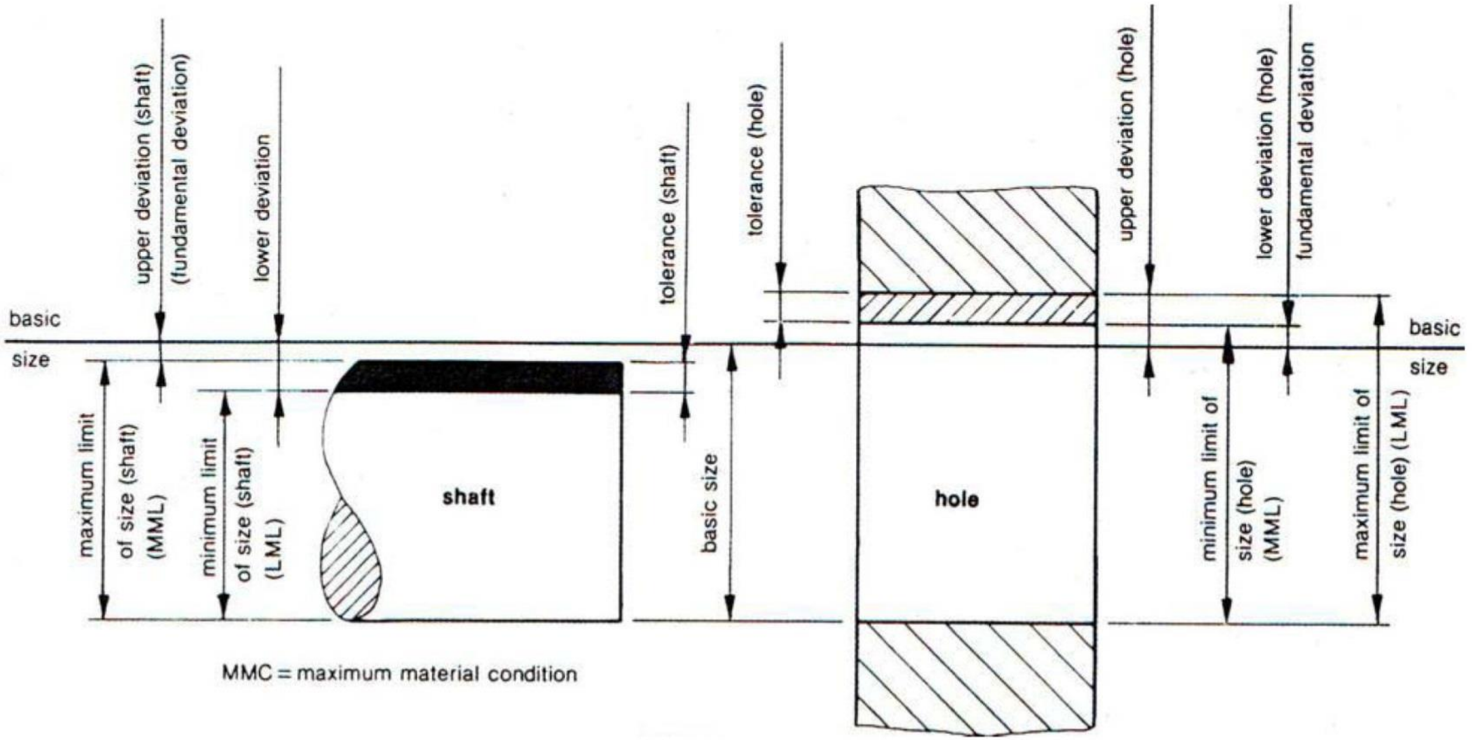
B) Transition fit: Fits under some conditions (there is an overlap).

- When you want a semi permanent fit (grub screws to connect motor shaft etc)

C) Interference fit: Never fits!

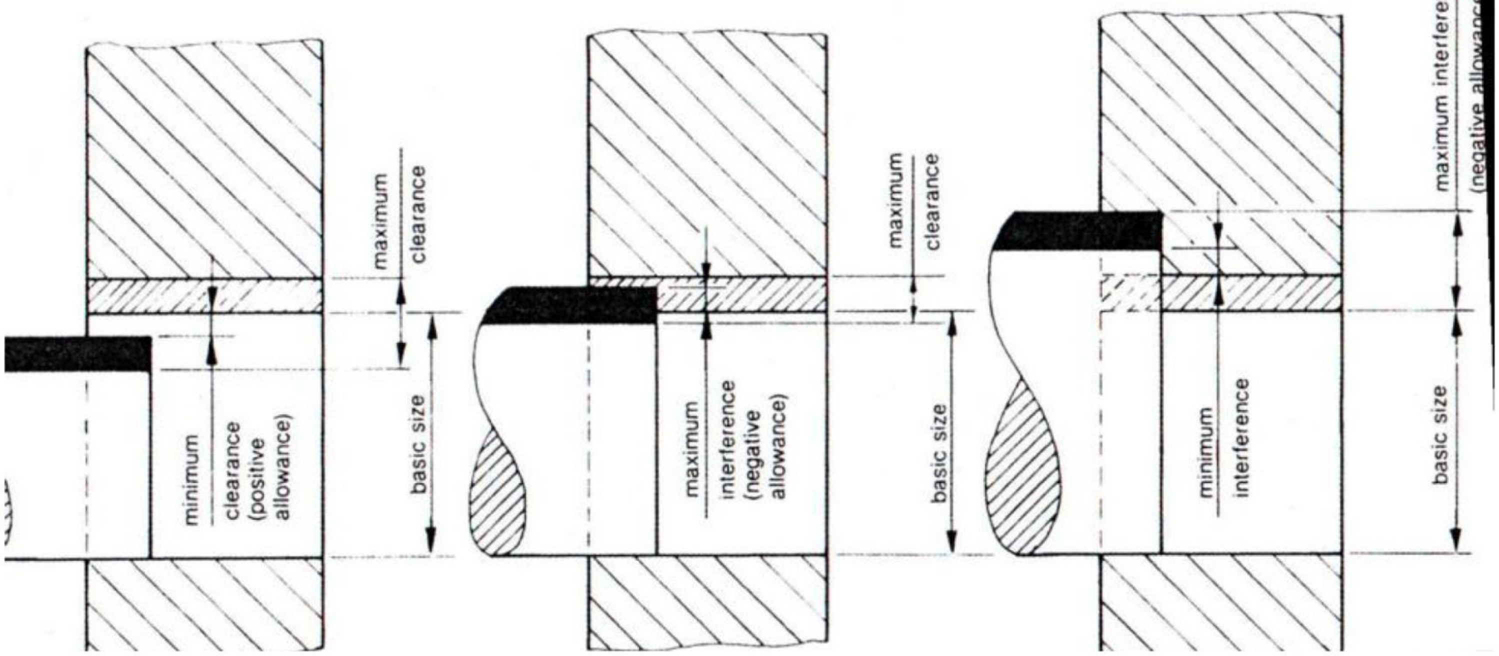
- We make it fit! By smashing them together both parts actually elastically deform a bit! The two components are then held together with extremely high friction forces. These are called friction fits or press fits and used for mating gears to shafts, bearings into housings, wheels on axles.

Designation of Shaft and Hole Sizes and Limits



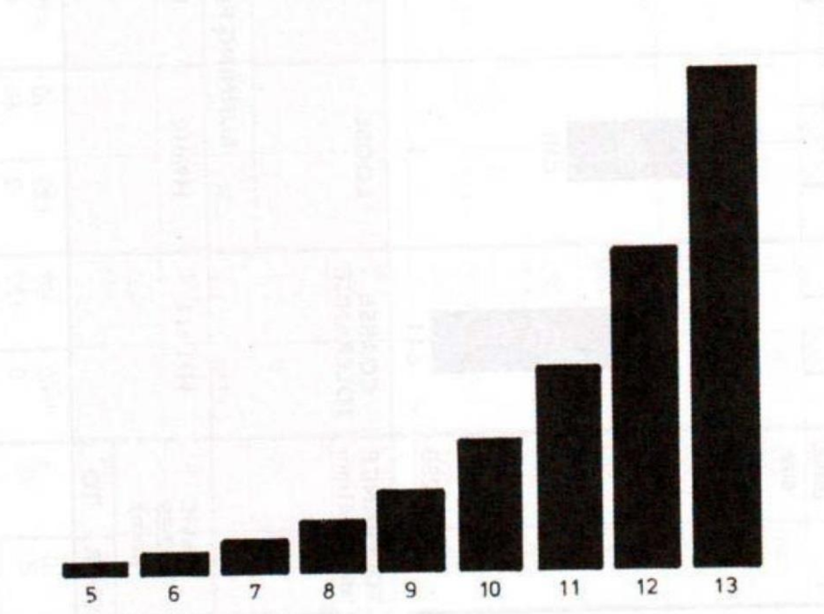
Designation of Shaft and Hole Sizes and Limits

- Clearance fit
- Transition fit
- Interference fit



Grades of Tolerances (ISO System)

- 18 grades,
  - Very fine –for lower numbers
  - Extremely coarse –for larger number
- Each grade is approximately 1.6 times as great as the grade below or finer than it



Deviations of Tolerances

- Deviation which closest to the basic size used to locate the tolerance zone wrt the basic size
- Hole-basis (tables 4.1a & b)
    - Hole is standard with zero deviation
        - is commonly used
        - easier to produce standard holes
        - easier to turn the shaft to suit
- Shaft-basis
    - Shaft is standard with zero deviation
    - desirable in some cases when a driving shaft has to have a number of different parts fitted to it

Deviations of Tolerances

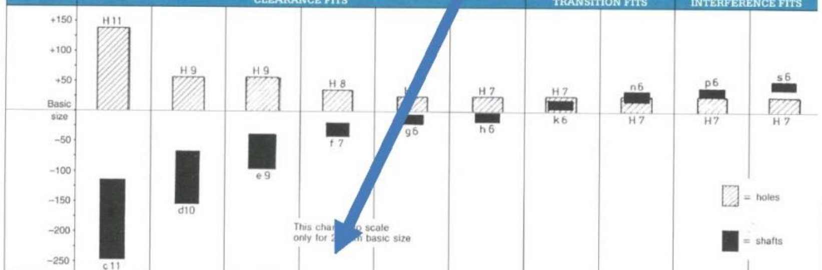
Clearance fit (Normal Running) H8/F7

Assume Basic size 30mm diameter

Hole tolerance: +0.39 max and 0.0 min

Shaft tolerance: -0.25 max and 0.50 min

TABLE 4.1(a) A selection of fits—hole-basis system (deviations)



TOLERANCE unit = 0.001mm		CLEARANCE FITS: COARSE TOLERANCE	CLEARANCE FITS: LOOSE (RUNNING FIT)	CLEARANCE FITS: EASY (RUNNING FIT)	CLEARANCE FITS: NORMAL RUNNING	CLEARANCE FITS: PRECISION RUNNING, SLIDING	CLEARANCE FITS: AVERAGE LOCATION	TRANSITION FITS: LIGHT (PUSH FIT)	TRANSITION FITS: HEAVY (PUSH FIT)	INTERFERENCE FITS: PRESS FIT (FERROUS)	INTERFERENCE FITS: HEAVY PRESS FIT (NON-FERROUS)
BASIC SIZES (mm) OVER	TO	H11/c11	H9/d10	H9/e9	H8/f7	H7/g6	H7/h6	H7/k6	H7/n6	H7/p6	H7/s6
0	3	+60 0 / −60 −120	+25 0 / −20 −60	+25 0 / −14 −39	+14 0 / −6 −16	+10 0 / −2 −8	+10 0 / 0 −6	+10 0 / +6 0	+10 0 / +10 4	+10 0 / +12 +6	+10 0 / +20 +14
3	6	+75 0 / −70 −145	+30 0 / −30 −78	+30 0 / −20 −50	+18 0 / −10 −28	+12 0 / −4 −12	+12 0 / 0 −8	+12 0 / +9 +1	+12 0 / +16 +8	+12 0 / +20 +12	+12 0 / +27 +19
6	10	+90 0 / −80 −170	+36 0 / −40 −98	+36 0 / −25 −61	+22 0 / −13 −28	+15 0 / −5 −14	+15 0 / 0 −9	+15 0 / +10 +1	+15 0 / +19 +10	+15 0 / +24 +15	+15 0 / +32 +23
10	18	+110 0 / −95 −205	+43 0 / −50 −120	+43 0 / −32 −75	+27 0 / −16 −34	+18 0 / −6 −17	+18 0 / 0 −11	+18 0 / +12 +1	+18 0 / +23 +12	+18 0 / +29 +18	+18 0 / +39 +28
18	30	+130 0 / −110 −240	+52 0 / −65 −149	+52 0 / −40 −92	+33 0 / −20 −41	+21 0 / −7 −20	+21 0 / 0 −13	+21 0 / +15 +2	+21 0 / +28 +15	+21 0 / +35 +22	+21 0 / +48 +35

OVER	TO	H11	c11	H9	d10	H9	e9	H8	f7	H7	g6	H7	h6	H7	k6	H7	n6	H7	p6	H7	s6
30	40	+160	−120 −280	+62	−80	+62	−50	+39	−25	+25	−9	+25	0	+25	+18	+25	+33	+25	+42	+25	+59
40	50	0	−130 −290	0	−180	0	−112	0	−50	0	−25	0	−16	0	+2	0	+17	0	+26	0	+43
50	65	+190	−140 −330	+74	−100	+74	−60	+46	−30	+30	−10	+30	0	+30	+21	+30	+39	+30	+51	+30	+72 +53
65	80	0	−150 −340	0	−220	0	−134	0	−60	0	−29	0	−19	0	+2	0	+20	0	+32	0	+78 +59
80	100	+220	−170 −390	+87	−120	+87	−72	+54	−36	+35	−12	+35	0	+35	+25	+35	+45	+35	+59	+35	+93 +71
100	120	0	−180 −400	0	−260	0	−159	0	−71	0	−34	0	−22	0	3	0	+23	0	+37	0	+101 +79
120	140	+250	−200 −450	+100 0	−145 −305	+100 0	−84 −185	+63 0	−43 −83	+40 0	−14 −39	+40 0	0 −25	+40 0	+28 +3	+40 0	+52 +27	+40 0	+68 +43	+40 0	+117 +92
140	160	0	−210 −460																		+125 +100
160	180		−230 −480																		+133 +108
180	200	+290	−240 −530	+115 0	−170 −355	+115 0	−100 −215	+72 0	−50 −96	+46 0	−15 −44	+46 0	0 −29	+46 0	+33 +4	+46 0	+60 +31	+46 0	+79 +50	+46 0	+151 +122
200	225	0	−260 −550																		+159 +130
225	250		−280 −570																		+169 +140
250	280	+320	−300 −620	+130 0	−190 −400	+130 0	−110 −240	+81 0	−56 −108	+52 0	−17 −49	+52 0	0 −32	+52 0	+36 +4	+52 0	+66 +34	+52 0	+88 +56	+52 0	+190 +158
280	315	0	−330 −650																		+202 +170
315	355	+360	−360 −720	+140 0	−210 −440	+140 0	−125 −265	+89 0	−62 −119	+57 0	−18 −54	+57 0	0 −36	+57 0	+40 +4	+57 0	+73 +37	+57 0	+98 +62	+57 0	+226 +190
355	400	0	−400 −760																		+244 +208
400	450	+400	−440 −840	+155	−230	+155	−135	+97	−68	+63	−20	+63	0	+63	+45	+63	+80	+63	+108	+63	+272 +232

UNSW SYDNEY

Deviations of Tolerances

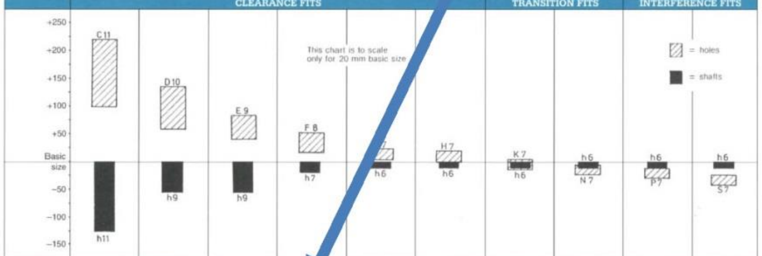
Clearance fit (Normal Running) H8/F7

Assume Basic size 30mm diameter

Hole tolerance: +0.64 max and 0.25 min

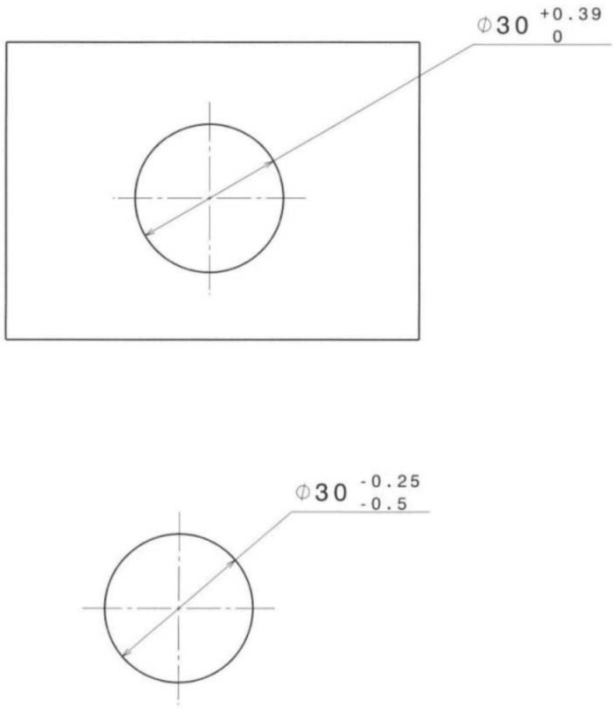
Shaft tolerance: -0.0 max and -0.25 min

TABLE 4.1(b) A selection of fits—shaft-basis system (deviations)

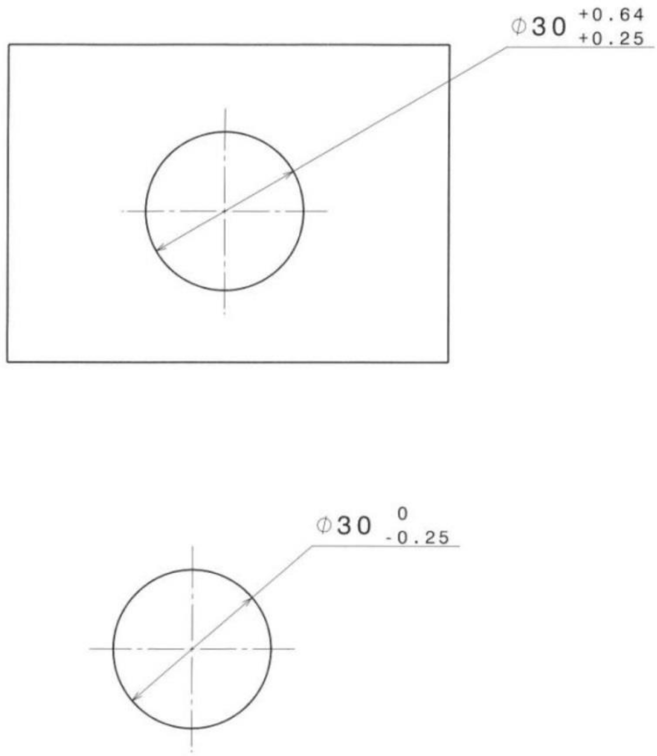


		CLEARANCE FITS												TRANSITION FITS				INTERFERENCE FITS			
TOLERANCE unit = 0.001mm		COARSE TOLERANCE		LOOSE		EASY		NORMAL RUNNING		PRECISION RUNNING, SLIDING		AVERAGE LOCATION		LIGHT		HEAVY		PRESS FIT (FERROUS)		HEAVY PRESS FIT (NON-FERROUS)	
				RUNNING FIT										PUSH FIT							
BASIC SIZES (mm)		C11/h11		D10/h9		E9/h9		F8/h7		G7/h6		H7/h6		K7/h6		N7/h6		P7/h6		S7/h6	
OVER	TO																				
0	3	+120 +60	0 -60	+60 +20	0 -25	+39 +14	0 -25	+20 +6	0 -10	+12 +2	0 -6	+10 0	0 -6	0 -10	0 -6	-4 -14	0 -6	-6 -16	0 -6	-14 -24	0 -6
3	6	+145 +70	0 -75	+78 +30	0 -30	+50 +20	0 -30	+28 +10	0 -12	+16 +4	0 -8	+12 0	0 -8	+3 -9	0 -8	-4 -16	0 -8	-8 -20	0 -8	-15 -27	0 -8
6	10	+170 +80	0 -90	+98 +40	0 -36	+61 +25	0 -36	+35 +13	0 -15	+20 +5	0 -9	+15 0	0 -9	+5 -10	0 -9	-4 -19	0 9	-9 -24	0 -9	-17 -32	0 -9
10	18	+205 +95	0 -110	+120 +50	0 -43	+75 +32	0 -43	+43 +16	0 -18	+24 +6	0 -11	+18 0	0 -11	+6 -12	0 -11	-5 -23	0 -11	-11 -29	0 -11	-21 -39	0 -11
18	30	+240 +110	0 -130	+149 +65	0 -52	+92 +40	0 -52	+53 +20	0 -21	+28 +7	0 -13	+21 0	0 -13	+6 -15	0 -13	-7 -28	0 -13	-14 -35	0 -13	-27 -48	0 -13
30	40	+280 +120	0 -160	+180	0	+112	0	+64	0	+34	0	+25	0	+7	0	-8	0	-17	0	-34	0
40	50	+290 +130	0 -160	+80	-62	+50	-62	+25	-25	+9	-16	0	-16	-18	-16	-33	-16	-42	-16	-59	-16
50	65	+330 +140	0 -190	+220	0	+134	0	+76	0	+40	0	+30	0	+9	0	-9	0	-21	0	-42 -72	0 -19
65	80	+340 +150	0 -190	+100	-74	+60	-74	+30	-30	+10	-19	0	-19	-21	-19	-39	-19	-51	-19	-48 -78	0 -19
80	100	+390 +170	0 -220	+260	0	+159	0	+90	0	+47	0	+35	0	+10	0	-10	0	-24	0	-58 -93	0 -22
100	120	+400 +180	0 -220	+120	-87	+72	-87	+36	-35	+12	-22	0	-22	-25	-22	-45	-22	-59	-22	-66 -101	0 -22
120	140	+450 +200	0 -250	+305	0	+185	0	+106	0	+54	0	+40	0	+12	0	-12	0	-28	0	-77 -117	0 -25
140	160	+460 +210	0 -250	+145	-100	+85	-100	+43	-40	+14	-25	0	-25	-28	-25	-52	-25	-68	-25	-85 -125	0 -25
160	180	+480 +230	0 -250																	-93 -133	0 -25
180	200	+530 +240	0 -290	+355	0	+215	0	+122	0	+61	0	+46	0	+13	0	-14	0	-33	0	-105 -151	0 -29
200	225	+550 +260	0 -290	+170	-115	+100	-115	+50	-46	+15	-29	0	-29	-33	-29	-60	-29	-79	-29	-113 -159	0 -29
225	250	+570 +280	0 -290																	-123 -169	0 -29
250	280	+620 +300	0 -320	+400	0	+240	0	+137	0	+62	0	+52	0	+16	0	-14	0	-36	0	-138 -190	0 -32
280	315	+650 +330	0 -320	+190	-130	+110	-130	+56	-52	+17	-32	0	-32	-36	-32	-66	-32	-88	-32	-150 202	0 32
315	355	+720 +360	0 -360	+440	0	+265	0	+151	0	+75	0	+57	0	+17	0	-16	0	-41	0	-169 -226	0 -36
355	400	+760 +400	0 -360	+210	-140	+125	-140	+62	-57	+18	-36	0	-36	-40	-36	-73	-36	-98	-36	-187 -244	0 -36
400	450	+840 +440	0 -400	+480	0	+290	0	+165	0	+83	0	+63	0	+18	0	-17	0	-48	0	-209 -272	0 -40

Deviations of Tolerances



H8f7
Hole-basis



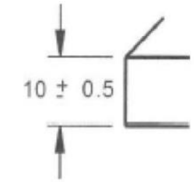
H8f7
Shaft-basis

Classes of Fits with Designations

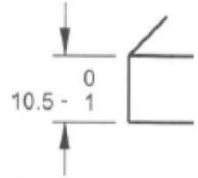
	Hole basis	Shaft basis
• Coarse tolerance	**H11c11**	**C11h11**
• Loose	**H9d10**	**D10h9**
• Easy	**H9e9**	**E9h9**
• Nominal running	**H8f7**	**F8h7**
• Precision running or sliding	**H7g6**	**G7h6**
• Average location	**H7h6**	**H7h6**
• Light location	**H7k6**	**K7h6**
• Heavy location	**H7n6**	**N7h6**
• Press fit (ferrous)	**H7p6**	**P7h6**
• Heavy press fit (non-ferrous)	**H7s6**	**S7h6**

Application of Tolerances

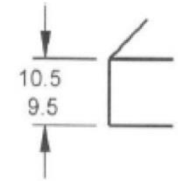
1. Bilateral tolerances



2. Unilateral tolerance



3. Limits of size



Assembly of Components

- A mechanical Assembly
  - A combination of "Fitting Together" components designed perform a function
- Each part/component has a finished size within a specified tolerance
- In the design of mechanical assemblies
  - Make sure the cumulative effect of the tolerance of assembled components controlled

Geometric Tolerances - Symbols

GEOMETRIC CHARACTERISTIC	SYMBOL	TYPE OF TOLERANCE	APPLICATION
Straightness	—	Form	For individual features
Flatness	⏥		
Circularity	○		
Cylindricity	⌭		
Profile of a line	⌒	Profile	For individual or related features
Profile of a surface	⌓		
Angularity	∠	Orientation	For related features
Perpendicularity	⊥		
Parallelism	//		
Position including concentricity and symmetry	⌖	Location	
Circular runout	↗	Runout	
Total runout	⌰		

Notes and Symbols

- A symbol has uniform meaning
- Symbols are an international language
  - No translation necessary
- Symbols are more eligible
  - Drawings are often copied with lower quality
- Symbols are compact – noes take more tiem and space

SURFACE B PERPENDICULAR TO DATUM A WITHIN .001

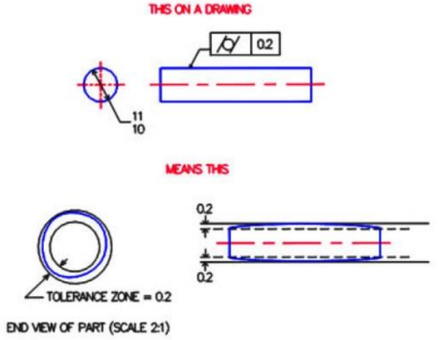
=

⊥	.001	A

Cylindricity

Cylindricity tolerance is used when cylindrical part features must have good circularity and straightness, like pins or camshafts. While circularity applies only to cross sections, cylindricity applies simultaneously to the entire surface.

An example of cylindricity tolerance is shown below. In the top figure, a shaft has a cylindricity tolerance applied to it. The boxed symbols can be read "this surface must lie between two concentric cylinders spaced 0.2 apart". The lower figure shows a sample part that meets this tolerance.

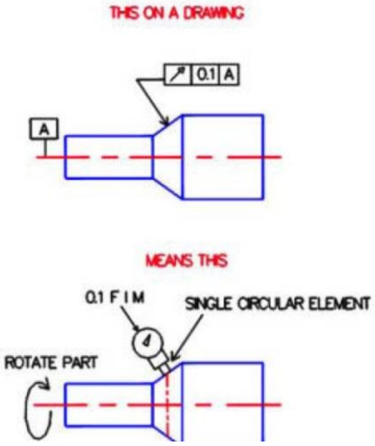


http://www.emachineshop.com/machine-shop/GD-T-Cylindricity-Definition/page606.html

Runout

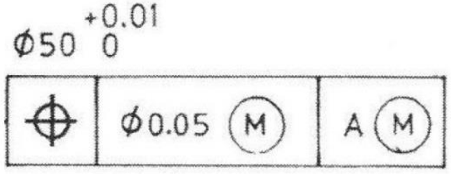
- R**unout tolerance** is used to control the location of a circular part feature relative to its axis. This is different than circularity, which controls overall roundness. Runout is usually applied to parts with circular cross sections that must be assembled like drill bits, segmented shafts, or machine tool components. Runout helps to limit the axis offset of two parts to ensure they can spin and wear evenly.

- The boxed symbols can be read "each circular element of this surface must have full indicator movement (FIM) of less than 0.1 relative to datum A". The bottom figure shows a sample measuremen cross section, but multiple measurements are required to verif

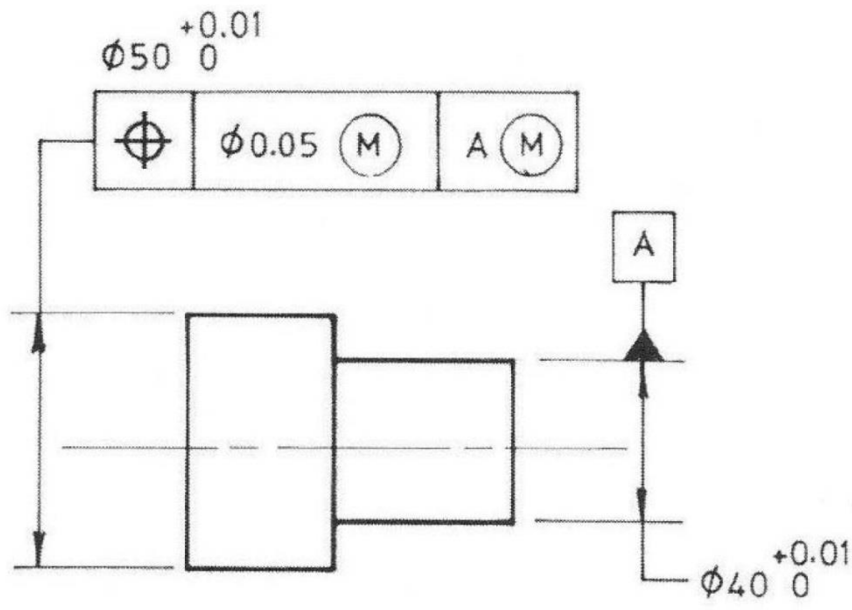


Displaying Geometry Tolerances

- Two methods:
  - The tolerance frame method
    - Used when there are no more than 3 simple groups of geometry tolerances
  - The tabular method
    - Used when the group of geometry tolerances are complex and more than 3
    - AS1100 Part 101



Geometric Tolerances - Example



Shaft concentricity and size tolerance, referred to a DATUM

Source: Dimensioning and Tolerancing – Leonard E. Farmer

Concluding Remarks

- Critical to familiarise yourself with the concepts and the type of tolerances
- For your project
  - Proper size tolerance is a “MUST”
  - A generic geometric and surface tolerance is desirable