Section 9: Dimensioning features recognition and interpretation

PURPOSE
This section aims to enable the student to extend their knowledge of Drawing Interpretation from Engineering Drawings produced to AS1100 standard.

Objectives

At the end of this section you should be able to:

- ☐ Interpret information on detail drawings of engineering components.
- ☐ Interpret information on detail drawings of engineering assemblies.

Completion guidance

Students should attempt this section if the material relates to the engineering discipline they are employed or intend to be employed in.

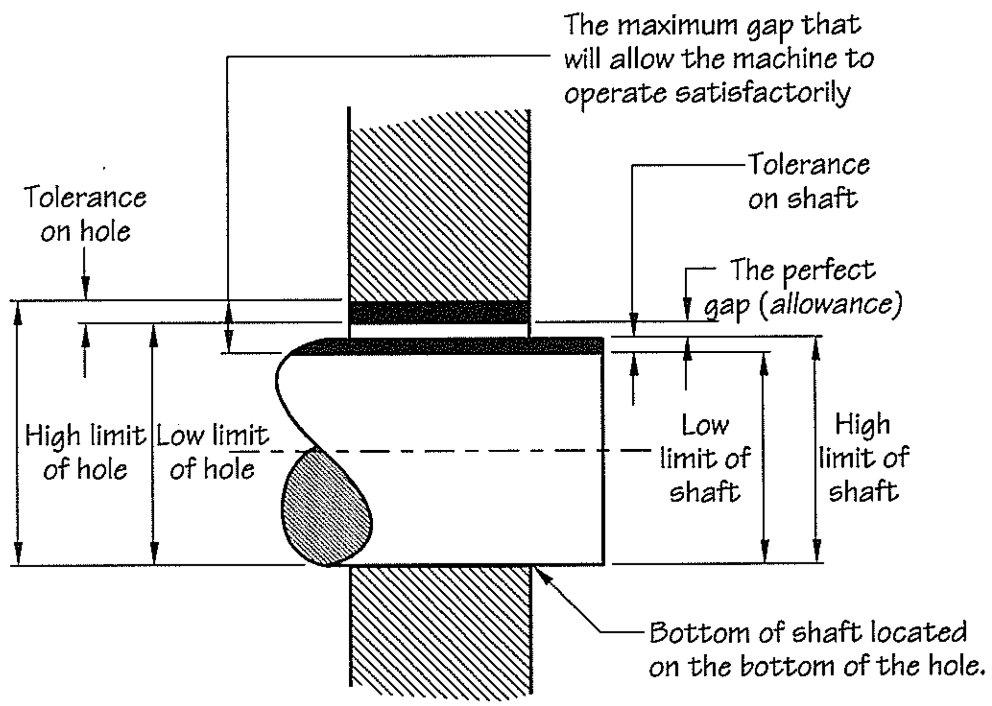
Dimensioning features recognition and interpretation

It would be ideal if industries could manufacture every component to exactly the designed size, that is, if a designer has worked out that the diameter of a shaft has to be 30mm in order to transmit the power required and every shaft could be machined to 30.00mm diameter and every bearing to 30.08mm diameter.

There are a number of reasons why this is virtually impossible and uneconomical to do. One is cost; the more accurate you try to make something the more careful you have to be , hence it takes more time and is costly. Another reason is the fact that machine tools such as lathes, milling machines, grinding machines etc. all have cutters or grinding wheels that are subject to wear, hence the size or the components will change as the cutters wear.

As a result of the above, along with expansion and contraction due to temperature changes, and many others, a designer has to allow for a range in size of both parts that fit together. That is, the size of a shaft will have an upper and lower limit of size and the bearing will have an upper and lower limit of sizes as well, giving a variation in quality of fit on assembly. See the diagram at the right.

The difference between the upper and lower limit of size for a component is called the tolerance, defined as *'the amount of variation permitted in the size of a component'.*



Unilateral tolerancing
(Tolerance in one direction)

Tolerances are in one direction only from the design size. (Design size 65mm)

$$65.0^{+0.5}_{-0}$$

By calculation
Upper limit of size is 65.5mm
Lower limit of size is 65.0mm

any size in this range is acceptable

Bilateral tolerancing
(Tolerance in two directions)

Tolerances are in both directions only from the design size. (Design size 73mm)

$$73.0^{+0.4}_{-0.3}$$

By calculation
Upper limit of size is 73.4mm
Lower limit of size is 72.7mm

any size in this range is acceptable

Tolerancing

You have already learnt about tolerancing in relation to a dimensioning technique. This section tells you how to use tolerances in manufacturing.

When we manufacture parts, it is impossible (as stated before) to achieve dimensional perfection because of:

- Human error
- Machine error
- Temperature variations
- Production costs
- Time

To allow for these things, we include a manufacturing tolerance in dimensions. A tolerance is the permissible variation of a dimension which will still allow us to make the article.

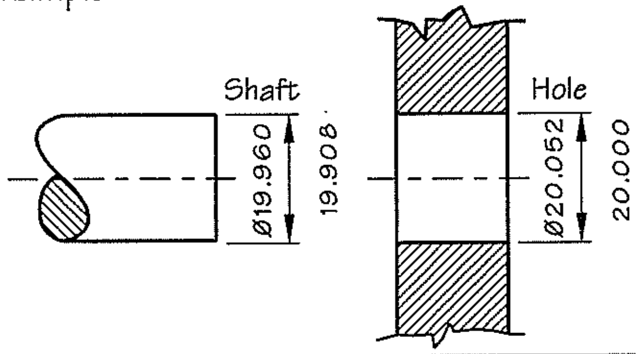
A broad generous tolerance is cheaper and easier to produce than a narrow, precise one. The smaller the tolerance, the more expensive the article is to make.

Definitions

Nominal size	An approximate size in round figures. We can use nominal sizes when we make general statements, and we don't have to mention tolerance.
Basic size	Is the size about which the manufacturing limits are fixed. (Often the basic size is the same as the nominal size.)
Actual size	Is the true, measured size of a manufactured part.
Limits of size	Are the biggest and smallest sizes which are acceptable for a given part. The maximum acceptable size is known as the upper limit. The minimum acceptable size is known as the lower limit.
Tolerance	Is the difference between the upper and lower limits of size.

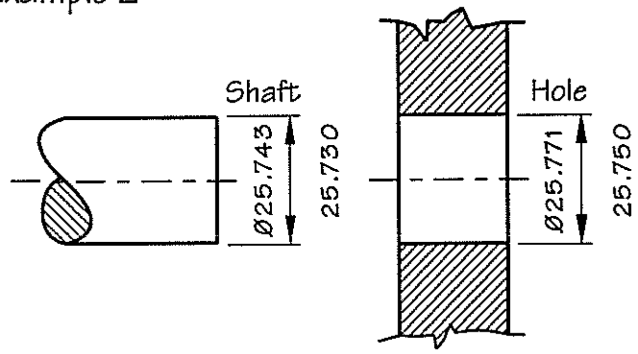
Exercise 9-1

Example 1



	Shaft	Hole
Nominal size -	20	20
Basic size -	20	20
Upper limit -	19.960	20.052
Lower limit -	19.908	20.00
Tolerance -	19.960 - 19.908 .052	20.052 20.000 .052

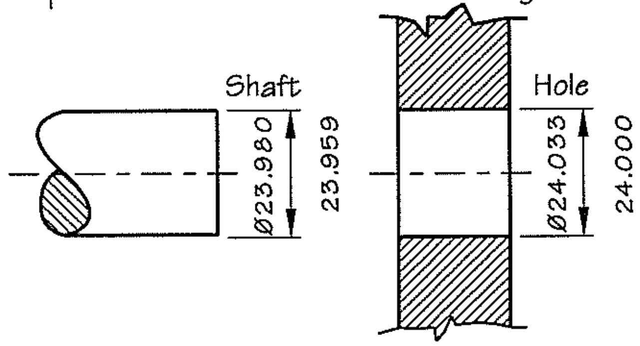
Example 2



	Shaft	Hole
Nominal size -	26	26
Basic size -	25.75	25.75
Upper limit -	25.743	25.771
Lower limit -	25.730	25.750
Tolerance -	25.743 - 25.730 .013	25.771 25.750 .021

Exercise 1

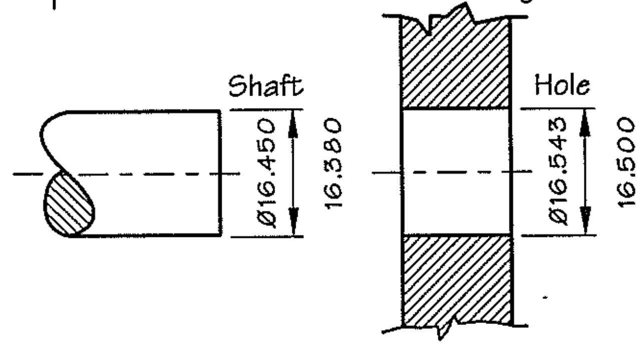
Complete the chart from the limits given



	Shaft	Hole
Nominal size -	24	24
Basic size -		
Upper limit -		
Lower limit -		
Tolerance -		

Exercise 2

Complete the chart from the limits given

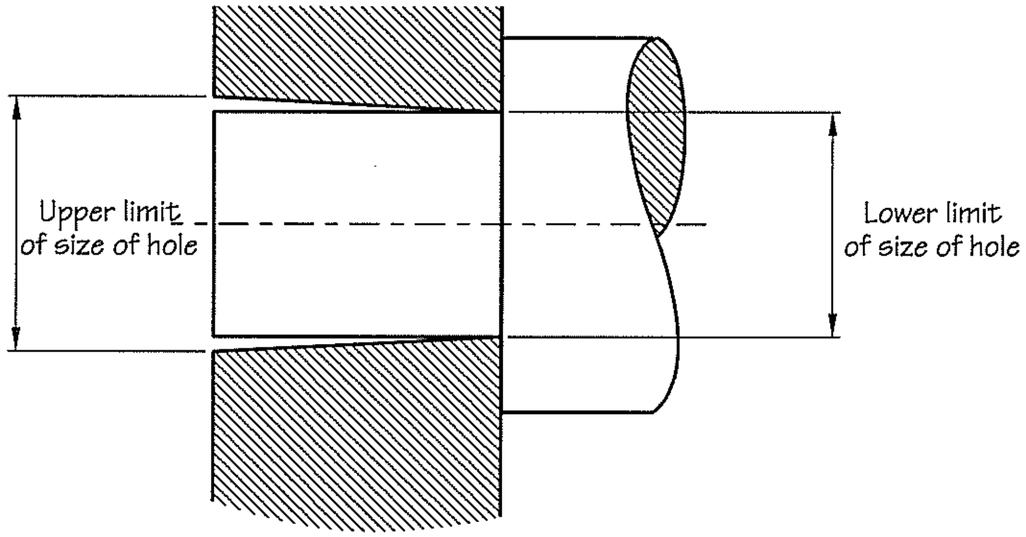


	Shaft	Hole
Nominal size -	16.5	16.5
Basic size -		
Upper limit -		
Lower limit -		
Tolerance -		

Geometric tolerances or geometry tolerances. (an introduction)

The overall performance of a mechanism is not only dependent on the quality of fit between mating parts. Very small tolerances play a significant role, however, if the surfaces in contact are relatively rough, rapid wear will usually result. Hence, if small tolerances are specified they must go hand in hand with specifying quality surface finish as well.

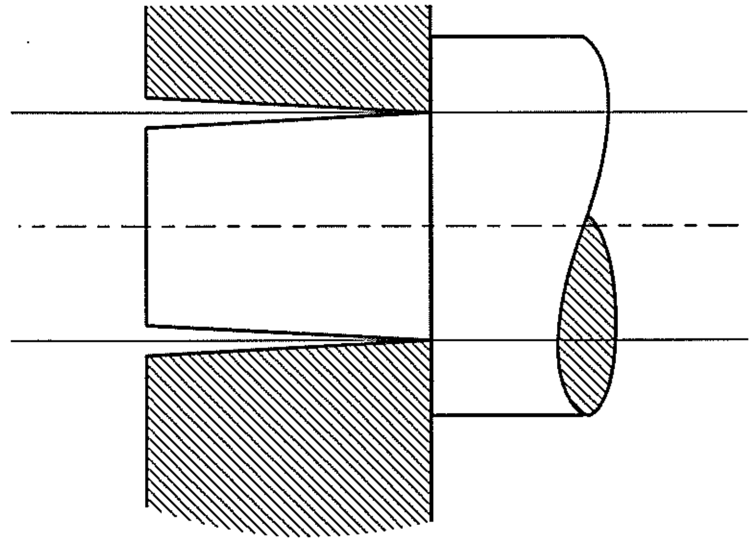
The tolerancing that has just been discussed, is called General Tolerancing. It looks good on paper because straight lines are drawn parallel to each other with drawing instruments. In actual practice, a shaft with say a 0.05mm tolerance may taper from one end to the other. Everywhere it is measured it is within tolerance, but tapered.



The above hole is within tolerance with regard to General Tolerancing, however, it is not parallel and the shaft and bearing would wear rapidly at the right hand end because it is touching at one point only and not all the way along.

Geometric (geometry) tolerancing is a way of specifying or saying that as well as being within its general tolerance, the hole or shaft must be parallel as well. This means, in the case of the bearing and shaft shown above, that the bearing and shaft will be in contact for the whole length of the bearing and will wear properly.

The illustration below takes the example to the left to the extreme possible unsatisfactory situation. Both parts, shaft and bearing, are tapered to the limit of their tolerance zone, but in opposite directions.



You will recognise when a geometric tolerance applies to a feature by a multi-part box, which may consist of two, three or four or more sections as shown below.

The boxes give the relation ship to another feature in the first box, which may be parallel to, at right angles to, concentric to etc., the degree of accuracy in the second box, and the feature it relates to in the third box.