The circular tolerance zone is 57 % larger in area than the square tolerance zone and this increase is achieved generally without prejudicing the function of the part.

It is not possible to assign tolerances to the two 30 mm location dimensions so that a circular tolerance zone results. However, if the diameter of the tolerance zone is specified, with its centre at the theoretically exact position of the hole centre, untoleranced location dimensions can be used. In the positional tolerancing system these were known as true position or basic dimensions and were followed by the abbreviation (TP). They are now shown in boxes (see 7.1), are theoretically exact and, in effect, locate the position of a tolerance zone.

As well as increasing the size of tolerance zones and simplifying the indication of location dimensions, positional tolerancing using true position dimensions has advantages when applied to groups of holes and other features. Since untoleranced dimensions are used, difficulties caused by the accumulation of tolerances are avoided and the calculation of tolerance values, based on the functional requirements of the components, is easy. (See clause **12**.) Furthermore, where the use of the maximum material principle is appropriate (see clause **11**) an additional increase in tolerance can be obtained and functional gauges, which simulate the mating part, can be used.

Positional tolerancing was applied to only one geometrical characteristic of a feature but the advantages of stating permitted tolerances directly led to an extension of its use to cover tolerances of form, such as flatness, and attitude, such as parallelism. The increased field of application of the system led to its name being changed to 'geometrical tolerancing'.

NOTE. The titles of the standards publications referred to in this document, together with those of related publications, are listed on the inside back cover.

0. Notes on the use of this document

0.1 The figures are complete only in so far as is necessary to illustrate the point under consideration. Linear dimensions and tolerance values on the figures are in millimetres.

0.2 The numerical values of dimensions and tolerances are examples only and are not quoted as recommended practices.

0.3 Wide tolerance zones and exaggerated feature deviations within them have been used in the interpretations of the examples to illustrate the principles clearly. They should not be considered typical.

0.4 The drawings given as interpretations of the examples are provided for explanatory purposes only and should not appear on component drawings.

0.5 FIRST ANGLE projection has been used for all illustrations drawn in orthographic projection. THIRD ANGLE projection could equally well have been used without affecting the principles established.

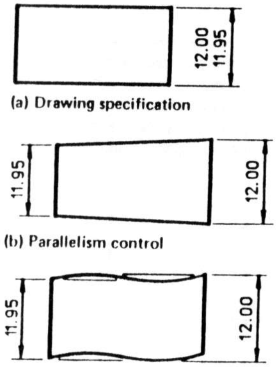
0.6 The corresponding clause numbers of BS 308 are shown in parentheses and in non bold type, after the bold type clause numbers of this document to facilitate cross-referencing between them. The BS 308 clause numbers refer to Part 3 : 1972 except where noted otherwise.

1. Scope

This publication discusses the principles set out in BS 308 : Part 3 : 1972 and is intended for use in engineering courses in colleges, polytechnics and universities, and in industrial training establishments, to introduce the subject of geometrical tolerancing.

2. (1.3) When geometrical tolerances are used

2.1 (1.3.1 and Part 2, 9.1, 9.2) A size tolerance applied to a dimension can effect some degree of control over deviations of form. For example, for the component shown in figure 1(a) the size tolerance has an effect on the parallelism of the upper and lower faces, as shown in figure 1(b), and on the flatness of those surfaces, as shown in figure 1(c).



(a) Drawing specification

(b) Parallelism control

(c) Flatness control

Figure 1. Effect of size tolerance on deviations of form

If, however, a different degree of control of form is required, geometrical tolerances of form should be specified in accordance with BS 308 : Part 3 : 1972. These form tolerances then take precedence over the form control implied by the size tolerance.

Positional deviations are also controlled by geometrical tolerances.

2.2 The use of geometrical tolerances prevents such notes as 'surfaces to be flat and parallel' appearing on drawings, with their inherent difficulties of interpretation. Furthermore, since the geometrical tolerancing symbols are internationally agreed (see ISO/R 1101), language difficulties cannot occur.

2.3 (1.3.2) Geometrical tolerances should be specified for all requirements critical to functioning and interchangeability except when it is known that the machinery and manufacturing techniques that will be used can be relied on to achieve the required standard of accuracy. The use of geometrical tolerances will permit satisfactory functioning and interchangeability when parts are made in different locations, on varying equipment and by personnel with varying experience.

2.4 Geometrical tolerances should be specified only where they are essential, otherwise the manufacturing and inspection costs for components can be increased. In any case tolerances should be as large as possible, subject to the design requirements being met.

2.5 (1.4.6) The use of geometrical tolerances does not imply that any particular method of production or inspection is to be used.

3. Features of a component

3.1 Single features. Figure 2 illustrates some of the single features that may be present on a component. Geometrical tolerances may be applied to these features. For example, an axis may have a straightness of positional tolerance, a face may have a flatness tolerance and a cylindrical surface may have a roundness tolerance.

3.2 Combinations of single features. The features shown in figure 2 are straight lines and surfaces that are plane, cylindrical and spherical. These single features may be combined to form other features such as slots, grooves and tongues. Thus, a tongue consists of a pair of parallel plane surfaces with another plane surface at right angles to them, and a median plane. Slots, grooves and tongues may need tolerances of position or symmetry.

4. (2.1) Characteristics of a feature and their symbols for use on drawings

The straightness of an axis, the flatness of a face and so on are characteristics of features. These characteristics are denoted on a drawing by symbols, as shown in table 1 on page 4. The table also shows that the characteristics fall into two main categories, those for single features and those for related features, and into subgroups for form, attitude, location and composite tolerances.

5. Additional symbols for use on drawings

The symbols for a boxed (theoretically exact) dimension, a circular or cylindrical tolerance zone, a datum feature and maximum material condition are shown in table 2 on page 4.

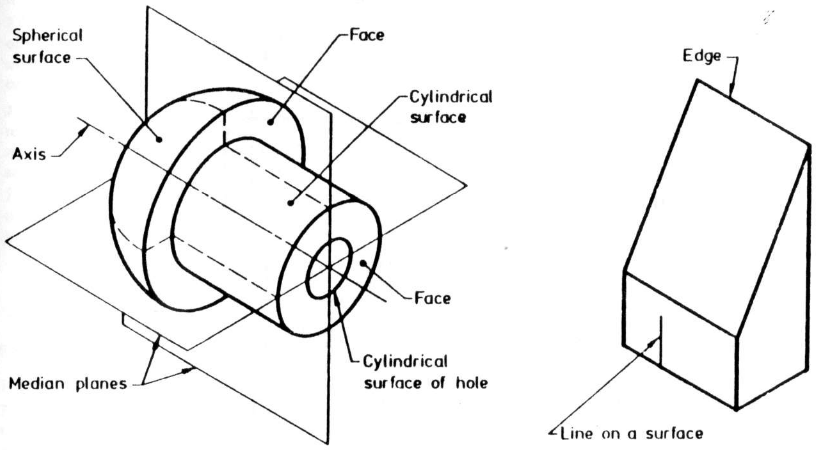


Figure 2. Single features of a component