Chapter 5

Limits, Fits, and Tolerances

Introduction

✓ No two parts can be produced with identical measurements by any manufacturing process.

✓ In any production process, regardless of how well it is designed or how carefully it is maintained, a certain amount of natural variability will always exist.

✓ These natural variations are random in nature and are the cumulative effect of many small, essentially uncontrollable causes.

✓ Usually, variability arises from improperly adjusted machines, operator error, tool wear, and/or defective raw materials.

2

Introduction

✓ Such characteristic variability is generally large when compared to the natural variability.

✓ This variability, which is not a part of random or chance cause pattern, is referred to as 'assignable causes'.

✓ Characteristic variations can be attributed to assignable causes that can easily be identified and controlled.

✓ If the process can be kept under control, that is, all the assignable and controllable causes of variations have been eliminated or controlled, the size variations will be well within the prescribed limits.

Introduction

✓Some variability in dimension within certain limits must be tolerated during manufacture, however precise the process may be.

✓The permissible level of tolerance depends on the functional requirements, which cannot be compromised.

✓No component can be manufactured precisely to a given dimension; it can only be made to lie between two limits, upper (maximum) and lower (minimum).

✓The designer has to suggest these tolerance limits, which are acceptable for each of the dimensions used to define shape and form, and ensure satisfactory operation in service.

Introduction

✓ When the tolerance allowed is sufficiently greater than the process variation, no difficulty arises.

✓ The difference between the upper and lower limits is termed *permissive tolerance.*

✓ For example, a shaft has to be manufactured to a diameter of 40 ± 0.02 mm.

✓ This means that the shaft, which has a basic size of 40 mm, will be acceptable if its diameter lies anywhere between the limits of sizes, that is, an upper limit of 40.02 mm and a lower limit of 39.98 mm. Then permissive tolerance is equal to 40.02 − 39.98 = 0.04.

Tolerances

✓ Tolerance can be defined as the magnitude of permissible variation of a dimension or other measured value from the specified value.

✓ It can also be defined as the total variation permitted in the size of a dimension, and is the algebraic difference between the upper and lower acceptable dimensions. It is an absolute value.

✓ The basic purpose of providing tolerances is to permit dimensional variations in the manufacture of components, adhering to the performance criterion as established by the specification and design.

Tolerances

✓ If high performance is the sole criterion, then functional requirements dictate the specification of tolerance limits; otherwise, the choice of setting tolerance, to a limited extent, may be influenced and determined by factors such as methods of tooling and available manufacturing equipment.

✓ The industry follows certain approved accuracy standards, such as ANSI (American National Standards Institute) and ASME (American Society of Mechanical Engineers), to manufacture different parts.
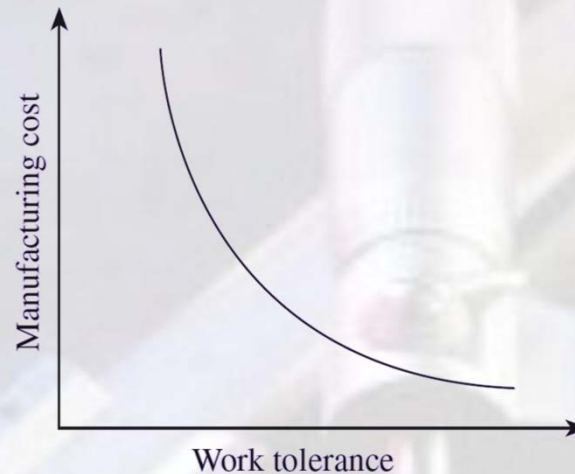
Manufacturing Cost and Work Tolerance

Fig. 3.2 Relationship between work tolerance and manufacturing cost

✓ It is very pertinent to relate the production of components within the specified tolerance zone to its associated manufacturing cost.

✓ As the permissive tolerance goes on decreasing, the manufacturing cost incurred to achieve it goes on increasing exponentially.

✓ When the permissive tolerance limits are relaxed without degrading the functional requirements, the manufacturing cost decreases.

Tolerances

Classification of Tolerance

Tolerance can be classified under the following categories:

1. Unilateral tolerance
2. Bilateral tolerance
3. Compound tolerance
4. Geometric tolerance

Unilateral Tolerance

When the tolerance distribution is only on one side of the basic size, it is known as unilateral tolerance.

In other words, tolerance limits lie wholly on one side of the basic size, either above or below it.
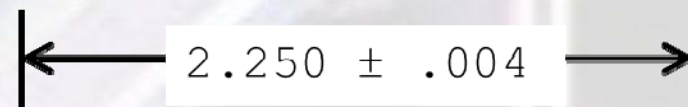
Example: $40^{+0.02}_{+0.01}$, $40^{+0.02}_{-0.00}$, $40^{-0.01}_{-0.02}$, $40^{+0.00}_{-0.02}$
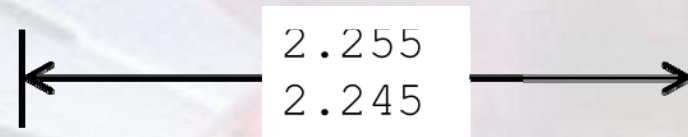
Tolerances

- **unilateral tolerance**

$$2.250 \begin{array}{l} +.000 \\ -.008 \end{array}$$

- **bilateral tolerance**

$$2.250 \pm .004$$

- **limit tolerance**

$$\begin{array}{l} 2.255 \\ 2.245 \end{array}$$

Tolerances

Bilateral Tolerance

When the tolerance distribution lies on either side of the basic size, it is known as bilateral tolerance.

In other words, the dimension of the part is allowed to vary on both sides of the basic size but may not be necessarily equally disposed about it.

Example:

$$40^{\pm 0.02}, \quad 40^{+0.02}_{-0.01}$$

Basic size
40 mm

(a)

(b)
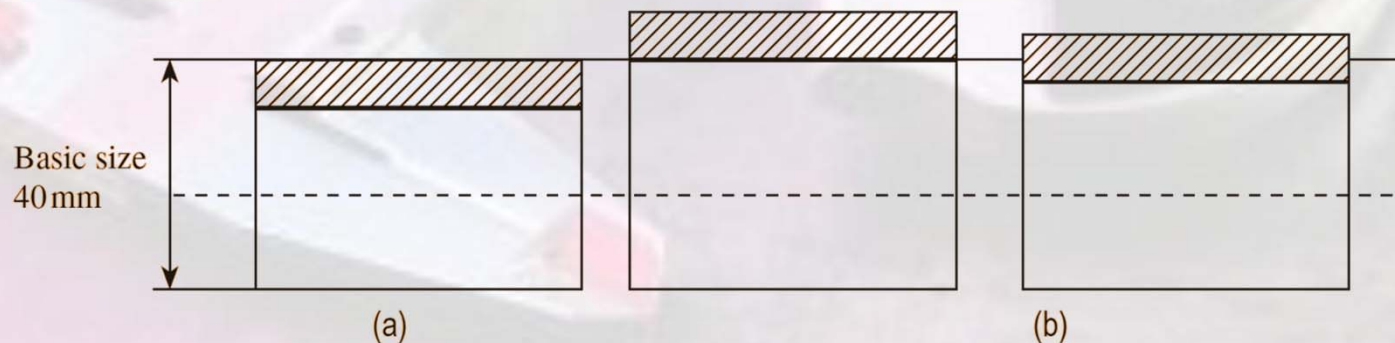
Fig. 3.3   Tolerances (a) Unilateral (b) Bilateral

Tolerances

Compound Tolerance

When tolerance is determined by established tolerances on more than one dimension, it is known as compound tolerance.

For example, tolerance for the dimension *R is determined by the combined effects of tolerance on 40 mm dimension, on 60º, and on 20 mm dimension. The tolerance obtained for dimension R is known as compound tolerance (Fig. 3.4). In practice, compound tolerance should be avoided as far as possible.*
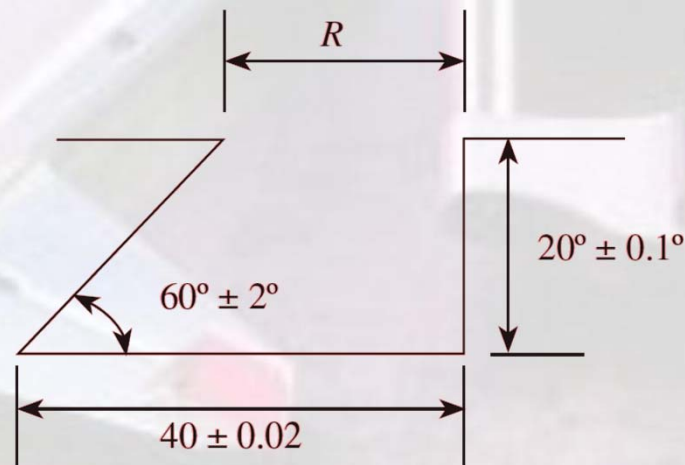

Fig. 3.4   Compound tolerance

Geometric Tolerance

✓ Geometric tolerances are used to indicate the relationship of one part of an object with another. Consider the example shown in Fig. 3.5.
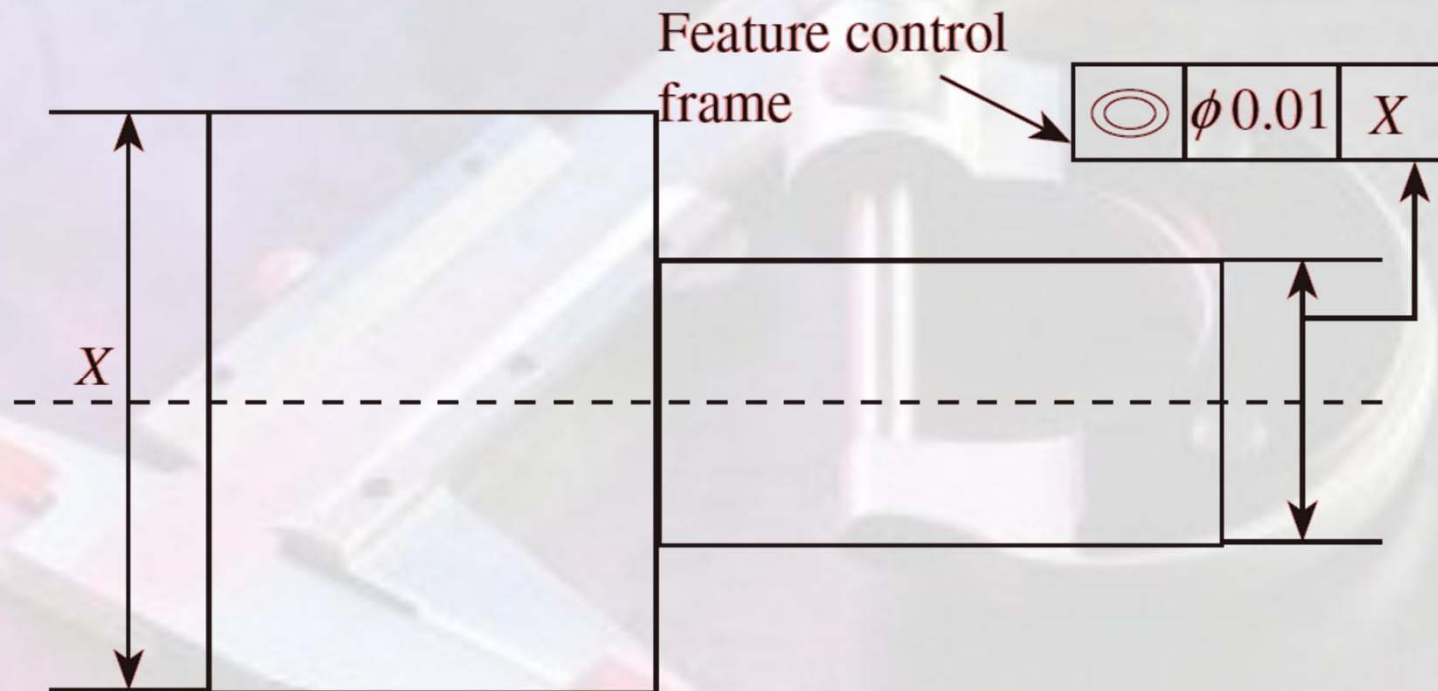
Fig. 3.5 Representation of geometric tolerance

Tolerances

✓ <u>Form tolerances:</u> Form tolerances are a group of geometric tolerances applied to individual features. They limit the amount of error in the shape of a feature and are independent tolerances. Form tolerances as such do not require locating dimensions. These include straightness, circularity, flatness, and cylindricity.

✓ <u>Orientation tolerances</u>: Orientation tolerances are a type of geometric tolerances used to limit the direction or orientation of a feature in relation to other features. These are related tolerances. Perpendicularity, parallelism, and angularity fall into this category.

✓ <u>Positional tolerances:</u> Positional tolerances are a group of geometric tolerances that controls the extent of deviation of the location of a feature from its true position. This is a three-dimensional geometric tolerance comprising position, symmetry, and concentricity.
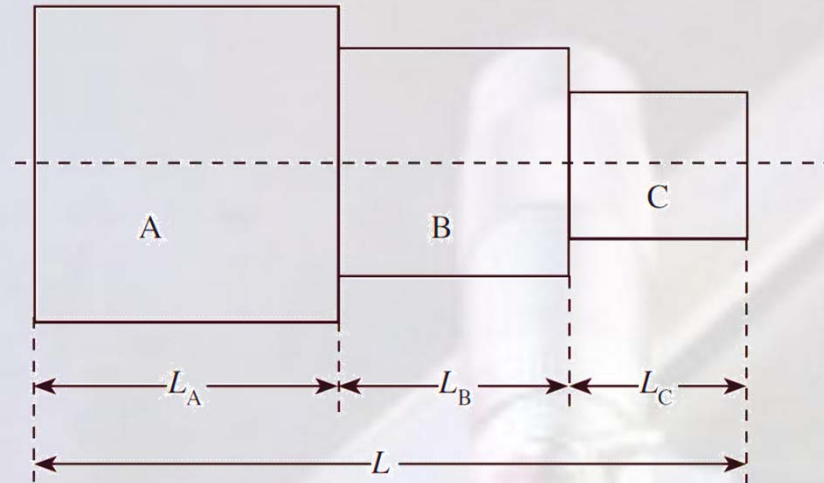
Fig. 3.6 Accumulation of tolerances

Consider the example shown if figure 3.6.

Let $L_A = 30^{+0.02}_{-0.01}$ mm, $L_B = 20^{+0.02}_{-0.01}$ mm and $L_C = 10^{+0.02}_{-0.01}$ mm

The overall length of the assembly is the sum of the individual length of components given as

$L = L_A + L_B + L_C$
$L = 30 + 20 + 10 = 60$ mm

Then, cumulative upper tolerance limit is 0.02 + 0.02 + 0.02 =0.06 mm and cumulative lower limit = – 0.01 – 0.01 – 0.01 = – 0.03 mm

Therefore dimension of the assembled length will be = $60 \, {}^{+\,0.06}_{-\,0.03}$ mm

It is essential to avoid or minimize the cumulative effect of tolerance build-up, as it leads to a high tolerance on overall length, which is undesirable.

If progressive dimensioning from a common reference line or a baseline dimensioning is adopted, then tolerance accumulation effect can be minimized. This is clearly illustrated in Fig. 3.7.
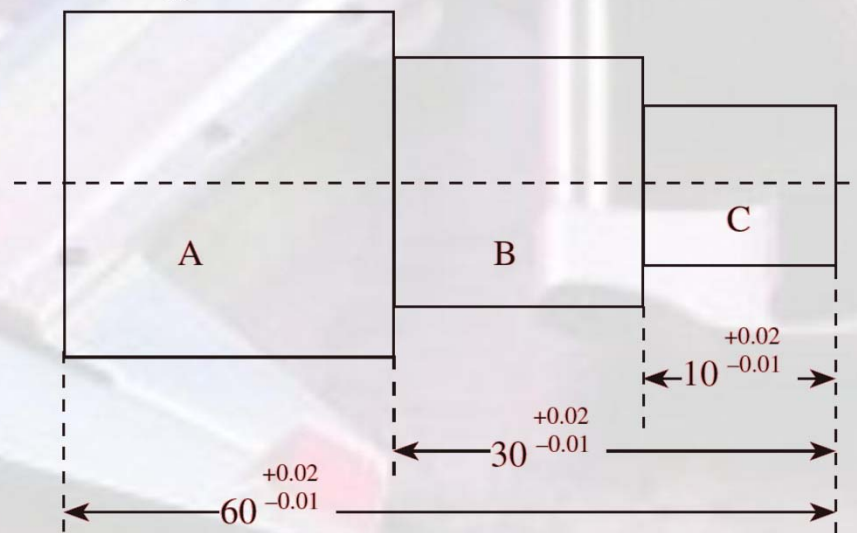

Fig. 3.7 Progressive dimensioning

Maximum and Minimum Metal Conditions

✓ Let us consider a shaft having a dimension of 40 ± 0.05 mm.

✓ The maximum metal limit (MML) of the shaft will have a dimension of 40.05 mm because at this higher limit, the shaft will have the maximum possible amount of metal.

✓ The shaft will have the least possible amount of metal at a lower limit of 39.95 mm, and this limit of the shaft is known as minimum or least metal limit (LML).

✓ Similarly, consider a hole having a dimension of 45 ± 0.05 mm.

✓ The hole will have a maximum possible amount of metal at a lower limit of 44.95 mm and the lower limit of the hole is designated as MML.

✓ For example, when a hole is drilled in a component, minimum amount of material is removed at the lower limit size of the hole. This lower limit of the hole is known as MML.

✓ The higher limit of the hole will be the LML. At a high limit of 45.05 mm, the hole will have the least possible amount of metal.

✓ The maximum and minimum metal conditions are shown in Fig. 3.8

FITS

Fits

✓ Manufactured parts are required to mate with one another during assembly.

✓ The relationship between the two mating parts that are to be assembled, that is, the hole and the shaft, with respect to the difference in their dimensions before assembly is called a *fit.*

✓ *An ideal fit is required for proper functioning of the mating parts. Three basic types of fits can be identified, depending on the actual limits of the hole or shaft:*
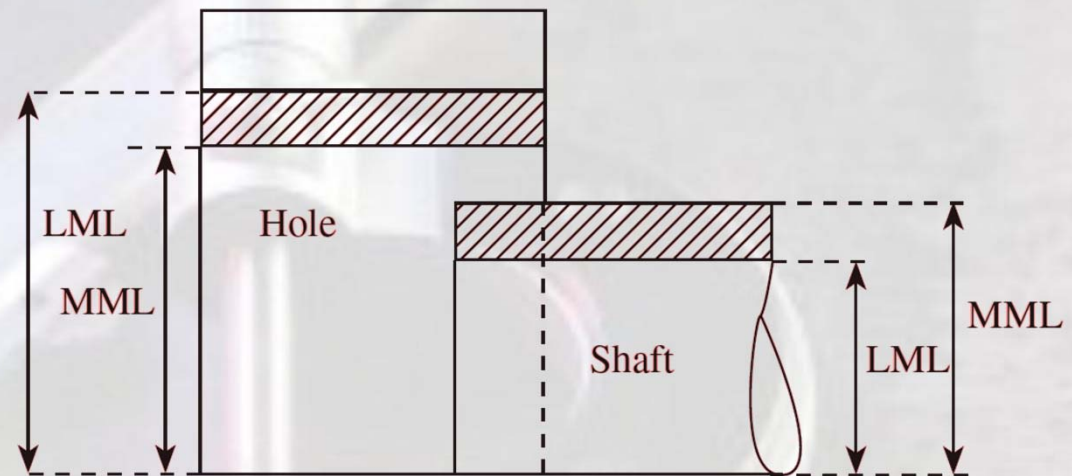
Fig. 3.8   MML and LML

FITS

1. Clearance fit
2. Interference fit
3. Transition fit

<u>Clearance fit</u>: The largest permissible diameter of the shaft is smaller than the diameter of the smallest hole.

✓ This type of fit always provides clearance. Small clearances are provided for a precise fit that can easily be assembled without the assistance of tools. When relative motions are required, large clearances can be provided, for example, a shaft rotating in a bush.

✓ In case of clearance fit, the difference between the sizes is always positive. The clearance fit is described in Fig. 3.9.
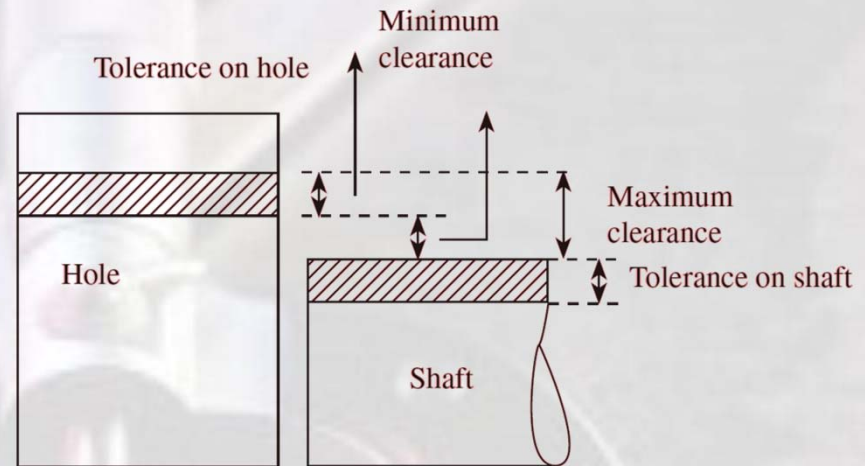
Fig. 3.9 Clearance fit

FITS

Interference fit: The minimum permissible diameter of the shaft exceeds the maximum allowable diameter of the hole.

✓
   This type of fit always provides interference. Interference fit is a form of a tight fit. Tools are required for the precise assembly of two parts with an interference fit.

✓ When two mating parts are assembled with an interference fit, it will be an almost permanent assembly, that is, the parts will not come apart or move during use. To assemble the parts with interference, heating or cooling may be required.

✓ In an interference fit, the difference between the sizes is always negative.

Allowance

✓ **Allowance:** An allowance is the intentional difference between the maximum material limits, that is, LLH and HLS (minimum clearance or maximum interference) of the two mating parts. It is the prescribed difference between the dimensions of the mating parts to obtain the desired type of fit.

✓ Allowance may be positive or negative. Positive allowance indicates a clearance fit, and an interference fit is indicated by a negative allowance.
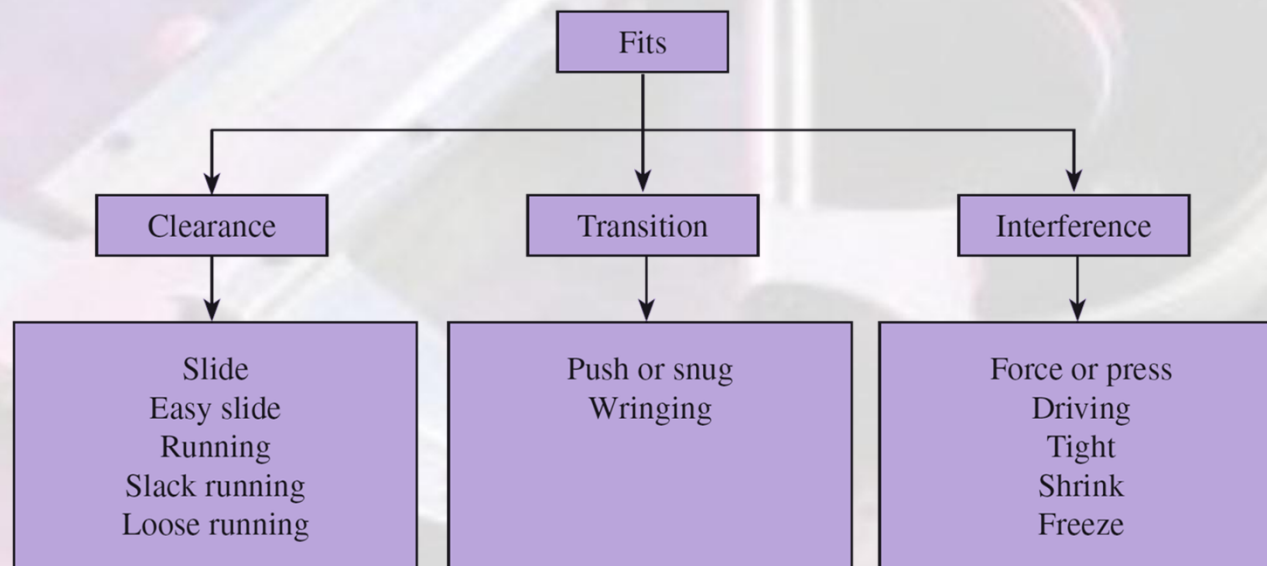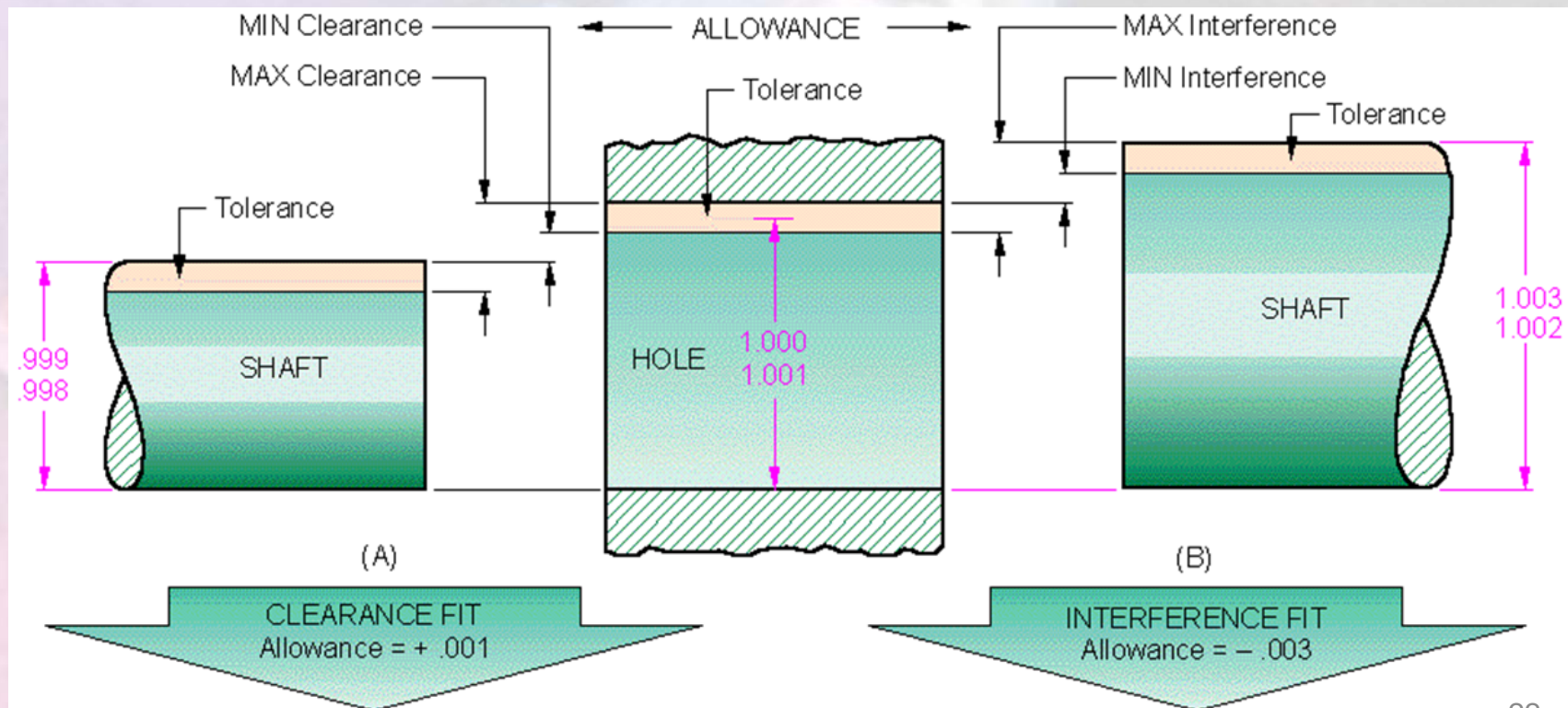
Allowance = LLH − HLS


Fig. 3.12 Detailed classification of fits

Fit Types

Type 1: Clearance fit = Occurs when two toleranced mating parts will always leave a space or clearance when assembled

Type 2: Interference fit = Occurs when two toleranced mating parts will always interfere when assembled

MIN Clearance ── ALLOWANCE ── MAX Interference
MAX Clearance ── Tolerance ── MIN Interference
Tolerance

Tolerance

.999 .998 — SHAFT

HOLE 1.000 1.001

SHAFT 1.003 1.002 — Tolerance

(A)

(B)

CLEARANCE FIT
Allowance = + .001

INTERFERENCE FIT
Allowance = − .003

Allowance always equals smallest hole minus largest shaft

22

Fit Types

● **Type 3: Transition fit** = Occurs when two toleranced mating parts are sometimes and interference fit and sometimes clearance fit when assembled.

23
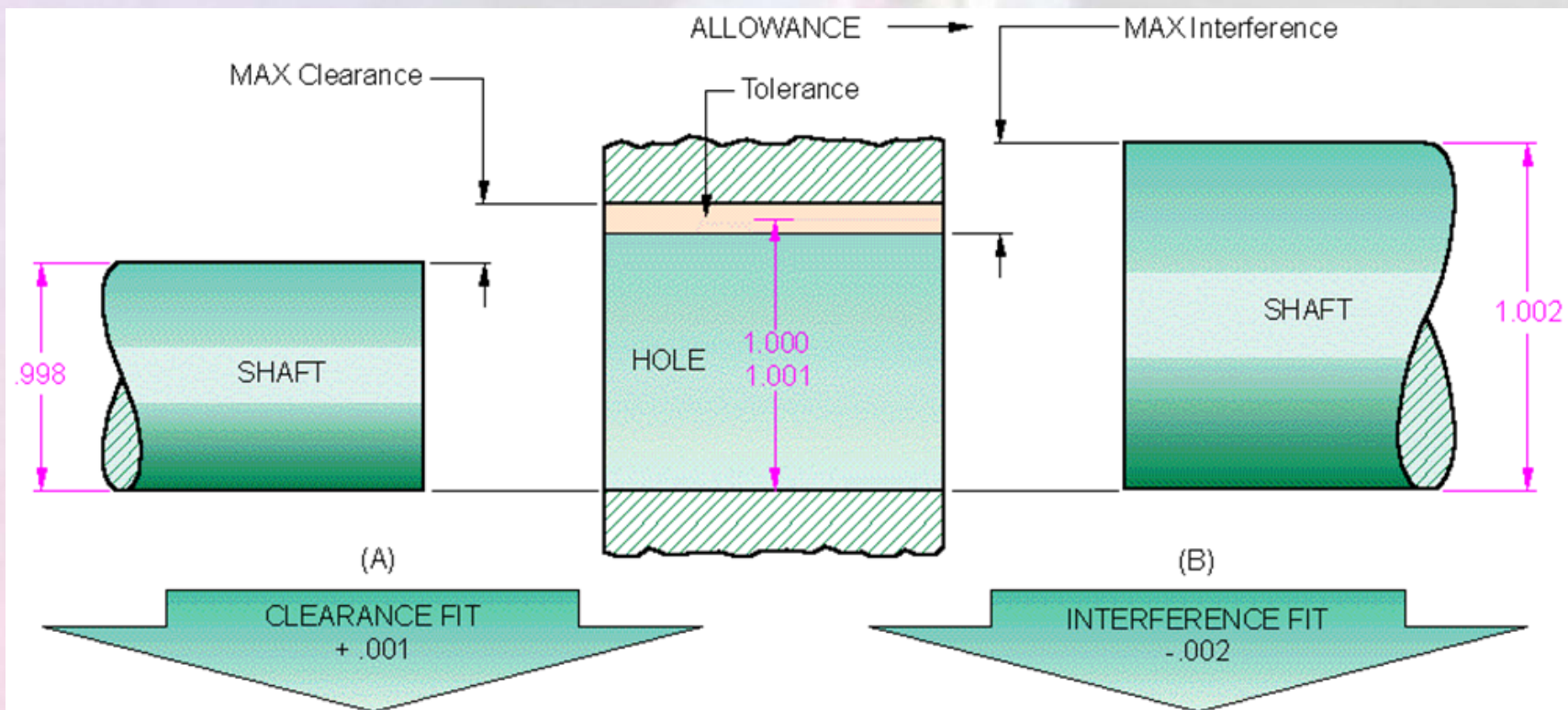
General Terminology in Fits

✓ <u>Basic size:</u> This is the size in relation to which all limits of size are derived. Basic or nominal size is defined as the size based on which the dimensional deviations are given. This is, in general, the same for both components.

✓ <u>Limits of size:</u> These are the maximum and minimum permissible sizes acceptable for a specific dimension. The operator is expected to manufacture the component within these limits. The maximum limit of size is the greater of the two limits of size, whereas the minimum limit of size is the smaller of the two.

✓ <u>Tolerance:</u> This is the total permissible variation in the size of a dimension, that is, the difference between the maximum and minimum limits of size. It is always positive.

✓ <u>Allowance:</u> It is the intentional difference between the LLH and HLS. An allowance may be either positive or negative.

Allowance = LLH − HLS

General Terminology in Fits

Grade: This is an indication of the tolerance magnitude; the lower the grade, the finer the tolerance.

Deviation: It is the algebraic difference between a size and its corresponding basic size. It may be positive, negative, or zero.

Upper deviation: It is the algebraic difference between the maximum limit of size and its corresponding basic size. This is designated as 'ES' for a hole and as 'es' for a shaft.

Lower deviation: It is the algebraic difference between the minimum limit of size and its corresponding basic size. This is designated as 'EI' for a hole and as 'ei' for a shaft.

Actual deviation: It is the algebraic difference between the actual size and its corresponding basic size.

Fundamental deviation: It is the *minimum difference between the size of a component and its basic size. This is identical to the upper deviation for shafts and lower deviation for holes.*

25

General Terminology in Fits

✓ <u>Zero line:</u> This line is also known as the line of zero deviation. The convention is to draw the zero line horizontally with positive deviations represented above and negative deviations indicated below. The zero line represents the basic size in the graphical representation.

✓ <u>Shaft and hole:</u> These terms are used to designate all the external and internal features of any shape and not necessarily cylindrical.

✓ <u>Fit:</u> It is the relationship that exists between two mating parts, a hole and a shaft, with respect to their dimensional difference before assembly.

General Terminology in Fits
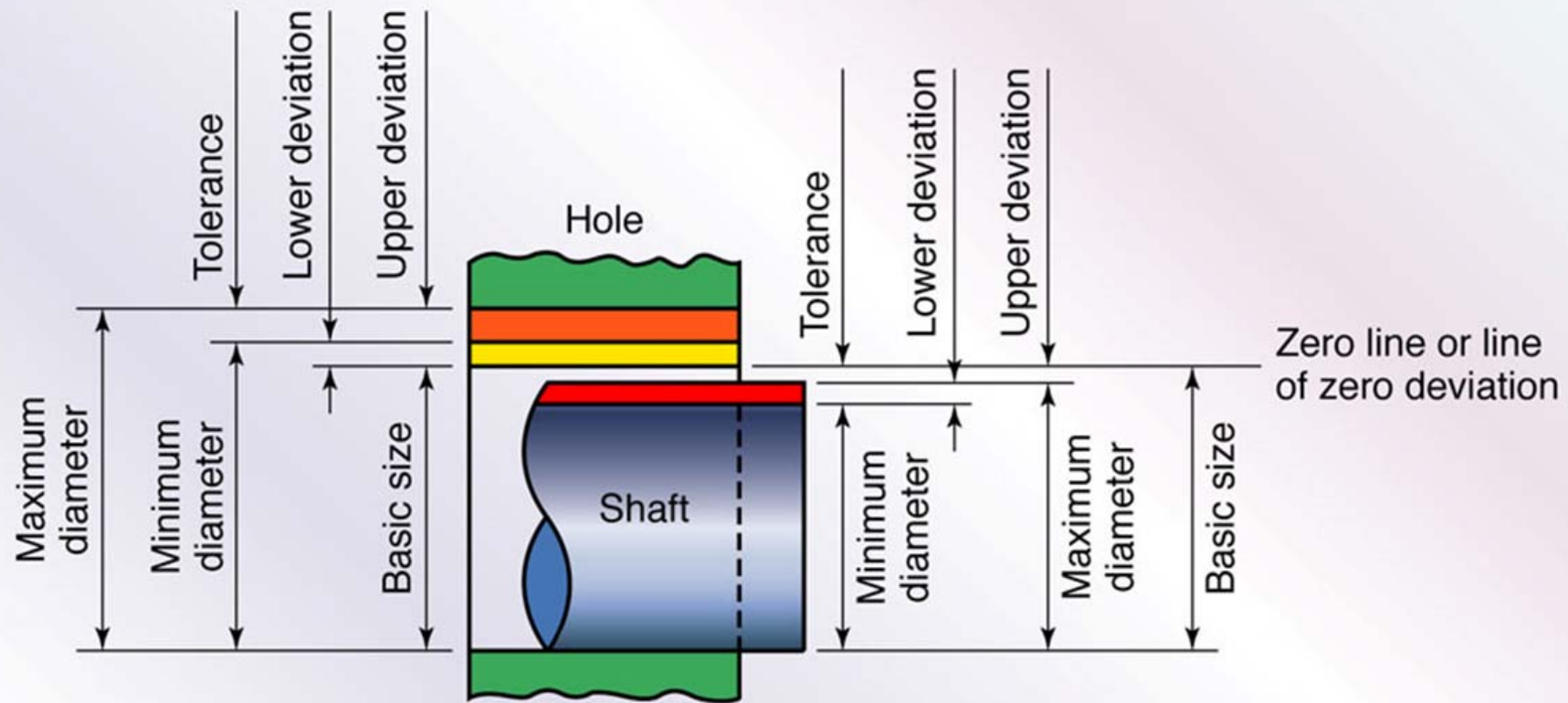
General Terminology in Fits

- Basic Hole System
  - The basic hole system is used to apply tolerances to a hole and shaft assembly.

Basic hole system
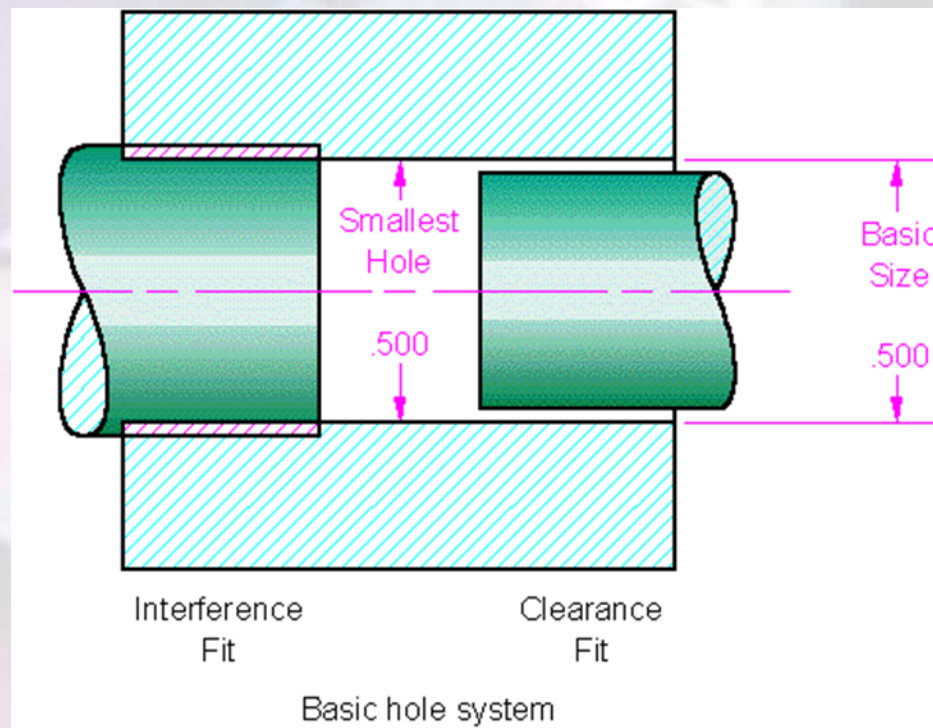
General Terminology in Fits

✓ <u>Tolerance symbols:</u> These are used to specify the tolerance and fits for mating components. For example, in 40 H8f7, the number 40 indicates the basic size in millimeters; capital letter H indicates the fundamental deviation for the hole; and lower-case letter f indicates the shaft. The numbers following the letters indicate corresponding IT grades.
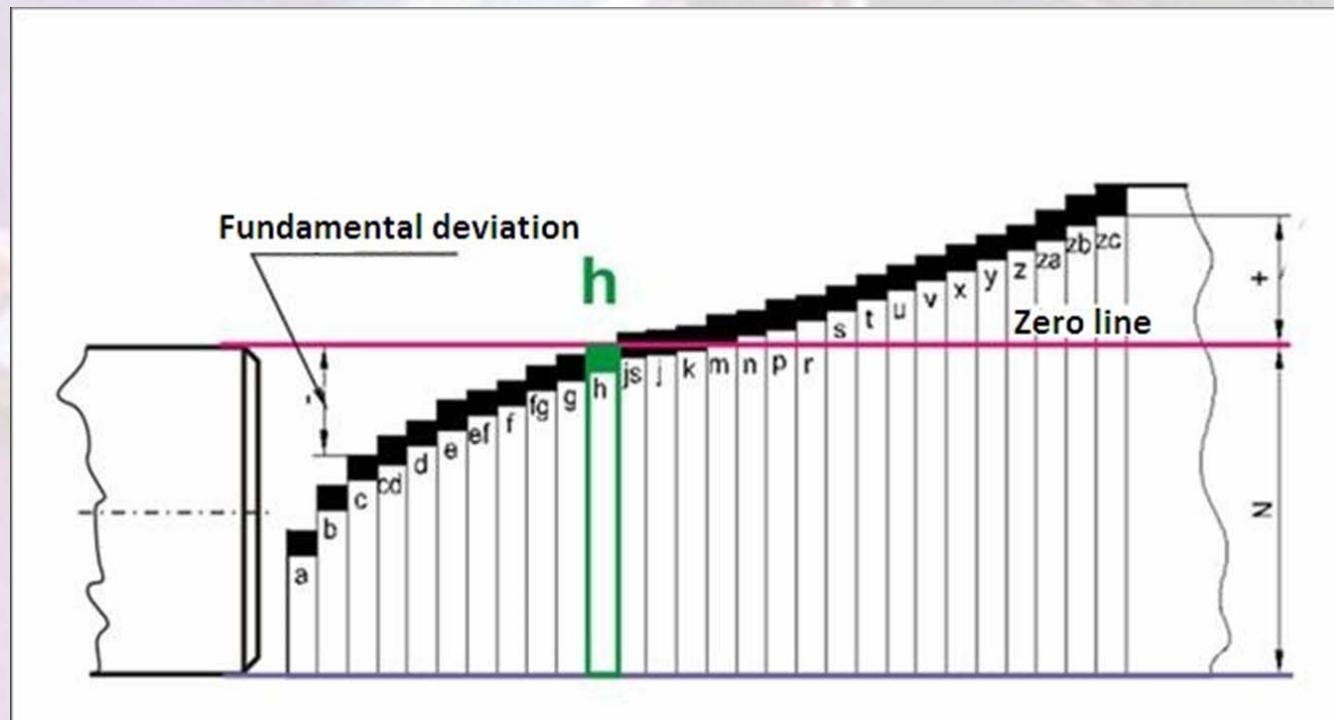
General Terminology in Fits

ISO Tolerances for Shafts (ISO 286-2)

	Nominal Shaft Sizes (mm)

	3	6	10	18	30	40	50	65	80	100	120	140	160	180	200	225	250	280	315	355
over	3	6	10	18	30	40	50	65	80	100	120	140	160	180	200	225	250	280	315	355
inc.	6	10	18	30	40	50	65	80	100	120	140	160	180	200	225	250	280	315	355	400

micrometres

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

30

General Terminology in Fits

The ISO System of Limits and Fits (referred to as the ISO system) is covered in national standards throughout the world, as shown by the following list:

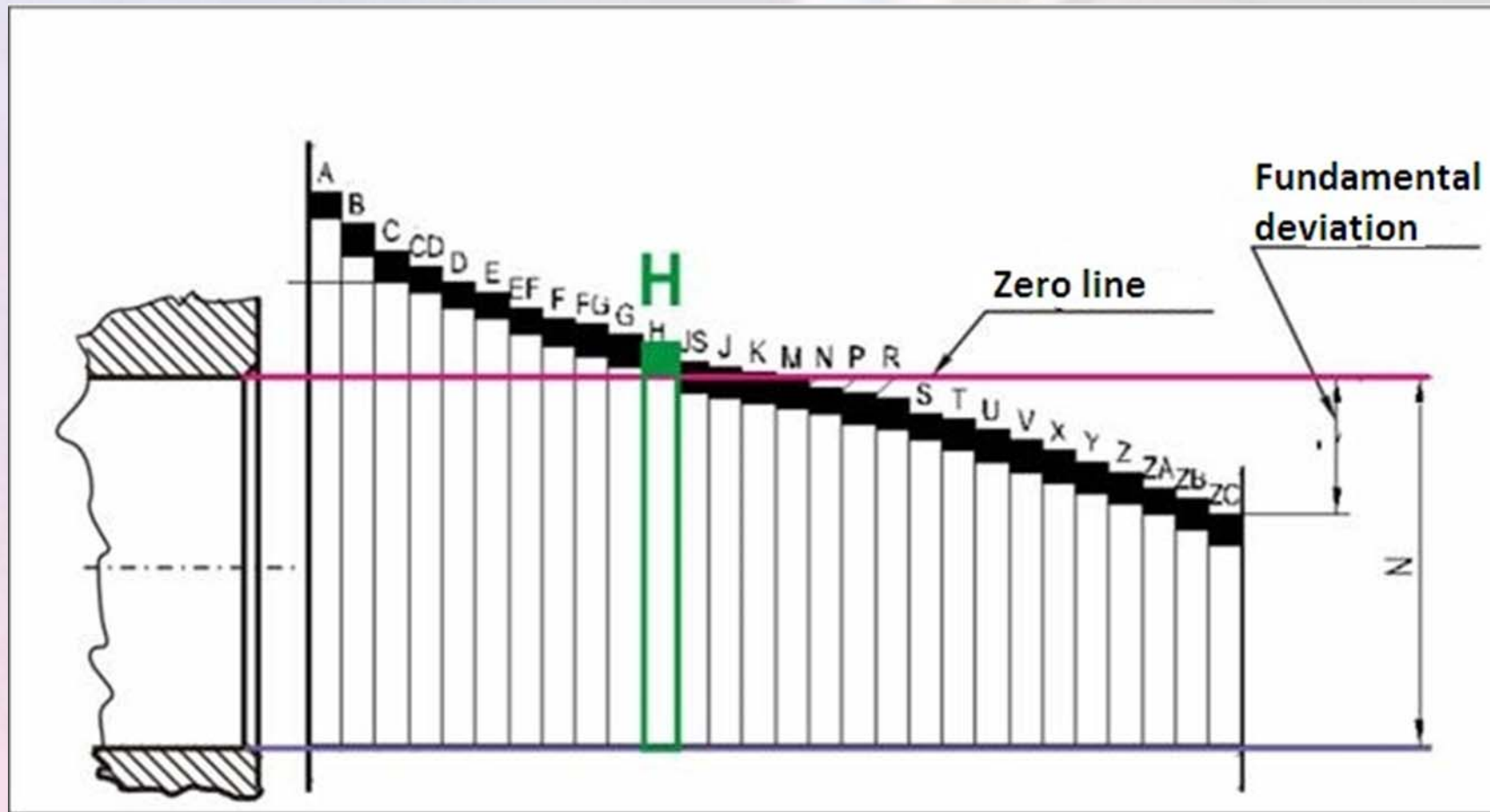Global ISO 286
USA ANSI B4.2
Japan JIS B0401
Germany DIN 7160//61
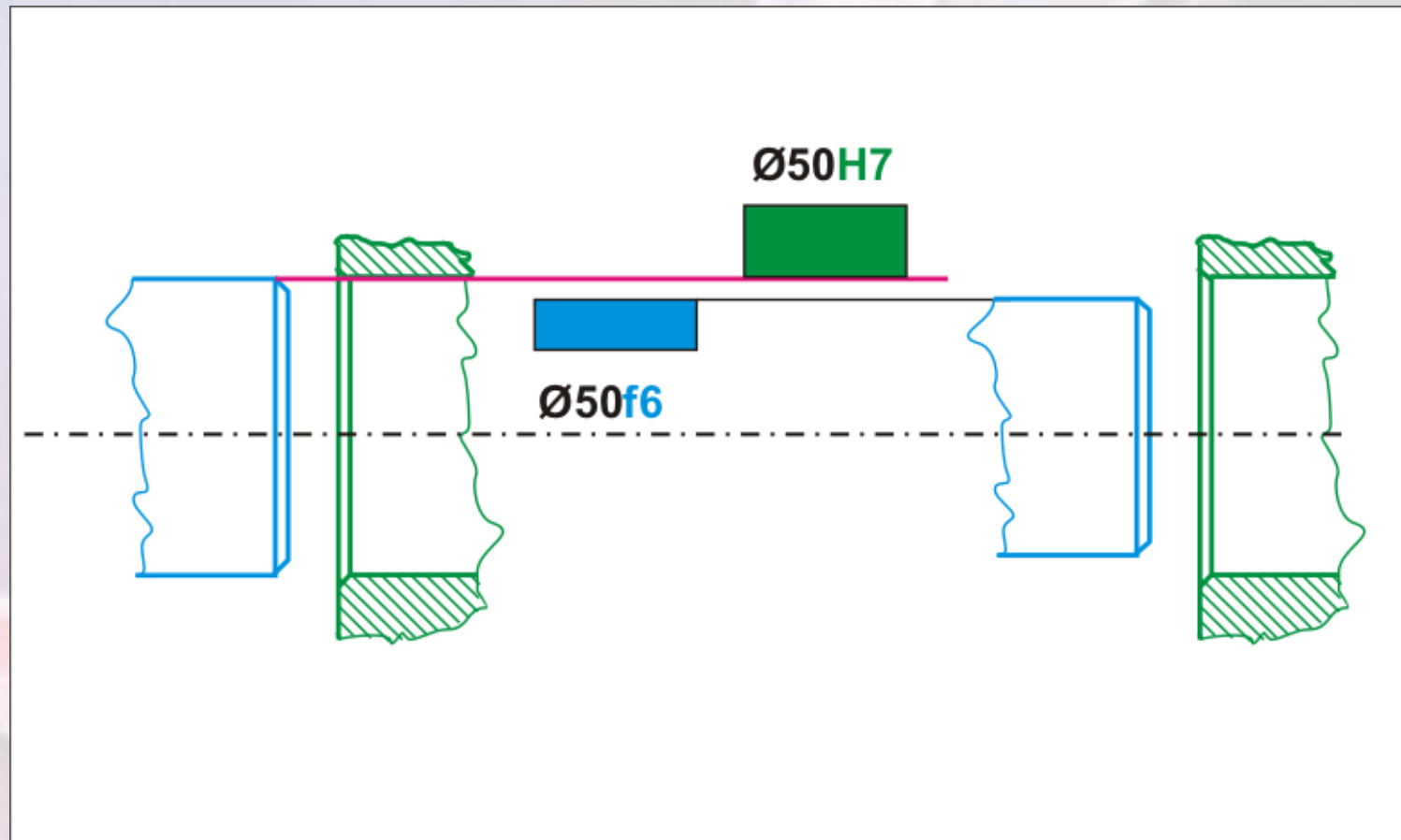France NF E 02-100-122
UK BSI 4500
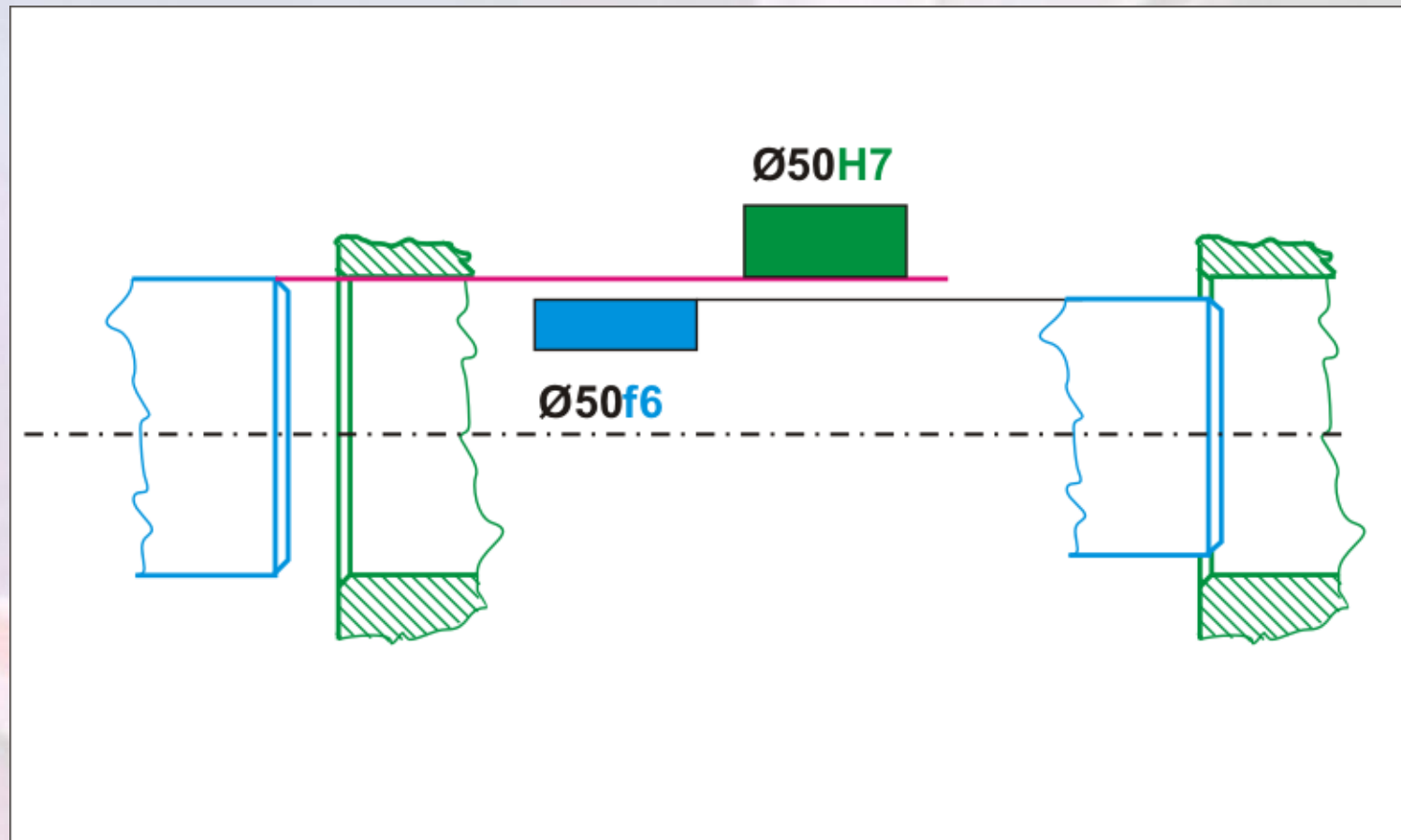Italy UNI 6388
Australia AS 1654
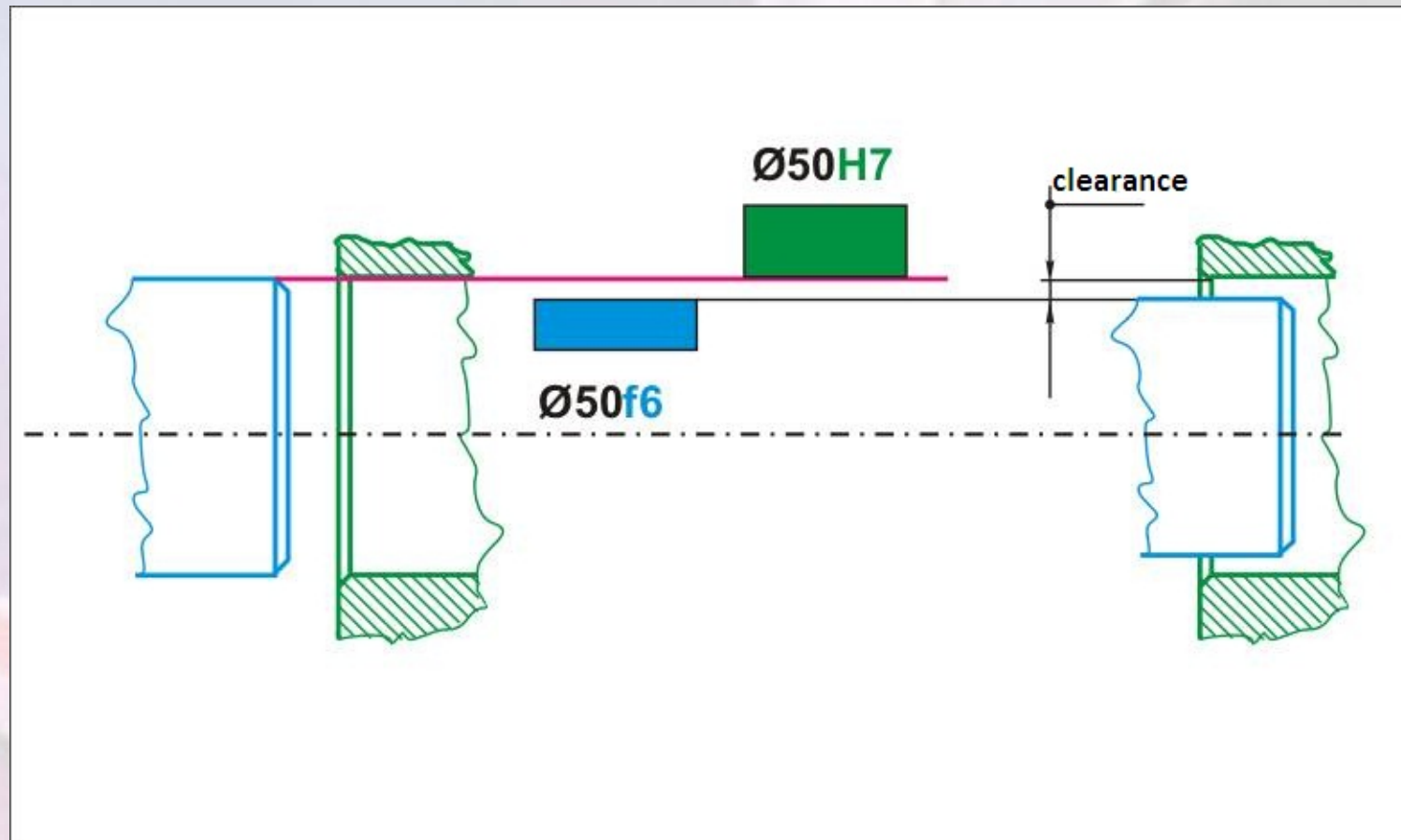
General Terminology in Fits
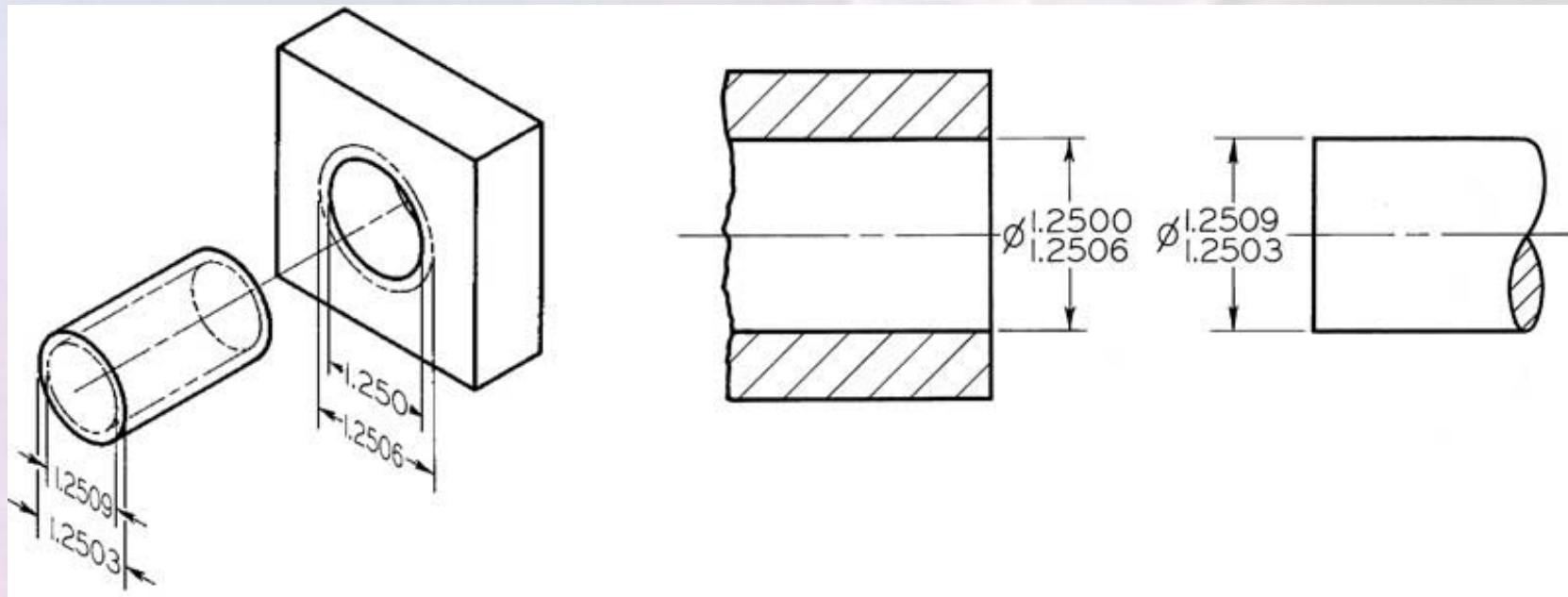
Clearance Fit (e.g.: H7/f6)

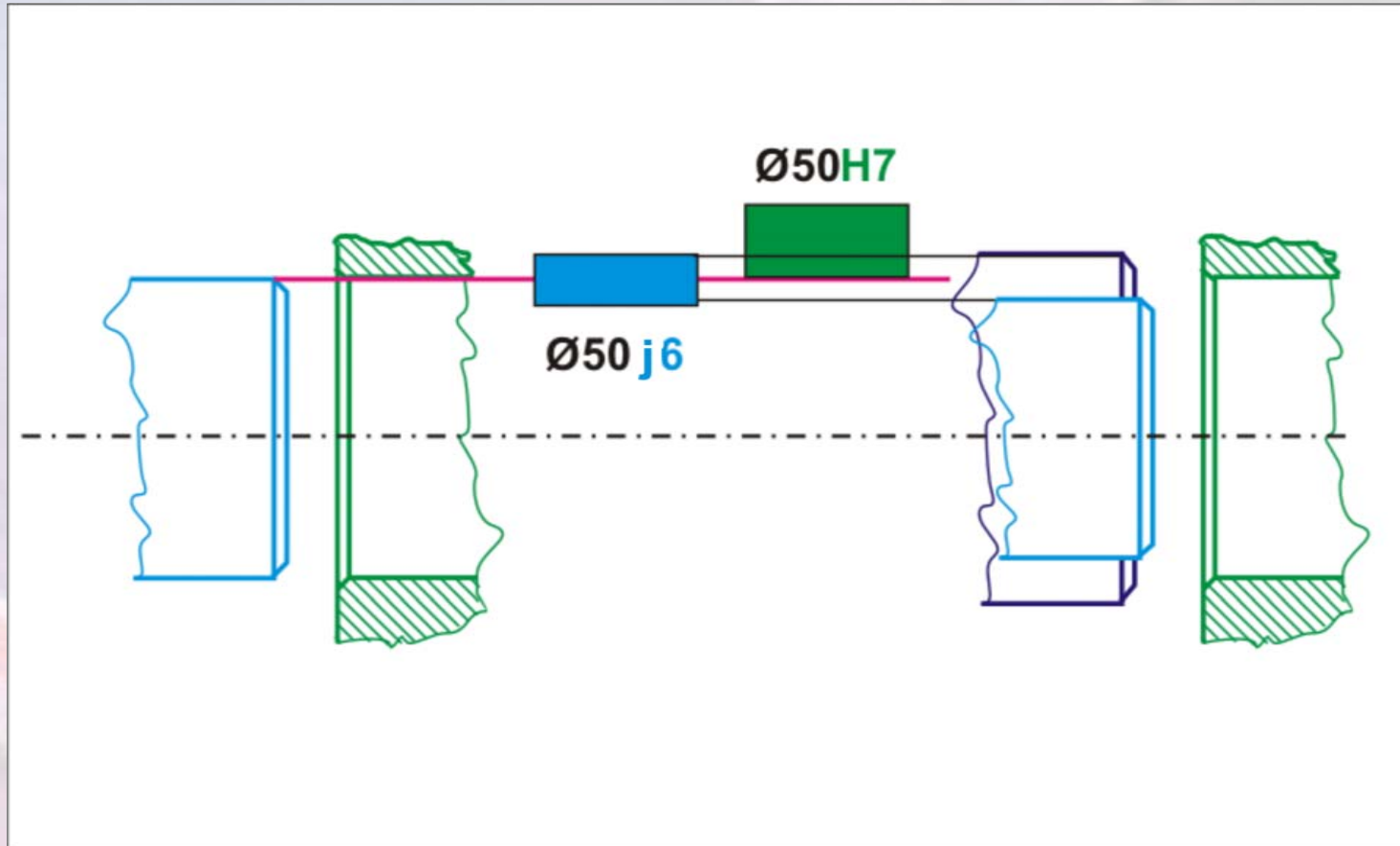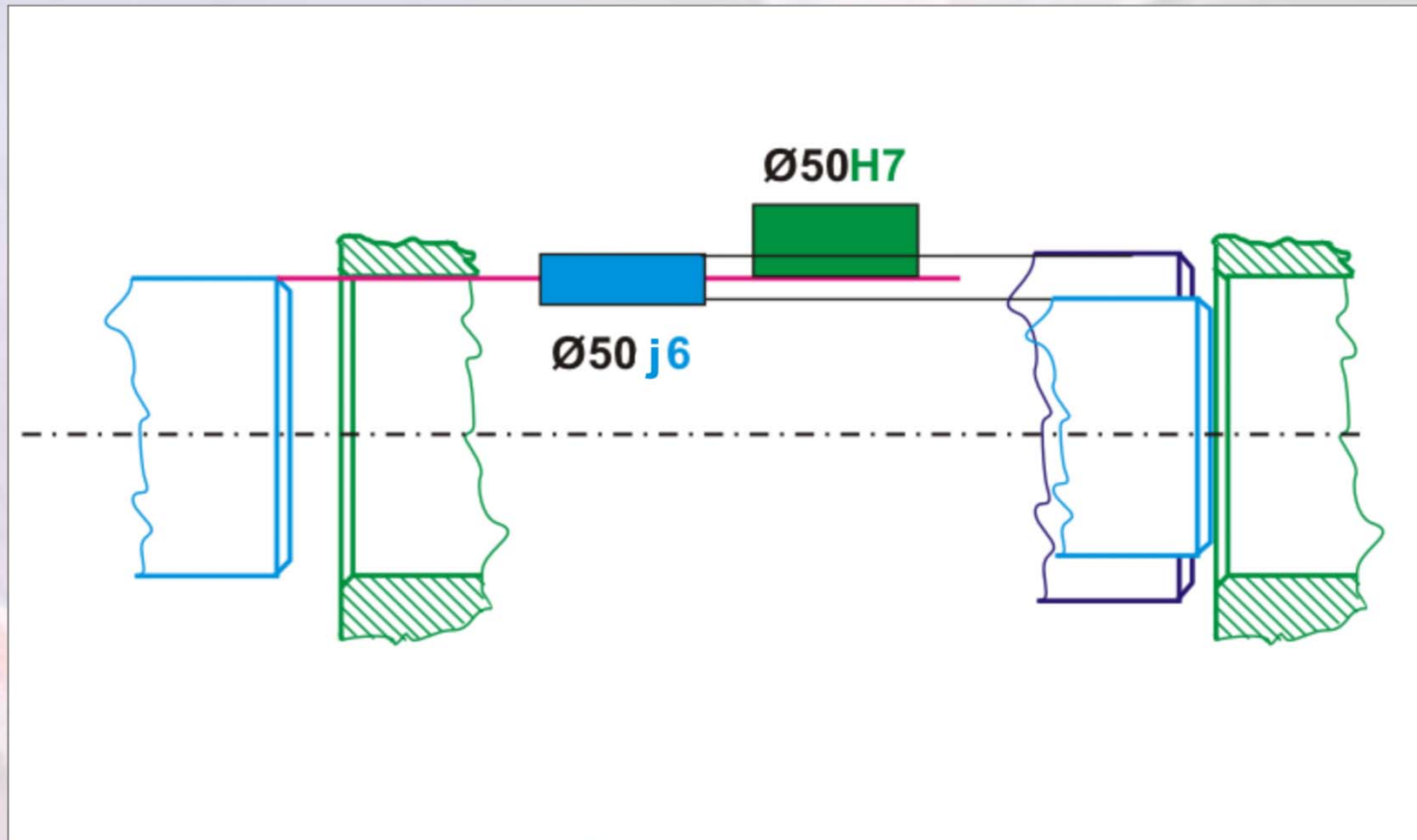# Clearance Fit (pl. H7/f6)

Clearance Fit (pl. H7/f6)

Transition Fit

Either a **clearance** or an **interference** may result depending on the exact value of the dimensions of the machined shaft and hole within the specified tolerance zones.
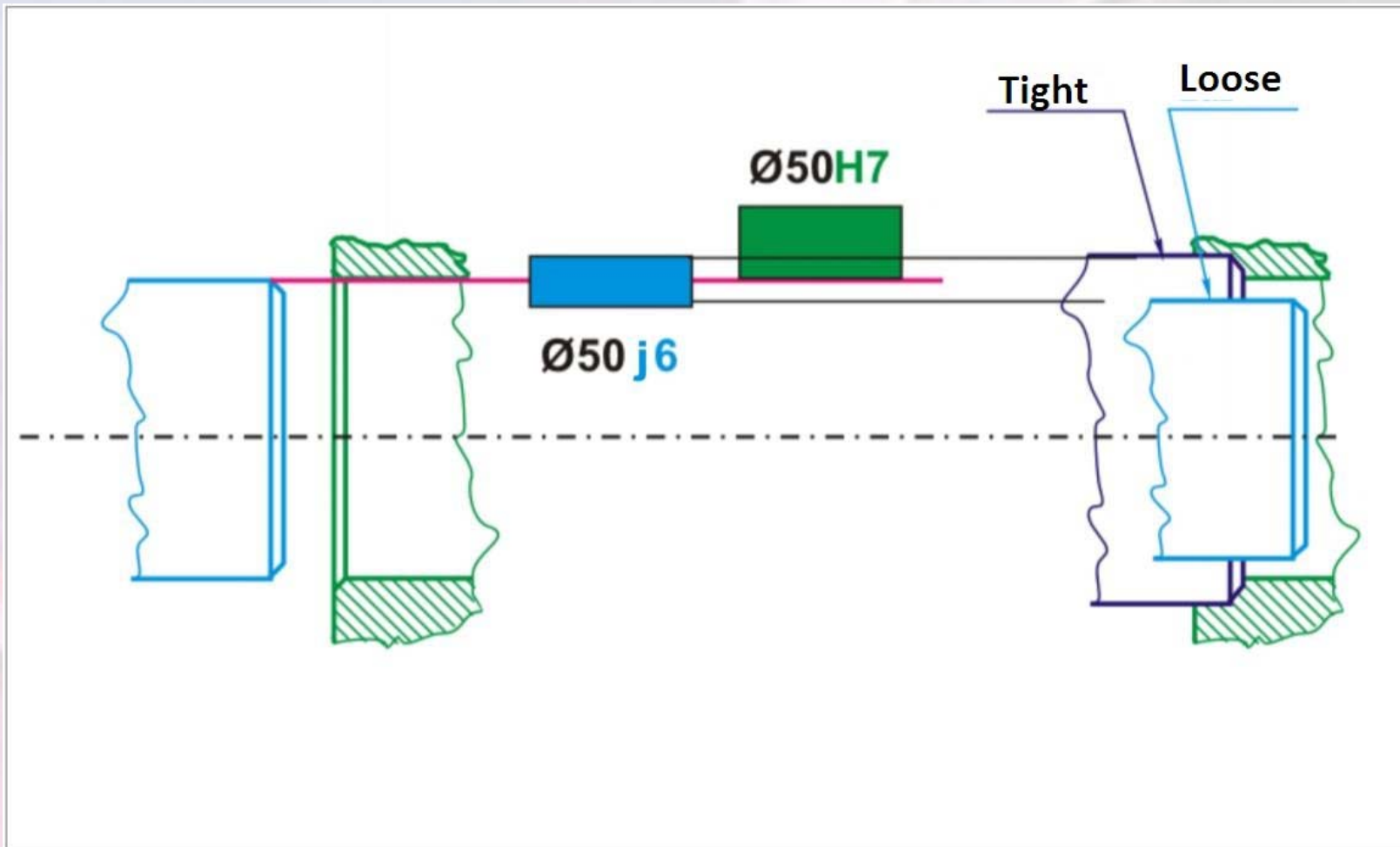
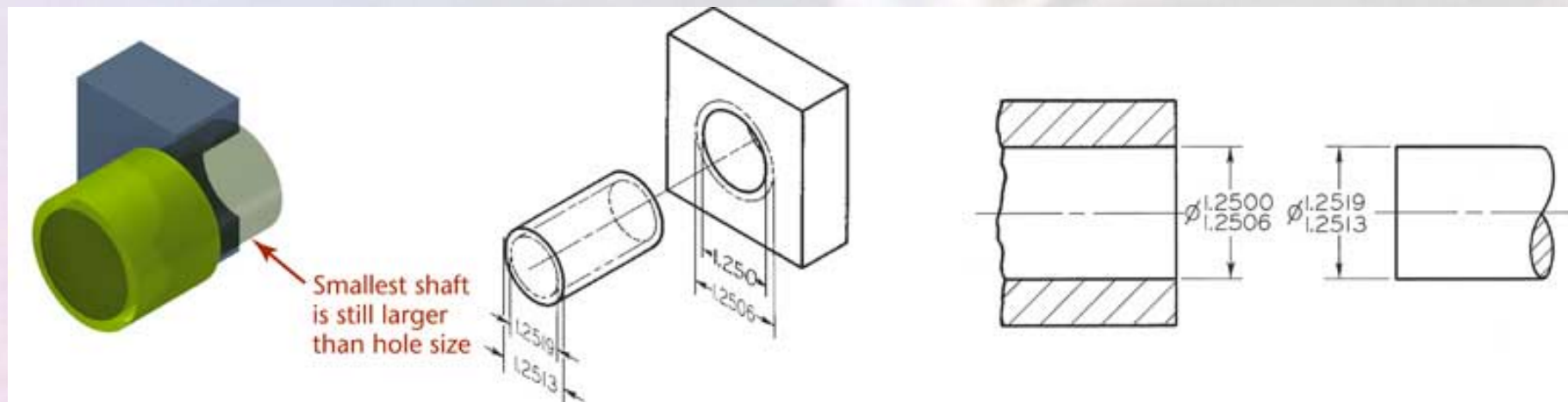# *Transition Fit (e.g.: H7/j6)*

Transition Fit (e.g.: H7/j6)

Transition Fit (e.g.: H7/j6)
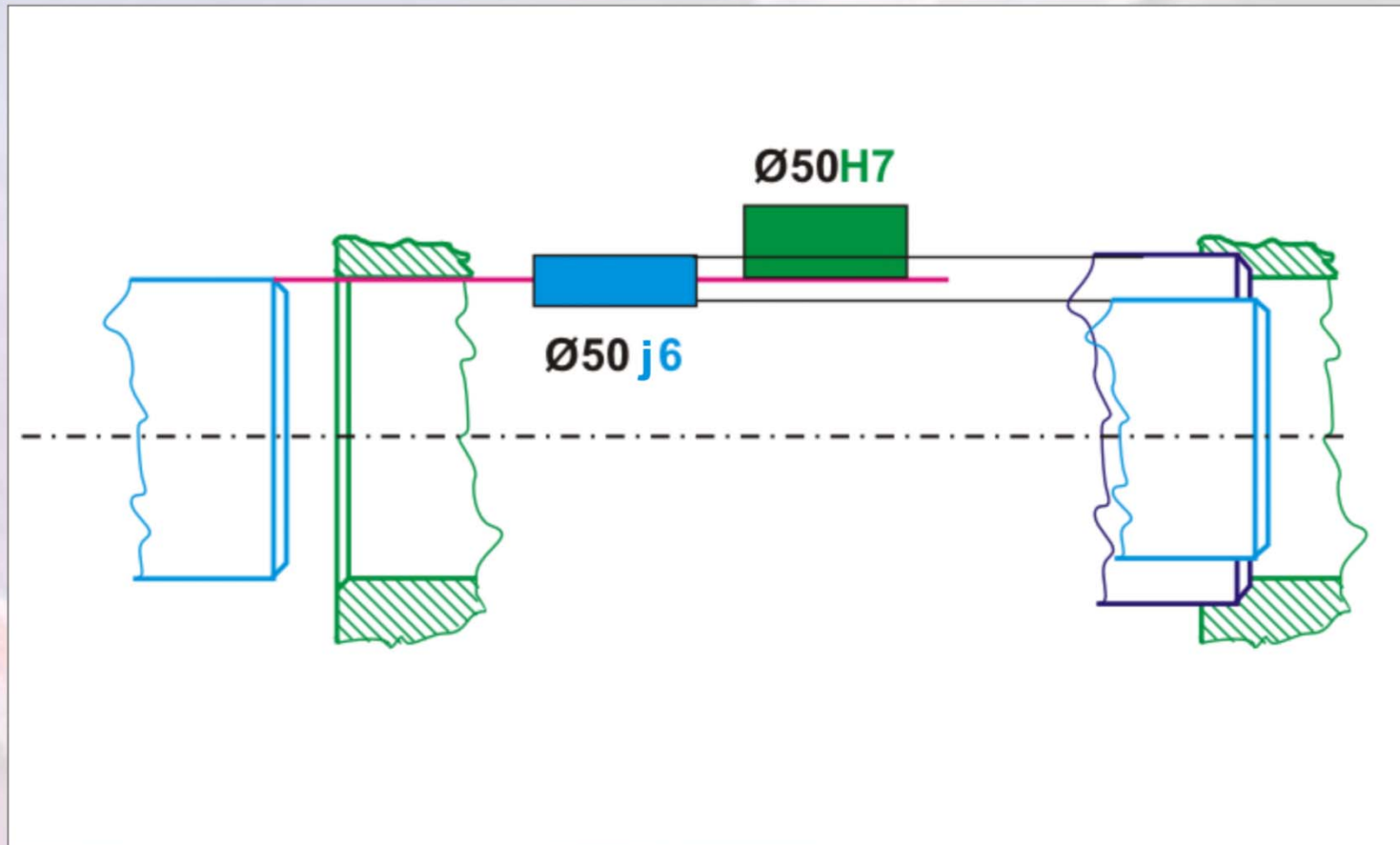
Transition Fit (e.g.: H7/j6)

Interference Fit
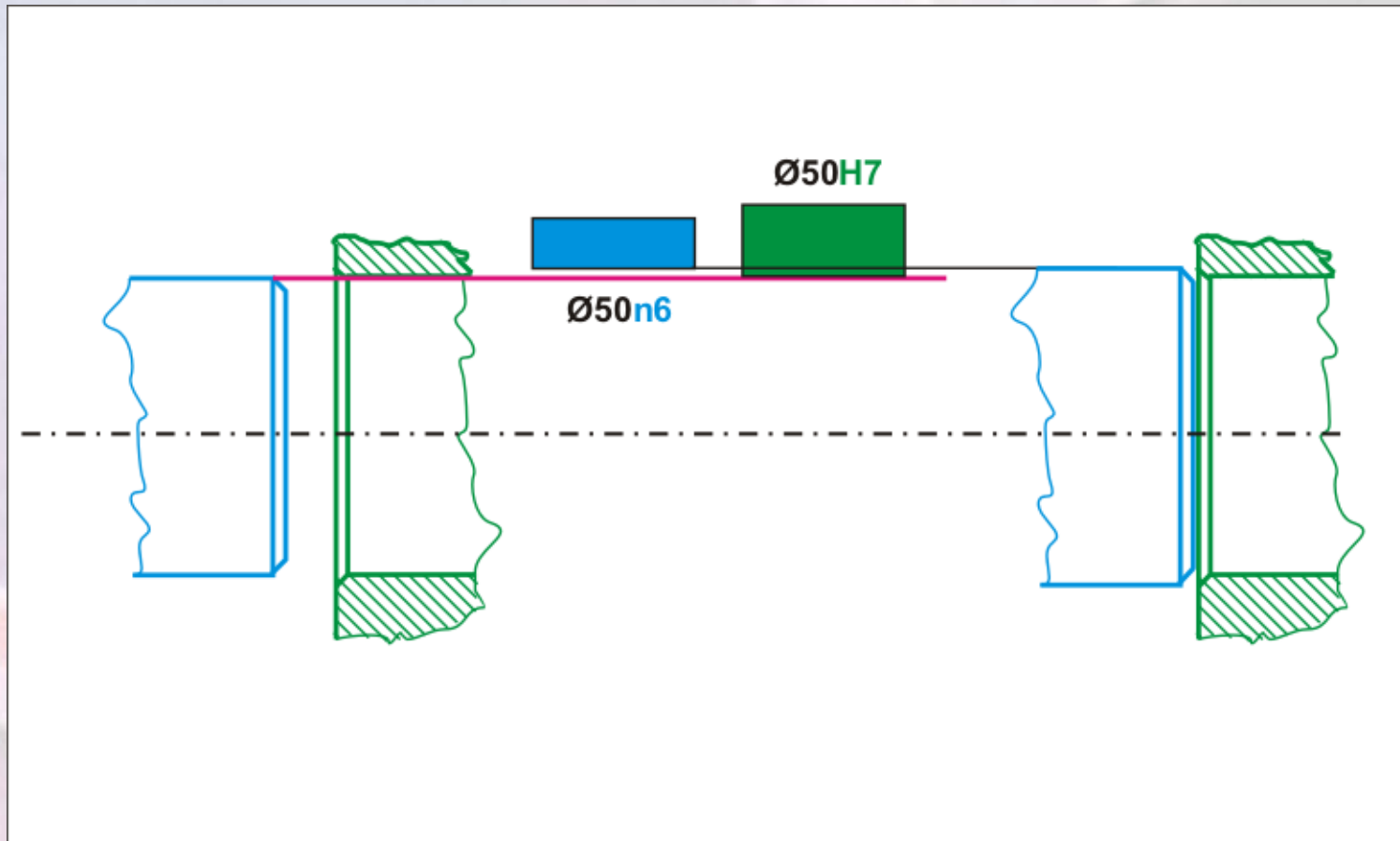

Smallest shaft is still larger than hole size

The mating parts have such limits that the lowest shaft diameter is larger than the largest hole diameter..
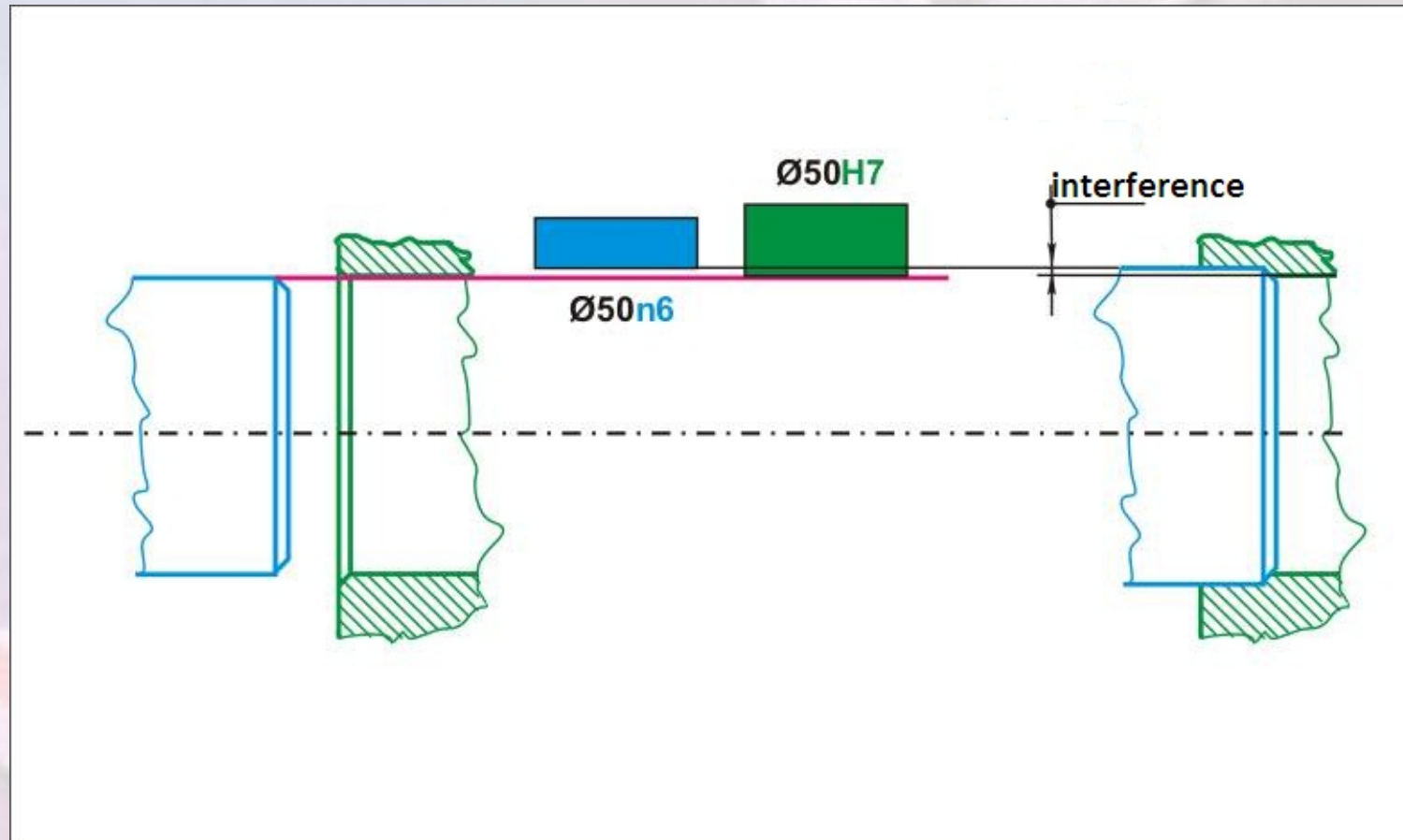
Interference Fit (H7/n6)
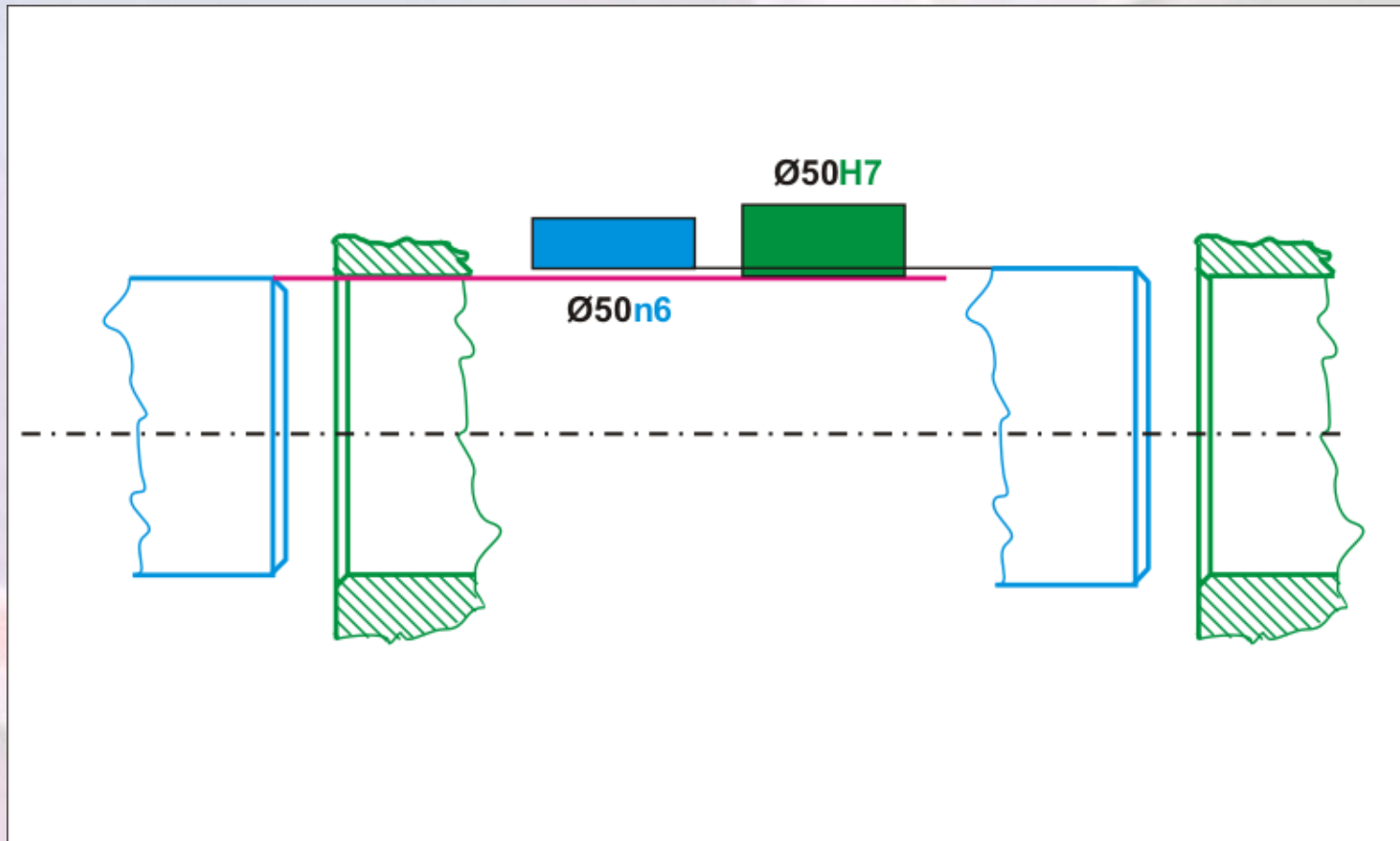
Interference Fit (H7/n6)

Interference Fit (H7/n6)

Example: Ø50H7/f6

- Online calculation of Fits

- http://www.tss.trelleborg.com/global/en/service/fits_tolerance/fits_tolerance_1.html

»ISO Fits & Tolerances is also available as a mobile app.

Units of measurement	⦿ Metric	○ Inches		
	Bore		**Shaft**	
Tolerance class	H ▾	7 ▾	f ▾	6 ▾

Nominal Size	50	Range between 1 - 3150 Millimeters

|	Bore	Shaft
Lower limit deviation	0 micrometer [µm]	-41 micrometer [µm]
Upper limit deviation	25 micrometer [µm]	-25 micrometer [µm]

⚑ Combination returns a **clearance fit**

Minimum interference	=	25 micrometer [µm]
Maximum interference	=	66 micrometer [µm]

	Hole	Shaft
Upper Limit	50.025	49.975
Lower Limit	50	49.959

45

Two ways of indicating tolerances on technical drawings

Limits of a dimension or the tolerance values are specified directly with the dimension.

Ø50 +0,05 -0,1

Ø50 h7
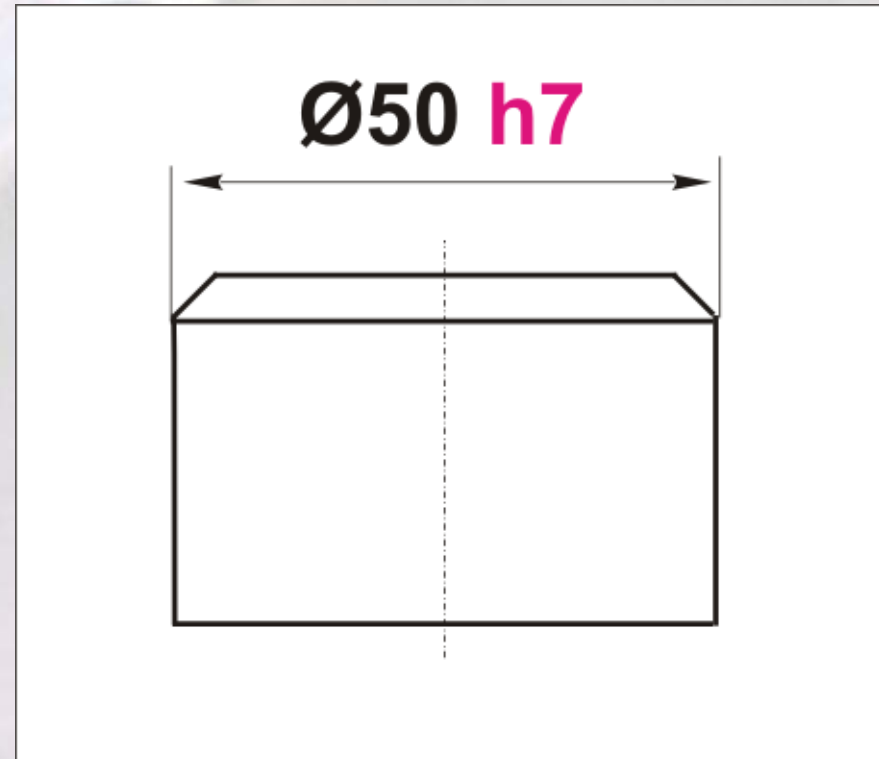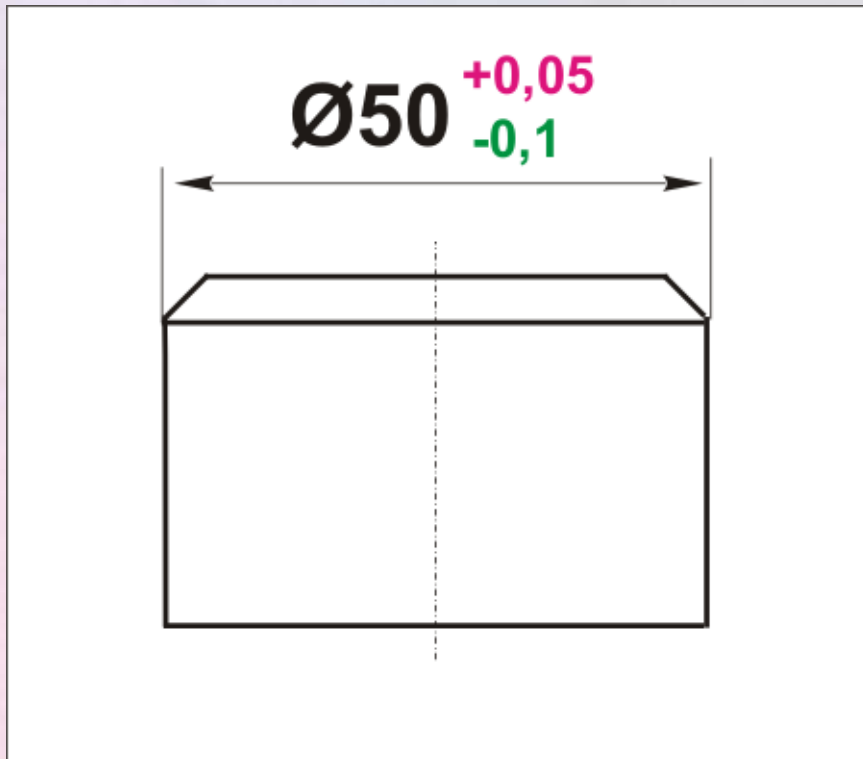
Indicating tolerances

The dimension is given by:

- a shape symbol,
- nominal size,
- a letter indicating the position of the tolerance zone in relation to zero line,
- a number indicating the width of the tolerance zone.
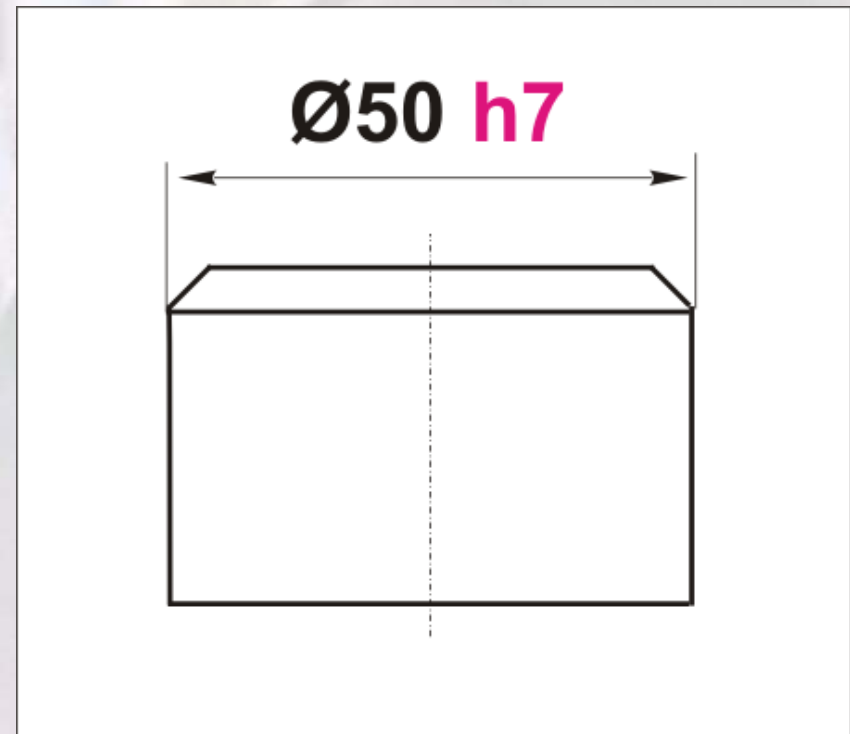
Specifying Fits in technical Drawing