Section 4

Basic Mechanical Design

4.1 Engineering abbreviations

The following abbreviations are in common use in engineering drawings and specifications.

Table 4.1 Engineering abbreviations in common use

Abbreviation	*Meaning*
A/F	Across flats
ASSY	Assembly
CRS	Centres
L or CL	Centre line
CHAM	Chamfered
CSK	Countersunk
C'BORE	Counterbore
CYL	Cylinder or cylindrical
DIA	Diameter (in a note)
Ø	Diameter (preceding a dimension)
DRG	Drawing
EXT	External
FIG.	Figure
HEX	Hexagon
INT	Internal
LH	Left hand
LG	Long
MATL	Material
MAX	Maximum
MIN	Minimum
NO.	Number
PATT NO.	Pattern number
PCD	Pitch circle diameter
RAD	Radius (in a note)
R	Radius (preceding a dimension)
REQD	Required
RH	Right hand
SCR	Screwed
SH	Sheet
SK	Sketch
SPEC	Specification

Engineers' Data Book, Fourth Edition. Clifford Matthews.

Table 4.1 (Cont.)

Abbreviation	*Meaning*
SQ	Square (in a note)
□	Square (preceding a dimension)
STD	Standard
VOL	Volume
WT	Weight

4.1.1 American terminology

In the USA, slightly different terminology is used, based on the published standard ASME Y14.5 *Dimensioning and Tolerancing*: 2009.

Table 4.2

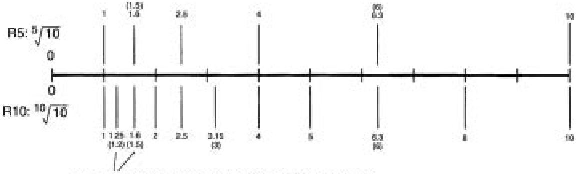
Abbreviation	*Meaning*
ANSI	American National Standards Institute
ASA	American Standards Association
ASME	American Society of Mechanical Engineers
AVG	Average
CBORE	Counterbore
CDRILL	Counterdrill
CL	Centre line
CSK	Countersink
FIM	Full indicator movement
FIR	Full indicator reading
GD&T	Geometric dimensioning and tolerancing
ISO	International Standards Organization
LMC	Least material condition
MAX	Maximum
MDD	Master dimension definition
MDS	Master dimension surface
MIN	Minimum
mm	Millimetre
MMC	Maximum material condition
PORM	Plus or minus
R	Radius
REF	Reference
REQD	Required
RFS	Regardless of feature size
SEP REQT	Separate requirement
SI	Systèmè International (the metric system)
SR	Spherical radius
SURF	Surface
THRU	Through
TIR	Total indicator reading
TOL	Tolerance

4.1.2 Preferred numbers and preferred sizes

Preferred numbers are derived from geometric series in which each term is a uniform percentage larger than its predecessor. The first five principal series (named the 'R' series) are shown in Table 4.3. Preferred numbers are taken as the basis for ranges of linear sizes of components, often being rounded up or down for convenience. Figure 4.1 shows the development of the R5 and R10 series.

Table 4.3

Series	*Basis*	*Ratio of terms (% increase)*
R5	$\sqrt[5]{10}$	1.58 (58%)
R10	$\sqrt[10]{10}$	1.26 (26%)
R20	$\sqrt[20]{10}$	1.12 (12%)
R40	$\sqrt[40]{10}$	1.06 (6%)
R80	$\sqrt[80]{10}$	1.03 (3%)



'Rounding' of the R5 and R10 series *numbers* (shown in brackets) gives series of preferred *sizes*

Figure 4.1

USEFUL STANDARD

BS 2045: 1982: Preferred numbers.

4.2 Datums and tolerances – principles

A *datum* is a reference point or surface from which all other dimensions of a component are taken; these other dimensions are said to be *referred to* the datum. In most practical designs, a datum surface is normally used, this generally being one of the surfaces of the machine element itself rather than an 'imaginary' surface. This means that the datum surface normally plays some important part in the operation of the elements – it is usually machined and may be a mating surface or a locating face between elements, or similar. Simple machine mechanisms do not *always* need datums; it depends on what the elements do and how complicated the mechanism assembly is.

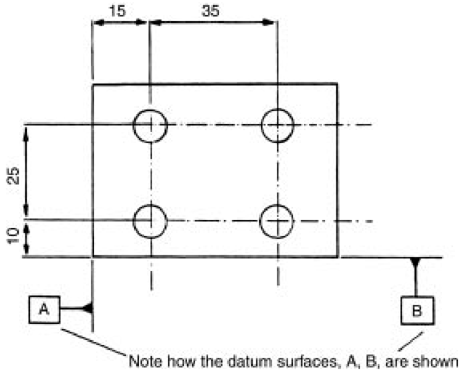


Figure 4.2

A *tolerance* is the allowable variation of a linear or angular dimension about its ‘perfect’ value. British Standard 8888: 2008 contains accepted methods and symbols.

4.3 Toleranced dimensions

In designing any engineering component it is necessary to decide which dimensions will be toleranced. This is predominantly an

BS 8888

	Toleranced characteristic
—	Straightness
▱	Flatness
○	Circularity
//	Parallelism
∠	Angularity
⊥	Squareness
◎	Concentricity
↗	Run-out
⌰	Total run-out

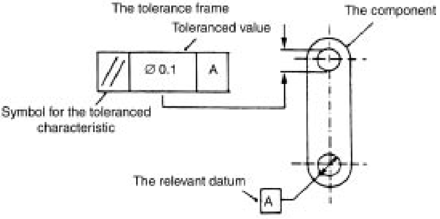


Figure 4.3

exercise in necessity – only those dimensions that *must* be tightly controlled, to preserve the functionality of the component, should be toleranced. Too many toleranced dimensions will increase significantly the manufacturing costs and may result in 'tolerance clash', where a dimension derived from other toleranced dimensions can have several contradictory values.

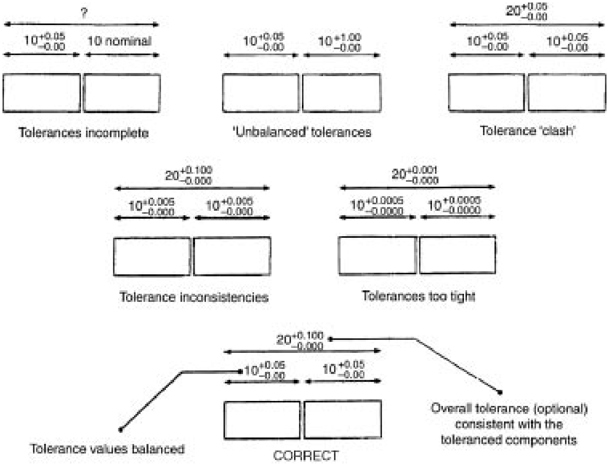


Figure 4.4

4.4 General tolerances

It is a sound principle of engineering practice that in any machine design there will only be a small number of toleranced features. The remainder of the dimensions will not be critical.

There are two ways to deal with this: first, an engineering drawing or sketch can be annotated to specify that a *general tolerance* should apply to features where no specific tolerance is

Table 4.4 Typical tolerances for linear dimensions

Dimension	*Tolerance*
0.6 mm–6.0 mm	± 0.1 mm
6 mm–36 mm	± 0.2 mm
36 mm–120 mm	± 0.3 mm
120 mm–315 mm	± 0.5 mm
315 mm–1000 mm	± 0.8 mm

mentioned. This is often expressed as ± 0.5 mm. Alternatively, the drawing can make reference to a 'general tolerance' standard such as BS EN 22768 which gives typical tolerances for linear dimensions as shown.

4.5 Holes

The tolerancing of holes depends on whether they are made in thin sheet (up to about 3 mm thick) or in thicker plate material. In thin material, only two toleranced dimensions are required:

- *Size* A toleranced diameter of the hole, showing the maximum and minimum allowable dimensions.
- *Position* Position can be located with reference to a datum and/or its spacing from an adjacent hole. Holes are generally spaced by reference to their centres.

For thicker material, three further toleranced dimensions become relevant: straightness, parallelism and squareness.

- *Straightness* A hole or shaft can be *straight* without being perpendicular to the surface of the material.
- *Parallelism* This is particularly relevant to holes and is important when there is a mating hole-to-shaft fit.
- *Squareness* The formal term for this is perpendicularity. Simplistically, it refers to the squareness of the axis of a hole to the datum surface of the material through which the hole is made.

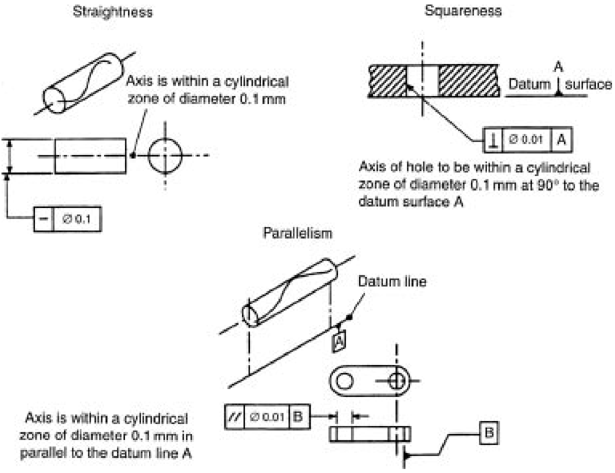
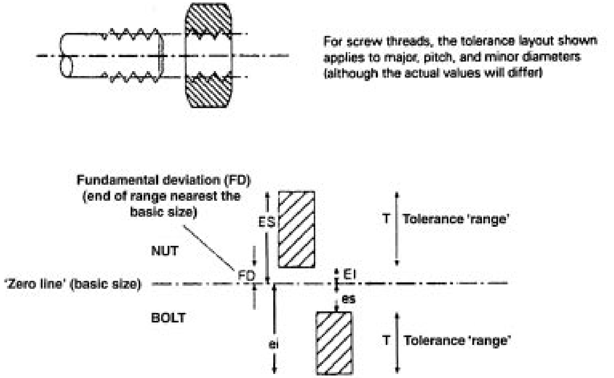


Figure 4.5 Straightness, parallelism, and squareness–BS 8888:2008

4.6 Screw threads

There is a well-established system of tolerancing adopted by British and International Standard Organizations and manufacturing industry. This system uses the two complementary elements of fundamental deviation and tolerance range to define fully the tolerance of a single component. It can be applied easily to components, such as screw threads, which join or mate together.

- *Fundamental deviation* (*FD*) is the distance (or 'deviation') of the nearest 'end' of the tolerance band from the nominal or 'basic' size of a dimension.
- *Tolerance band* (or 'range') is the size of the tolerance band, i.e. the difference between the maximum and minimum acceptable size of a toleranced dimension. The size of the tolerance band, and the location of the FD, governs the system of limits and fits applied to mating parts.



FD is designated by a letter code, e.g., g, H
Tolerance range (T) is designated by a number code, e.g., 5, 6, 7

Commonly used symbols are:
EI – lower deviation (nut)
ES – upper deviation (nut)
ei – lower deviation (bolt)
es – upper deviation (bolt)

Figure 4.6

Tolerance values have a key influence on the costs of a manufactured item so their choice must be seen in terms of economics as well as engineering practicality. Mass-produced items are competitive and price sensitive, and over-tolerancing can affect the economics of a product range.

4.7 Limits and fits

4.7.1 Principles

In machine element design there is a variety of different ways in which a shaft and hole are required to fit together. Elements such as bearings, location pins, pegs, spindles, and axles are typical examples. The shaft may be required to be a tight fit in the hole, or to be looser, giving a clearance to allow easy removal or rotation. The system designed to establish a series of useful fits between shafts and holes is termed *limits and fits.* This involves a series of tolerance grades so that machine elements can be made with the correct degree of accuracy and be interchangeable with others of the same tolerance grade.

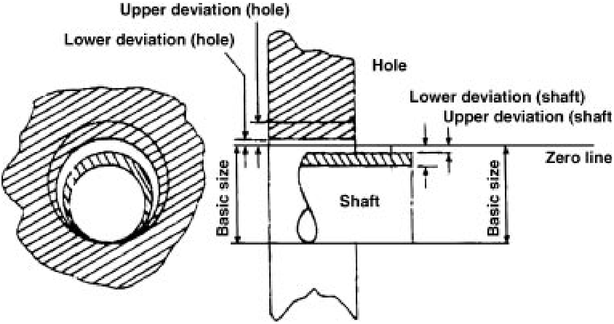


Figure 4.7

The British Standard BS 4500 /BS EN 20286 'ISO limits and fits' contains the recommended tolerances for a wide range of engineering requirements. Each tolerance grade is designated by a combination of letters and numbers, such as IT7, which would be referred to as grade 7.

Figure 4.7 shows the principles of a shaft/hole fit. The 'zero line' indicates the basic or 'nominal' size of the hole and shaft (it is the same for each) and the two shaded areas depict the tolerance zones within which the hole and shaft may vary. The hole is conventionally shown above the zero line. The algebraic difference between the basic size of a shaft or hole and its actual size is known as the *deviation.*

- It is the deviation that determines the nature of the fit between a hole and a shaft.
- If the deviation is small, the tolerance range will be near the basic size, giving a tight fit.
- A large deviation gives a loose fit.

Various grades of deviation are designated by letters, similar to the system of numbers used for the tolerance ranges. Shaft deviations are denoted by small letters and hole deviations by capital letters. Most general engineering uses a 'hole-based' fit in which the larger part of the available tolerance is allocated to the hole (because it is more difficult to make an accurate hole) and then the shaft is made to suit, to achieve the desired fit.

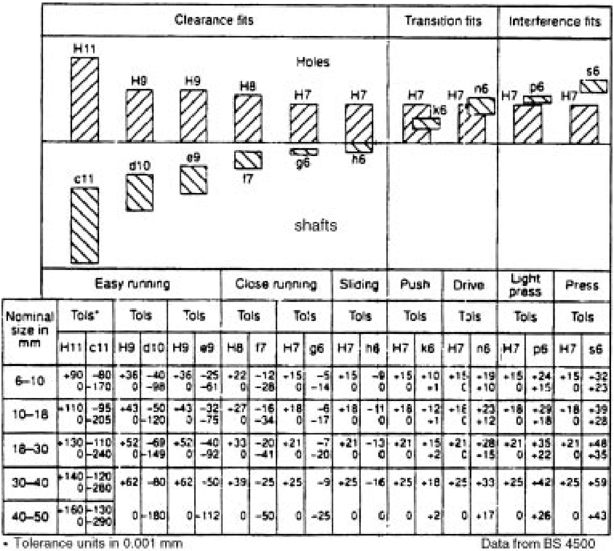
4.7.2 Common combinations

There are seven popular combinations used in general mechanical engineering design:

1. *Easy running fit:* H11–c11, H9–d10, H9–e9. These are used for bearings where a significant clearance is necessary.
2. *Close running fit:* H8–f7, H8–g6. This only allows a small clearance, suitable for sliding spigot fits and infrequently used journal bearings. This fit is not suitable for continuously rotating bearings.
3. *Sliding fit:* H7–h6. Normally used as a locational fit in which close-fitting items slide together. It incorporates a very small clearance and can still be freely assembled and disassembled.
4. *Push fit:* H7–k6. This is a transition fit, mid-way between fits that have a guaranteed clearance and those where there is metal interference. It is used where accurate location is required, e.g. dowel and bearing inner-race fixings.
5. *Drive fit:* H7–n6. This is a tighter grade of transition fit than the H7–k6. It gives a tight assembly fit where the hole and shaft may need to be pressed together.
6. *Light press fit:* H7–p6. This is used where a hole and shaft need permanent, accurate assembly. The parts need pressing together but the fit is not so tight that it will overstress the hole bore.
7. *Press fit:* H7–s6. This is the tightest practical fit for machine elements such as bearing bushes. Larger interference fits are possible but are only suitable for large heavy engineering components.

4.8 Surface finish

Surface finish, more correctly termed ‘surface texture’, is important for all machine elements that are produced by machining processes such as turning, grinding, shaping, or honing. This applies to surfaces which are flat or cylindrical. Surface texture is covered by its own technical standard, BS 1134: 2010 *Assessment of surface texture*. It is measured using the parameter R_a which is



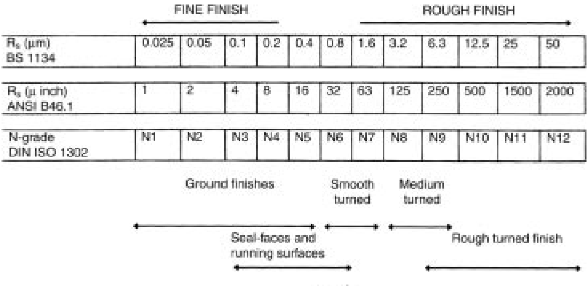
Nominal size in mm	Easy running						Close running				Sliding		Push		Drive		Light press		Press	
	Tols*		Tols		Tols		Tols		Tols		Tols		Tols		Tols		Tols		Tols	
	H11	c11	H9	d10	H9	e9	H8	f7	H7	g6	H7	h6	H7	k6	H7	n6	H7	p6	H7	s6
6–10	+90 0	-80 -170	+36 0	-40 -98	+36 0	-25 -61	+22 0	-12 -28	+15 0	-5 -14	+15 0	-9 0	+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	-11 0	+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	-13 0	+21 0	+15 +2	+21 0	+28 +15	+21 0	+35 +22	+21 0	+48 +35
30–40	+140 0	-120 -280	+62	-80	+62	-50	+39	-25	+25	-9	+25	-16	+25	+18	+25	+33	+25	+42	+25	+59
40–50	+160 0	-130 -290	0	-180	0	-112	0	-50	0	-25	0	0	0	+2	0	+17	0	+26	0	+43

* Tolerance units in 0.001 mm

Data from BS 4500

Figure 4.8 Metric Equivalents

a measurement of the average distance between the median line of the surface profile and its peaks and troughs, measured in micrometres (μm). There is another system from a comparable standard, ISO 1302, which uses a system of N-numbers – it is simply a different way of describing the same thing.



R_a (μm) BS 1134	0.025	0.05	0.1	0.2	0.4	0.8	1.6	3.2	6.3	12.5	25	50
R_a (μ inch) ANSI B46.1	1	2	4	8	16	32	63	125	250	500	1500	2000
N-grade DIN ISO 1302	N1	N2	N3	N4	N5	N6	N7	N8	N9	N10	N11	N12

A prescribed surface finish is shown on a drawing as $\overset{1.6}{\sqrt{}}$ – on a metric drawing this means 1.6 μm R_a

Figure 4.9

4.8.1 Choice of surface finish: ‘rules of thumb’

- Rough turned, with visible tool marks: N10 (12.5 μm R_a)
- Smooth machined surface: N8 (3.2 μm R_a)
- Static mating surfaces (or datums): N7 (1.6 μm R_a)
- Bearing surfaces: N6 (0.8 μm R_a)
- Fine ‘lapped’ surfaces: N1 (0.025 μm R_a)

Finer finishes can be produced but are more suited for precision application such as instruments. It is good practice to specify the surface finish of close-fitting surfaces of machine elements, as well as other BS 8888 parameters such as squareness and parallelism.