7.6 Fits and clearances

610201 MAINTENANCE PRACTICES for AMEL

Presented by Aircraft Maintenance Training Division (AMTD)

The Knowledge level indicators are defined on 3 levels as follows:

Level 1	A familiarisation with the principal elements of the subject.
Objectives:	
(a)	The applicant should be familiar with the basic elements of the subject.
(b)	The applicant should be able to give a simple description of the whole subject, using common words and examples.
(c)	The applicant should be able to use typical terms.
Level 2	A general knowledge of the theoretical and practical aspects of the subject and an ability to apply that knowledge.
Objectives:	
(a)	The applicant should be able to understand the theoretical fundamentals of the subject.
(b)	The applicant should be able to give a general description of the subject using, as appropriate, typical examples.
(c)	The applicant should be able to use mathematical formulae in conjunction with physical laws describing the subject.
(d)	The applicant should be able to read and understand sketches, drawings and schematics describing the subject.
(e)	The applicant should be able to apply his knowledge in a practical manner using detailed procedures.
Level 3	A detailed knowledge of the theoretical and practical aspects of the subject and a capacity to combine and apply the separate elements of knowledge in a logical and comprehensive manner.
Objectives:	
(a)	The applicant should know the theory of the subject and interrelationships with other subjects.
(b)	The applicant should be able to give a detailed description of the subject using theoretical fundamentals and specific examples.
(c)	The applicant should understand and be able to use mathematical formulae related to the subject.
(d)	The applicant should be able to read, understand and prepare sketches, simple drawings and schematics describing the subject.
(e)	The applicant should be able to apply his knowledge in a practical manner using manufacturer's instructions.
(f)	The applicant should be able to interpret results from various sources and measurements and apply corrective action where appropriate.

Certification statement and objectives

These Study Notes comply with the syllabus of EASA Regulation (EU) No. 1321/2014 Annex III (Part-66) Appendix I, including the amendment Regulation (EU) 2023/989, and the associated Knowledge Levels as specified below.

Objective	Part-66 Ref.	Knowledge Levels		
		A	B1 B3	B2 B2L
Fits and clearances	**7.6**	1	2	1
Drill sizes for bolt holes, classes of fits;				
Common system of fits and clearances;				
Schedule of fits and clearances for aircraft and engines;				
Limits for bow, twist and wear;				
Standard methods for checking shafts, bearings and other parts.				

Limits and fits

General

To ensure that assemblies function correctly, their parts must fit together predictably. No component can be manufactured to an exact size, so the designer must decide on appropriate upper and lower limits for each dimension.

Accurately toleranced dimensioned features usually take much more time to manufacture correctly and increase production costs significantly. Good engineering practice finds the optimum balance between the required accuracy for the component's function and the minimum manufacturing cost.

In hand and machine fitting, the term 'fitting' means putting parts together so that they touch or join with each other in such a way that either one part can turn inside another, one will slide upon another, or the parts will hold tightly together so that they cannot move upon each other. Examples of fitted parts are shafts fitted to a bearing, a piston running in a cylinder, a propeller splined to its shaft or a bolt fitting into a nut. Parts may be machined, filed, ground lapped or scraped to achieve the fit required.

Most fitting was at one time carried out by hand and required great skill and judgment. With high-developed machine tools capable of producing precision work of accuracy and uniformity, hand-fitting has become superseded. However, it still has a part to play in teaching basic skills and in many cases where individual work is required.

Interchangeability

Individual parts are made in significant quantities by carefully planned processes using special tools, jigs and fixtures using modern methods. The use of gauges and templates maintains uniformity in size and shape. By this uniformity, the parts can be manufactured as exact replicas of each other, and therefore each similar part can be interchanged.

Dimensions

Mass production has long been the basis of the approach to the most economical manufacturing methods. The complete replacement of a defective item is common practice in the maintenance of aircraft and aerospace components.

For this reason, limits are imposed on the manufacturing processes to ensure that, if any two mating parts are manufactured to the dimensions as stated on the relevant drawings, then the parts assemble without the need for further significant adjustments and in the least time possible.

The limits are based on the allowances and tolerances imposed on the dimensions of the manufactured parts. These dimensions are given the accuracy required by the designer of the respective parts.

The basic size of a dimension is given with tolerances expressed as a plus/minus range.

Allowances

An allowance is a difference in a dimension necessary to give a particular ‘class of fit’ between two parts. If, for example, (and using a typical limit system), a shaft was required to locate a corresponding hole in a component. Then, to assist in the manufacturer’s economy, the hole or the shaft is made as accurately as possible to the nominal size and an allowance is applied to the associated item. The term ‘shaft’ also includes bolts and pins.

If the shaft is constant and the hole varies in size, then the system is shaft-based. If the hole is constant and the shaft varies in size, the system is hole-based. The hole-based system is the one in more general use.

The item dimensioned to include the allowance also has high and low limits and tolerance. The correct allowance would be the difference between the high limit of the shaft and the hole’s low limit.

Definitions

With the application of the limits and fits system, the interchangeability of components can be maintained even when different workers make parts in different factories. For general use, the following terms are used:

Dimension is a feature of any component, such as the length or diameter of which the size is stated.

Nominal size is the size referred to as a matter of convenience.

Actual size is the measured size.

Limits of size are the two extreme permissible sizes of a dimension or part.

Tolerance is the difference between the high and the low limits of the size of a dimension. The tolerance provides for the error in machining and permits sufficient accuracy to be obtained without any finishing operations.

Unilateral is where the maximum and minimum Tolerance dimensions are together above or below a nominal size, as shown in these two examples:

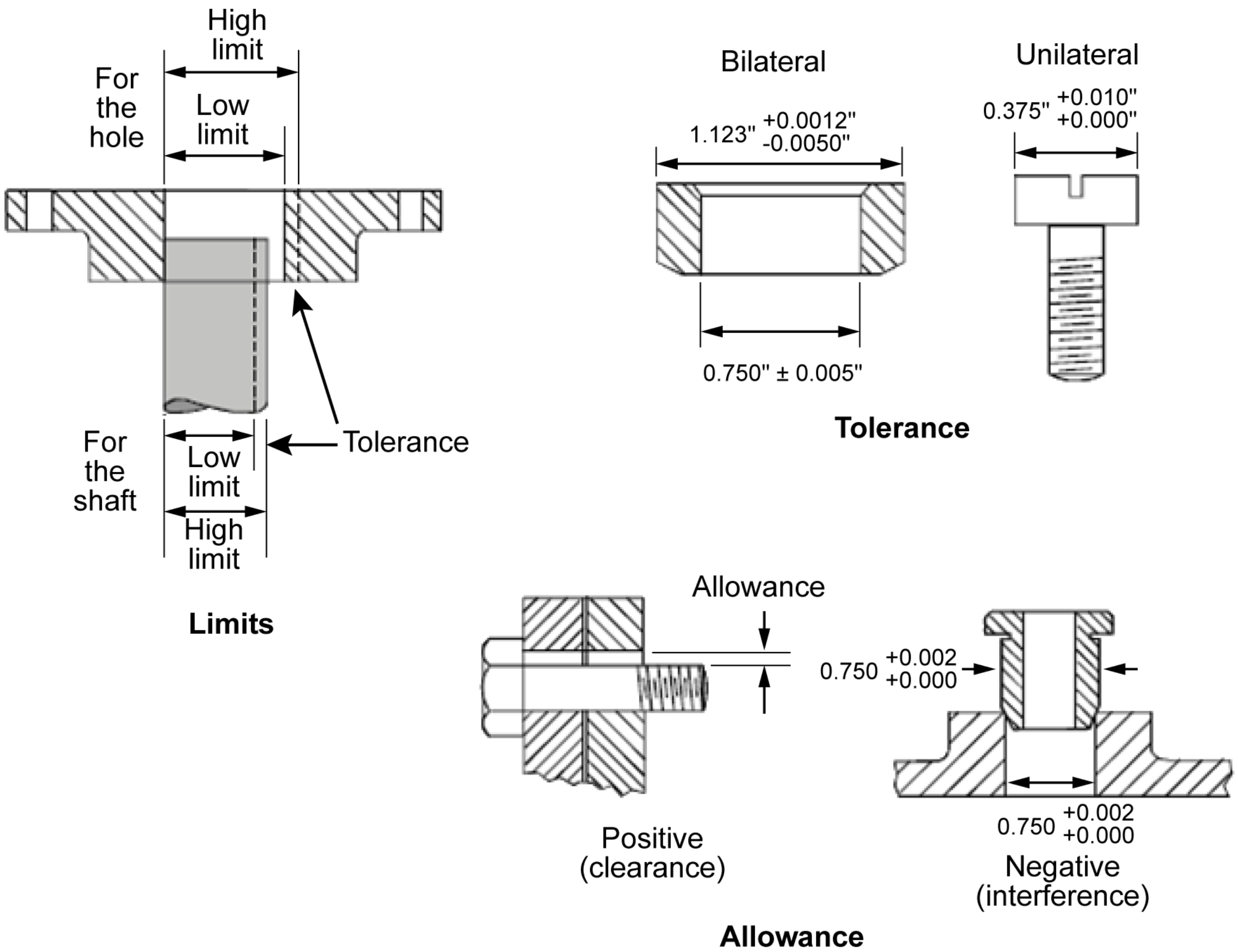
$1.000"\ ^{+0.002"}_{-0.001"}$ $\quad$ $1.000"\ ^{-0.001"}_{-0.000"}$

Bilateral is where the maximum and minimum Tolerance dimensions are respectively one above and one below a nominal size as:

$1.000"\ ^{+0.0015"}_{-0.0015"}$ $\quad$ 1.000" ±0.0015"

Deviation is the algebraic difference between size and the corresponding nominal size.

Allowance is a prescribed difference between the high limit for a shaft and the low limit for a hole to provide a particular type of fit. An allowance may be either positive or negative per the type of fit; a positive allowance results in a clearance fit, and a negative allowance in an interference fit. Tolerance and allowance are separate and distinct terms and should not be confused.



Limits, tolerance and allowance

Fits

Up to this point, dimensions and tolerances have been considered to apply to any individual engineering part. When it becomes necessary to fit two such parts together, the tolerances to be applied to each part must be carefully worked out before any fit can be achieved. To do this, it is convenient to consider all fits associated with circular holes and shafts. However, in practice, tolerance limits and the resulting fits equally apply to other mating parts.

A fit is defined as the difference, before assembly, between the sizes of the two parts which are to be assembled. Using various assembly methods (some gentle, some extremely forceful), it is possible in engineering to obtain a fit whether the shaft is smaller than the hole or not. A clearance between the parts and assembly is relatively easy when the shaft is smaller.

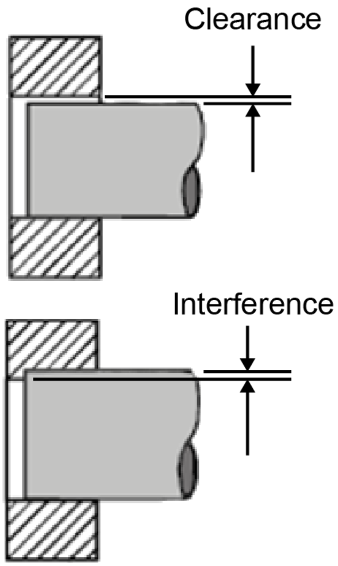
When the reverse is true, interference is said to occur, and force of some kind is needed to complete the assembly. Three main fit classes emerge from these two basic situations: clearance, transition, and interference.

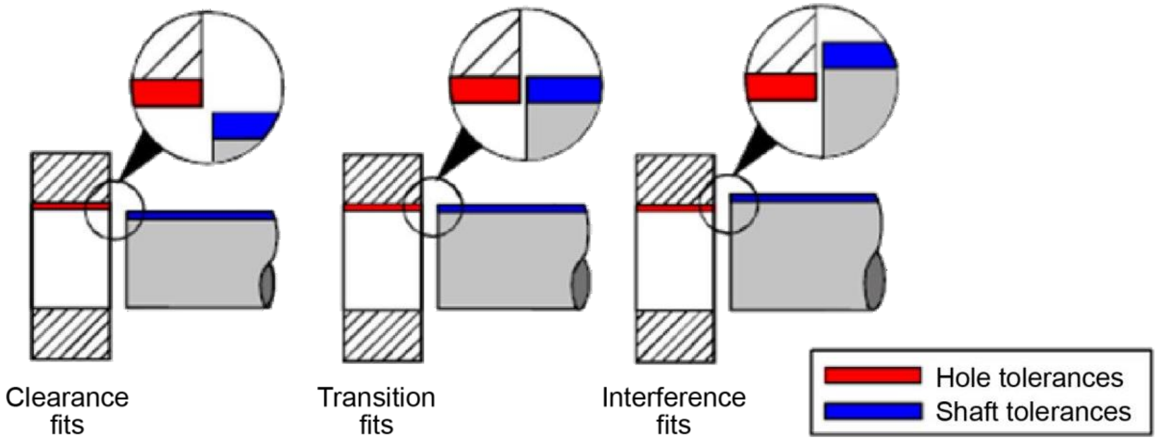
The definitions of these are:

Clearance fit (A) A fit that always provides a positive clearance, or in technical terms, where the hole's tolerance zone is entirely above that of the shaft.

Transition fit (B) A fit which may provide either a clearance or interference or where the tolerance zones of the hole and the shaft overlap.

Interference fit (C) A fit that always provides an interference where the hole’s tolerance zone is entirely below that of the shaft.





Fits definitions

Tolerances

Tolerance is the total variation in the size of a part or feature on a part. To find the tolerance of a part or feature, subtract the smallest size from the largest.

If a dimension is specified, in millimetres, as 10 ± 0.02, the part is acceptable if the dimension is manufactured to an actual size between 9.98 and 10.02 mm. Below are examples of ways to define such limits for a linear dimension.

Example:
Given a dimension of

2.50 ± 0.05

the largest size would be 2.55, the smallest size would be 2.45, and the tolerance would be 0.10

Unilateral and bilateral tolerances

- **Bilateral tolerance** is the amount of variation above or below the design size.

  Example:

  3.75 ± 0.05

  which means the feature or parts can be as small as 3.70 or as large as 3.80

- **Unilateral tolerance** is the variation in one direction, either above or below the design size.

  Example:

  3.75 +0.05/-0.00

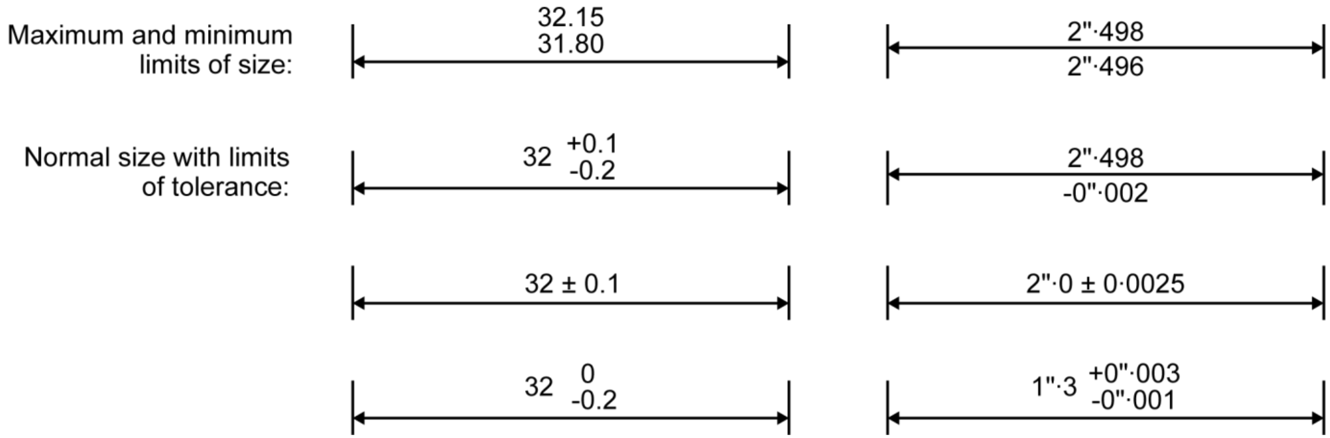
  which means the feature or parts can be as small as 3.75 or as large as 3.8

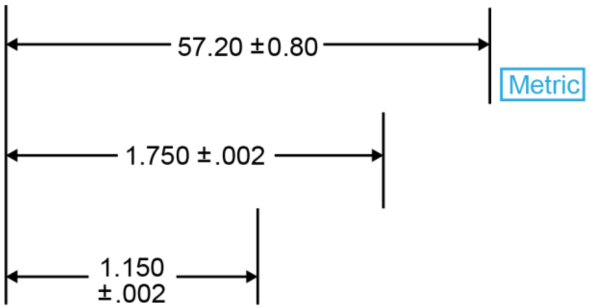
  Example: With sheet material, such as patch plates used in certain repairs, the dimensions quoted in the repair scheme usually have tolerance in one direction only, the nominal size being the lower limit. In effect, the patch plate must never be below the nominal size, although it can be slightly over according to the repair scheme in the manual.

Baseline and continued tolerances

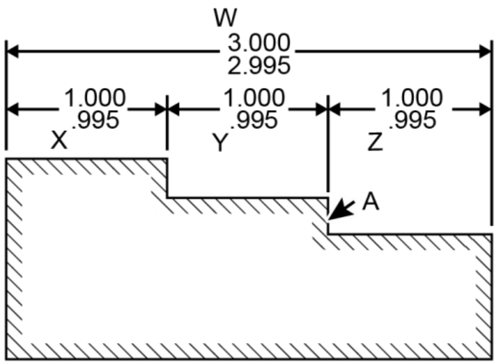
Baseline tolerances do not ‘stack-up’.

Continued (or ‘chained’) tolerances ‘stack up’ and are cumulative.

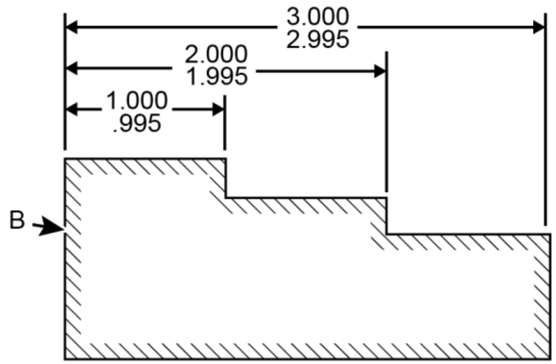




Equal bilateral tolerances



(a) Cumulative tolerances



(b) Baseline tolerances

Drill sizes for holes

The size of the hole to be drilled depends on its purpose. A hole drilled for a rivet with a specific diameter would differ from those drilled to take a screw thread or the plain shank of a bolt of the same diameter.

The hole for a rivet is generally drilled slightly oversize (the amount of oversize depends on the rivet size). The generous oversize allows clearance for the rivet to ‘swell’ when formed. The table below shows a few examples of the drill sizes for various rivet sizes.

Similarly, the size of a hole to accommodate a shaft depends on the shaft’s size and how the hole/shaft combination is to be used.

Additionally, if the hole is to be reamed, it must be drilled slightly smaller than its nominal size to allow for the metal to be removed by the reamer.

Hole sizes for fitting a bolt vary depending on the application and structural significance. For example, a precision bolt’s plain shank fitted into a primary structural component is most likely to be an interference or transition fit. This means the hole is drilled slightly smaller than the bolt’s shank diameter.

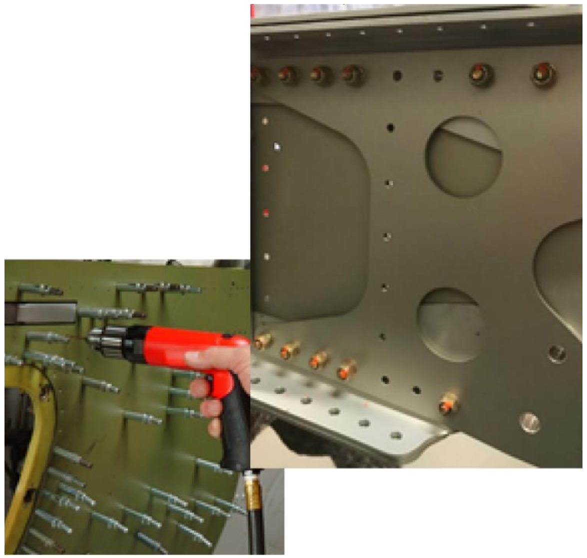
Bolt holes to be threaded are also drilled undersize before the tapping process. For standard fit bolts, the table below provides some common examples of drill sizes for threaded holes in accordance with the bolt size.

Drill sizes, as discussed in Module 7.3, are fixed and can be found on charts that list each standard drill size, together with other columns such as clearance and tapping sizes. These charts may also include equivalent sizes in metric, fractional, letter, and number/letter systems.

Rivet diameter (in)	Drill size	
	Pilot	Final
3/32	3/32 (0.0937)	# 40 (0.098)
1/8	1/8 (0.125)	# 30 (0.1285)
5/32	5/32 (0.1562)	# 21 (0.159)
3/16	3/16 (0.1875)	# 11 (0.191)
1/4	1/4 (0.250)	F (0.257)

Bolt or screw designation	Drill size through hole	Tap size	Drill size threaded hole
6-32	#27	6-32	#33
8-32	#18	8-32	#29
AN3	#10	10-32	#21
AN4	1/4 in or letter F	1/4 in-28	#3
AN5	5/16 in	5/16 in-24	Letter I
AN6	3/8 in	3/8 in-24	Letter Q
AN7	7/16 in	7/16 in-20	Letter W
AN8	1/2 in	1/2 in-20	7/16 in

Drill and tap sizes for rivets, bolts and screws



Drill	Decimal	Drill	Decimal	Drill	Decimal	Drill	Decimal
80	.0135	43	.089	8	.199	25/64	.3906
79	.0145	42	.0935	7	.201	X	.397
1/64	.0156	3/32	.0938	13/64	.2031	Y	.404
78	.016	41	.096	6	.204	13/32	.4063
77	.018	40	.098	5	.2055	Z	.413
76	.020	39	.0995	4	.209	27/64	.4219
75	.021	38	.1015	3	.213	7/16	.4375
74	.0225	37	.104	7/32	.2188	29/64	.4531
73	.024	36	.1065	2	.221	15/32	.4688
72	.025	7/64	.1094	1	.228	31/64	.4844
71	.026	35	.110	A	.234	1/2	.500
70	.028	34	.111	15/64	.2344	33/64	.5156
69	.0292	33	.113	B	.238	17/32	.5313
68	.031	32	.116	C	.242	35/64	.5469
1/32	.0313	31	.120	D	.246	9/16	.5625
67	.032	1/8	.1250	1/4 (E)	.250	37/64	.5781
66	.033	30	.1285	F	.257	19/32	.5938
65	.035	29	.136	G	.261	39/64	.6094
64	.036	28	.1405	17/64	.2656	5/8	.625
63	.037	9/64	.1406	H	.266	41/64	.6406
62	.038	27	.144	I	.272	21/32	.6563
61	.039	26	.147	J	.277	43/64	.6719
60	.040	25	.1495	K	.281	11/16	.6875
59	.041	24	.152	9/32	.2813	45/64	.7031
58	.042	23	.154	L	.290	23/32	.7188
57	.043	5/32	.1563	M	.295	47/64	.7344
56	.0465	22	.157	19/64	.2969	3/4	.750
3/64	.0469	21	.159	N	.302	49/64	.7656
55	.052	20	.161	5/16	.3125	25/32	.7813
54	.055	19	.166	O	.316	51/64	.7969
53	.0595	18	.1695	P	.323	13/16	.8125
1/16	.0625	11/64	.1719	21/64	.3281	53/64	.8281
52	.0635	17	.173	Q	.332	27/32	.8438
51	.067	16	.177	R	.339	55/64	.8594
50	.070	15	.180	11/32	.3438	7/8	.875
49	.073	14	.182	S	.348	57/64	.8906
48	.076	13	.185	T	.358	29/32	.9063
5/64	.0781	3/16	.1875	23/64	.3594	59/64	.9219
47	.0785	12	.189	U	.368	15/16	.9375
46	.081	11	.191	3/8	.375	61/64	.9531
45	.082	10	.1935	V	.377	31/32	.9688
44	.086	9	.196	W	.386	63/64	.9844

Classes of threaded fasteners

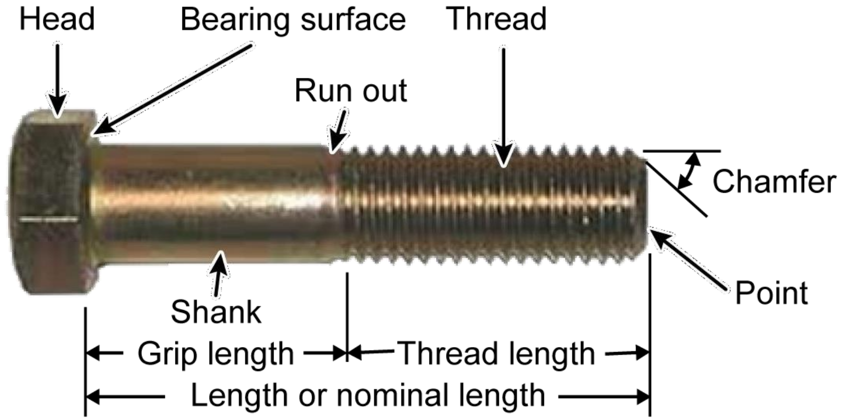
Threads are designated by class of fit. The class of a thread indicates the tolerance allowed in manufacturing. Class 1 is a loose fit, Class 2 is a free fit, Class 3 is a medium fit, and Class 4 is a close fit. Aircraft bolts are almost always manufactured in Class 3, medium fit.

A Class 4 fit requires a wrench to turn the nut onto a bolt, whereas a Class 1 fit can easily be turned with the fingers. Generally, aircraft screws are manufactured with a Class 2 thread fit for easy assembly.

Bolts and nuts are also produced with right-hand and left-hand threads. A right-hand thread tightens when turned clockwise; a left-hand thread tightens when turned counterclockwise.

Class of fit	Type of fit
1	Loose
2	Free
3	Medium
4	Close
5	Tight

Class of fits for bolts



Nomenclature threaded fasteners
(standard hex bolt illustrated)

Common systems for fits and clearances

General

There are many instances in aircraft engineering where one part engages with or fits into another. A cylindrical shaft or pin fitting into a hole is the most common example. Depending on the functional requirements, the ‘fit’ between the two parts may need to be tight or loose. But these terms (tight and loose) are inexact. They are given precision and consistency by a standard series of prescribed tolerance conditions and values called limits and fits.

In engineering, a ‘fit’ refers to the clearance between two mating parts. The choice of an engineering fit determines whether the two parts can move relative to each other if a clearance fit or act as a whole in case of a tight interference fit.

While limits and fits apply to all sorts of mating parts, their primary use is for regulating the sizes of mating shafts and holes for the best performance.

Standards of fits and clearances

Both ISO and ANSI have standardised fits in three classes:

- clearance;
- transition; and
- interference.

Each class has various options for choosing the correct one for a specific application.

The tolerance values depend on many practical considerations, including function, the need for interchangeability of parts, and cost.

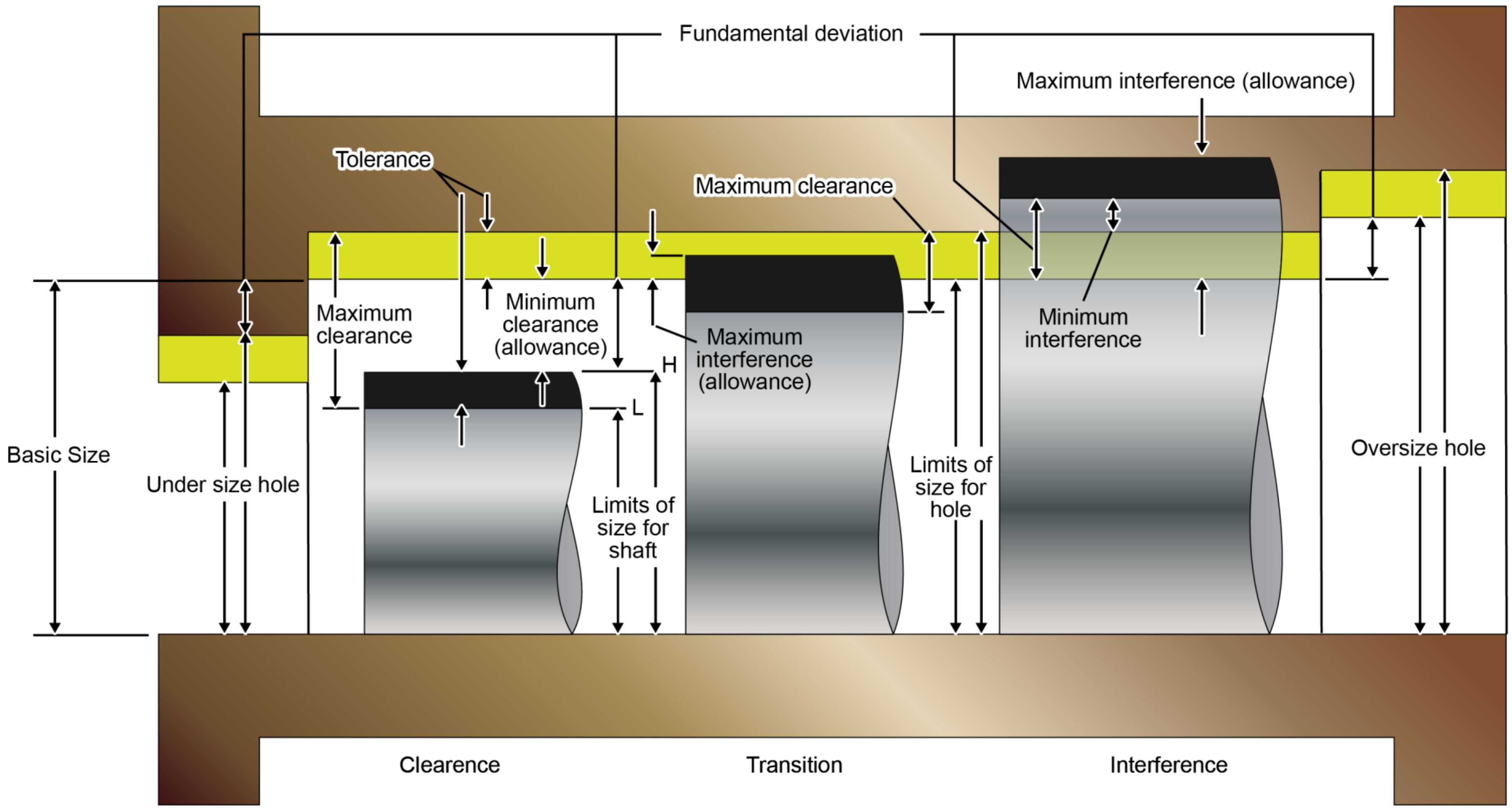
Various standard fits have been devised around the world to ensure consistent specifications. The British Standards System (BS 1916-1&2:1953 and -3:1963), devised by the British Standards Institute (BSI), has a comprehensive system designed to cater to all work classes. The system provides for 21 types of holes designated by capital letters A, B, C, D… and 21 types of shafts designated by small letters: a, b, c, d… from nominal diameters of 0.04 inch up to 19.69 inches. Each type of hole or shaft is provided with 16 grades of accuracy designated by numbers 1 – 16.

In the United States, The American National Standards Institute (ANSI) has developed a standard for inch units using symbols (RCx for running and sliding fits and FNx for force and shrink fits). The ANSI standard for metrics follows the ISO standard.

The American National Standards Institute (ANSI) gives this definition of fit:

“The standard that is mostly used in Europe is the ISO metric one, which was fully defined in BS4500 and its later European equivalent BS EN 20286 ISO Limits & Fits.”

Thus the ‘ISO 286 standard’ established by the International Organization for Standardization (ISO) is very similar to that of the BS. The system provides 28 types of holes and shafts with 20 grades of accuracy.



Disposition of limits and tolerances (basic hole)

Newall system

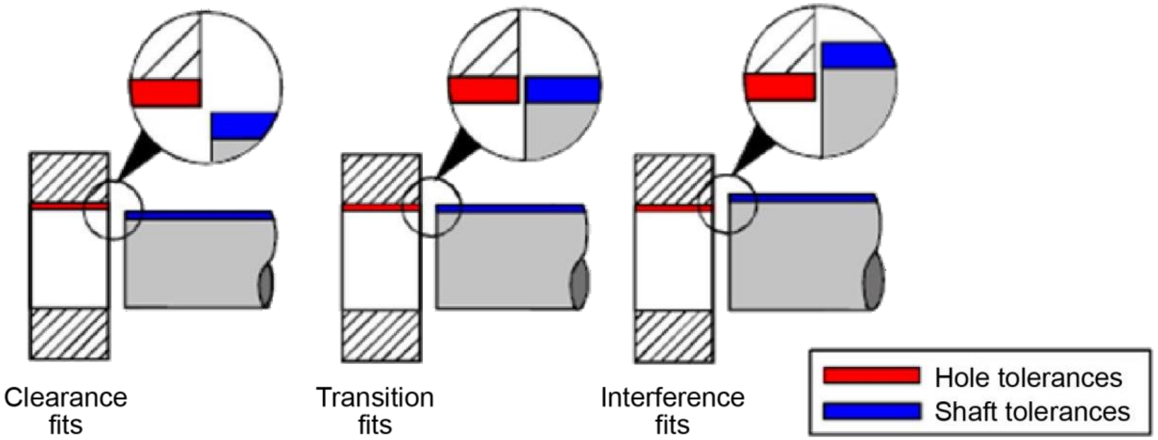
This is the first standard that evolved in Great Britain to standardise limits and fits and is still used to a certain extent. However, all the fits provided by this system can be obtained with approximately the same values using more current systems.

The Newall system provides a range of clearance, transition, and interference fits for sizes up to 12 inches. A hole basis system stipulates two grades of holes, specified with bilateral tolerances and 6 grades of shaft tolerances.

It is the earliest of all the systems and is extremely simple. It specifies too few fits, and those listed do not enforce modern ideas regarding their basic deviations. However, this served a useful purpose in the past but is not considered suitable for modern production.

The Newall system is founded on the ‘hole’ basis. Provision is made in the hole size for production quality error, and the variation to obtain the required fit quality is allowed on the shaft size.

The holes are classified as Class A and Class B fits. Class A holes are manufactured to a closer tolerance than Class B holes. The table shows how the shafts are classified using the letters F, P, D, X, Y, and Z.



Class of fit	Type of fit	Remarks
Interference F	Force	Mechanical pressure is required for assembly and once assembled, no dismantling is likely to be required.
D	Driving	These are a little less tight than Force Fit and one part can be driven into the other.
Transition P	Push	Slight manual effort is required to assemble the parts. Suitable for detachable or locating parts but not for moving parts.
Clearance X, Y and Z	Running	Suitable for various types of moving parts. Class Z provides the finest fit

Newall system of fits

Limits and fits systems

Shaft basis – A limit system is on a shaft basis when the shaft is the constant member, and the different fits are obtained by varying the hole size.

Hole basis – A limit system is on a hole basis when the hole is the constant member, and different fits are obtained by varying the shaft sizes.

Most of the existing systems of limits and fits are on a hole basis. This is because holes are generally produced to size by tools of fixed dimensions, such as reamers, and are not so readily varied in size, whereas shafts can be easily varied in size during manufacture. A system using the fixed hole and the variable shaft is the British Standard BS 4500 system (partially superseded by BS EN 20286).

The British Standard system also complies with the International Standards Organisation (ISO) requirements to produce international tolerance grade bands or IT numbers. Each IT number or band has the same relative accuracy level, but the values vary depending on the nominal size. Although 18 defined tolerance grade bands for each size group, only IT6 to IT11 are used for preferred fits (smaller grade numbers indicate smaller tolerance zones).

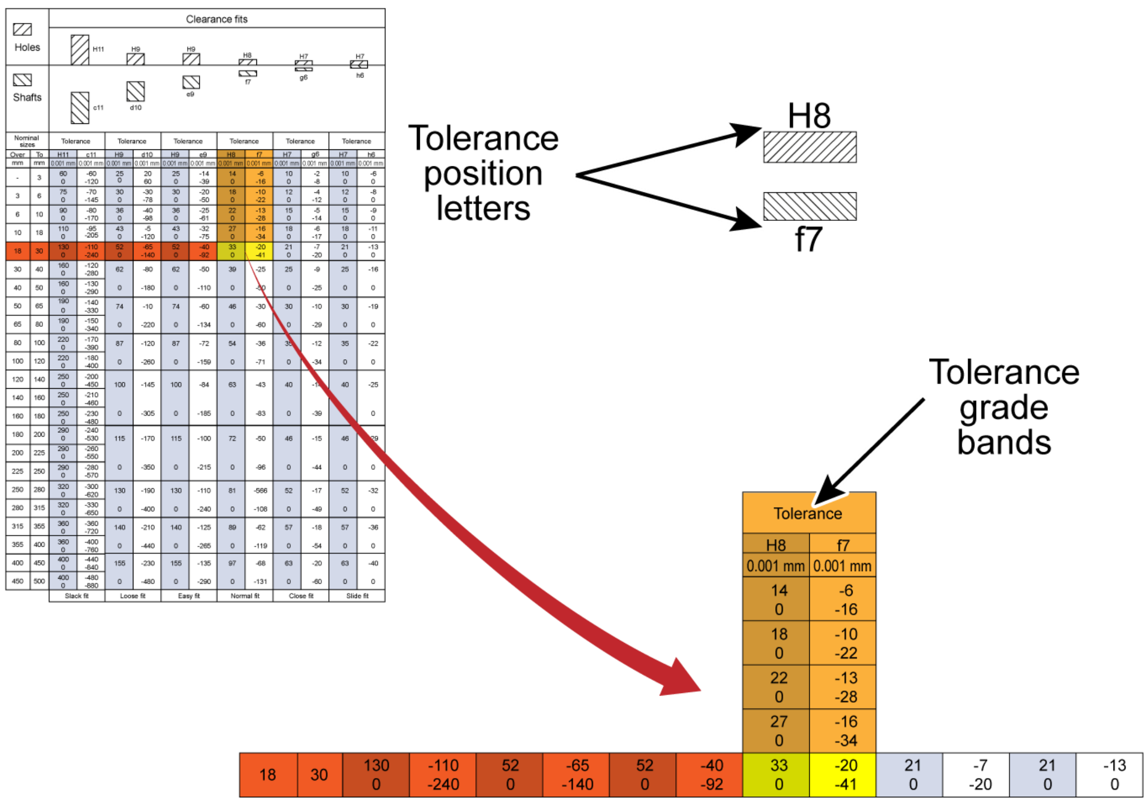
Tolerance position letters are used, capitals for hole (internal) dimensions, and lowercase for shaft (external) dimensions. These indicate the fundamental deviation that locates the tolerance zone relative to the nominal size.

In a shaft basis system of fits, the fundamental deviation is ‘f’ relating to a nominal shaft. In a hole basis system of fits, the fundamental deviation is ‘H’, relating to nominal hole size.

Example: A 20 mm nominal diameter journal/shaft is to have clearance but a close, accurate running fit. An H8/f7 fit is suitable.

From the BS chart, for a 20 mm diameter nominal size (18-30 mm), the H8 limits are +0.033 and -0.000 mm, and the f7 limits are -0.020 and -0.041 mm.

Hence the hole diameter should be between 20.000 and 20.033 mm, and the shaft diameter should be between 19.959 and 19.980 mm.


Clearance fits
Holes
Shafts
Tolerance position letters
H8
f7
Tolerance grade bands
Tolerance
H8
f7
0.001 mm
0.001 mm
14 0
-6 -16
18 0
-10 -22
22 0
-13 -28
27 0
-16 -34
18
30
130 0
-110 -240
52 0
-65 -140
52 0
-40 -92
33 0
-20 -41
21 0
-7 -20
21 0
-13 0

ISO limits and fits

Fits have been standardised and can be taken directly from those tabulated in the BS 4500 standard, ISO limits and fits.

The BS 4500 standard refers to tolerance symbols of letters followed by numbers. The BS Data Sheet BS 4500A, shown on the following two pages, shows a range of fits derived, using the hole basis, from the following tolerances.

Holes: H11 H9 H8 H7

Shafts: c11 d10 e9 f7 g6 k6 n6 p6 s6

Remember:

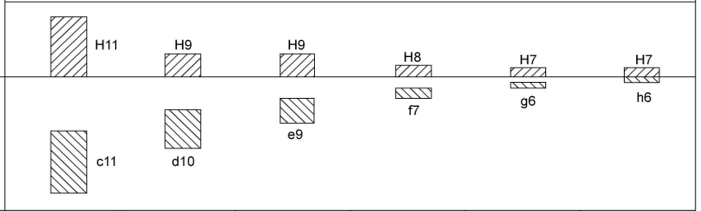
- capital letters always refer to holes; lowercase letters always refer to shafts; and
- the greater the number, the greater or wider the tolerances.

The selection of a pair of these tolerances gives you the fit. The number of possible combinations is vast. BS 4500 helps to standardise this and offers a range of fits suitable for most engineering applications.

Clearance fits

Holes

Shafts

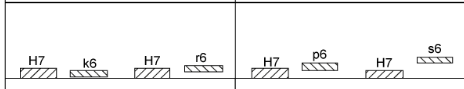


Nominal sizes		Tolerance		Tolerance		Tolerance		Tolerance		Tolerance		Tolerance	
Over	To	H11	c11	H9	d10	H9	e9	H8	f7	H7	g6	H7	h6
mm	mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm
-	3	60 0	-60 -120	25 0	20 60	25 0	-14 -39	14 0	-6 -16	10 0	-2 -8	10 0	-6 0
3	6	75 0	-70 -145	30 0	-30 -78	30 0	-20 -50	18 0	-10 -22	12 0	-4 -12	12 0	-8 0
6	10	90 0	-80 -170	36 0	-40 -98	36 0	-25 -61	22 0	-13 -28	15 0	-5 -14	15 0	-9 0
10	18	110 0	-95 -205	43 0	-5 -120	43 0	-32 -75	27 0	-16 -34	18 0	-6 -17	18 0	-11 0
18	30	130 0	-110 -240	52 0	-65 -140	52 0	-40 -92	33 0	-20 -41	21 0	-7 -20	21 0	-13 0
30	40	160 0	-120 -280	62	-80	62	-50	39	-25	25	-9	25	-16
40	50	160 0	-130 -290	0	-180	0	-110	0	-50	0	-25	0	0
50	65	190 0	-140 -330	74	-10	74	-60	46	-30	30	-10	30	-19
65	80	190 0	-150 -340	0	-220	0	-134	0	-60	0	-29	0	0
80	100	220 0	-170 -390	87	-120	87	-72	54	-36	35	-12	35	-22
100	120	220 0	-180 -400	0	-260	0	-159	0	-71	0	-34	0	0
120	140	250 0	-200 -450	100	-145	100	-84	63	-43	40	-14	40	-25
140	160	250 0	-210 -460										
160	180	250 0	-230 -480	0	-305	0	-185	0	-83	0	-39	0	0
180	200	290 0	-240 -530	115	-170	115	-100	72	-50	46	-15	46	-29
200	225	290 0	-260 -550										
225	250	290 0	-280 -570	0	-350	0	-215	0	-96	0	-44	0	0
250	280	320 0	-300 -620	130	-190	130	-110	81	-566	52	-17	52	-32
280	315	320 0	-330 -650	0	-400	0	-240	0	-108	0	-49	0	0
315	355	360 0	-360 -720	140	-210	140	-125	89	-62	57	-18	57	-36
355	400	360 0	-400 -760	0	-440	0	-265	0	-119	0	-54	0	0
400	450	400 0	-440 -840	155	-230	155	-135	97	-68	63	-20	63	-40
450	500	400 0	-480 -880	0	-480	0	-290	0	-131	0	-60	0	0
		Slack fit		Loose fit		Easy fit		Normal fit		Close fit		Slide fit	

Transition fits / Interference fits

Holes

Shafts



Tolerance		Tolerance		Tolerance		Tolerance			
H7	k6	H7	r6	H7	p6	H7	s6	Over	To
0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	mm	mm
10 0	6 0	10 0	10 0	10 0	12 0	10 0	20 0	-	3
12 0	9 0	12 0	16 8	12 0	20 12	12 0	27 19	3	6
15 0	10 1	15 0	19 10	15 0	24 15	15 0	32 20	6	10
18 0	12 0	18 0	23 0	18 0	29 0	18 0	39 0	10	18
21 0	25 2	21 0	23 15	21 0	35 22	21 0	48 35	18	30
25	18	25	33	25	42	25	59	30	40
0	2	0	17	0	26	0	7	40	50
30	21	30	39	30	51	30 0	72 0	50	65
0	2	0	20	0	32	30 0	78 59	65	80
35	25	35	45	35	59	35 0	93 91	80	100
0	3	0	23	0	37	35 0	101 79	10	120
0	23	40	52	40	68	40 0	117 92	120	140
						40 0	125 100	140	160
0	3	0	27	0	43	40 0	133 108	160	180
46	33	46	60	46	79	46 0	151 122	180	200
						46 0	159 130	200	225
0	4	0	31	0	50	46 0	169 140	225	250
52	36	52	66	52	88	52 0	190 158	250	280
0	4	0	34	0	56	52 0	202 170	280	315
57	40	57	73	57	98	57 0	226 190	315	355
0	4	0	37	0	62	57 0	244 208	355	400
63	45	63	80	63	108	63 0	272 232	400	450
0	5	0	40	0	0	63 0	292 252	450	500
Push fit		Drive fit		Press fit		Force fit			

Selected ISO fits - hole basis. Extract from BS 4500, data sheet 4500A

Determining working limits

Consider an example of a shaft and housing used in a linkage.

- Type of fit: ‘normal’ clearance fit.
- Basic or nominal size: Dia. 40mm

We determine the actual working limits, and the range of allowable sizes, for the shaft and the hole in the housing.

Look along the bottom of the ISO Fits Data Sheet 4500A and locate “Normal Fit”. We use this pair of columns to extract our tolerances.

The tolerances indicated are:

- 1st column H8 for the hole (uppercase H)
- 2nd column f7 for the shaft (lowercase f)

The actual tolerances depend upon the basic or nominal diameter as well as the class of fit. So, locate 40 mm in the left-hand nominal sizes column. The 30 – 40 or 40 – 50 range is acceptable. Read across and note the tolerance values for the hole and the shaft, as shown below.

For the hole diameter, we have a tolerance of:
+0.039mm -0.000mm

For the shaft diameter, we have a tolerance:
of: -0.025mm -0.050mm

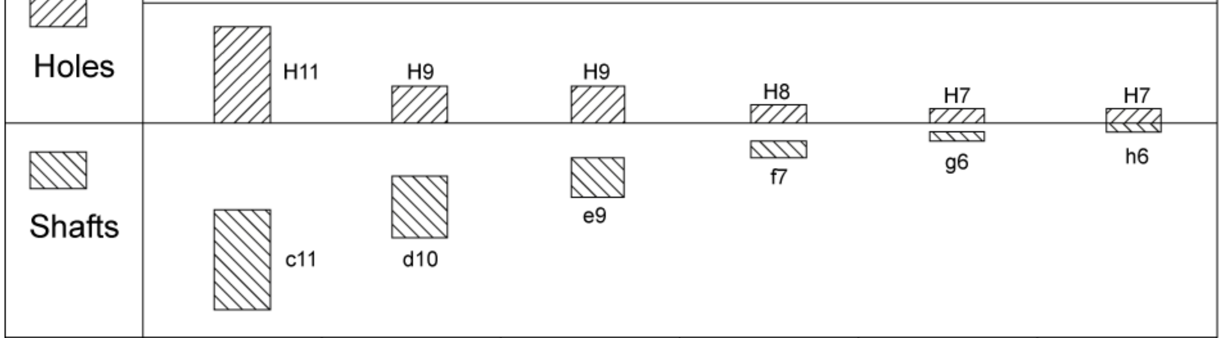
These tolerance values are added to the nominal size to obtain the actual allowable sizes.

Note that this is a clearance fit. As long as the hole and shaft are manufactured within the specified tolerances, the hole is always either slightly oversize or spot on the nominal size, and the shaft is always slightly under-size. This ensures that there is always a free clearance fit.

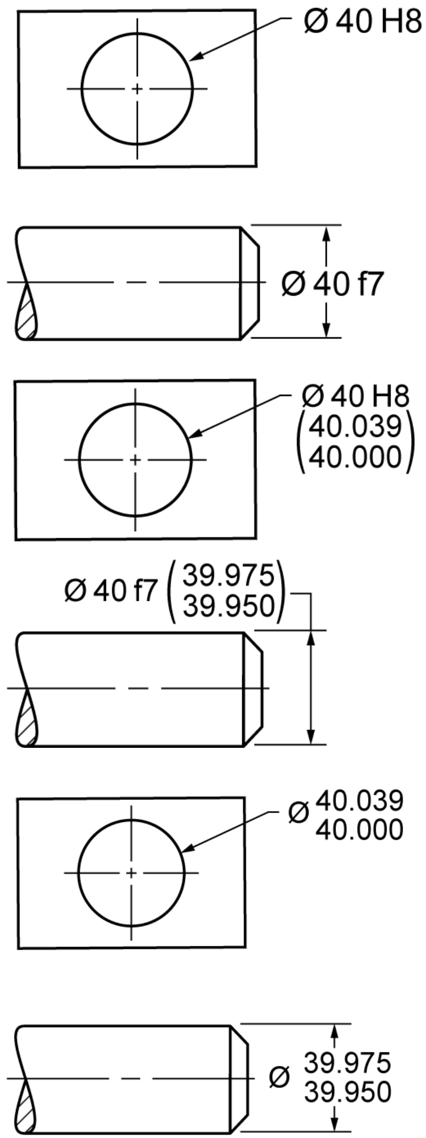
These tolerances may be expressed on a drawing in several ways:

1) Simply as the nominal size with the tolerance class. This is not always preferred, as the machine operator has to calculate the working limits.
2) The nominal size with the tolerance class as above and the calculated working limits are included.
3) The calculated working limits only.

Clearance fits



Nominal sizes		Tolerance		Tolerance		Tolerance		Tolerance		Tolerance		Tolerance	
Over	To	H11	c11	H9	d10	H9	e9	H8	f7	H7	g6	H7	h6
mm	mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm
-	3	60 0	-60 -120	25 0	20 60	25 0	-14 -39	14 0	-6 -16	10 0	-2 -8	10 0	-6 0
3	6	75 0	-70 -145	30 0	-30 -78	30 0	-20 -50	18 0	-10 -22	12 0	-4 -12	12 0	-8 0
6	10	90 0	-80 -170	36 0	-40 -98	36 0	-25 -61	22 0	-13 -28	15 0	-5 -14	15 0	-9 0
10	18	110 0	-95 -205	43 0	-5 -120	43 0	-32 -75	27 0	-16 -34	18 0	-6 -17	18 0	-11 0
18	30	130 0	-110 -240	52 0	-65 -140	52 0	-40 -92	33 0	-20 -41	21 0	-7 -20	21 0	-13 0
30	40	160 0	-120 -280	62	-80	62	-50	39	-25	25	-9	25	-16
40	50	160 0	-130 -290	0	-180	0	-110	0	-50	0	-25	0	0
50	65	190 [illegible]	-140 [illegible]	74	-10	74	-60	46	-30	30	-10	30	-19
400	450	[illegible] 0	[illegible] -840	155	-230	155	-135	97	-68	63	-20	63	-40
450	500	400 0	-480 -880	0	-480	0	-290	0	-131	0	-60	0	0
		Slack fit		Loose fit		Easy fit		Normal fit		Close fit		Slide fit	



‘Basic hole’ and ‘basic shaft’ systems

The principles of fit are illustrated in the figure below.

A specified fit is achieved in practice by controlling the relative size and tolerance of each of the two mating parts.

The main figure shows the condition where the basic size of the hole remains constant, and the basic size of the shaft varies to achieve the desired fit condition. This is the most commonly used condition for fits, termed **basic hole system**.

With the basic hole system, the minimum hole is taken as the basic size, an offset from this is assigned, and tolerances are applied on both sides of and away from this offset.

A less-used alternative is the **basic shaft system,** in which the maximum shaft is taken as the basic size.

The following terms apply for ISO metric fits and to BS4500.

Basic size, or dimension, is the theoretical size of the part (hole or shaft) from which the application of offsets and tolerances derives the actual size limits. For a fit, the value of basic size is the same for the two mating parts.

Deviation is the difference between the basic size and the hole or shaft permitted toleranced size; it comprises 2 components:

- **Upper deviation** is the difference between the part’s basic and permitted maximum sizes.
- **Lower deviation** is the difference between the part’s basic and minimum permitted size.

Tolerance is the difference between a part’s permitted minimum and maximum sizes.

Actual size is the measured size of the finished part.

Nominal size is the designation used for convenient general identification and is usually expressed in common round values. The actual size is usually slightly different from the Nominal.

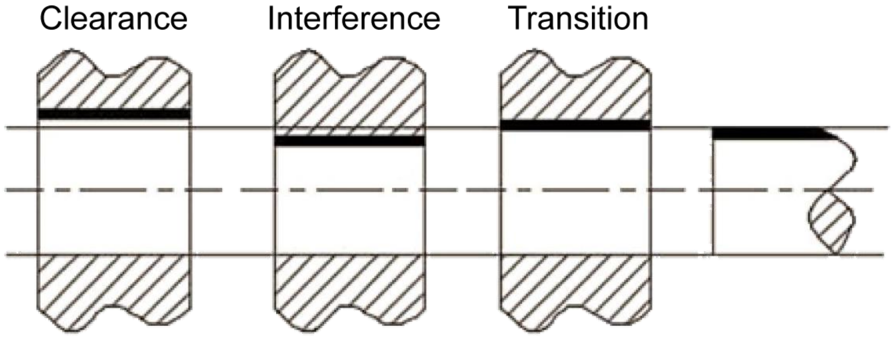
Allowance refers to the mating condition of the two parts. It represents the tightest permissible fit and is simply the smallest hole minus the largest shaft. This difference is positive (minimum clearance) for clearance fits, while for interference fits, it is negative (maximum interference).

For example, if a shaft with a nominal diameter of 10 mm is to have a sliding fit within a hole, the shaft might be specified with a tolerance range from 9.964 to 10 mm (i.e., a zero fundamental deviation but a lower deviation of 0.036 mm) and the hole might be specified with a tolerance range from 10.04 mm to 10.076 mm (0.04 mm fundamental deviation and 0.076 mm upper deviation).

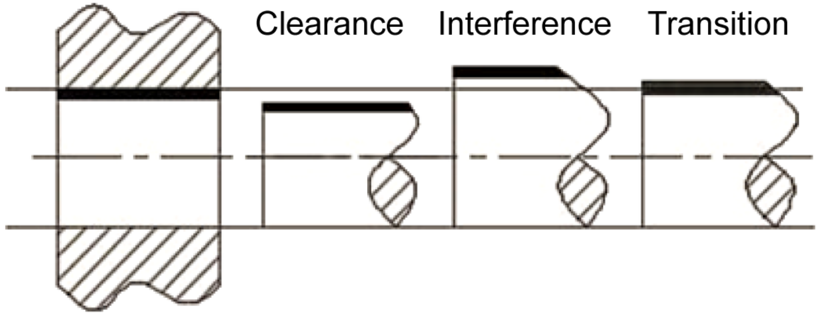
This would provide a clearance fit between 0.04 mm (largest shaft paired with the smallest hole, called the maximum material condition; MMC) and 0.112 mm (smallest shaft paired with the largest hole, least material condition; LMC).

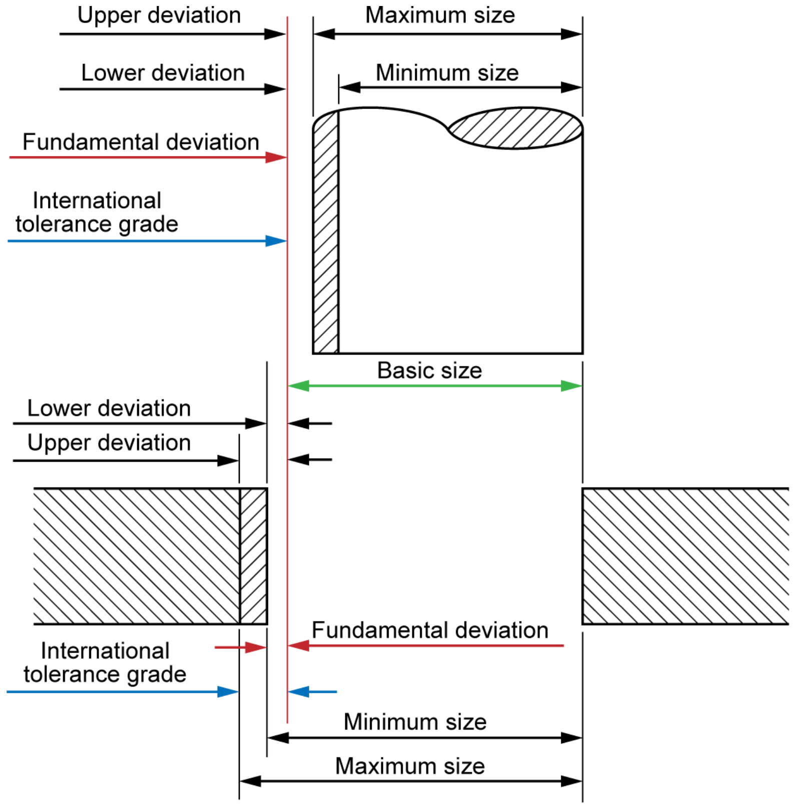
In this case, the size of the tolerance range for both the shaft and hole is chosen to be the same (0.036 mm), meaning that both components have the same International Tolerance grade, but this need not be the case in general.

Shaft system



Hole system





Size nomenclature

International tolerance grades

A system of standardised tolerances called International Tolerance grades is often used when designing mechanical components. The standard (size) tolerances are divided into two categories: hole and shaft.

They are labelled with a letter (capitals for holes and lowercase for shafts) and a number, for example, H7 (hole, tapped hole, or nut) and h7 (shaft or bolt). H7/h6 is a common standard tolerance that gives a tight fit.

The tolerances work in such a way that for a hole H7 means that the hole should be made slightly larger than the base dimension (in this case, for an ISO fit 10 +0.015 −0, meaning that it may be up to 0.015 mm larger than the base dimension, and 0 mm smaller).

The actual amount bigger/smaller depends on the base dimension. For a shaft of the same size, h6 would mean 10 +0 −0.009, which means the shaft may be as small as 0.009 mm smaller than the base dimension and 0 mm larger.

This standard tolerances method is known as limits and fits and can be found in ISO 286.

The table below summarises the international tolerance (IT) grades and the general applications of these grades:

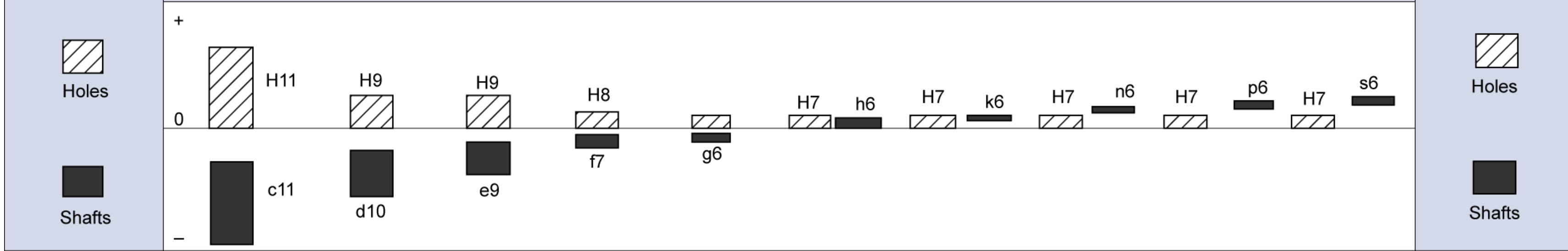
Nominal sizes mm		H7		H8		H9		H11	
Over	Up to and including	ES +	EI	ES +	EI	ES +	EI	ES +	EI
...	3	10	0	14	0	25	0	60	0
3	6	12	0	18	0	30	0	75	0
10	18	18	0	27	0	43	0	110	0
18	30	21	0	33	0	52	0	130	0
30	50	25	0	39	0	62	0	160	0
50	80	30	0	46	0	74	0	190	0
80	120	35	0	54	0	87	0	220	0
120	180	40	0	63	0	100	0	250	0
180	250	46	0	72	0	115	0	290	0
250	315	52	0	81	0	130	0	320	0
315	400	57	0	89	0	140	0	360	0
400	500	63	0	97	0	155	0	400	0

British Standard limits of tolerance for selected holes (upper and lower deviations) BS 4500:1969

ES = upper deviation, EI = lower deviation.

The dimensions are given in 0.001 mm, except for the nominal sizes, which are in millimeters.

	Measuring tools									Material								
IT grade	01	0	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
							Fits							Large manufacturing tolerances				



Basic size (mm)		Upper and lower deviations for tolerance class (values µm)																				Basic size (mm)	
		H11	c11	H9	d10	H9	e9	H8	f7	H7	g6	H7	h6	H7	k6	H7	n6	H7	p6	H7	s6		
Above	Up to and incl.	+	–	+	–	+	–	+	–	+	–	+	–	+	–	+	–	+	–	+	–	Above	Up to and incl.
0	3	60 0	60 120	25 0	20 60	25 0	14 39	14 0	6 16	10 0	2 8	10 0	6 0	10 0	6 0	10 0	10 4	10 0	12 6	10 0	20 14	0	3
3	6	75 0	70 145	30 0	30 78	30 0	20 50	18 0	10 22	12 0	4 12	15 0	9 0	15 0	10 1	15 0	19 10	15 0	24 15	15 0	32 23	3	6
6	10	90 0	80 170	36 0	40 96	36 0	25 61	22 0	13 28	15 0	5 14	18 0	11 0	18 0	12 1	18 0	23 12	18 0	29 18	18 0	39 28	6	10
10	18	110 0	95 205	43 0	50 120	43 0	32 75	27 0	16 34	18 0	6 17	21 0	13 0	21 0	15 2	21 0	28 15	21 0	35 22	21 0	48 35	10	18
18	30	130 0	110 240																			18	30
30	40	160 0	120 280	62 0	80 180	62 0	50 112	39 0	25 15	25 0	9 25	25 0	16 0	25 0	18 2	25 0	33 17	25 0	42 26	25 0	59 43	30	40
40	50	160 0	130 290																			40	50
50	65	190 0	140 330	74 0	100 220	74 0	60 134	46 0	30 60	30 0	10 29	30 0	19 0	30 0	21 2	30 0	39 20	30 0	51 32	30 0	72 53	50	65
65	80	190 0	150 340																	30 0	78 59	65	80
80	100	220 0	170 390	87 0	120 260	87 0	72 159	54 0	36 71	35 0	12 34	35 0	22 0	35 0	25 3	35 0	45 23	35 0	5 37	35 0	93 71	80	100
100	120	220 0	180 400																	35 0	101 79	100	120
120	140	250 0	200 450																	40 0	117 92	120	140
140	160	250 0	210 400	100 0	145 305	100 0	84 185	62 0	43 83	40 0	14 39	40 0	25 0	40 0	28 3	40 0	52 27	40 0	68 43	40 0	125 100	140	160
160	180	250 0	230 480																	40 0	133 106	160	180
180	200	290 0	240 530																	46 0	151 122	180	200
200	225	290 0	260 550	115 0	170 355	115 0	100 215	72 0	50 96	46 0	15 44	46 0	29 0	46 0	33 4	46 0	90 31	46 0	79 50	45 0	159 130	200	225
225	250	290 0	280 570																	46 0	169 140	225	250
250	280	320 0	300 630	130 0	190 400	130 0	110 240	81 0	56 108	52 0	17 49	52 0	32 0	52 0	36 4	52 0	66 34	52 0	88 56	52 0	190 158	250	280
280	315	320 0	330 650																	52 0	202 170	280	315
315	355	360 0	360 720	140 0	210 440	140 0	125 265	89 0	62 119	57 0	18 54	57 0	36 0	57 0	40 4	57 0	73 37	57 0	96 62	57 0	226 190	315	355
355	400	360 0	400 760																	57 0	224 208	355	400
400	450	400 0	440 840	155 0	230 480	155 0	135 290	97 0	68 131	63 0	20 60	63 0	40 0	60 0	45 5	63 0	80 40	63 0	106 68	63 0	272 232	400	450
450	500	400 0	480 880																	63 0	292 252	450	500

Selected ISO 286 fits (BS 4500) – hole basis

Standard values for designated tolerances

With engineering fits, the tolerance is always shown in an alphanumeric code. For example, a hole tolerance may be H7.

The number shows the international tolerance grade (ISO 286). A tolerance class determines a range of values. The final measurement can vary from the base measurement.

The capital letter signifies that we are dealing with a hole. When indicating tolerance for a shaft, the letter is lowercase.

Each tolerance condition is characterised by its offset from the basic size and the extent (range) of the tolerance values.

The principles above are made specific by a prescribed set of tolerance values. Refer to the table below. Note specific examples such as H8, c11. These letter/number pairings specify the tolerance, termed a tolerance zone.

The tolerance zone refers to the relationship of tolerance to the basic size.

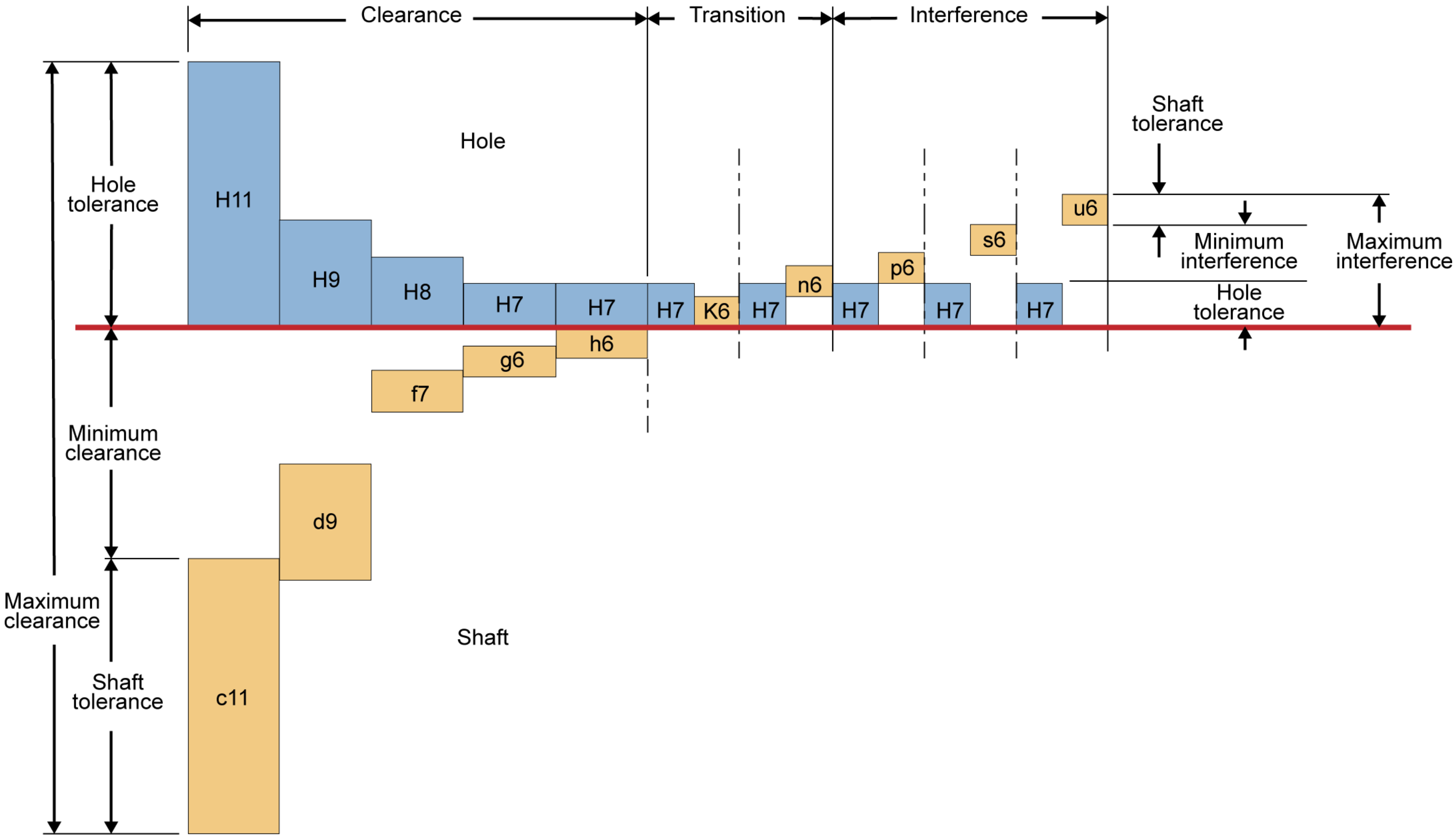
The offset is termed by ISO as the fundamental deviation and is indicated by a letter. To distinguish holes from shafts, holes are always designated with upper case letters and shafts with lower case. The full range spans letters ‘A’ – ‘Z’, centred at ‘H’ (and lowercase for shafts). For holes, ‘A’ yields a hole well above the basic size and ‘V’ well below. For shafts, this is reversed.

The range of the tolerance values is specified by the international tolerance grade (IT) and is indicated by a number from 0 – 16. Each number provides a consistent level of accuracy within the grade, with 0 giving a very tight tolerance range and 16 a slack tolerance range.

Thus, the combination of fundamental deviation and tolerance grade uniquely defines a tolerance zone.

Hole basis	Shaft basis	Type of fit	Description
H11/c11	C11/h11	Clearance fit	Loose running fit for wide commercial tolerances or allowances on external members.
H9/d9	D9/h9		Free running fit not for use where accuracy is essential, but good for large temperature variations, high running speeds, or heavy journal pressures.
H8/f7	F8/h7		Close running fit for running on accurate machines and for accurate moderate speeds and journal pressures.
H7/g6	G7/h6		Sliding fit not intended to run freely, but to move and turn freely and locate accurately.
H7/h6	H7/h6		Locational clearance fit provides snug fit for locating stationary parts; but can be freely assembled and disassembled.
H7/k6	K7/h6	Transition fit	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.
H7/p6	P7/h6	Interference fit	Locational interference fit for parts requiring rigidity and alignment with prime accuracy of location but without special bore pressure requirements.
H7/s6	S7/h6		Medium drive fit for ordinary steel parts or shrink fits on light sections, the tightest fit usable with cast iron.
H7/u6	U7/h6		Force fit suitable for parts which can be highly stressed or for shrink fits where the heavy pressing forces required are impractical.

Fits definitions



Standardised by BS4500: ISO units and fits

Clearance fits

With a clearance fit, the shaft is always smaller than the hole. This enables easy assembly and leaves room for sliding and rotational movement.

When the shaft diameter is at its minimum and the hole diameter at its maximum, we have a situation of maximum clearance. When the shaft diameter is at its maximum and the hole diameter at its minimum, we have a situation of minimum clearance.

Clearance fits come in 6 sub-categories. Starting from the loosest:

- loose running;
- free running;
- close running;
- sliding;
- close clearance; and
- locational clearance

Loose running fit

Fit with the largest clearance. Suitable for applications where accuracy is not of the utmost importance and contamination may be a problem.

Example uses in engineering: fits exposed to dust contamination, corrosion, thermal and mechanical deformations, e.g. pivots and latches.

Example fits: H11/c11, H11/a11, H11/d11 (all hole-basis), C11/h11, A11/h11, D11/h11 (all shaft-basis).

Using a 25 mm diameter, an H11/c11 fit gives a minimum clearance of 0.11 mm and a maximum clearance of 0.37 mm. The shaft diameter can fall between 24.76 and 24.89 mm, while the minimum hole size is 25 mm and the maximum 25.13 mm.

Free running fit

Suitable where no special requirements apply to the accuracy of matching parts. It leaves room for movement in environments with heavy temperature fluctuations, high running speeds and heavy plain bearing pressures.

Example uses in engineering: applications where maintaining a film of oil lubrication is important. For example, the shaft and plain bearing fit with little rotational movement.

Example fits: H9/d9, H9/c9, H9/d10 (all hole-basis), D9/h9, D9/h8, D10/h9 (all shaft-basis).

Using a 25 mm diameter, an H9/d9 fit gives a minimum clearance of 0.065 mm and a maximum clearance of 0.169 mm.

Close running fit

Close-running fits are a good choice for applications that require smaller clearances and moderate accuracy. Suitable for withstanding medium speeds and pressures.

Example uses in engineering: machine tools, sliding rods and machine tool spindles.

Example fits: H8/f8, H9/f8, H7/f7 (all hole-basis), F8/h6, F8/h7 (all shaft-basis).

Using a 25 mm diameter, an H8/f7 fit gives a minimum clearance of 0.020 mm and a maximum clearance of 0.074 mm.

Sliding fit

It leaves a small clearance for high accuracy while maintaining ease of assembly. Parts can turn and slide quite freely.

Example uses in engineering: guiding of shafts, sliding gears, slide valves, automobile assemblies, clutch discs and parts of machine tools.

Example fits: H7/g6, H8/g7 (all hole-basis), G7/h6 (shaft-basis).

Using a 25 mm diameter, an H7/g6 fit gives a minimum clearance of 0.007 mm and a maximum clearance of 0.041 mm.

Locational clearance fit

Location clearance fits provide minimal clearance for high accuracy requirements. Provides a snug fit for stationary parts. The assembly does not need any force, and the mating parts can turn and slide freely with lubrication, helping with assembly by hand.

Example uses in engineering: roller guides and guiding shafts.

Example fits: H7/h6, H8/h7, H8/h9, H8/h8 (all hole-basis).

Using a 25 mm diameter, an H7/h6 fit gives a minimum clearance of 0.000 mm and a maximum clearance of 0.034 mm.

Transition fits
A transition fit encompasses two possibilities. The shaft may be slightly bigger than the hole, requiring some force to create the fit. At the other end of the spectrum is a clearance fit with a little bit of room for movement.

Specifying a transition fit means that both outcomes are possible even inside a single batch.

Transition fits come in two forms – similar fit and fixed fit.

Similar fit
It leaves a small clearance or creates a slight interference. Assembly is possible using a rubber mallet.

Example uses: hubs, gears, pulleys and bearings.

Example fits: H7/k6 for hole-basis and K7/h6 for shaft-basis

Using a 25 mm diameter, an H7/k6 fit gives a maximum clearance of 0.019 mm and a maximum interference of 0.015 mm.

Fixed fit
It leaves a small clearance or creates a slight interference. Assembly is possible using light force.

Example uses in engineering: driven bushes and armatures on shafts.

Example fits: H7/n6 for hole-basis and N7/h6 for shaft-basis.

Using a 25 mm diameter, an H7/n6 fit gives a maximum clearance of 0.006 mm and a maximum interference of 0.028 mm.

Interference fits

Interference fits are also known as press fits or friction fits. These types of fits always have the same principle of having a larger shaft compared to the hole size.

The assembly stage requires force, sometimes lubrication, heating of the hole and freezing the shaft. These help to increase/decrease the hole and shaft sizes, respectively, to make for an easier process.

The interference helps secure the relative positioning of the shaft and hub even during rotation, making this fit good for transmitting rotational speed and power.

Press fit

Minimal interference. Assembly can be performed with cold pressing.

Example uses in engineering: hubs, bushings and bearings.

Example fits: H7/p6 for hole-basis, P7/h6 for shaft-basis.

Using a 25 mm diameter, an H7/p6 fit gives a minimum interference of 0.001 mm and a maximum interference of 0.035 mm.

Driving fit

Needs higher assembly forces for cold pressing. Another way is by using hot pressing. This interference fit is more prominent than with a press fit.

Example uses in engineering: permanent mounting of gears, shafts and bushes.

Example fits: H7/s6 for hole-basis, S7/h6 for shaft-basis.

Using a 25 mm diameter, an H7/s6 fit gives a minimum interference of 0.014 mm and a maximum interference of 0.048 mm.

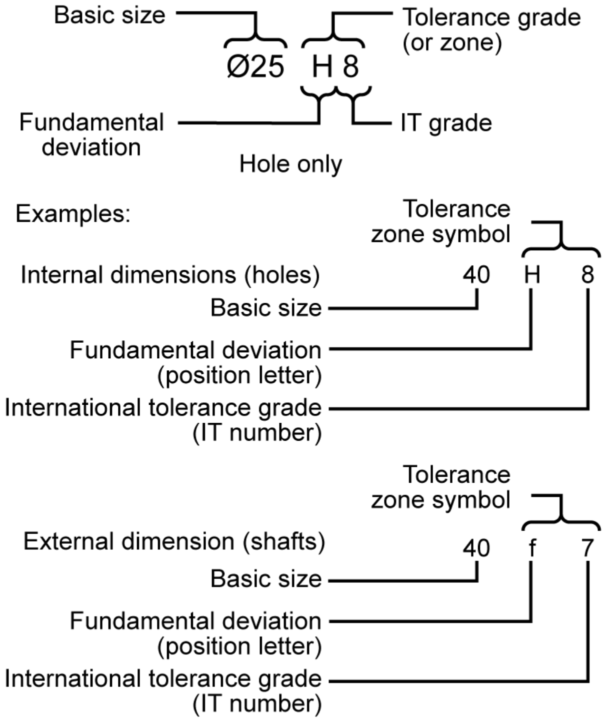
Forced fit

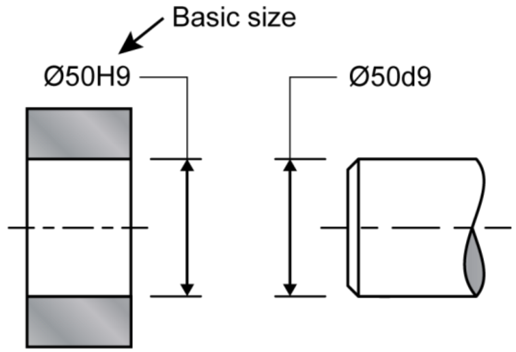
High interference fit. Assembly requires heating the part with a hole and freezing the shaft to force the mating parts together. Disassembly can result in broken parts.

Example uses in engineering: shafts and gears.

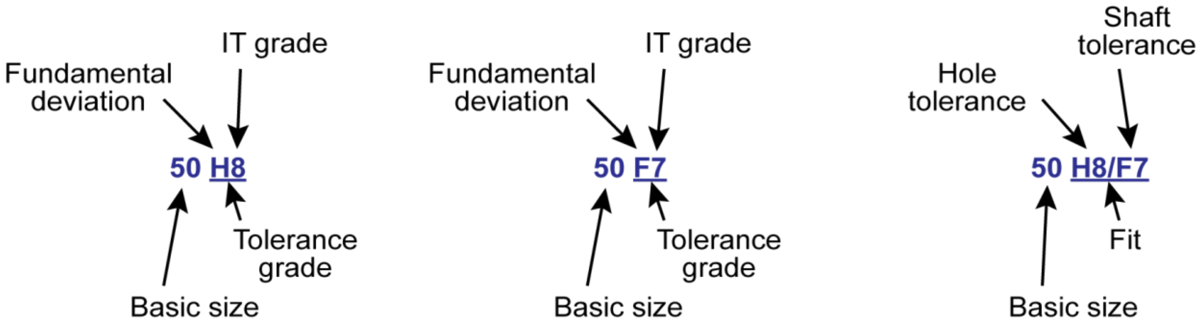
Example fits: H7/u6 for hole-basis, U7/h6 for shaft-basis

Using a 25 mm diameter, an H7/u6 fit gives a minimum interference of 0.027 mm and a maximum interference of 0.061 mm.





Basic hole method – metric



(a) Hole (b) Shaft (c) Fit

Hole and shaft tolerance system examples

Limits for ovality, bow, twist and wear

General
Wear occurs at any time that there is motion between two parts. This motion can be intentional, such as when a shaft rotates in a plain (journal) bearing or when a roller moves back and forth over a track. Wear can also be accidental, where two parts that should be immovable chafe together.

If the parts are intended to move together, the maintenance documentation will have a schedule of fits and clearances based on the limit system issued for each mechanism used on the aircraft.

If the parts are not intended to move together, it relies on inspection procedures to discover the problem and repair schemes to be initiated to prevent a recurrence.

During regular servicing or overhauling of aircraft components, dimensional checks may be carried out with the aid of various precision measuring instruments, e.g. micrometer, Vernier calliper, and dial test indicator. Wear limits may be found in the appropriate repair or overhaul manual.

The schedule of fits and clearances contains tables, which specify the limits on wear and other characteristics such as:

- ovality (of a hole or shaft);
- bow of a shaft; and
- twist of a shaft.

Limits for wear
The four dimensions, typically covered in wear tables, are:

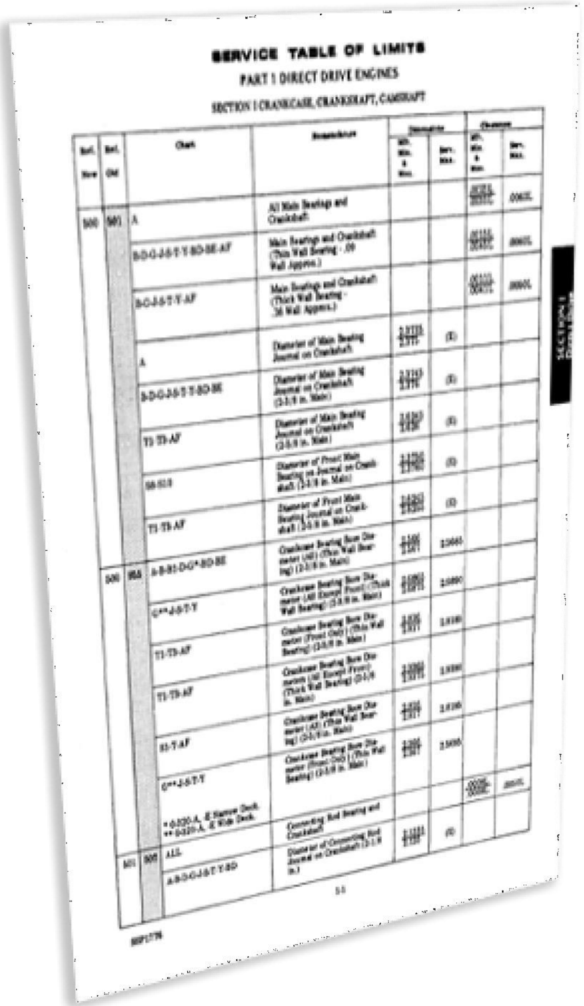
- dimension new;
- permissible worn dimension;
- clearance new; and
- permissible worn clearance.

‘Dimension new’ relates to the size of the part when new and shows the relevant tolerances.

‘Permissible worn dimension’ refers to the size a part may wear before it must be rejected as unserviceable. Parts, which are not worn beyond this size, can be used again, providing a suitable mating part is chosen to keep the clearance within the permissible figure. This frequently involves choosing a new part to mate with the worn part.

‘Clearance new’ is the desired clearance in limit form. Interference fits are quoted as negative clearances.

‘Permissible worn clearance’ refers to the maximum allowable clearance when reassembling the component.


SERVICE TABLE OF LIMITS
PART 1 DIRECT DRIVE ENGINES
SECTION I CRANKCASE, CRANKSHAFT, CAMSHAFT

The overhaul manual provides the limits of wear

Limits for ovality
This usually occurs due to surface wearing, through friction or linear movement. Ovality and can apply equally to holes and shafts.

Holes may be tested for ovality using Go/No-Go gauges, internal micrometers, or callipers, as previously discussed in the tools module.

Shafts may be checked for wear by measuring them using a micrometer. However, roundness cannot be measured by numerous diameter measurements at different points around the circumference. Certain shapes appear round when measured in this manner, although they may not be. Testing the shaft's roundness may be achieved by placing it on a V-block and rotating it under a scribing block or a dial test indicator (this same method may be used to check shafts for bow).

If two or more diameters of a shaft have a common axis and must be true to one another, they are said to be concentric. Concentricity may be tested by placing the shaft in V-blocks or between centres and rotating it under a dial test indicator.

It is essential to test for the ovality of a shaft before testing it for bow, as the results may be suspect if bow is tested first.

Bow in a shaft can be determined, in a workshop, by utilising V-blocks, a surface gauge and a dial test indicator (DTI) in conjunction with a surface table.

Limits for bow
When dealing with structural tubes, it is vital that the ends are square with each other and that the centreline of the complete shaft or tube is straight.

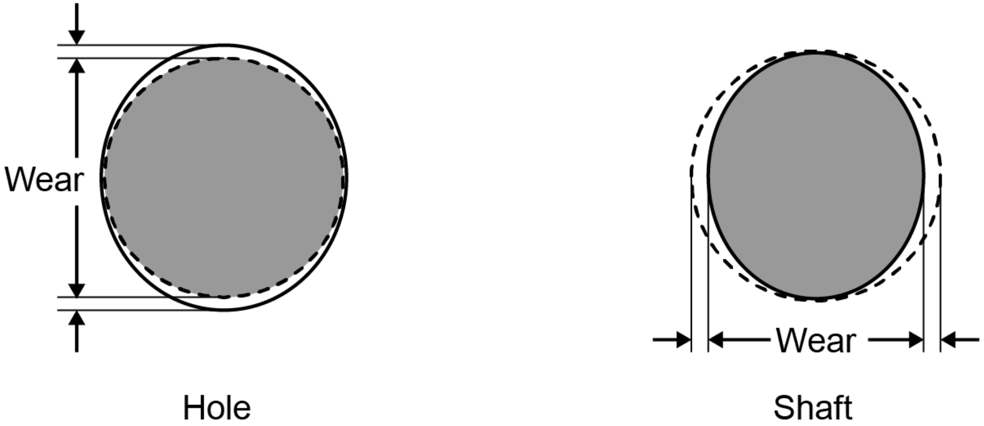
The item is bowed if the shaft's centre line is not straight.

All cylindrical items, both tubular and solid, can be given a limit to the amount of bow permitted.

To measure the amount of bow in a structural member (e.g. a strut), a straight edge and a set of feelers can be used, providing no protruding fittings prevent the straight edge from being applied directly along the surface of the member. The straight edge should be placed along the entire length of the member parallel to its axis. By inserting feeler gauges at the point of maximum clearance, the amount of bow can be calculated by the formula:

$$\frac{\text{Clearance measured by feeler gauges}}{\text{Length of member}}$$

Generally, a maximum bow of a structural member of 1:600 is usually acceptable unless otherwise stated in the repair manual.



Ovality of a hole or a shaft


Measuring journal wear

Limits for twist

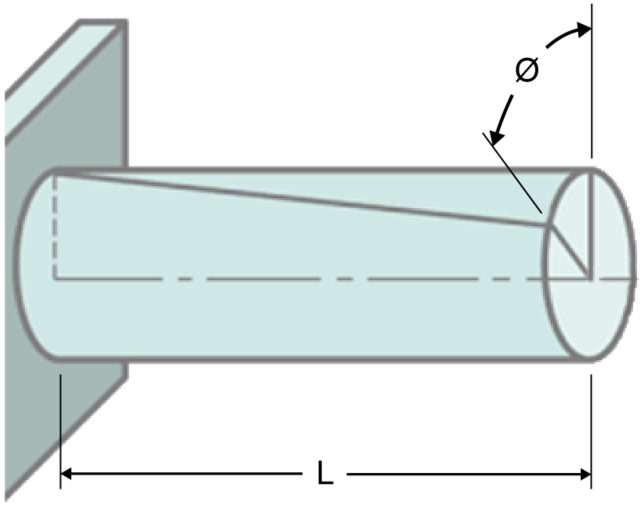
Twist is the result of applied torsion on circular or square-sectioned shafts. If the twist disappears because of removing the force, the shaft was loaded below its elastic limit. If the shaft remains twisted after the removal of the load, then it has been loaded above its elastic limit.

The action of a shaft, of whatever section, carrying a torque load is to twist in proportion to the torque applied. The result of cyclic loading of shafts is that, at certain times, the shafts must be checked for a permanent twist. If the shaft has a square section, it can be checked for a twist on a surface table using a DTI mounted on a surface gauge.

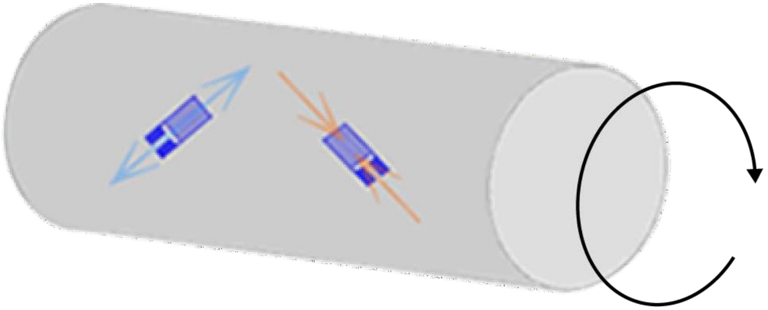
Solid or tubular shafts that must be checked for twist may have witness marks or lines engraved or etched at each end of the shaft. The shafts can be checked by mounting the shaft in V-blocks and then locating these marks in the horizontal position.

Using strain gauges, measuring the amount of twist to which a shaft is subjected whilst in operation or rotation is possible. These emit varying amounts of electric current under strain, indicating the load applied on a calibrated instrument.

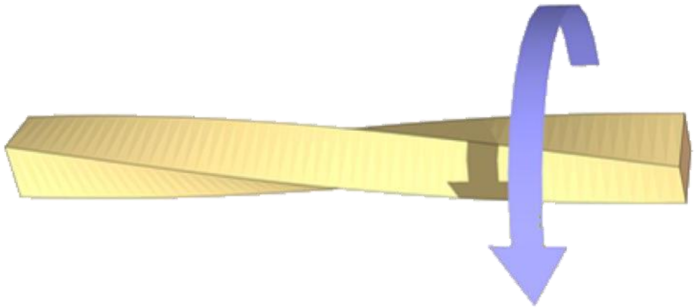
The aircraft or equipment designer sets all limits regarding the distortion of parts and records them in the relevant manuals. The methods used to measure the distortion will either be standard procedures, such as using a DTI and surface table or will have a particular procedure included in the manuals.



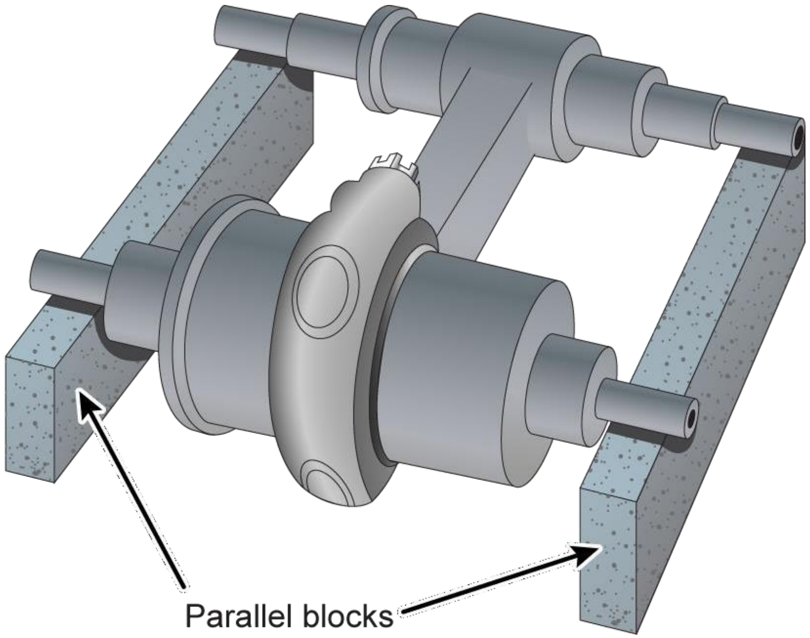
Shaft twist



Strain gauges bonded to a torque shaft



Shaft twist



Checking connecting rod parallelism and squareness

Standard measurement for checking bearings

Principal dimensions

Bearings have three principal dimensions, the inside diameter (ID), the outside diameter (OD) and the width.

Measure the ID

Insert the outer anvils of the calliper (the ones on top of the picture below) into the bore as shown and open the calliper until it is a good fit but not tight; now, read the value on the calliper. Without a calliper, a good-quality steel ruler will suffice.

Measure the OD

To measure the outside diameter of a bearing, place the calliper's jaws around it and close it until it's a good fit but not tight; now, read the value from the calliper.

Measure the width

In the same manner as you measured the outside diameter, measure the width of the bearing.

Bearing internal clearance

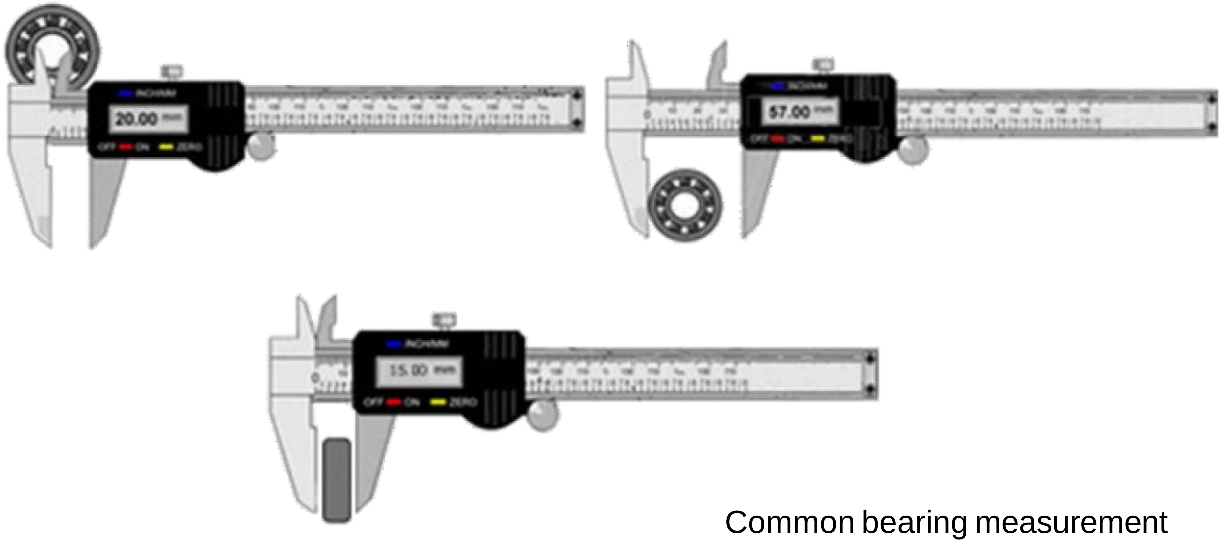
Internal clearance or radial play is the amount of looseness between the balls and raceways of a bearing.

Internal clearance is the play within a ball bearing. It is the geometrical clearance between the inner ring, outer ring and ball. It is a critical factor in bearing selection that directly impacts bearing life. It is often overlooked, particularly how interference fits reduce it.

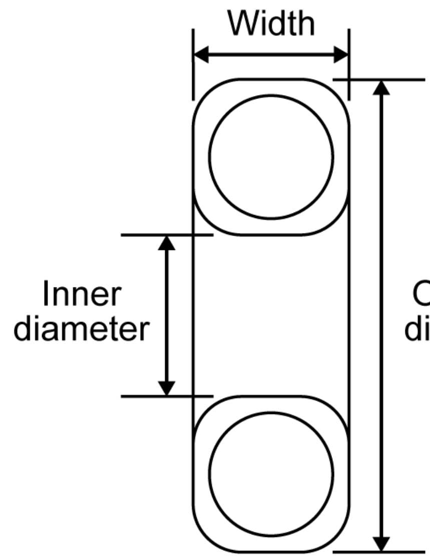
Radial clearance is the play between the ball and raceway perpendicular to the bearing axis. Axial clearance is the play parallel to the bearing axis and is typically at least ten times greater than the radial clearance. Generally, internal radial clearance is reduced by 80% of the interference fit amount.

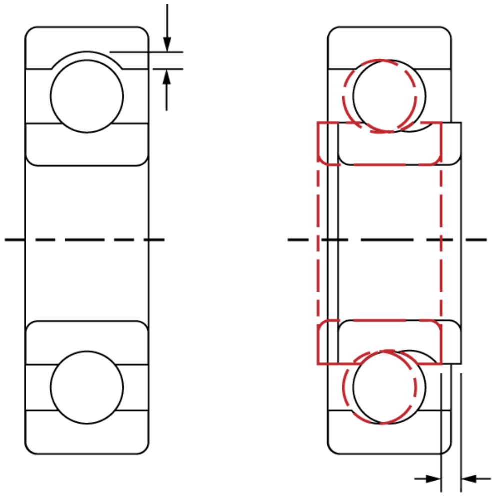
Too little or too much internal clearance significantly influences heat, vibration, noise, and fatigue life.

In extreme applications that see high or low temperatures, this clearance must be considered in the overall design to compensate for the thermal expansion and contraction of housings and shafts.



Common bearing measurement





Radial clearance Axial clearance

Bearing internal clearance

Aligned bearings

The two bearings that support a shaft (all the bearings, if there are more than two) must be coaxial with minimum offset and angularity. The clearance in the bearings can accommodate slight deviations. Once the bearing clearance is fully consumed, a hard stop is encountered with metal pressing against metal. The shaft then deflects and is no longer straight, and the bearings suffer accelerated wear.

The need for precise bearing alignment is well understood by aircraft and engine builders. They control it by line-boring housings, using arbours and fixtures for maintaining alignment during assembly in a clean environment using heat or cold, or a combination of these, to allow insertion without galling metal. In the field, alignment is less controlled during bearing replacements.

Fortunately, there are two simple tests for judging bearing alignment. The first is to rotate the shaft by hand and see that it spins freely. It should spin with minimal friction and no binding during a 360° rotation. The second test requires a DTI; this is used to check the parallelism of the faces and their perpendicularity to the shaft. The two test setups are shown below. One checks that the face of the outer ring is perpendicular to the shaft axis. The indicator is fixed to the shaft with a clamp, and the indicator tip reads on the face of the outer ring as the shaft is rotated. The total indicator run-out should be no more than that specified in the aircraft or engine manual.

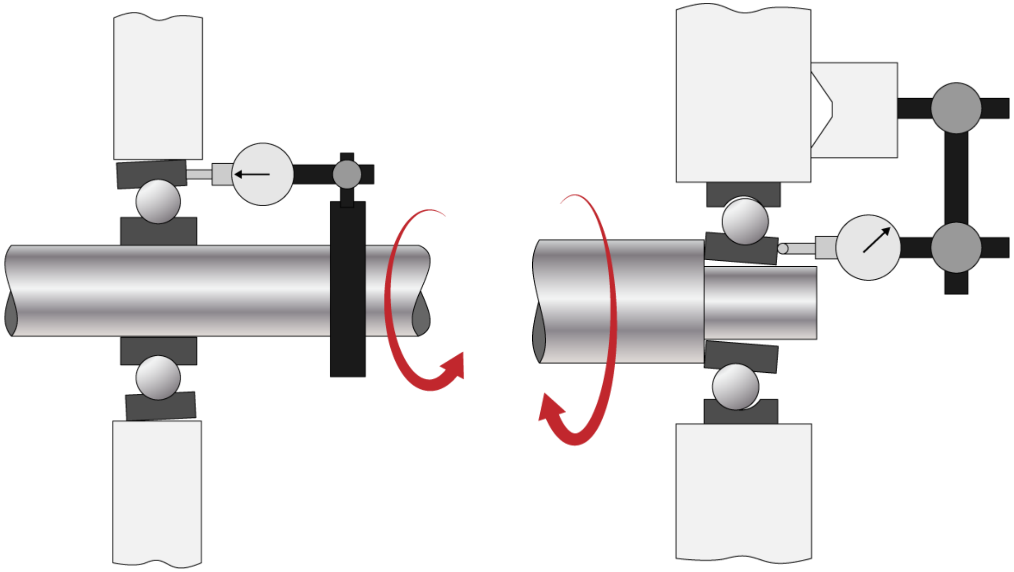
The subsequent test checks that the face of the inner ring is not canted on the shaft or that the shaft is not bent. The DTI is affixed to the housing or a stationary part, and the tip of the indicator reads on the face of the inner ring as the shaft is rotated. The acceptable reading must again be within limits.

These DTI measurements are not always possible if the rings are not accessible, but if they can be done, then adjustments with a punch and hammer, shimming, or repositioning pillow blocks can be done in place while observing the dial indicator. Aligned bearing operation is cool and quiet, and they have a normal life.

A severe condition of bearing misalignment can be introduced during operation as things heat up. The thermal expansion forces can bend the shaft and distort the housing or the frame. Thermal expansion can be measured, but it is often calculated and predicted. We can do nothing to stop thermal expansion. The best we can hope for is accommodating it by providing room for metal to grow. This is an engineering function for the designer, but the mechanic must be aware of it.

Floating or expansion bearings provide this clearance for thermal growth, and their proper setup is part of bearing alignment. If not correctly done, the machine may operate when cold, but vibration and noise worsen as temperatures rise. This is the key indicator to stop the machine before it is too late.

A straight shaft with aligned bearings rotates freely by hand and balances well.



Measurement of bearing alignment

Standard measurement for checking shafts

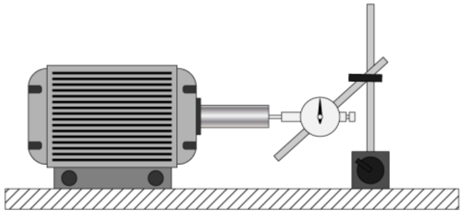
If the centre line of a rotating shaft, such as a crankshaft of an engine, is not straight, this is called ‘run-out’.

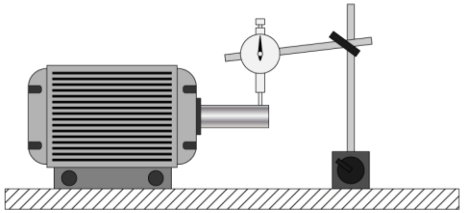
When the shaft is rotating, especially at high speed in a bowed state, there is the risk of vibration, which can lead to mechanical failures, loosening of fasteners and (most critical of all) fatigue.

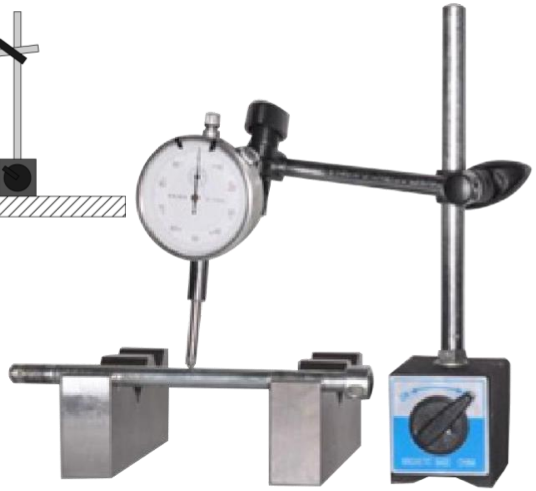
The shaft is placed on a pair of V-blocks, and a DTI is set up on its centre position. The shaft is then slowly rotated through 360°. The total deflection of the pointer on the DTI is the total run-out of the shaft. The circular run-out is calculated by dividing the total run-out by two.

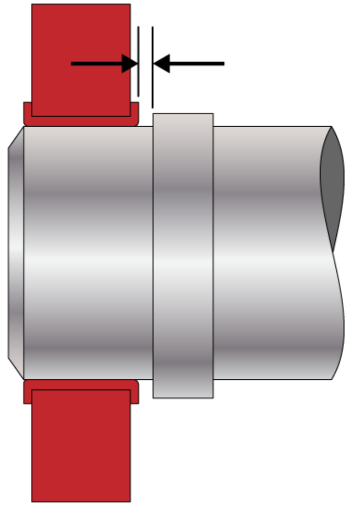
End float is associated with the run-out of a shaft and is often measured simultaneously. End float is the axial play in a rotating shaft, often caused by wear or misfit of the shaft bearings.

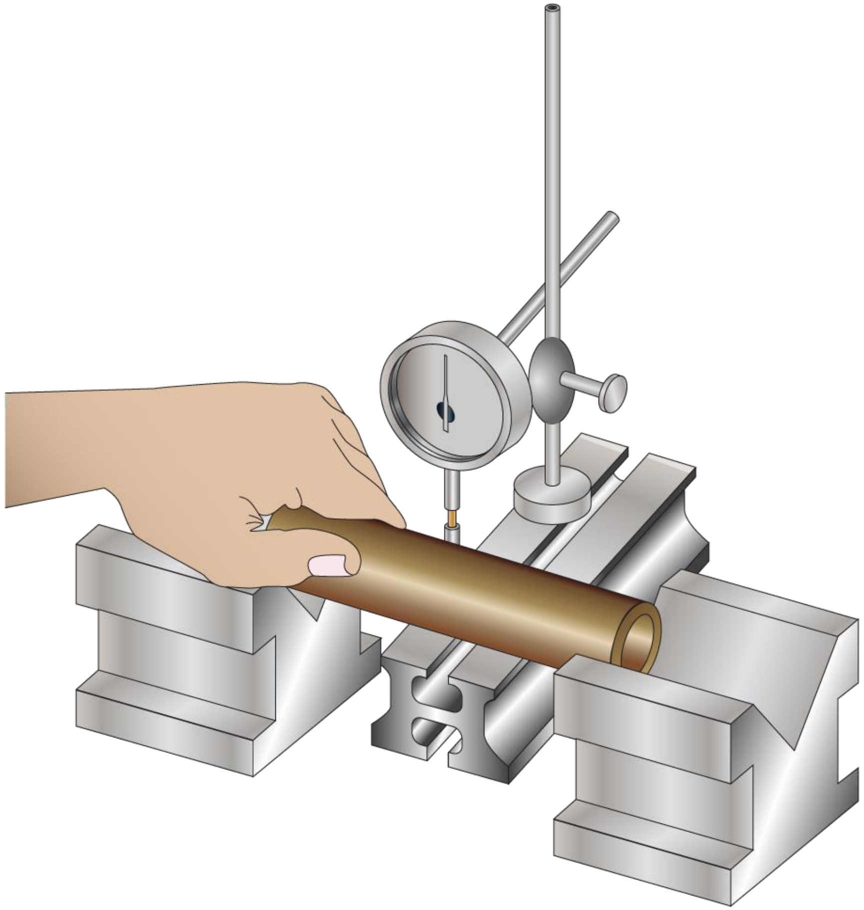
The diagrams below show the measurements of an electric motor shaft’s end float and run-out.



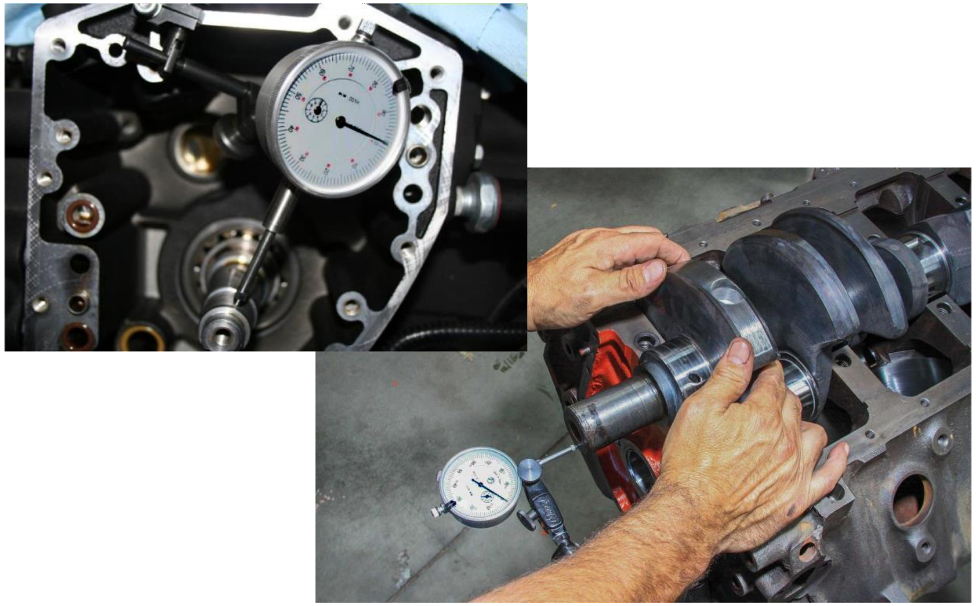








Checking a piston pin for bends



Geometric dimensioning and tolerancing (GD & T)

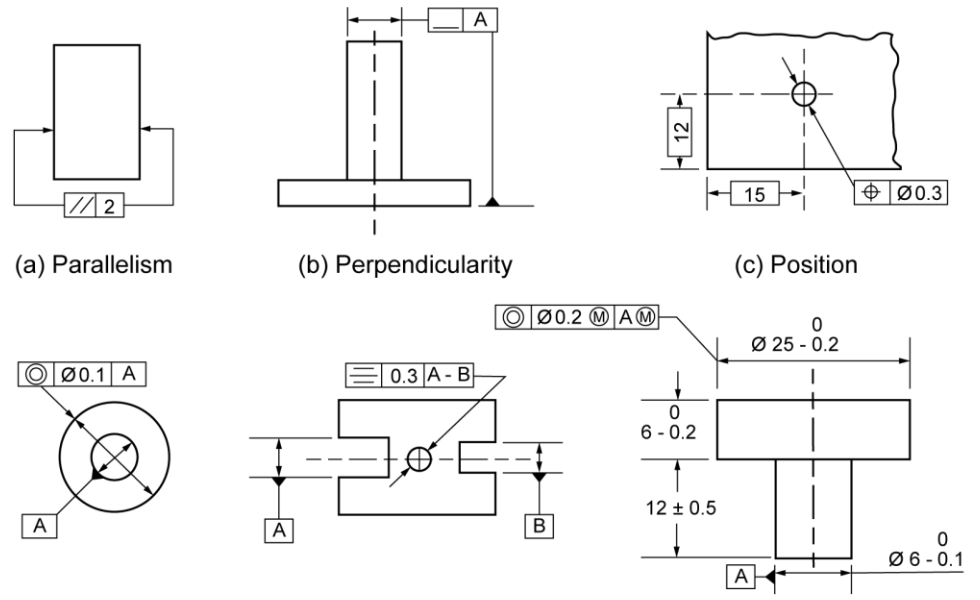
GD & T uses symbols to express dimensioning and tolerancing information found on the drawing sheet. These standardised symbols simplify the text information required for dimensioning and tolerancing.

It is sometimes necessary to place tolerances on both geometric features and dimensions to adequately control the shape of a part. On older drawings, this was done by annotating the feature to be toleranced, e.g. POSN TOL, and by adding notes to the drawing to specify the tolerance and the method of checking. On newer drawings, the international system recommended in BS 8888 is used, and this method is outlined in the following paragraphs.

Information relating to a particular geometric tolerance is enclosed within a rectangular frame on the drawing, with an arrow from the frame indicating the location of the feature to which the tolerance applies. Suppose the tolerance is related to a particular datum. In that case, a leader line is drawn from the frame to the datum position, or the datum is referenced separately and identified by a letter in the frame. Unless the datum is a dimension, it is defined by a solid equilateral triangle.

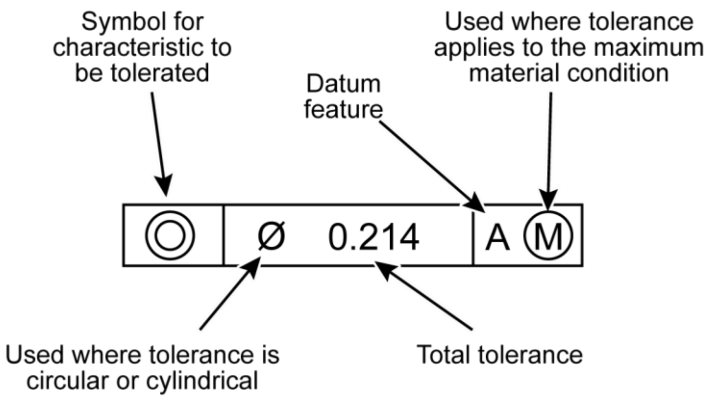
As a guide to interpreting a geometric tolerance, reference may be made to detail (e) below. This indicates that a symmetry tolerance of 0.3 mm is required with respect to datum features A and B. This tolerance indicates that the axis of the hole must be between two parallel planes, 0.3 mm apart, which are symmetrically disposed about the common median plane of the slots at the end of the part. The hole could also, if necessary, be marked to indicate a symmetry tolerance at 90° to the plane specified, and the tolerance for this could be different.

The symbol (M) in the tolerance frame detailed below indicates that the tolerance applies only to the maximum material condition of the dimension or datum feature and may be greater at the actual finished size.



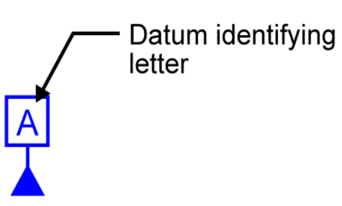
(a) Parallelism (b) Perpendicularity (c) Position

(d) Concentricity (e) Symmetry (f) Completely dimensioned part

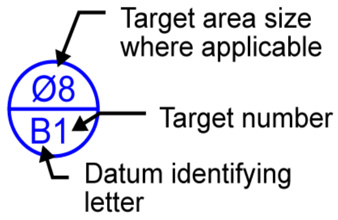


Tolerance frame

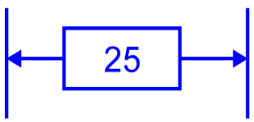
	Type of tolerance	Characteristic	Symbol
For individual features	Form	Straightness	—
		Flatness	⏥
		Circularity (roundness)	○
		Cylindricity	⌭
For individual or related features	Profile	Profile of a line	⌒
		Profile of a surface	⌓
For related features	Orientation	Angularity	∠
		Perpendicularity	⊥
		Parallelism	//
	Location	Position	⌖
		Concentricity	◎
	Run-out	Circular run-out	↗
		Total run-out	⌰
Other	General	Symmetry (obsolete)	⌯
		Datum feature	-A-
		Datum target	Ø6 A1



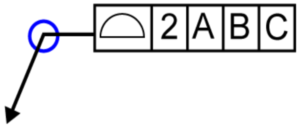
Datum feature symbol



Datum feature symbol

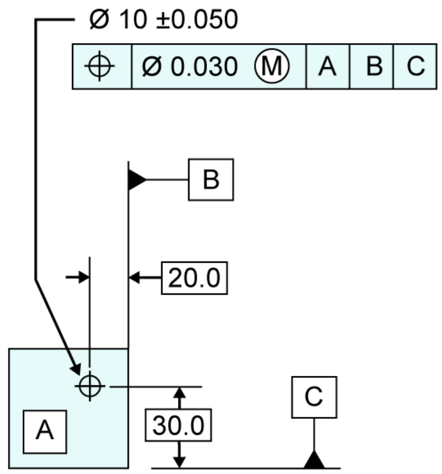


Basic dimension symbol

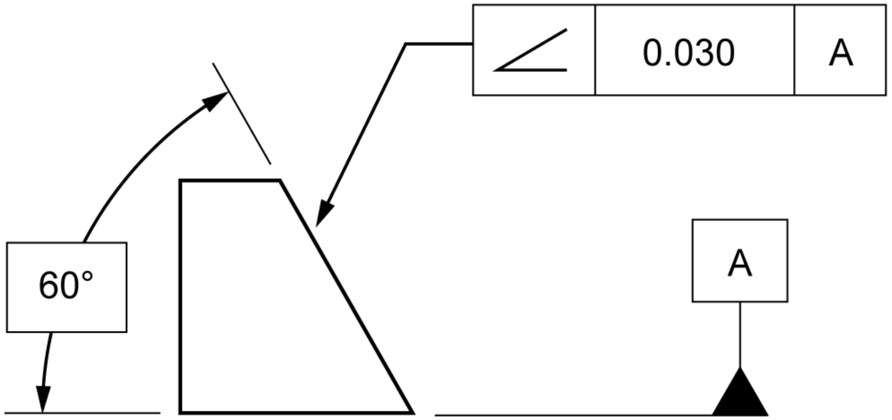


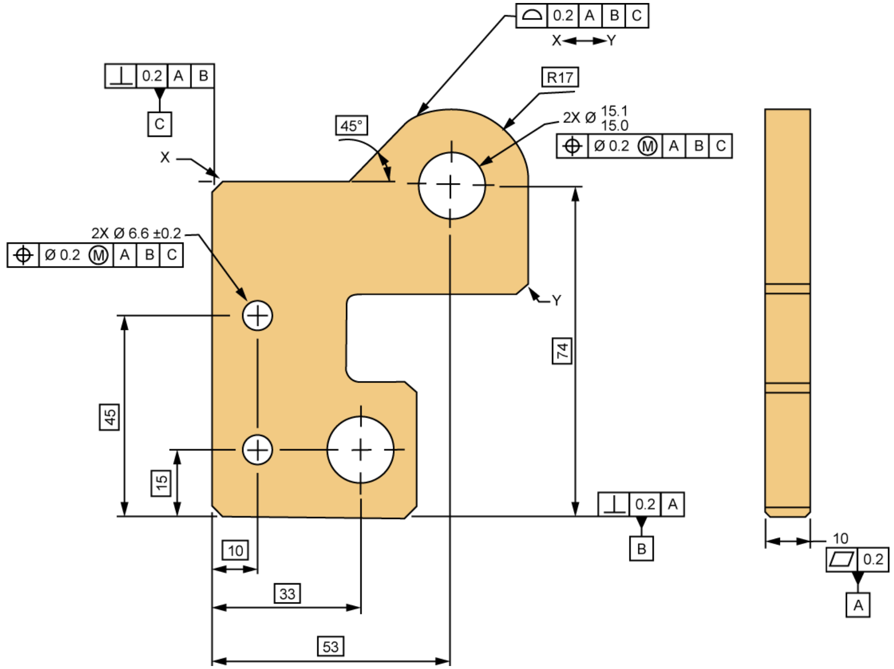
All-around symbol

Datum and other symbols

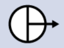
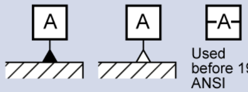
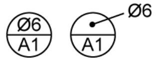


Common examples





Examples of tolerance features

Description	Symbols	Reference
Max material condition (MMC)	M	ISO 2692/ASME
Least material condition (LMC)	L	ISO 2692/ASME
Regardless of feature size (RFS)	S	ISO ASME
Spherical diameter (SD)	SØ	ISO ASME
Radius	R	ISO ASME
Diameter	Ø	ISO ASME
Envelope requirement	E	ISO ASME
Free State condition (non rigid parts)	F	ISO 10579/ASME
Minor diameter	LD	ISO ASME
Major diameter	MD	ISO ASME
Pitch diameter	PD	ISO ASME
Common zone	CZ	ISO ASME
Exact dimension	$\boxed{60}$	ISO ASME
Dimension origin		ISO ASME
All round (profile)		ISO NONE/ASME
Dayton feature indication	 Used before 1994 ANSI	ISO ASME
Dayton target indication		ISO ASME
Projected tolerance zone	P	ISO 10578/ASME

Other symbols in common use

Geometric tolerancing modifiers
The modifiers (MMC, LMC, RFS) clarify implied tolerances. There are three directly implied modifiers to the tolerance value. These are:

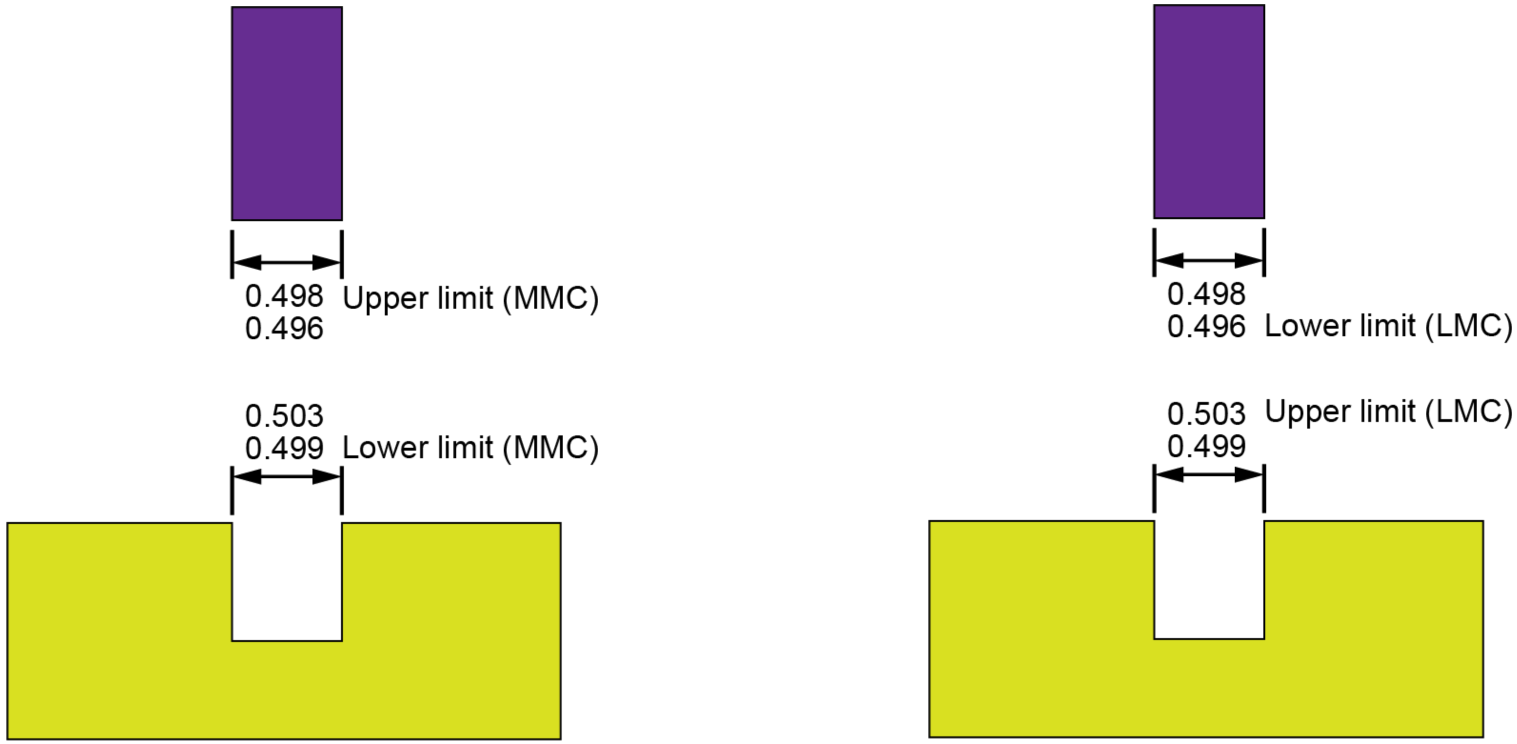
- regardless of feature size (RFS);
- maximum material condition (MMC); and
- least material condition (LMC)

The symbols for them can be found in the previous chart.

Regardless of feature size (RFS): RFS is the default modifier. RFS is used when the size feature does not affect the specified tolerance. So if no modifier symbol is shown in the feature control frame, RFS is the default modifier.

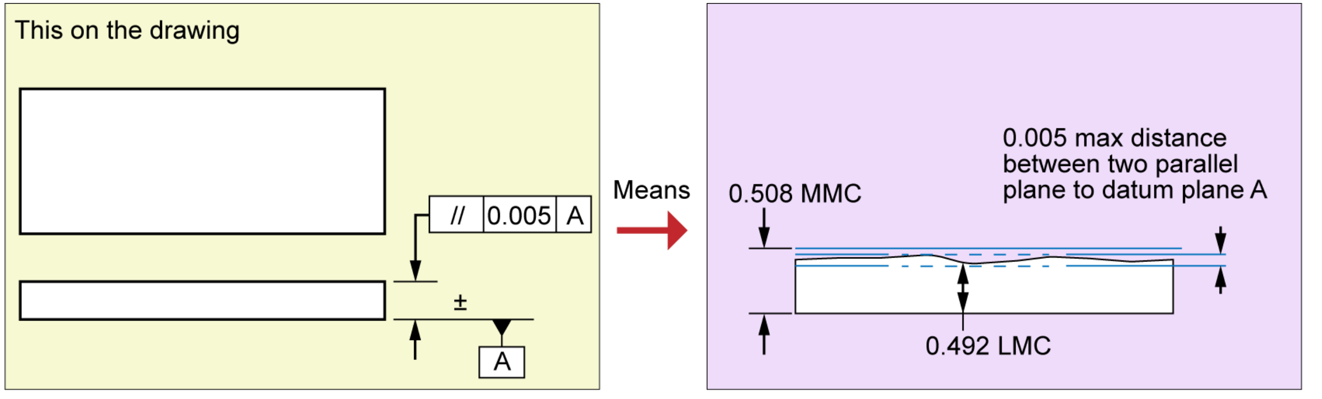
Maximum material condition (MMC): MMC can constrain the tolerance on given dimensions. MMC can be defined as the condition of a part feature where the maximum amount of material is required. MMC is also used to maintain clearance and fit between shafts and holes. The given tolerance is applied with MMC as the maximum shaft diameter and minimum hole diameter.

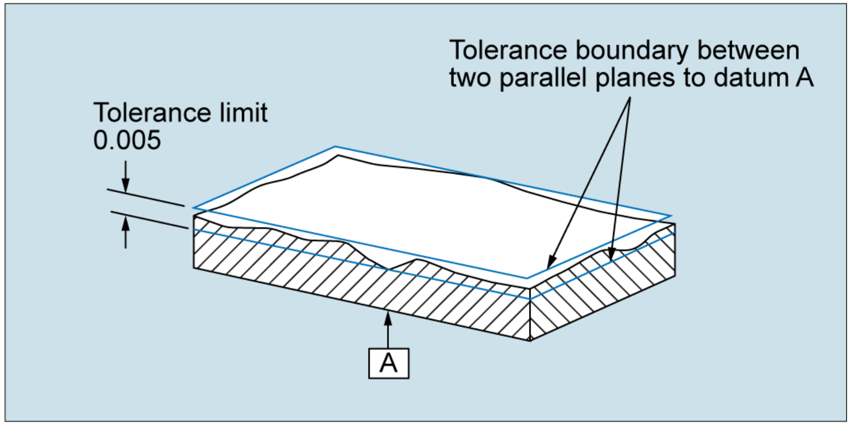
Least material condition (LMC): LMC condition where a size feature contains the least/minimum amount of material within the stated limits of size. LMC can be defined as the condition of a part feature where the least/minimum amount of material is required. With LMC, The given tolerance is applied as Least/Minimum shaft diameter and Maximum hole diameter.


0.498 Upper limit (MMC)
0.496
0.503
0.499 Lower limit (MMC)
0.498
0.496 Lower limit (LMC)
0.503 Upper limit (LMC)
0.499

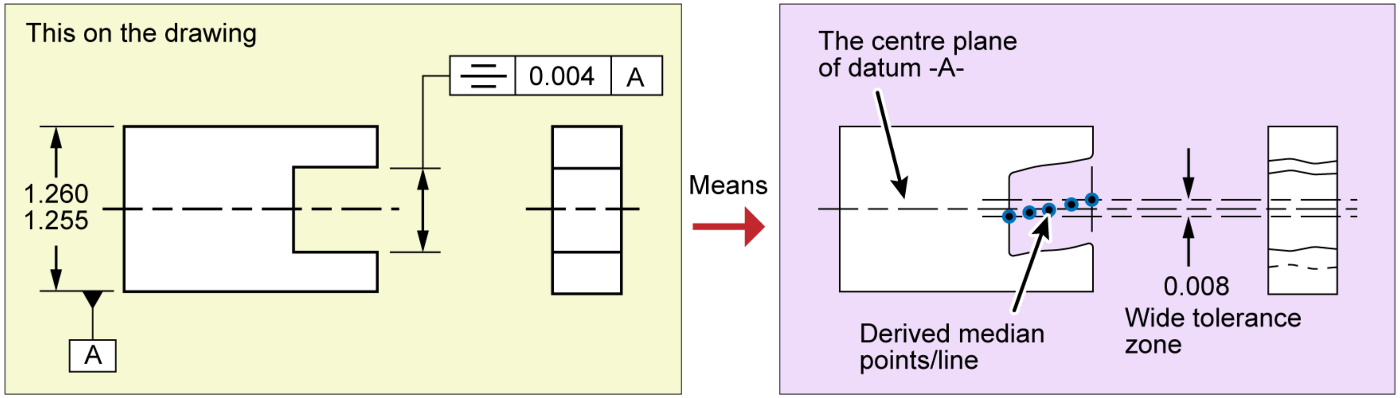
MMC and LMC

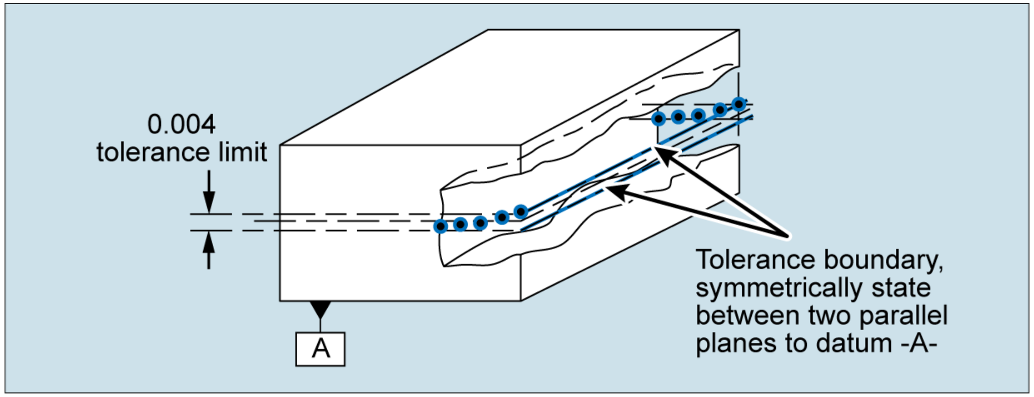
Examples of uses and applications of GD & T



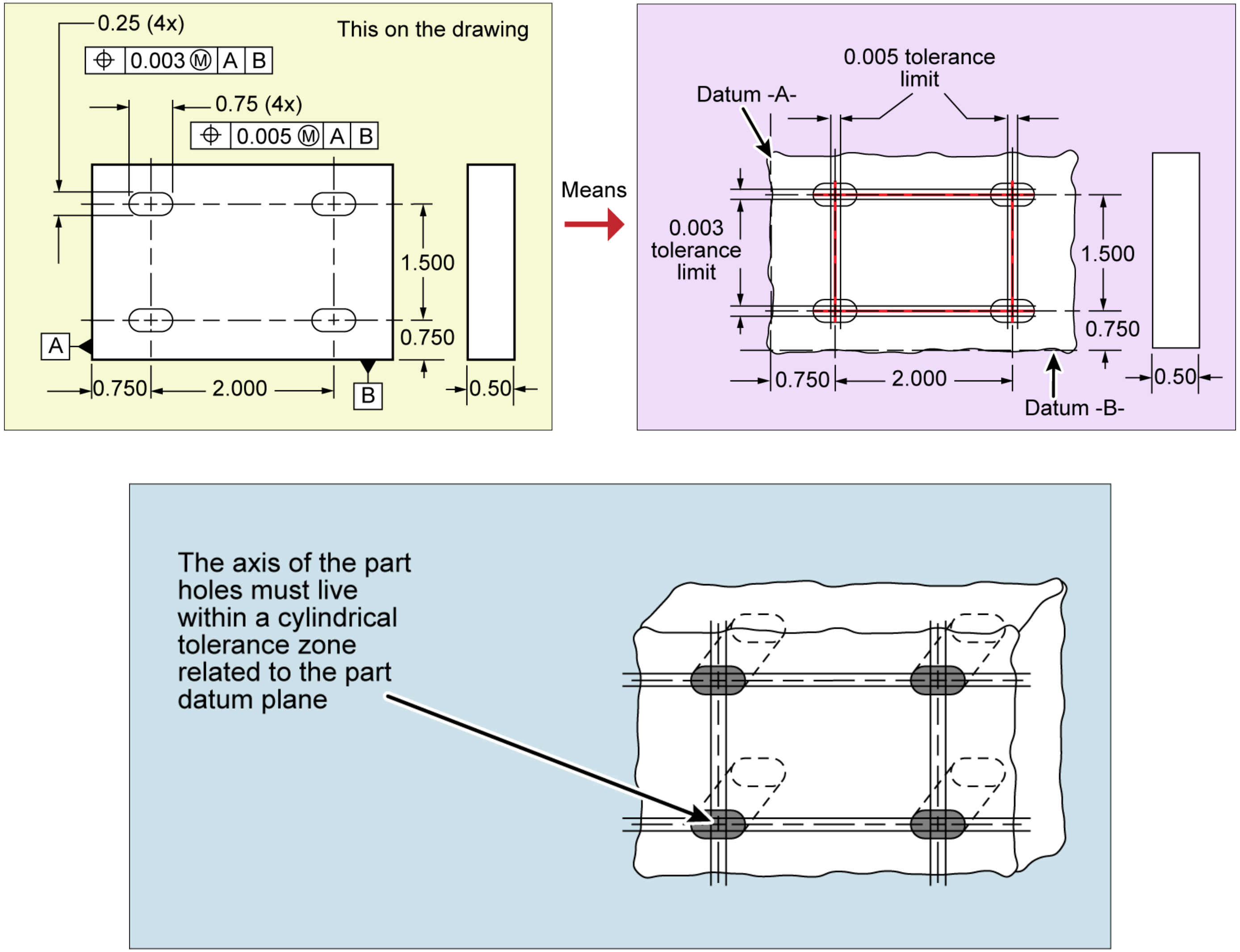


Parallelism tolerance

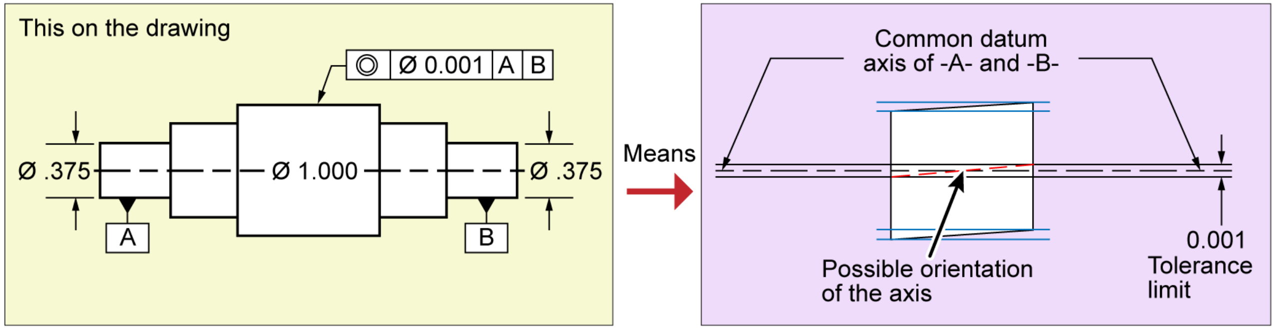




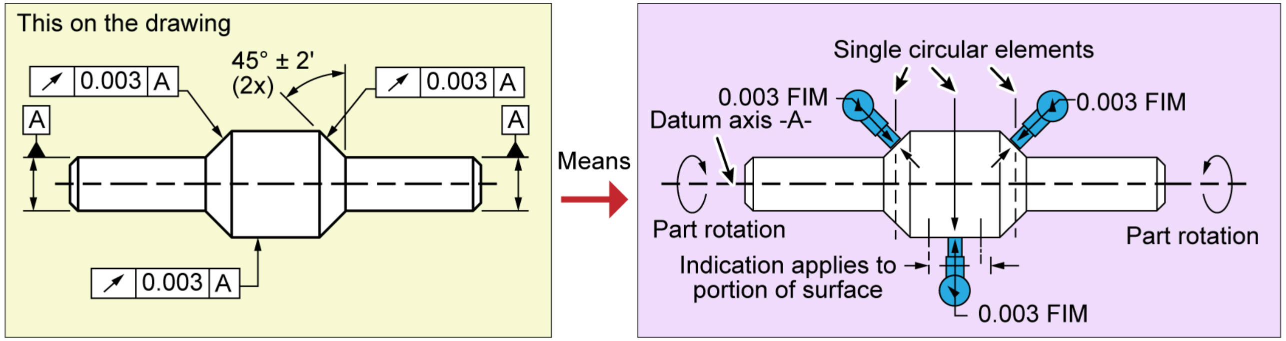
Symmetry tolerance



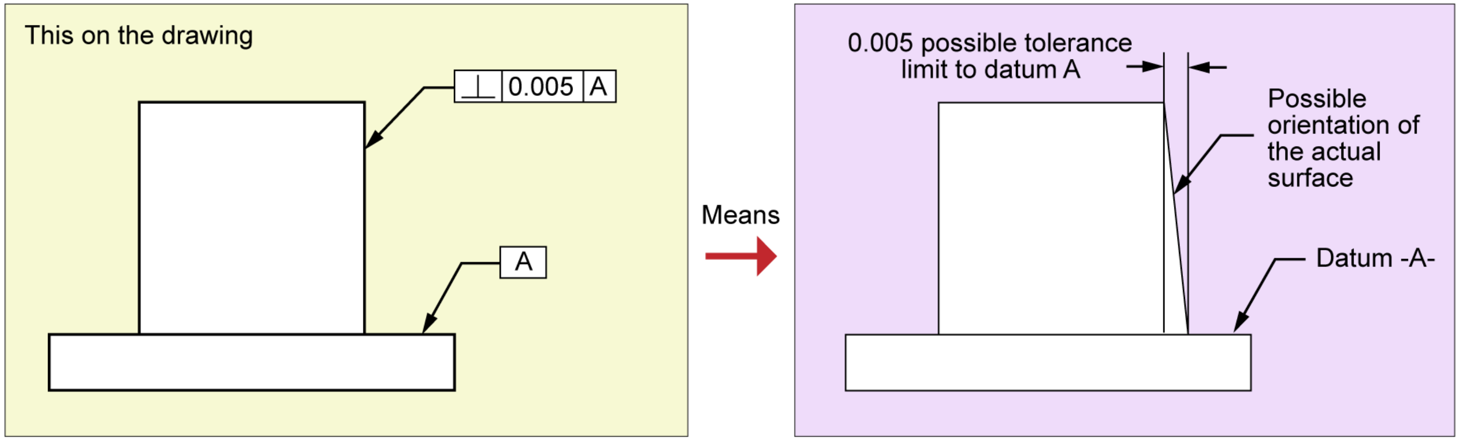
Hole position tolerance



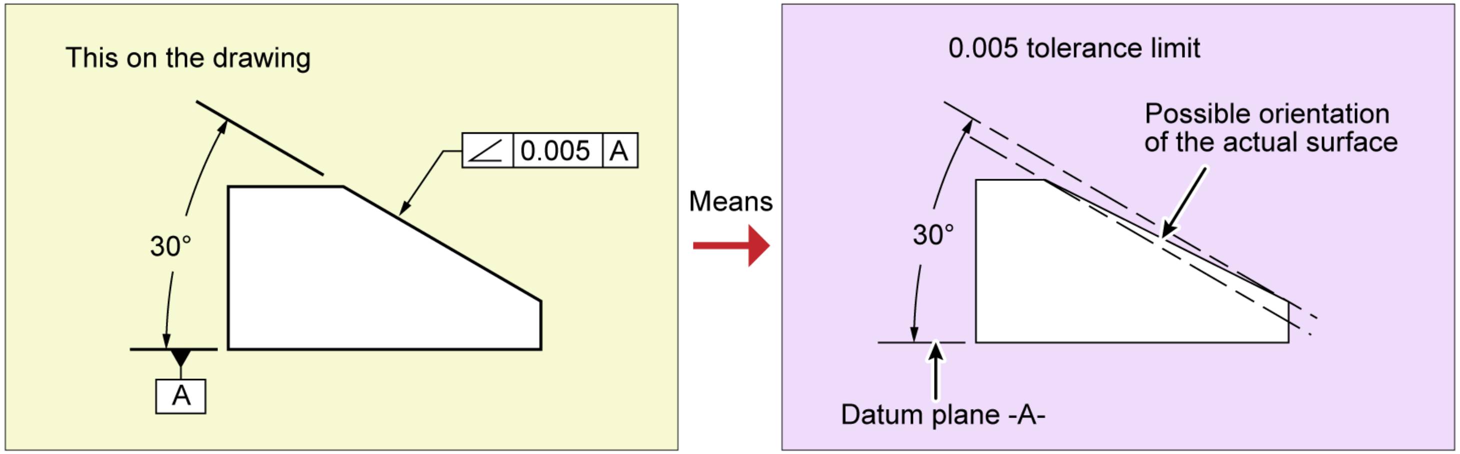
Concentricity tolerance



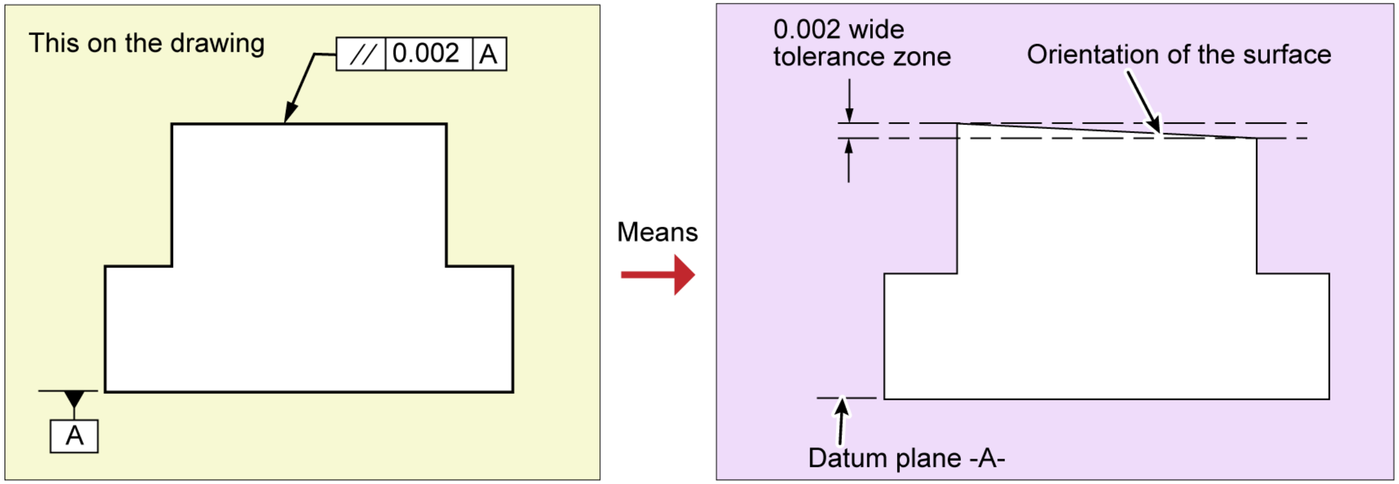
Circular runout tolerance



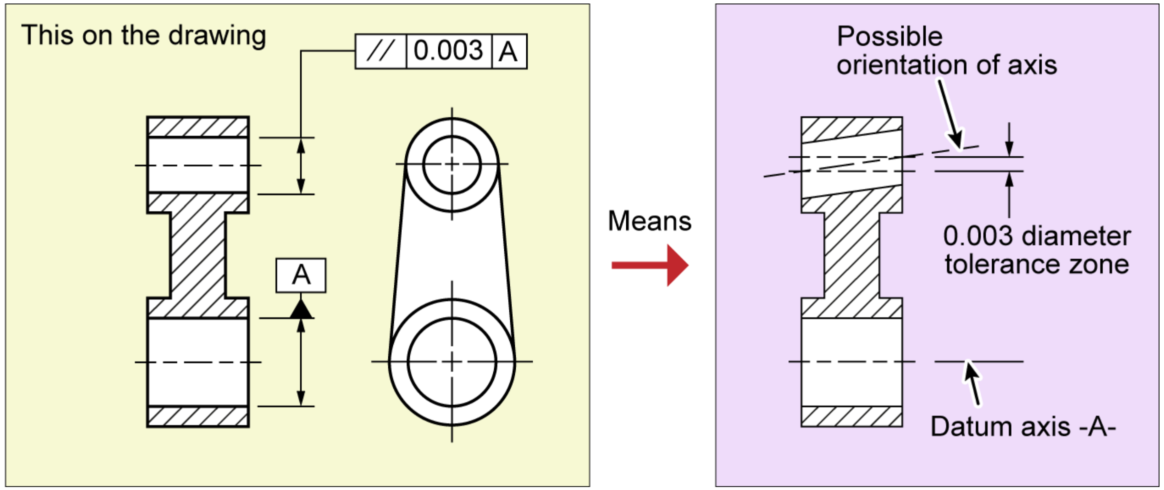
Perpendicularity tolerance

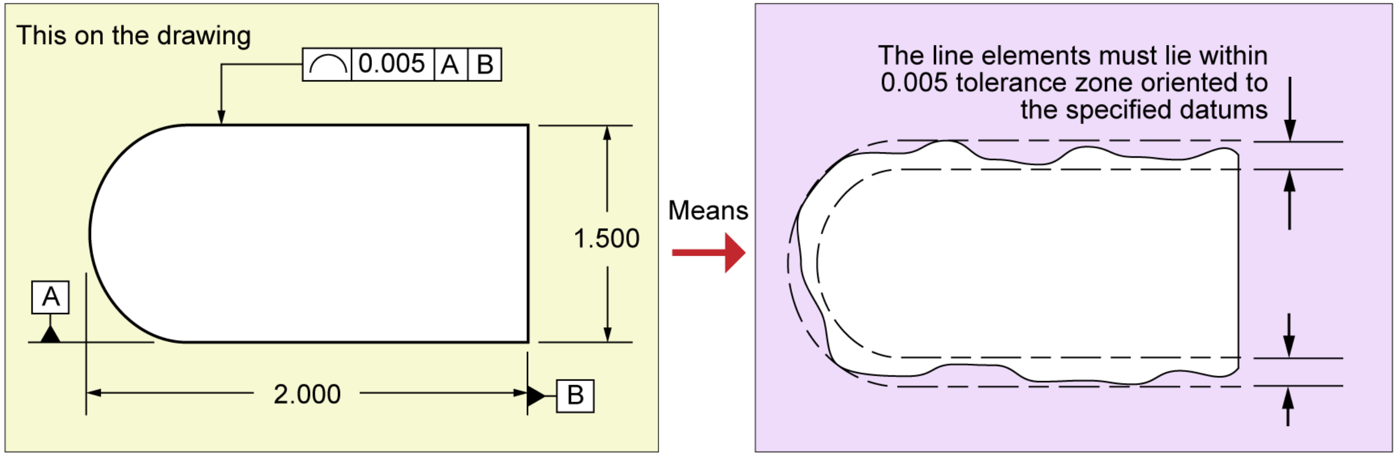


Angularity tolerance

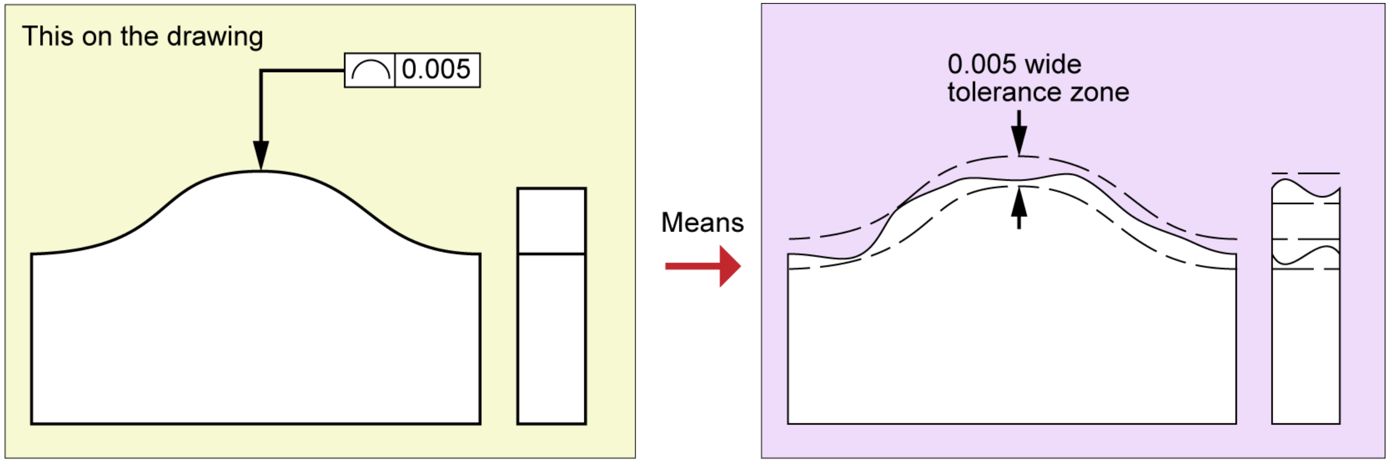


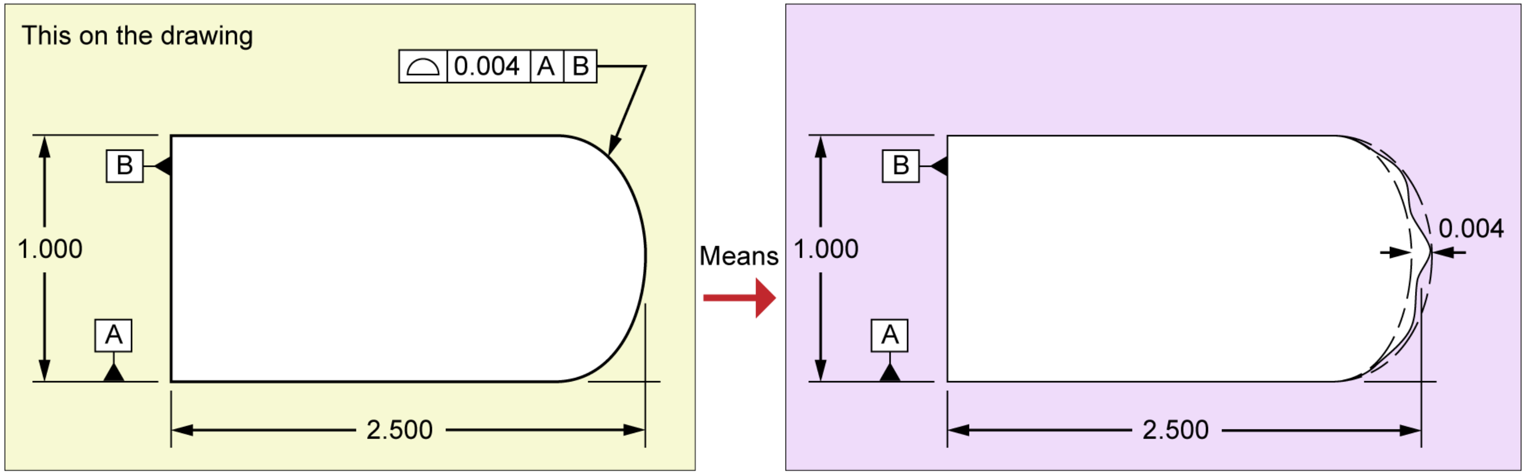
Parallelism tolerance

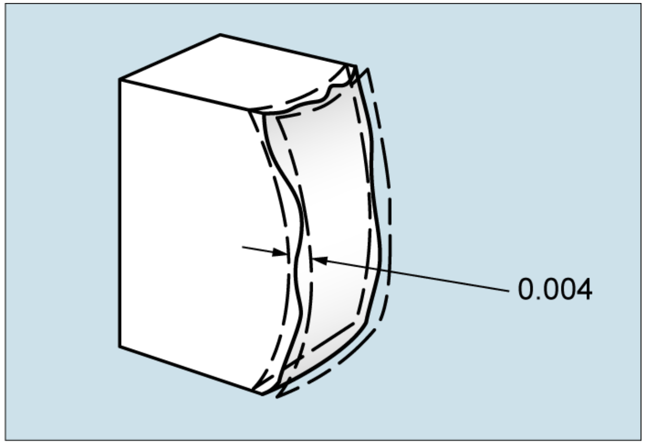




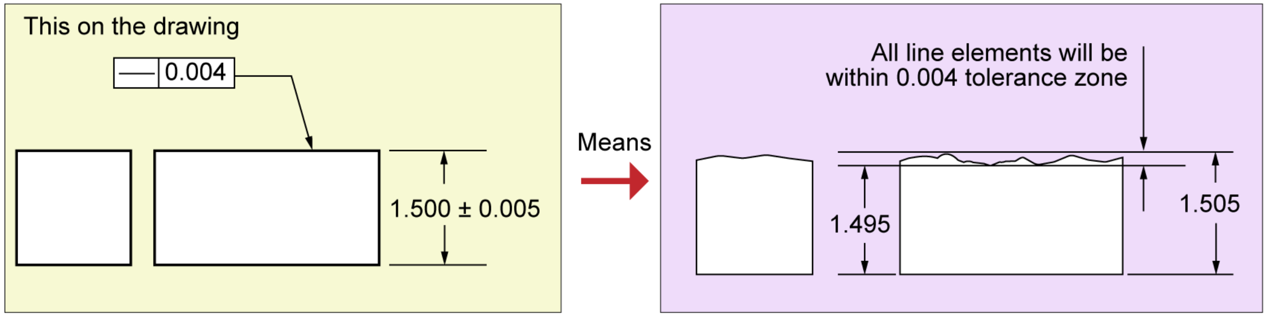
Profile of a line tolerance



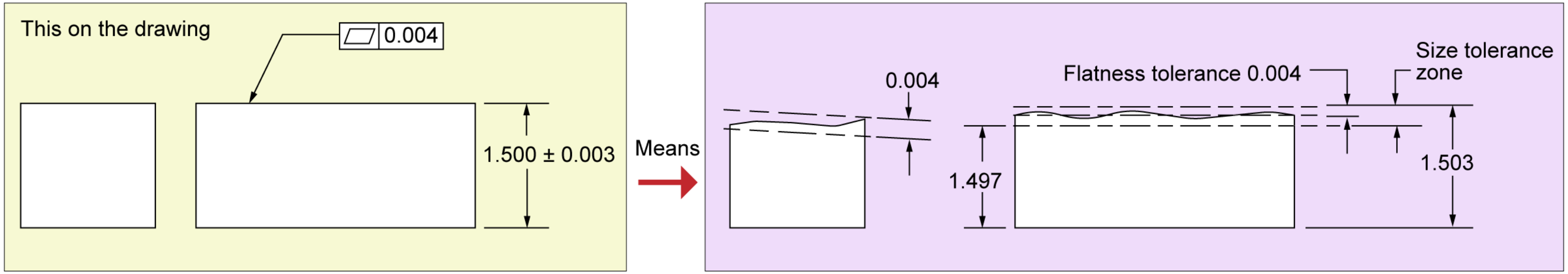




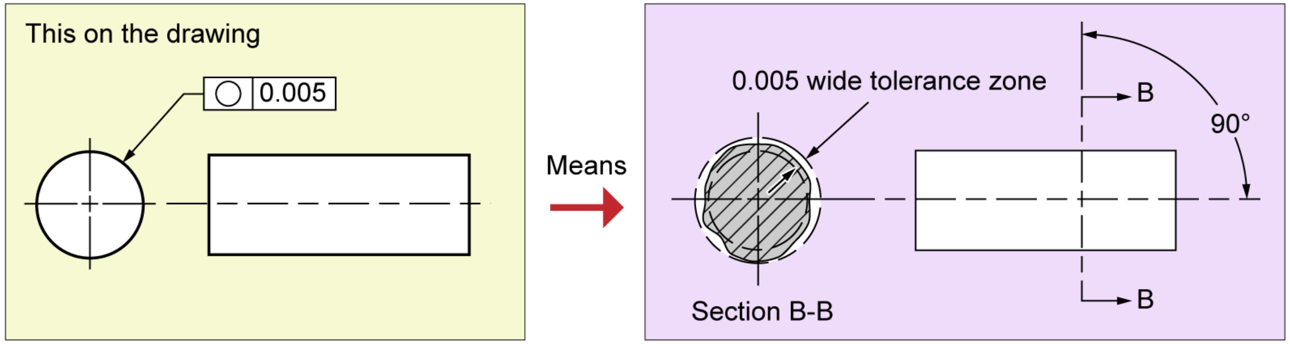
Profile of a surface tolerance



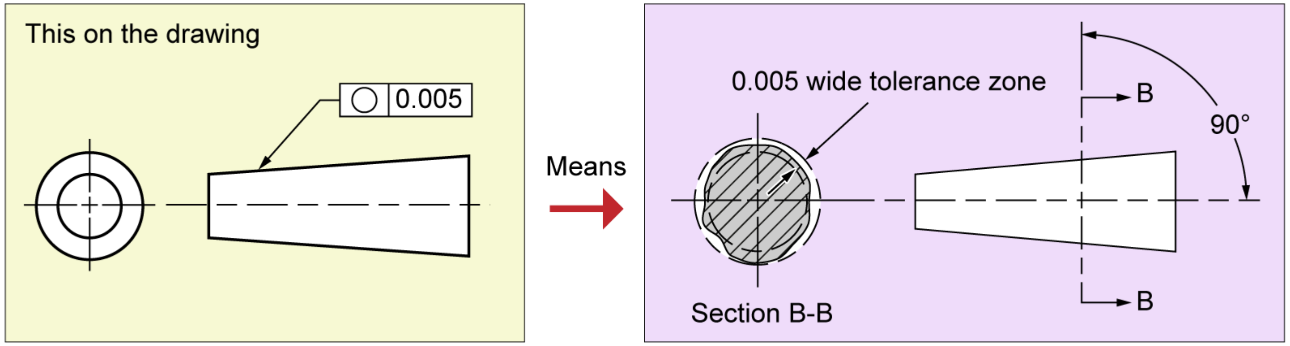
Straightness tolerance

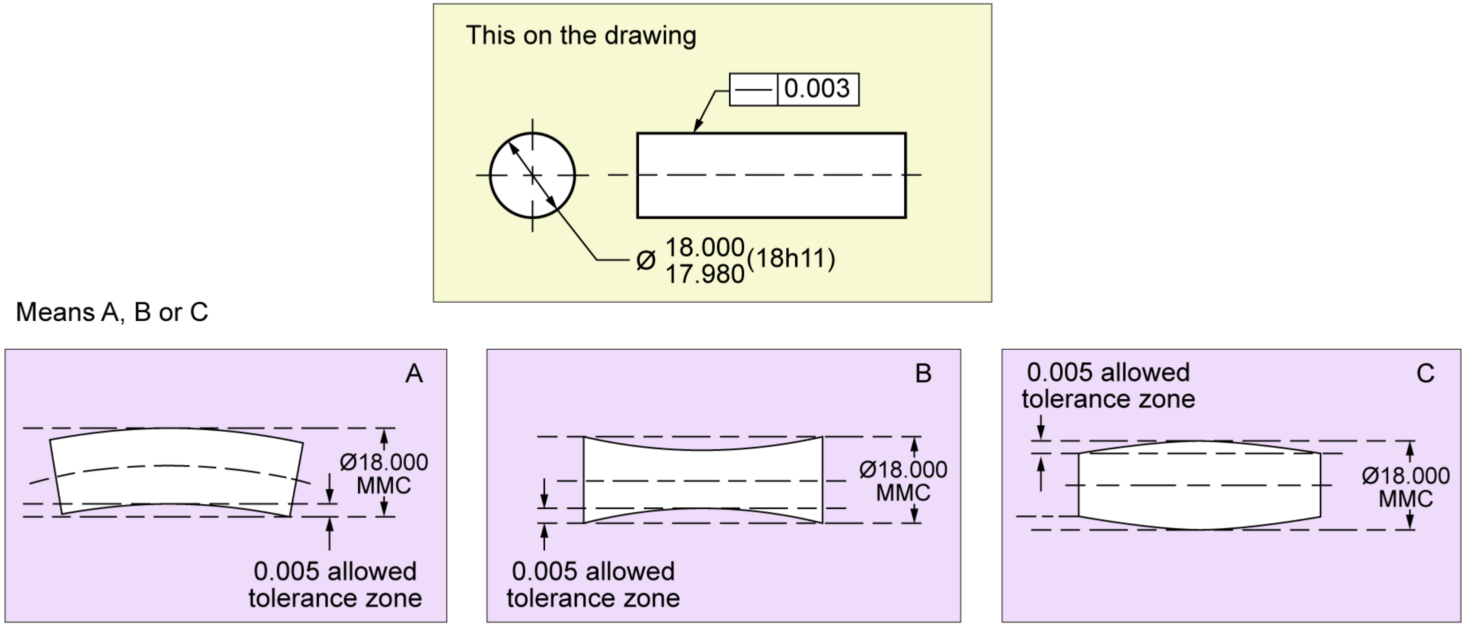


Flatness tolerance

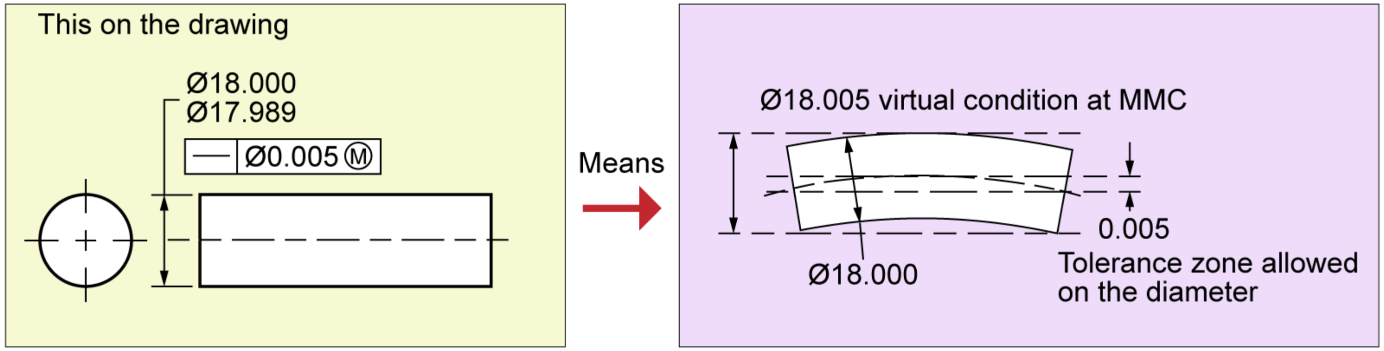


Circularity (roundness) tolerance

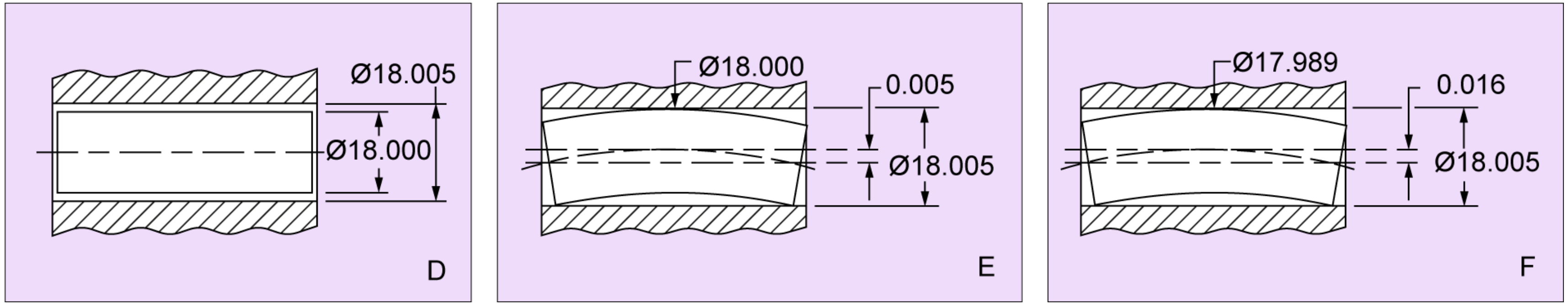




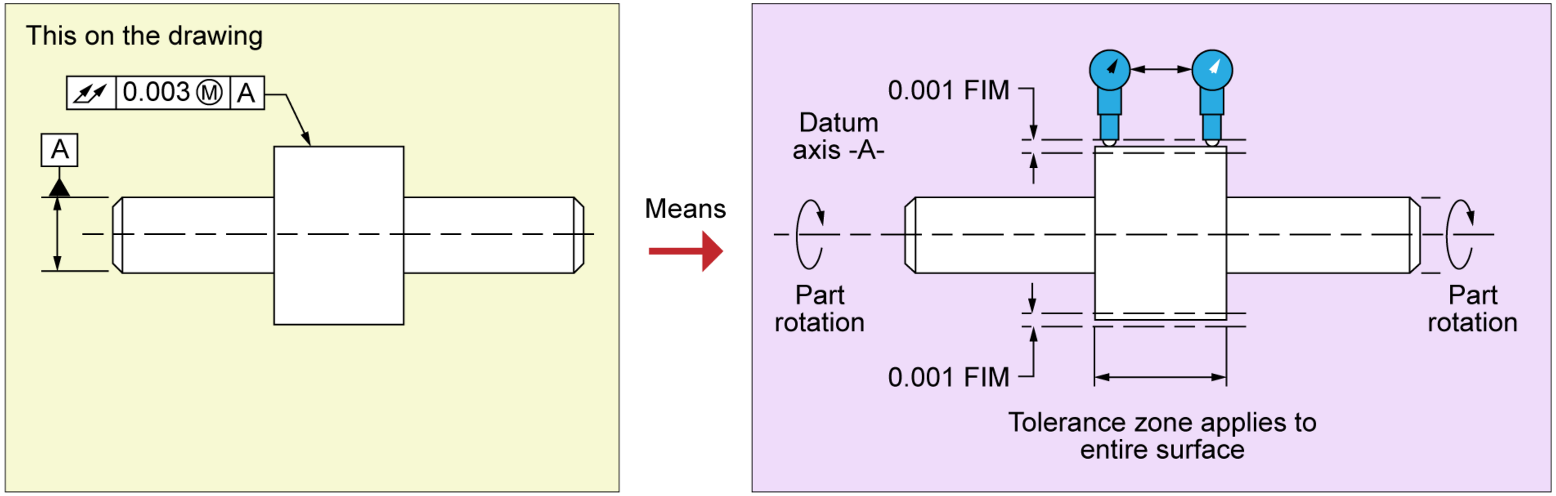
Straightness tolerance

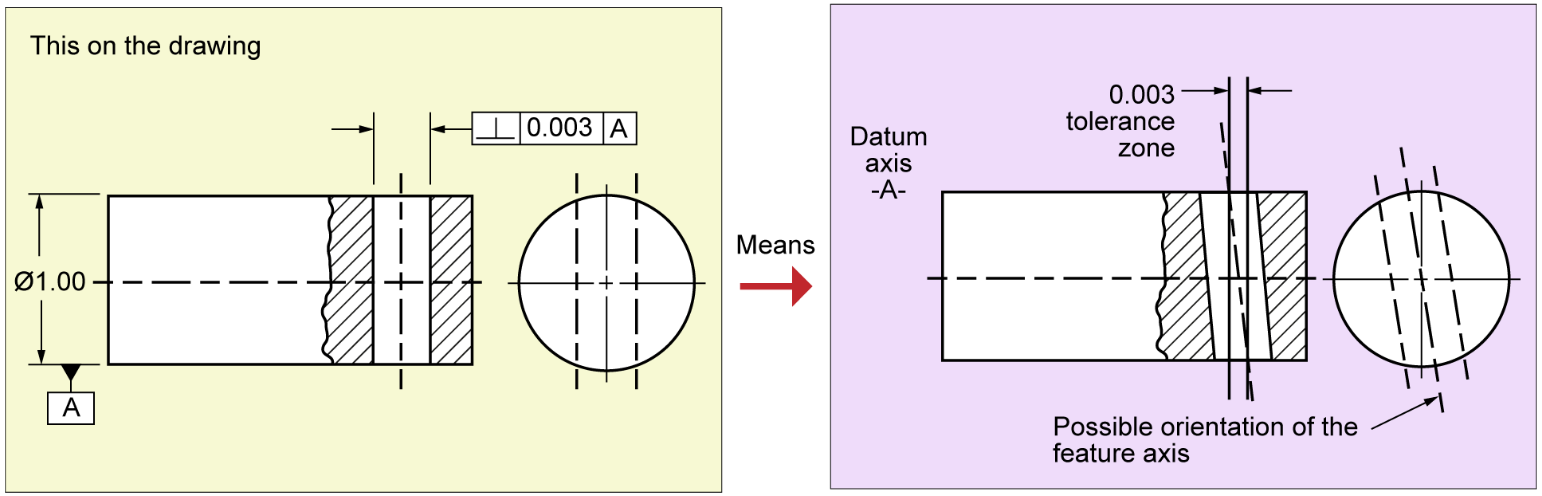


Acceptance boundaries D, E or F



Feature size	Diameter tolerance zone allowed
18.000	0.005
17.999	0.006
-	-
-	-
-	-
17.989	0.016





Total runout and perpendicularity tolerance