LIMITS, TOLERANCES AND FITS

9.1. GENERAL

With all the advancement in the machine tool technology, it is not possible to achieve dimensional perfection due to the following reasons : Temperature changes, tool wear, deflections and vibrations of the machine and the work and human error. Even if the dimension is to be maintained within a very close degree of accuracy, lot of time will be consumed resulting in increased cost of manufacture, Fig. 9.1, and even then there would still be small variation. Other drawbacks of manufacturing a component exactly to a a nominal size are:

(*i*) Components will have to be produced individually which is slow and costly.

(*ii*) Highly intensive skilled labour is required, which is also costly.

(*iii*) Automation and its accompanying benefits can not be realised.

(*iv*) Replacement points have to be made the same way and so hold up other processes.

(*v*) Some fits will be better than other, and the variation will be unknown.

In mass production where the work has to be done in a set competitive time, greater variations will result. This fact is recognised and certain variations are allowed in the sizes of the machine elements or parts. This system of manufacture in which the dimensions of a part lie within some specified limits leads to "Interchangeable manufacture". Interchangeable part manufacture is a major feature of modern serial and mass production. The term "interchangeable manufacture" implies that the parts which go into assembly may be selected at random from large number of parts. These parts which have been produced with all their dimensions within their specified limits, need not be made in the same shop or even in the same company. This system of interchangeable part manufacture, that is, the system of limits and fits results in the following advantages.

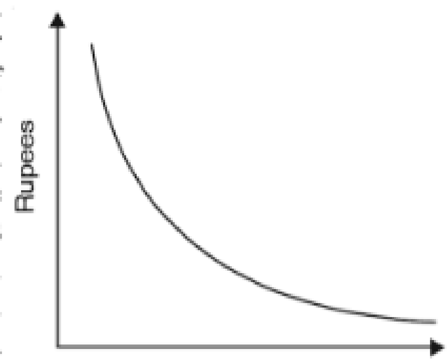


Fig. 9.1. Cost of Production vs Degree of Accuracy.

1. The components will assemble together at random.
During assembly, individual fitting is not necessary. This results in reduction in cost of production because the elimination of fitting reduces the time required to build the product.

2. Components can be manufactured in large batches or lots and all treated alike.

3. Machine tools which have been developed for quantity production enable the components to be manufactured more rapidly using cheaper labour.

4. Repair of existing machines or products is simplified because component parts can be easily replaced.

5. Parts can be checked by gauging which considerably simplifies inspection procedure.

9.2. TERMINOLOGY FOR LIMITS AND FITS

1. The type of fit which is required between the mating parts.

2. The tolerance which is to be allowed upon each dimension.

Types of fits will be discussed in the next section. Below, we give terminology for limits and fits.

Shaft. The term shaft refers not only to the diameter of a circular shaft but also to any external dimension on a component.

Hole. The term hole refers not only to the diameter of a circular hole but also to any internal dimension on a component.

When an assembly is made of two parts, one is known as male (enveloped) surface and the other one as female (enveloping) surface. The male surface is referred to as 'Shaft' and the female surface as 'Hole'

Basic Size. Basic size or Nominal size is the standard size for the part and is the same both for the hole and its shaft. This is the size which is obtained by calculation for strength.

Actual Size. Actual size is the dimension as measured on a manufactured part. As already noted the actual size will never be equal to the basic size and it is sufficient if it is within predetermined limits.

Limits of Size. These are the maximum and minimum permissible sizes of the part.

Maximum Limit. Maximum limit or High limit is the maximum size permitted for the part.

Minimum Limits. Minimum limit or Low limit is the minimum size permitted for the part.

Tolerance. Tolerance is the difference between maximum limit of size and minimum limit of size.

Allowance. Allowance is an intentional difference between the maximum material limits of mating parts. For shaft, the maximum material limit will be its high limit and for hole, it will be its low limit.

If the shaft is smaller than hole, the allowance is positive, but if the shaft is larger than the hole, it is negative.

The above definitions are explained in Fig. 9.2.

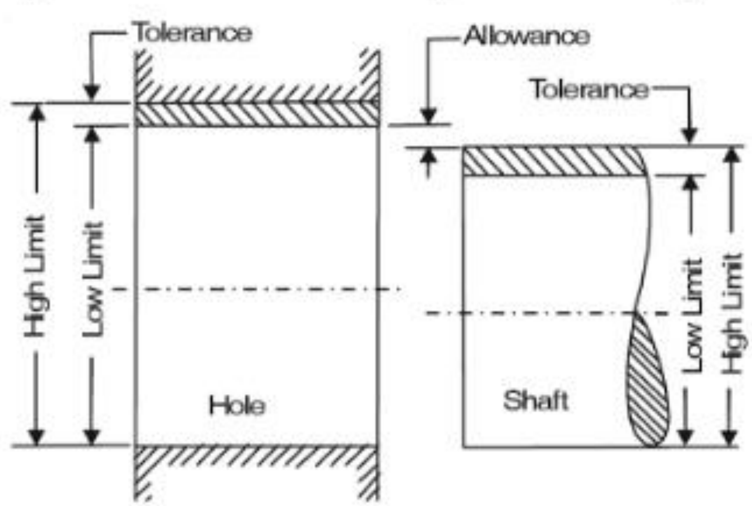


Fig. 9.2. Limits and Tolerance.

Deviation. It is the algebraic difference between a size (actual, maximum, etc.) and the corresponding basic size.

Actual Deviation. It is the algebraic difference between an actual size and the corresponding basic size.

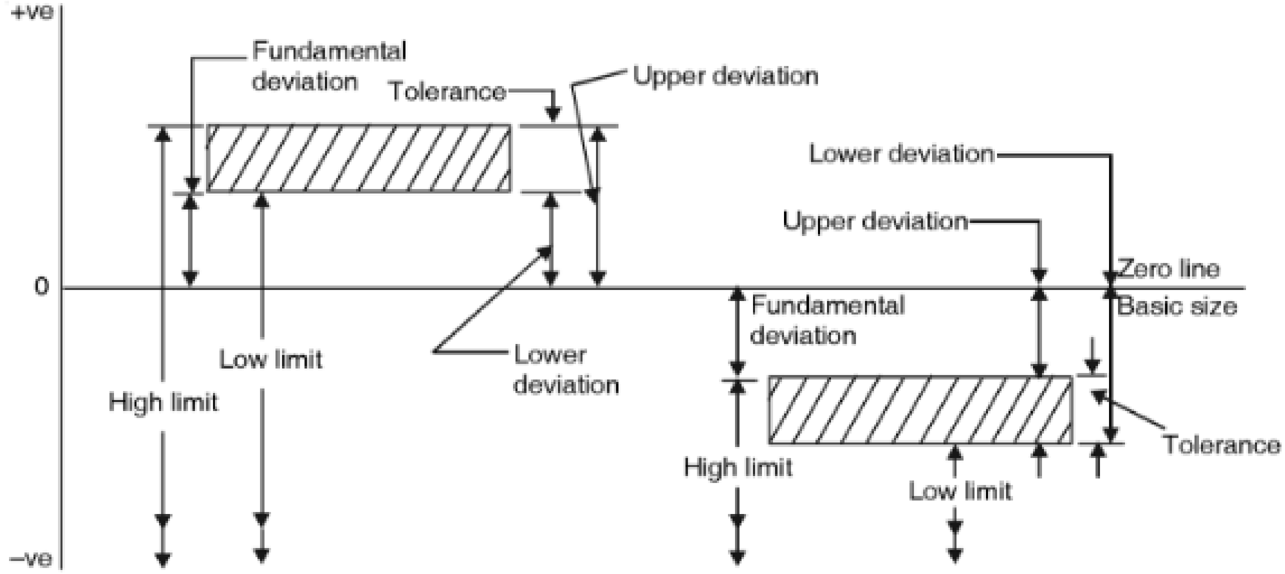


Fig. 9.3. Deviations.

Upper Deviation. It is the algebraic difference between the maximum limit of size and the corresponding basic size. It is a positive quantity when the maximum limit of size is greater than the basic size and a negative quantity when the maximum limit of size is less than the basic size.

Lower Deviation. It is the algebraic difference between the minimum limit of size and the corresponding basic size. It is a positive quantity when the minimum limit of size is greater than the basic size and a negative quantity when the minimum limit of size is less than the basic size.

Mean Deviation: - It is the arithmatic mean deviation between the upper deviation and Lower deviation.

Zero Line. It is a straight line to which the deviations are referred to in a graphical presentation of limits and fits. It is a line of zero deviation and represents the basic size. When the zero line is drawn horizontally, positive deviations are shown above and the negative deviations below this line.

Fundamental Deviation. This is the deviation, either the upper or the lower deviation, which is the nearest one to the zero line for either a hole or a shaft. It fixes the position of the tolerance zone in relation to the zero line.

Tolerance Zone. It is the zone bounded by the two limits of size of a part in the graphical presentation of tolerance. It is defined by its magnitude and by its position in relation to the zero line.

These terms are explained in Fig. 9.3.

Unilateral Limits. In the method of presenting the limits, both the limits of size are on the same side of the zero line. That is, the permitted tolerance is stated or indicated as wholly + ve or wholly –ve, e.g., $30\text{ mm}\,{}^{+0.13}_{+0.00}$ or $100\text{ mm}\,{}^{-0.12}_{-0.26}$. One of the limits of the size may be the basic size.

Bilateral Limits. Here, one of the limits of size is on one side of the zero line and the other limit of size is on the other side of the zero line, *i.e.*, the permitted tolerance is indicated partly + ve and partly negative, *e.g.*, $90\text{ mm}\,{}^{+0.010}_{-0.025}$.

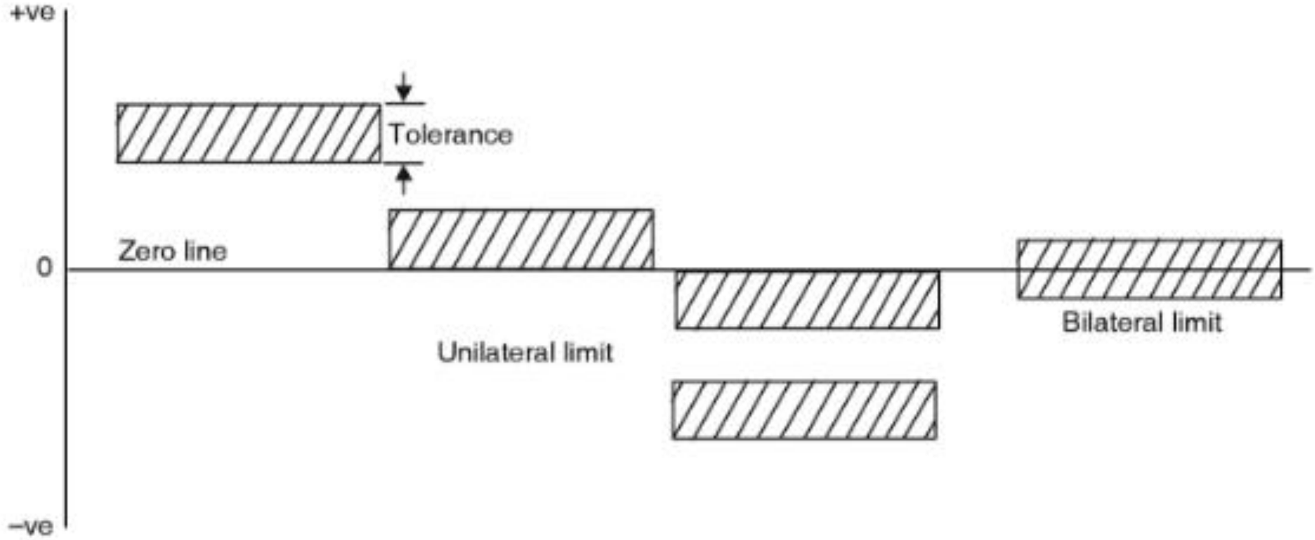


Fig. 9.4. Unilateral and Bilateral Limits.

9.2.1. Fits. The fit between two mating parts is the relationship existing between them with respect to the amount of play or interference which is present when they are assembled together. According the fit may result either in movable joint or a permanent joint. For example, a shaft running in a bush can move in relation to it and so forms a moving joint, whereas a pulley mounted on the shaft forms a fixed joint. The nature of joint or "fit" is characterised by the presence and size of 'clearance' (for movable joints) or 'interference' (for fixed joints). There are three basic types of fits.

1. **Clearance Fit.** In clearance fit or running fit, the shaft is always smaller than hole. A positive allowance exists between the largest possible shaft and the smallest possible hole, *i.e.*, when the shaft and hole are at their maximum metal conditions. The tolerance zone of the hole is entirely above that of the shaft.

Minimum Clearance. It is the difference between the maximum size of shaft and minimum size of hole.

Maximum Clearance. It is the difference between the minimum size of shaft and maximum size of hole.

2. **Interference, Press or Force Fit.** In this type of fit, the shaft is always larger than the hole. The tolerance zone of the shaft is entirely above that of the hole.

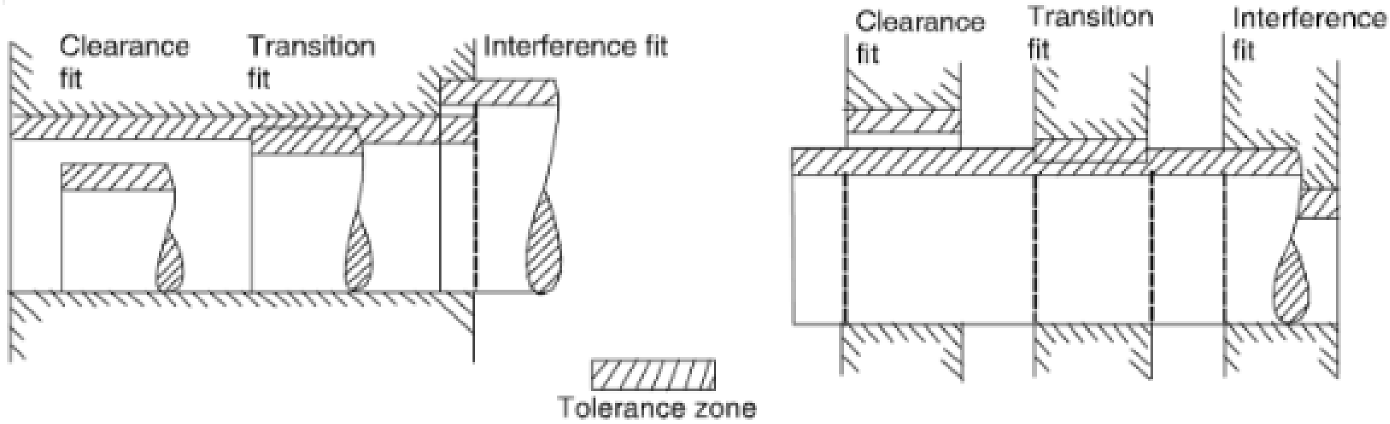
Minimum Interference. It is the difference between the maximum size of hole and the minimum size of shaft prior to assembly.

Maximum Interference. It is the difference between the minimum size of hole and the maximum size of shaft prior to assembly.

3. **Transition or Sliding Fit.** It occurs when the resulting fit due to the variations in size of male and female components due to their tolerance, varies between clearance and interference fits. The tolerance zones of shaft and hole overlap.

The various allowances for different fits may be obtained in two ways.

Hole Basis System. In this system, the hole is kept constant and the shaft diameter is varied to give the various types of Fits, Fig. 9.5. The basic size of the hole is taken as the low limit of size of the hole, i.e., the Maximum Metal Condition (MMC), of the hole. The high limit of size of the hole and the two limits of size for the shaft are then selected to give the desired fit.



(*a*) Hole basis system (*b*) Shaft basis system

Fig. 9.5. Fits.

It is clear, therefore, that in this system, the actual size of a hole that is within the tolerance limits is always more than the basic size; it can equal the basic size as a particular case but can never be less. In the `Basic Hole System', the holes get the letter `H' and shafts get different letters to decide the position of tolerance zone to obtain a desired fit.

Shaft Basis System. Here, the shaft is kept constant and the size of hole is varied to give the various fits, Fig. 9.5. The basic size of the shaft is taken as one of the limits of size (maximum limits) for the shaft, i.e. its Maximum Metal Condition (MMC). The other shaft limit of size and the two limits of size for the hole are then selected to give the desired fit.

It is clear, therefore, that in this system, the actual size of a shaft that is within the tolerance limits is always less than the basic size. As a particular case, it can equal the basic size but can never be larger. In the `Basic Shaft System', the shaft gets the letter `*h*' and holes get different letters to decide the position of tolerance zone to obtain a desired fit.

From a manufacturing point of view, it is preferable to use the "hole basis"system, because it is economical. This is because a great many holes are produced by standard fixed size tools, such as, twist drills, reamers, core drills, taps, broaches, etc. The advantages of using fixed size tools is that the machine need not be set up to obtain the proper size of the hole, setting up operations can consequently be made quicker and cheaper. Subsequently, the shaft sizes are more readily variable about the nominal size by means of turning or grinding operation. It is easier and more convenient to manufacture shafts of varying sizes than holes of varying sizes, as given above. The hole basis system is preferred, because it lessons the range of cutting and measuring tools for machining of holes, which are more expensive than tools to machine shafts. Also, the control of the size and shape of holes is more complicated and less accurate than the control of shafts. Applications : Machine and engine building, locomotive, construction.

The shaft basis system is more advantageous in certain cases, for example, this system can be efficiently applied for long shafts machined to the same size over their full lengths (smooth drawn shafts, shafts ground on centreless grinding machines etc.), if the shaft is to mate with at least two parts having holes that require different types of fit. Examples of "shaft basis" system are : the mating of a piston pin with both the piston and the connecting rod, and the outer rings of antifriction bearings with various bores in housings, electric motors, power transmission and products made from bright drawn bars.

It has been found in practice that a number of different fits of each basic type of fit are required which can provide different degrees of tightness or freedom between the mating parts.

The most commonly used fits of clearance type are : (1) Slide fit (2) Easy slide (3) running fit (4) slack running fit and (5) loose running fit.

1. **Slide fit.** Slide fits have a very small clearance, the minimum clearance being zero. Due to this, a sliding fit is close to the group of transition fits. They are employed when the mating parts move slowly relative to each other (for example, the tail stock spindle in a lathe, the feed movement of the spindle quill in a drilling machine etc.), or for mounting purposes (for example, stopper rings, keyed change gears etc.)

2. **Easy slide fit.** An easy slide fit provides for a small guaranteed clearance. It serves to ensure alignment between the shaft and hole and is supplied for slow and non-regular motion, for example, spindle of lathes and dividing heads, piston and slide valves, spigot or location fits.

3. **Running fits.** Running fits have appreciable clearance. It is employed in engineering for rotation at moderate speeds (shafts, gears, pulleys, couplings, crank shafts in their main bearings, throttles in the valve sleeves of a steam and air power forging hammer, bearings of small electric motors and pumps etc., gear box bearings). The clearance provides sufficient space for lubrication between mating friction surfaces.

4. **Slack running fits.** A slack running fit has considerable clearance which may be required as compensation for mounting errors, as in multi-support shafts (Cam shafts of I.C. engines) or shafts with widely spaced supports or if the bearings are very long (shafts of centrifugal pumps, shafts in the drives of cylindrical grinding machines etc.)

5. **Loose running fits.** These fits have the largest clearance and are employed for rotation at very high speeds and if misalignment of the mating parts may occur in assembly (shafts in specially long bearings (Plummer blocks), idle pulleys on their shafts.

Interference fits (shrink, heavy drive and light drive) are used for fixed permanent joints in which no additional fixing elements are needed. Elastic strains developed on the mating surfaces during the process of assembly prevent relative movement of the mating parts. Example of interference fits are : steel tyres on railway car wheels, gears on the intermediate shafts of trucks, bushing in the gear of a lathe head stock, pump impeller on shaft, drill bush in jig plate and cylinder linear in block etc.

Transition fits lie midway between clearance and interference fits. The use of transition fits does not guarantee either interference or clearances. Their main use is to ensure proper location of mating parts that are to be repeatedly disassembled and reassembled. In transition fits, torque is fitted between mating parts by means of additional fastening elements, such as keys, cotters, pins etc., which prevent relative motion between mating parts. The same purpose is served by spline shafts and bushings. The selection of a transition fit (force, tight, wringing or push) usually depends on how often the given joint is to be disassembled and reassembled during regular operation.

1. **Force fit.** Force fits are employed for mating parts that are not to be disassembled during their total service life, for example, gears on the shafts of a concrete mixer, forging machine etc.

2. **Tight fit.** Tight fits provide less interference than force fits. These fits are employed for mating parts that may be replaced while overhauling the machine, for example, stepped pulleys on the drive shafts of a conveyor, cylindrical grinding machine etc.

3. **Wringing fit.** A wringing fit provides, either, zero interference or a clearance. These find use where parts can be replaced without difficulty during minor repairs, for example, gears of machine tools.

4. **Push fit.** This fit is characterised by a clearance. It is employed for parts that must be disassembled during operation of a machine, for example, change gears slip bushing etc.

9.3. MEANING OF LIMITS

In the case of limits on hole, more is meant than simply the maximum and minimum size of the hole. The error of the hole may be as shown in Fig. 9.6 and yet a simple measurement of diameter would not indicate the lack of axial truth. The correct interpretation of a pair of limits can thus be stated as follows :

1. **Hole.** The upper limit refers to the greatest diameter at any point in hole, the lower limit refers to the diameter of the inscribed circle or cylinder which will just pass through the hole.

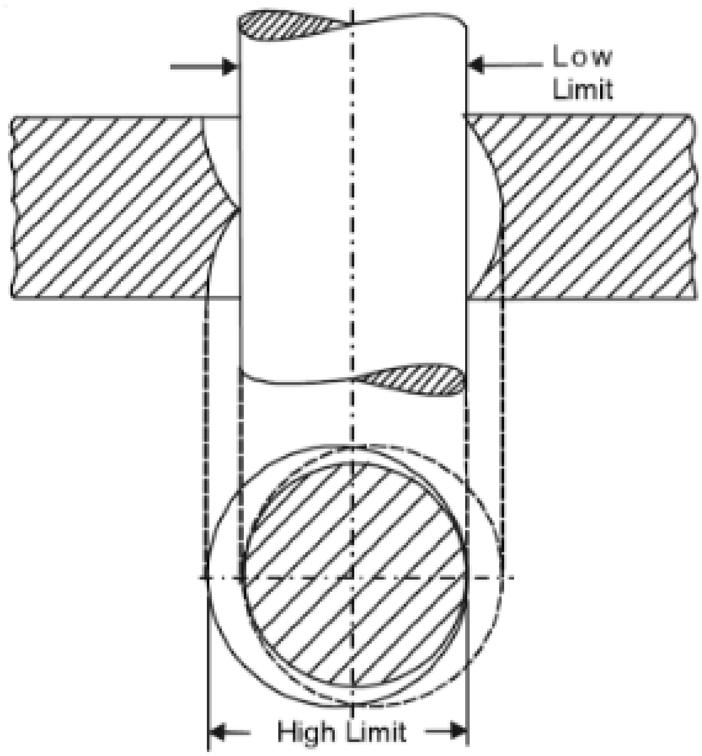


Fig. 9.6. Limits on Hole.

2. **Shaft.** The upper limit refers to the escribed ring which will just pass over the shaft, the lower limit refers to the minimum diameter at any point on the shaft, Fig. 9.7.

Although, normally limits refer to the extreme dimensions which may occur on a component, a more recent outlook on the matter is that the limits refer to the dimensions inside which all but a small percentage of parts must lie. Thus probability effect is taken into account. However, the definition given above still holds good.

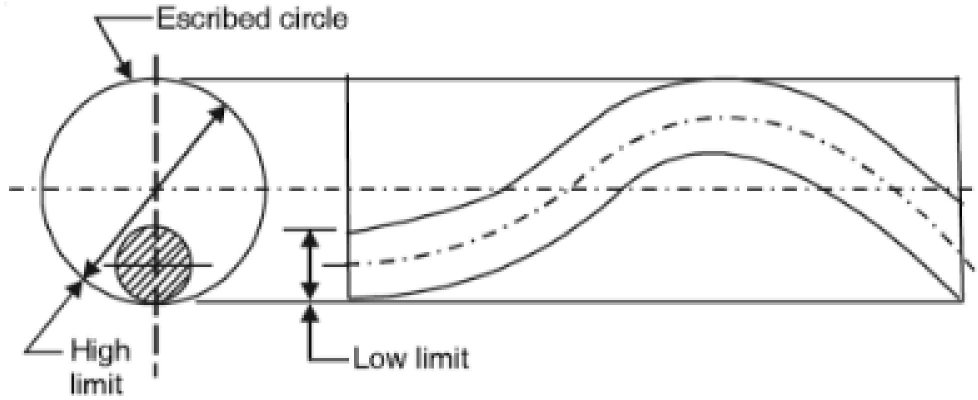


Fig. 9.7. Limits on Shaft.

The most important function of limits, is however, to enable interchangeable components to be produced. General limits are required to control general aspects of the specifications. Special limits are necessary with interchangeable manufacture.

(*i*) to control the fits between mating parts.

(*ii*) to maintain the desired clearance.

9.4. GENERAL LIMITS OF TOLERANCE

As mentioned before, tolerances are necessary to attain manufacturing control. The amount of tolerance to be given on a part depends upon :

(*i*) the function of the product, *i.e.*, the allowance desired in the fit.

(*ii*) manufacturing process available.

(*iii*) cost of production and assembly.

Cost of production and assembly increases as the tolerance is decreased. This increase in cost is due to :

(*a*) need for more sophisticated machinery.

(*b*) higher skills required.

(*c*) increased attention to inspection and handling.

The variation of relative cost of production as a function of tolerances will be similar to as shown in Fig. 9.1.

Therefore, tolerance that should be assigned to dimension is that which gives an economic balance between cost and quality. No part should be made with a greater of accuracy than is required by its use in a given mechanism or machine.

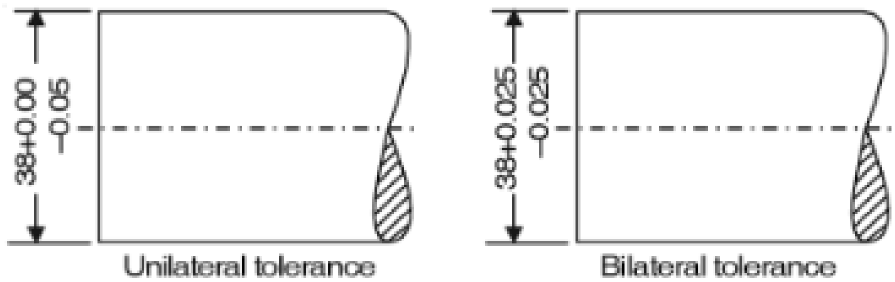
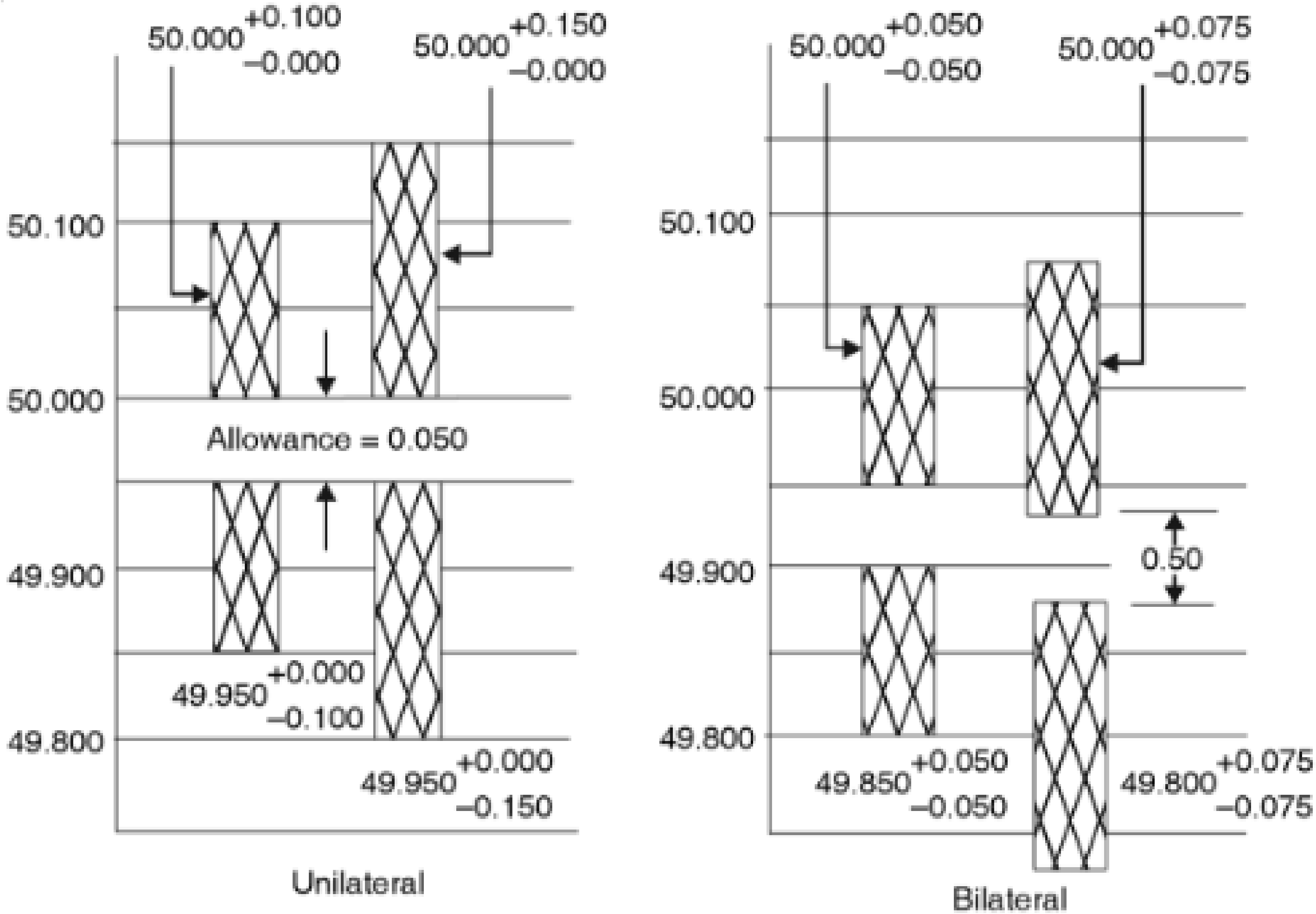


Fig. 9.8. Method of assigning tolerances

As already mentioned under article 9.2, manufacturing tolerance may be assigned to a dimension in two different manners :

1. Unilateral system
2. Bilateral system.

For unilateral tolerance, it is more critical for a certain dimension to deviate in one direction that in another. This system is also more satisfactorily and realistically applied to certain machining processes where it is common knowledge that dimensions will most likely deviate in one direction. For example, in drilling a hole with a standard size drill, the drill will most likely produce an oversize rather than an undersize hole. The operator machines to the lower limit of the hole (upper limit of a shaft), knowing fully well that he still has the whole tolerance left for machining before the parts are rejected. Also in this system the tolerance can be revised without affecting the allowance or clearance conditions between mating parts, *i.e.*, without altering the type of fit, Fig. 9.9. This system is most commonly used in industrial application. For interchangeable parts, unilateral system should always be used.



50.000 +0.100 −0.000
50.000 +0.150 −0.000
50.100
50.000
Allowance = 0.050
49.900
49.950 +0.000 −0.100
49.800
49.950 +0.000 −0.150
Unilateral
50.000 +0.050 −0.050
50.000 +0.075 −0.075
50.100
50.000
0.50
49.900
49.800
49.850 +0.050 −0.050
49.800 +0.075 −0.075
Bilateral

Fig. 9.9

GAUGES AND GAUGE DESIGN

10.1. INTRODUCTION

There are several methods available for the control of dimensions of components in a system of limits and fits. Each component, for example, be measured with an instrument giving a suitable accuracy and this method is often adopted, particularly for closely limited work. The method used for the majority of the work in quantity production is the system of limit gauging. This has the advantages that it can be operated in many cases by quite unskilled persons.

Gauges are inspection tools of rigid design, without a scale, which serve to check the dimensions of manufactured parts. Gauges do not indicate the actual value of the inspected dimensions of the component. They are only used for determining whether the inspected part has been made within the specified limits. A workman checking a component with a gauge does not have to make any calculations or to determine the actual dimensions of the part. Gauges are easy to employ. This is one reason for their wide application in engineering. Gauges differ from measuring instruments in the following respects :

(*i*) no adjustment is necessary in their use.

(*ii*) they usually are not general-purpose instruments but are specially made for some particular part, which is to be produced in sufficiently large quantities.

Gauging is used in preference to measuring when quantities are sufficiently high, because it is faster and easier with resulting lower costs.

10.2. PLAIN GAUGES

Plain gauges are used in checking plain, that is, unthreaded holes and shafts.

10.2.1. Classification of Plain Gauges. Plain gauges are classified in the following ways.

1. According to type.
2. According to purpose.
3. According to form of tested surface, and
4. According to design.

1. **According to type.** (*a*) Standard Gauges (*b*) Limit Gauges.

(*a*) **Standard Gauges.** Every gauge is almost a copy of the part example, a bushing is to be made which is to mate with a shaft. In this case, shaft is the mating part. The bushing is checked by a gauge which in so far as the form of its surface and its size is concerned is a copy of the mating part, that is, the shaft.

If a guage is made as an exact copy of the opposed (mating) part, in so far as the dimensions to be checked are concerned, it is called a 'standard gauge'. The first gauges to be developed were the standard gauges. The first standard gauges were the opposed (mating) parts themselves. When a component is assembled with its mating part, a (mating) part itself. However, such individual fittings are not convenient or even possible in mass production conditions. Moreover, the two parts to be assembled might be in production in two different shops or even at two different plants. Therefore, it is more proper to use, as a checking tool, not the mating part, but its exact copy as far as the tested dimension is concerned.

Such a standard gauge has two drawbacks :

(*i*) The quality of the manufactured part will depend upon the freedom with which it mates with the standard gauge. The judgement of this freedom is a relative thing and it usually creates misunderstanding between the purchaser and the manufacture.

(*ii*) A standard gauge cannot be used to check an interference fit. For example, if a bushing (Fig. 10.1) of 50 mm diameter is to be made for assembly with a shaft of 50.1 mm diameter, then the standard gauge diameter will be 50.1 mm of opposed part. Such a gauge will not pass into a properly produced bushing of diameter 50 mm.

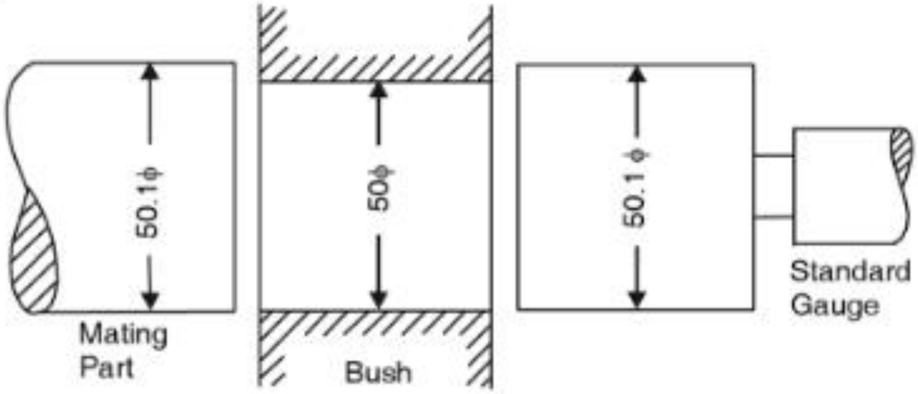


Fig. 10.1. Application of a Standard Gauge.

(*b*) **Limit Gauges.** The system of limit gauges is very widely used in industries. Limit gauges are made to the limits of the dimensions of the part to be tested. As there are two limits of the dimensions of a part, high and low, two gauges are needed to check each dimension of the part. The part is checked by successively assembling each of the gauges with it. Since the dimensions of a properly manufactured part must be within the prescribed limits, one of the gauges called a "Go Gauge" should pass through or over the part, while the other gauge called a "Not Go Gauge" should not pass through or over a part, Fig. 10.2. Gauges should pass through or over a part under their own weight and the part and the gauge must be at the same temperature.

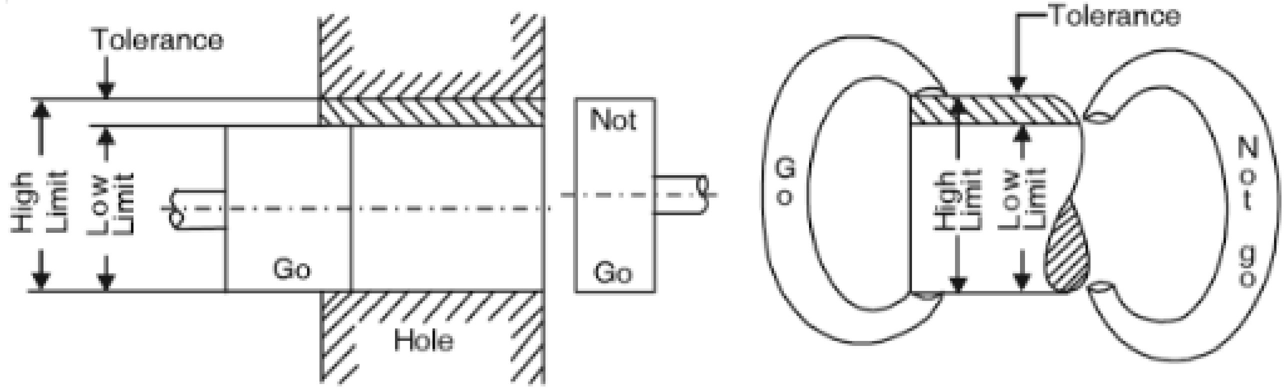


Fig. 10.2. Limit Gauges.

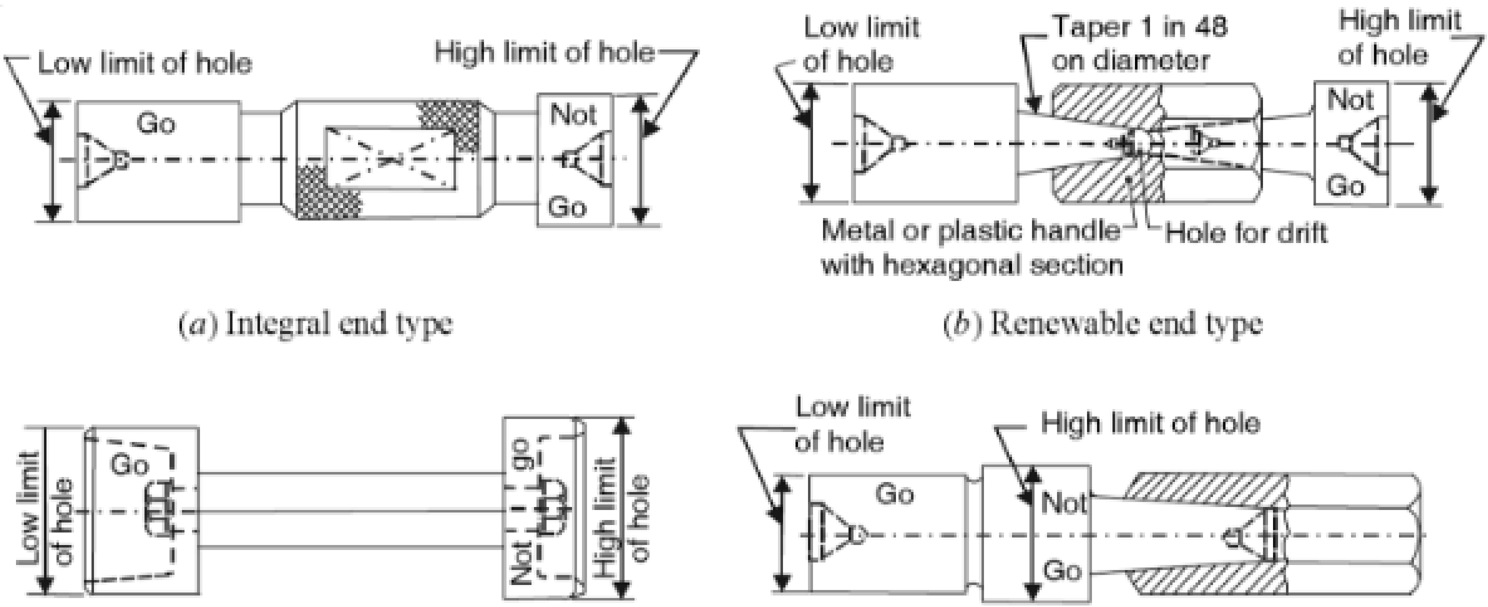
2. **According to Purpose.** According to purpose, the gauges may be classified as : (*a*) Workshop gauge or Working gauge (*b*) Inspection gauge (*c*) Purchase inspection gauge, and (*d*) Reference or Master gauge.

(*a*) **Workshop Gauge.** Workshop gauge or the manufacturing gauge is used by the machine operator to check the dimensions of the parts as they are being produced. These gauges usually have limits within those of the component being inspected. They are designed so as to keep the size of the part near the centre of the limit tolerance.

(*b*) **Inspection Gauge.** Inspection gauges are those used by inspectors in the final acceptance of manufactured parts when finished. These gauges are made to slightly larger tolerances than the workshop gauges so as to accept work slightly nearer the tolerance limit than the workshop gauges. This is to ensure that work which passes the working gauge will be accepted by the inspection gauge also.

(*c*) **Purchase Inspection Gauge.** The need of such gauges arises when the products of other plants are to be accepted. The purchaser must remember that the parts may have been made and checked by working gauges worn to the maximum permissible degree. Therefore the 'Go' side of the purchase inspection gauge must be designed accordingly. Thus, nominal size of "Go" purchase inspection gauge will be equal to the lower limit of the hole. 'No-Go' purchase inspection gauge design is similar to "No-Go" working gauge.

(*d*) **Reference or Master Gauges.** Reference or master gauges are used only for checking the size and condition of other gauges. Reference gauges are the reverse or opposite in form to working or inspection gauges. Due to the expenditure involved, reference gauges are seldom used and gauges are checked by universal measuring instruments optometers, comparators etc. or gauge blocks (for snap gauges).



(*a*) Integral end type

(*b*) Renewable end type

(*c*) Hollow end type

(*d*) Progressive type

Fig. 10.3. Plug Gauges.

3. **According to the form of tested surface.** According to the form of the tested surface, the gauges are of two types : Gauges for checking the holes and gauges for checking the shafts. Gauges for checking the holes are called "Plug Gauges" and those for checking the shafts are called "Snap or Gap gauges and Ring gauges".

(*a*) **Plug Gauge.** Plug gauges (Fig. 10.3) are used to check the holes. The `go' plug gauge is the size of the low limit of the hole while the `not go' plug gauge corresponds to the high limit of hole.

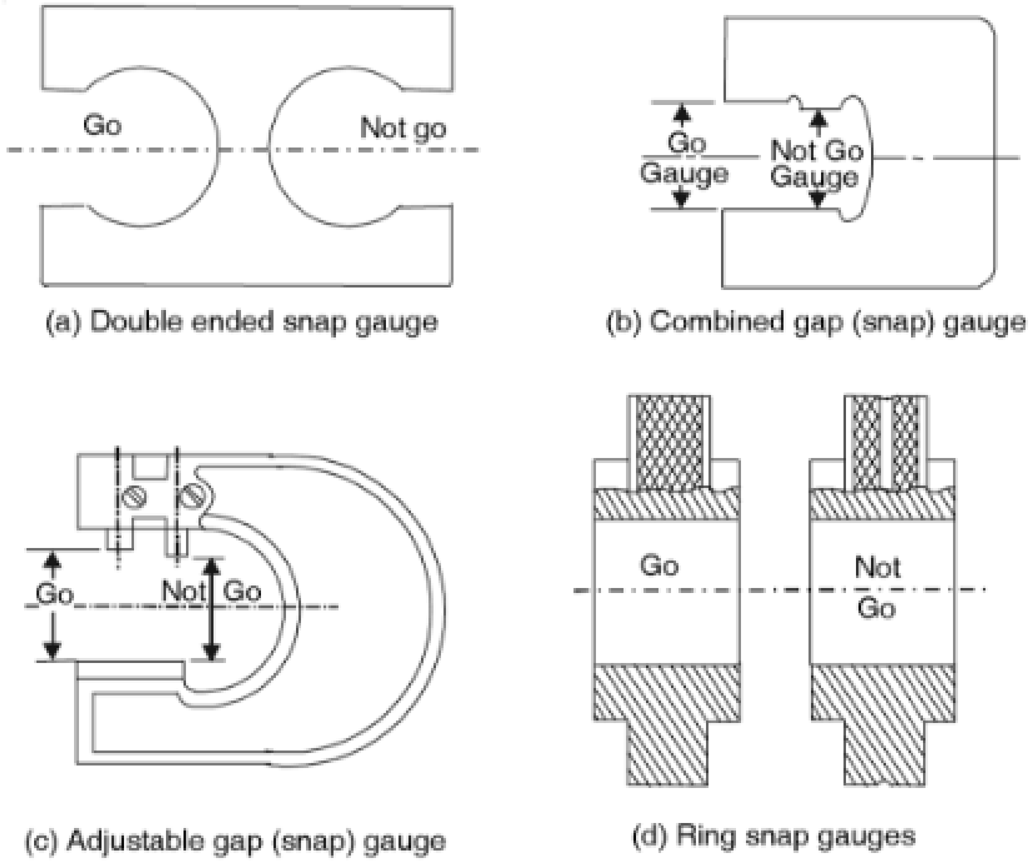
(*b*) **Snap, Gap or Ring Gauge.** These gauges are used for gauging the shafts and male components. The 'Go' snap gauge is of a size corresponding to the high (maximum) limit of the shaft, while the 'Not Go' gauge corresponds to the low (minimum limit). The various snap, gap and ring gauges are shown in Fig. 10.4. Gap gauges can also check lengths or widths flangs thickness etc.

4. **According to Design.** According to design, the gauges may be classified as :

(*i*)	(*a*)	Single limit	(*b*)	Double limit
(*ii*)	(*a*)	Single ended	(*b*)	Double ended
(*iii*)	(*a*)	Fixed	(*b*)	Adjustable
(*iv*)	(*a*)	Integral end	(*b*)	Renewable end
(*v*)	(*a*)	Solid end	(*b*)	Hollow end

(*i*) **Single limit and Double limit Gauge.** The gauge cannot be made exactly according to the dimensions it has to check.

The dimensions of 'Go' and 'Not Go' sides of the gauge will vary over a small range. Depending upon the manner of this variation, the gauge can be single or unilateral limit gauge and double or bilateral limit gauge. For example, if the low limit of hole is 6.25 cm and the high limit is 6.27 cm and the tolerance on gauge manufacture is 0.002 cm, then



(a) Double ended snap gauge

(b) Combined gap (snap) gauge

(c) Adjustable gap (snap) gauge

(d) Ring snap gauges

Fig. 10.4. Gauges of Shaft.

In Single limit (unilateral) system :

Size of 'Go' Plug gauge (For maximum metal condition of hole)

$$= 6.25 \begin{array}{l} +0.002 \\ -0.000 \end{array} \text{ cm}$$

Size of 'Not Go' Plug gauge $= 6.27 \begin{array}{l} +0.000 \\ -0.002 \end{array}$ cm

In Double limit (Bilateral) system

Size of 'Go' Plug gauge $= 6.25 \begin{array}{l} +0.001 \\ -0.001 \end{array}$ cm

Size of 'Not Go' Plug gauge $= 6.27 \begin{array}{l} +0.001 \\ -0.001 \end{array}$ cm

This has been made clear in Fig. 10.5.

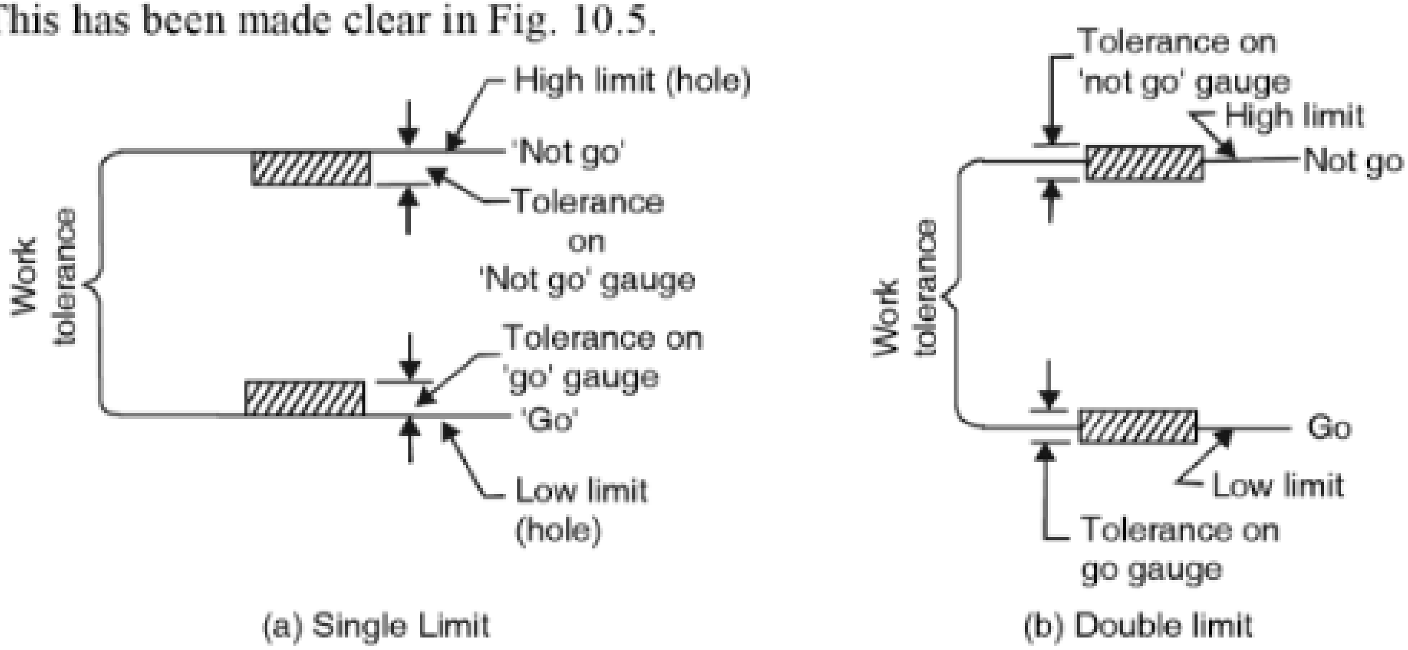


Fig. 10.5. Single Limit and Double Limit Gauge.

(*ii*) **Single end and double end gauge.** A double end limit gauge [Fig. 10.3 (*a*) to (*c*)], [Fig. 10.4 (*a*)] has the 'Go' side at one end and the 'Not Go' side at the other end of the gauge. Each end of the gauge is applied to the component so as to determine its acceptability. A single end gauge or progressive gauge [Fig. 10.3 (*d*)] and [Fig. 10.4 (*b*) and (*c*)] has both the 'Go' and 'Not Go' sides at the same end so that a part may be checked with one movement.

(*iii*) **Fixed and Adjustable Gauge.** In the case of fixed gauges, no change can be made in the size range whereas in adjustable gauge [Fig. 10.4 (*c*)], a small change can be made in the size range. Fixed gauges are usually less expensive initially, but they have the disadvantage of not permitting adjustment to compensate for wear. Adjustable gauges will be expensive but they can be used over a small range of different settings. They have forged steel frame into which anvils are fitted.

(*iv*) **Integral and Renewable End gauge.** In integral end type gauge [Fig. 10.3 (*a*), the 'Go' and 'Not Go' end of the gauge are integral with the gauge handle. In renewable end type gauge [Fig. 10.3 (*b*), (*c*)], the ends are detachable from the handle so that they be replaced separately when worn, and to economise in cost, as the ends are made of superior material as compared to that of the handle, which can be hoxagonal of plastic. It will reduce the weight and prevent the transfer of heat from the hands.

(*v*) **Solid end and Hollow end type gauge.** Plug gauges with diameter less than 63.5 mm are made with solid ends [Figs. 10.3 (*a*), (*b*) and (*d*)], but those with diameter larger than 63.5 mm are designed with hollow ends (Fig. 10.3 c) to limit excessive weight. Alternatively, the two ends may be attached to separate handles for ease of handling. This design can also be adopted for lighter plug gauges. But mostly, the 'Go' and 'Not Go' ends are combined in one unit as shown in Fig. 10.3. This makes the use of the gauge convenient and also it ensures that both ends of the gauge are kept together with no risk of misplacing one or the other.

10.3. DESIGN OF LIMIT GAUGES

The design of a limit gauge must ensure proper inspection of the part for which it is intended. The following points and factors must be kept in mind while designing the limit gauges :

1. Limit gauge tolerance :

(*a*) Manufacturing tolerance

(*b*) Wear allowance

and the disposition of these tolerances with respect to the work tolerance.

2. Taylor's principle of gauge design.

3. Fixing of gauging elements (ends) with handle.

4. Provision of Pilot.

5. Provision of Pilot.

6. Correct centring.

7. Materials

8. Hardness and Surface finish

9. Rigidity

10. Alignment

1. **Limit Gauge Tolerances**

(*a*) **Manufacturing tolerance.** We know that as in any other manufacturing process, in gauge making also it is economically impractical to attempt to make 'Go' and 'Not Go' gauges exactly to the two limits of the work tolerance. Thus it is necessary that permissible deviations in accuracies must be assigned for gauge manufacture. Gauge maker's tolerance or manufacturing tolerance should be kept as small as possible so that a large proportion of the work tolerance is still available for the manufacturing process. However the small the gauge tolerance, the more the gauge will cost.

There is no universally accepted policy for the amount of gauge tolerance. However, the following norms are generally accepted : Limit gauges are made 10 times more accurate than the tolerances they are going to control. That is, the tolerance on each gauge whether 'Go' or 'Not Go', is 1/10th of the work tolerance. For example, if the work tolerance is 10 units, then the manufacturing tolerance for `Go' and `Not Go' gauge each will be 1 unit. This makes it possible, although the probability is small, for the work tolerance available in the shop to be cut down to 80% of the specified tolerance. The amount of tolerance on inspection gauges is generally 5% of the work tolerance. Tolerance on reference or master gauges is generally 10% of the gauge tolerance.

Allocation of Manufacturing Tolerance. As already discussed, there are two systems for the allocation of manufacturing tolerance, unilateral system and bilateral system, Fig. 10.6.

(*i*) **Unilateral System.** In this system, the gauge tolerance zone lies entirely within the work tolerance zone. Due to this, the work tolerance zone becomes smaller by the sum of the gauge tolerances. But this ensures that every component passed by such a gauge regardless of the amount of gauge size variation will be within the work tolerance zone.

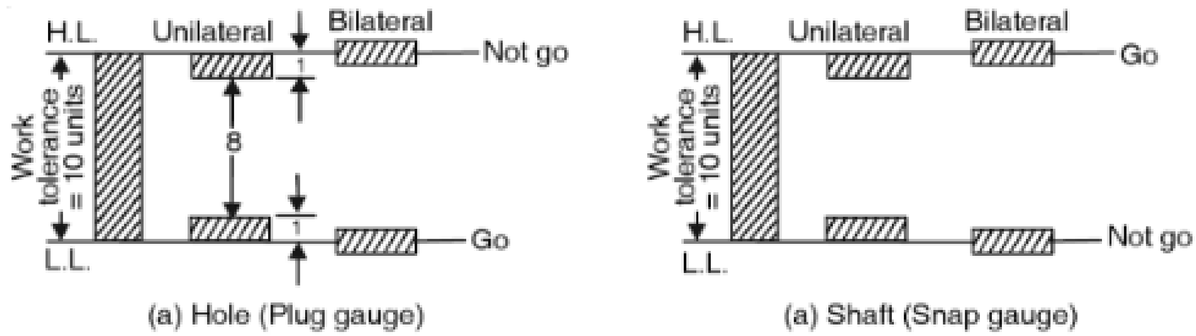


Fig. 10.6. Allocation of Manufacturing Tolerance.

For example, let the size of the hole to be tested be 25 ± 0.02 mm.

Therefore,

High limit of hole = 25.02 mm

Low limit of hole = 24.98

Work tolerance = 0.04

∴ Gauge tolerance = 10% of tolerance

= 0.004 mm

Dimension of 'Go' Plug/gauge

$$= 24.98\ \text{mm}\ {}^{+0.004}_{-0.000}$$

Dimension of 'Not Go' Plug gauge

$$= 25.02\ \text{mm}\ {}^{+0.000}_{-0.004}$$

The disadvantage of this system is that certain components may be rejected as being outside the working limits when they are not. However, the unilateral system has found wider use in industry than the bilateral system.

(*ii*) **Bilateral System.** In this system, the 'Go' and 'Not Go' gauge tolerance zones are bisected by the high and low limits of the work tolerance zone, Fig. 10.6. Taking the example as above,

$$\text{Dimension of 'Go' Plug gauge} = 24.98^{+0.002}_{-0.002} \text{ mm}$$

$$\text{Dimension of 'Not Go' Plug gauge} = 25.02^{-0.002}_{-0.002} \text{ mm}$$

The disadvantages of this system are that components which are within working limits can be rejected and parts which are outside the working limits can be accepted. But we already know (Chapter 9) that the percentage of such parts is very small, if the process is under control.

Another way of providing manufacturing tolerance in the unilateral system is shown in Fig. 10.7 in which the manufacturing tolerance is disposed opposite to the direction of wear, both for Go gauge and No Go gauge.

In the modern limit systems, unilateral system of providing tolerances is preferred, because in a hole basis system, the basic size will always be the Go size of a limit gauge, which is very convenient and practical.

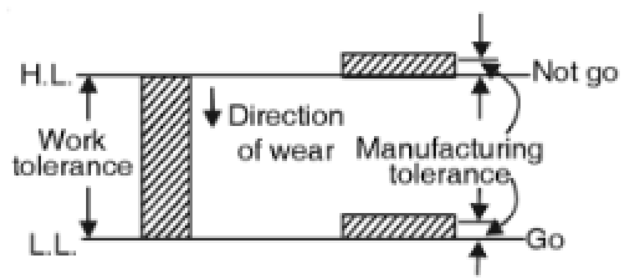


Fig. 10.7. Manufacturing Tolerance (Plug Gauge)

(*b*) **Wear Allowance.** Mostly the measuring surfaces of 'Go' gauges which constantly rub against the surfaces of the parts in inspection are subjected to wear and loose their initial size. 'Not Go' gauges are not subjected to so much wear as 'Go' gauges and there is considerable wear on 'Go' gauges only. The size of go plug gauge is reduced while that of go snap gauge increases. It is of course desirable to prolong the service life of the gauges, and therefore a special allowance of metal, known as wear allowance is added to the go gauge in a direction opposite to wear. Wear allowance is usually taken as 5% of work tolerance. Wear allowance is applied to a nominal go gauge diameter before gauge tolerance is applied. Taking the example discussed above,

Wear allowance = 5% of work tolerance

= 0.002 mm

Nominal size of go plug gauge = 24.98 + 0.002 = 24.982 mm

$\therefore$ Dimensions of Go Plug gauge = 24.982 + 0.004 − 0.000 mm

Dimension of Not Go Plug gauge = 25.02 + 0.000 − 0.004 mm

} Unilateral System

This is shown in Fig. 10.8

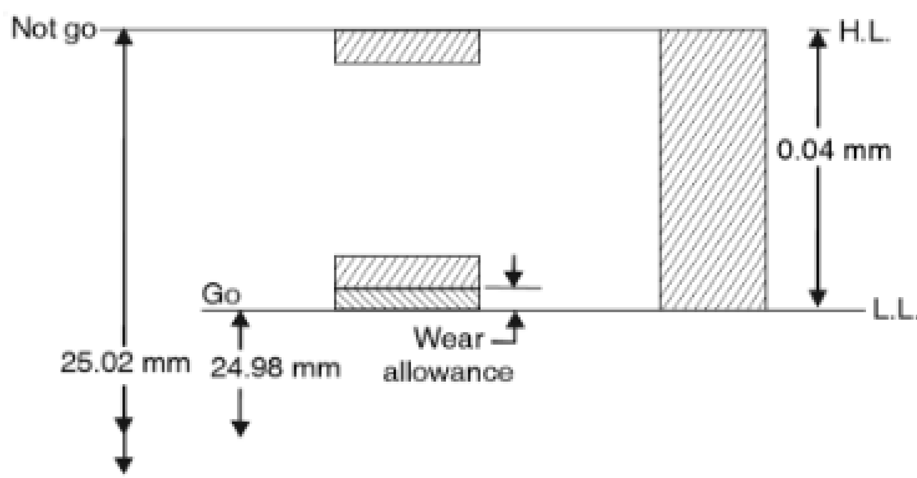


Fig. 10.8. Application of Wear Allowance.

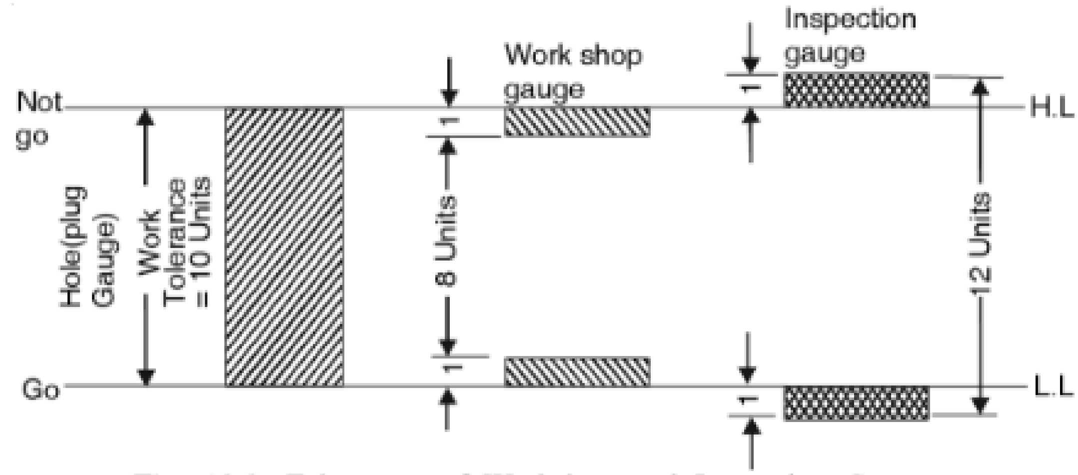


Fig. 10.9. Tolerances of Workshop and Inspection Gauges.

According to British Standards, wear allowance is provided when the work tolerance is greater than 0.09 mm. When the work tolerance is small and no wear allowance is provided, gauges should be of specially has working material in order to give a reasonable life. The method of providing tolerances on the workshop gauge and the inspection gauge according to British system is shown in Fig. 10.9. The tolerance on the workship gauge is arranged to fall inside the work tolerance (unilateral system) while on the inspection gauge it falls outside the work tolerance. The effect of these two types of gauges is that although the workshop gauge tends to cut down the tolerance available in the shop, it also ensures that all work passed by the working gauge will automatically pass the inspection gauge. It is theoretically possible for work which is slightly limits to be passed by the inspection gauge, although in practice very little trouble is experienced in this way. In this system, it may also happen that after wear a shop gauge may become inspection gauge (not a very desirable feature). Fig. 10.10 shows a revised system. In this system, the disadvantages of the inspection gauge are reduced by reducing the tolerance zone of the inspection gauge while the workshop gauge tolerance remains the same.

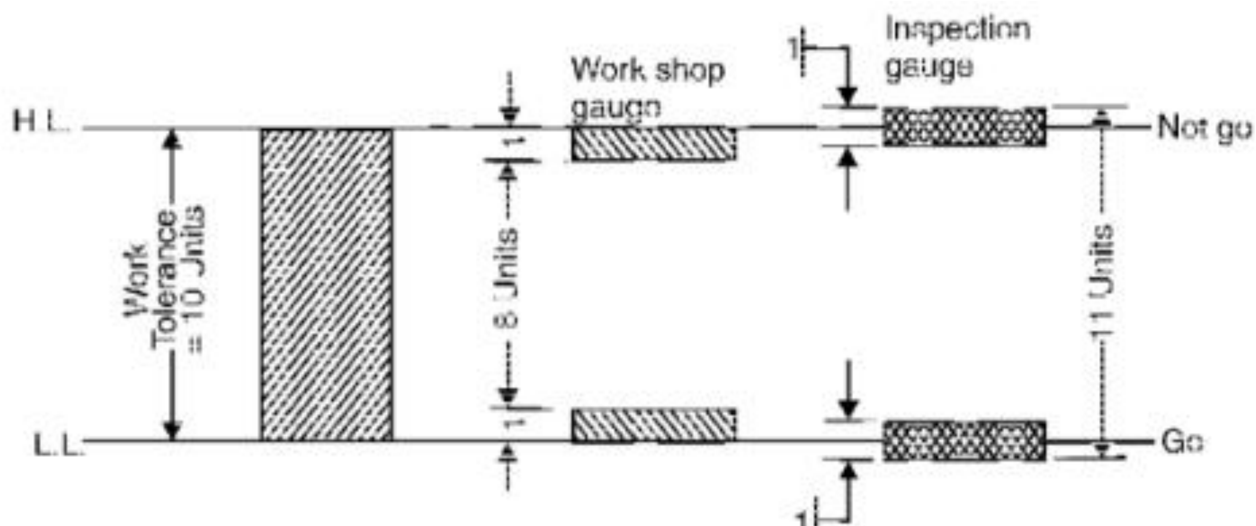


Fig. 10.10. Tolerances of Workshop and Inspection Gauges (Revised).

2. **Taylor's Principle.** This principle states that the Go gauge should always be so designed that it will cover the maximum metal condition (MMC) of as many dimensions as possible in the same limit gauge, whereas a Not Go gauge to cover the minimum metal condition of one dimension only, as shown in Fig. 10.2. According to this rule, a Go plug gauge should have a full circular section and be of full length of the hole, it has to check. In addition to control the diameter at any one point this ensures that any lack of straightness or parallelism of the hole will prevent the entry of full length Go plug gauge. For example, let us assume that a bushing is to be inspected. The bush is to mate with a shaft. The shaft is, therefore, the opposed part in relation to the bushing. Therefore the form of 'Go' plug gauge should exactly coincide with the form of the shaft. For this purpose, the "Go" plug gauge must be of adequate length, not less than the length of the future association of bushing and shaft. If this condition is not satisfied, part inspection with the gauge may prove to be defective or even entirely wrong. For instance, let us assume that the bush being inspected has a curved axis and a short 'Go' plug gauge is employed (Fig. 10.11). The short plug gauge will pass through all the curves of the bent bushing. This will lead to the erroneous conclusion that the workpiece is within the prescribed limits. Actually, such a bent bushing cannot mate properly with its opposed part. A 'Go' plug gauge of adequate length will not pass through a bent, curved bushing, and the error will be revealed. So, it will check geometrical shape as well. A long 'Go' plug gauge will check cylindrical surface, not in one direction, but in a number of sections simultaneously. Generally, the length of 'Go' plug gauge will check cylindrical surface, not in one direction, but in a number of sections simultaneously. Generally, the length of 'Go' plug gauge should not be less than 1.5 times the diameter of the hole. Length of 'Not Go' gauge is kept smaller than 'Go' gauge.

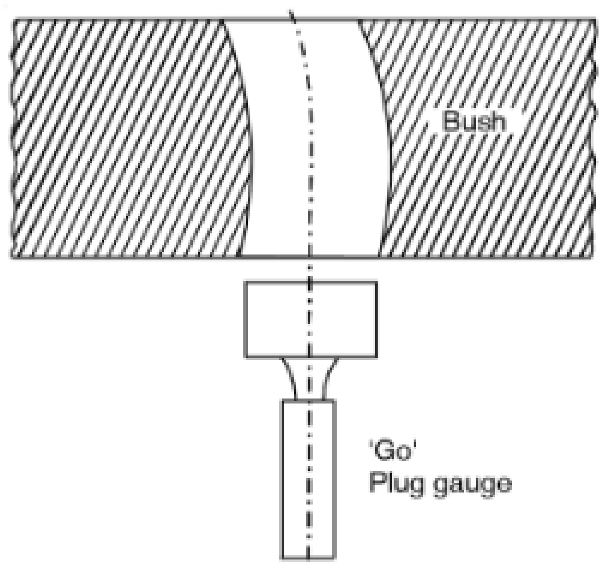


Fig. 10.11. Checking a Bush with a Curved Axis.

Now let us take the case of checking an oval hole by a cylindrical 'Not Go' gauge, Fig. 10.12. As the faces of the plug gauge and the hole under inspection overlap (hatched portion) the plug will obviously not enter the hole. This will again lead to erroneous conclusion that the part is within the prescribed limits. It will be more appropriate to make the 'Not go' gauge in the form of a pin or bar, shown with dashed lines. Turning such a pin gauge about the axis of the bushing will reveal the improper form of the hole.

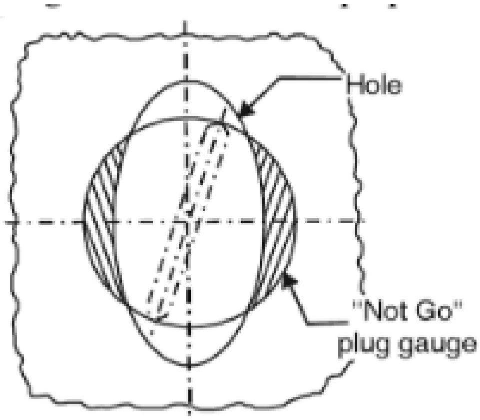


Fig. 10.12. Checking an Oval Hole.

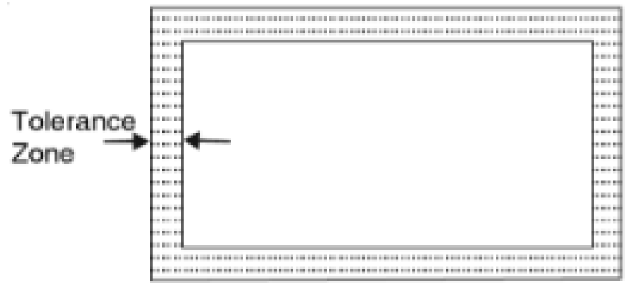


Fig. 10.13. Rectangular Hole.

Now let us consider the case of a rectangular hole (Fig. 10.13) which is to be checked for both linear and geometric features. Firstly, let us consider an error of geometry. For example, the corners of the rectangular hole are not square. Only a full form 'Go' gauge will indicate that the part is wrong. If pin gauges, made to the low limit of the hole (MMC), are used to check the hole they will enter the hole (Fig. 10.14) and the error will remain undetected. This will lead to wrong conclusion that the hole is satisfactory, when actually it is not.

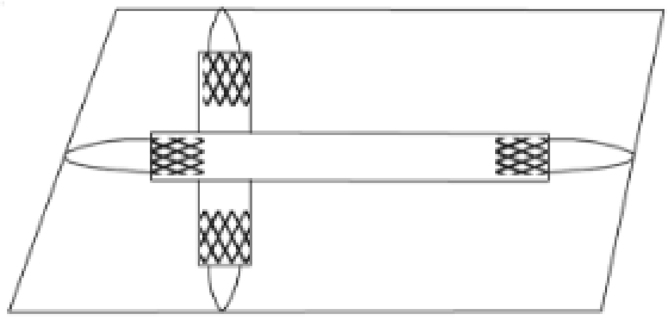


Fig. 10.14. Pin Gauges.

Next, let us consider an error of size, say for example, the length of hole is outside the high limit. A full form 'Not Go' gauge made to high limit of hole will not detect this error. As the width of hole is within limits, the gauge will not enter the hole, even though the length is outside limits. Again, this gauge will indicate that the hole is satisfactory when it is not. However, a 'Not Go' gauges of pin type to check the width and length of the hole.

From above, the design of various types of gauges based on Taylor's principle can be summarised as follows :

(*i*) **Circular Holes.** For checking circular holes 'Go Plug Gauge' should have a minimum length equal to the length of the hole or the length of engagement of the gauge with associated component, which ever is smaller. The 'Not Go Plug Gauge' will not be of full form. It should be pin gauge which would check the upper limit of hole (minimum metal condition) across any diameter at any position along the length of the hole. As already explained, turning such a gauge about the axis of hole will reveal the defect if there is any, because, even if the pin gauge accepts the hole along one axis, it will reject the hole when used along the other axis, as shown in Fig. 10.15.

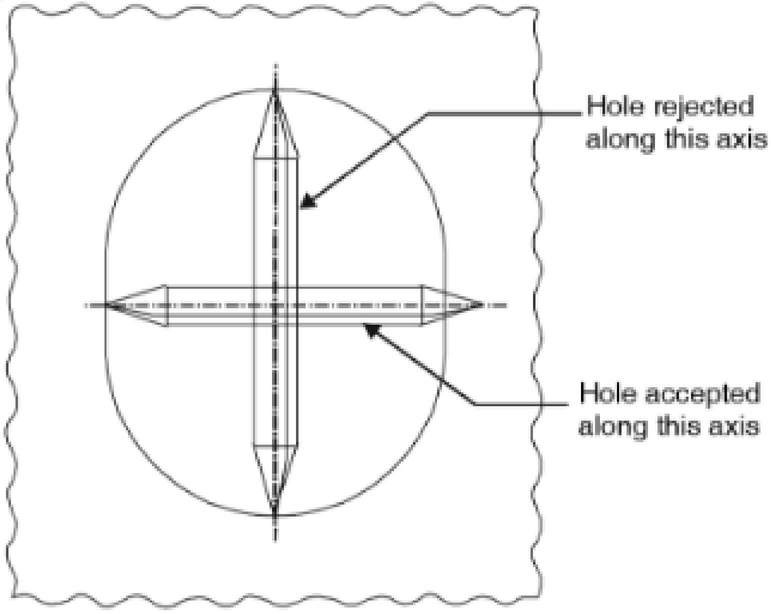


Fig. 10.15

(*ii*) **Circular Shafts.** To check the circular shafts, the ring gauge should be used for the 'Go Gauge' whose minimum length should be equal to the length of shaft or the length of engagement of the 'Ring Go Gauge' with the associated component, whichever is shorter. The 'Not Go Gauge' will not be ring gauge, but it should be in the form of a Snap Gauge or Gap Gauge, so that it is able to reject the shaft which is not circular, as shown in Fig. 10.16

Note. In many cases, shaft inspection with ring gauges would be difficult, for example, in checking stepped shafts or crank shafts. For this reason, "Go" snap gauges are much used for shafts, though their design is not according to rules.

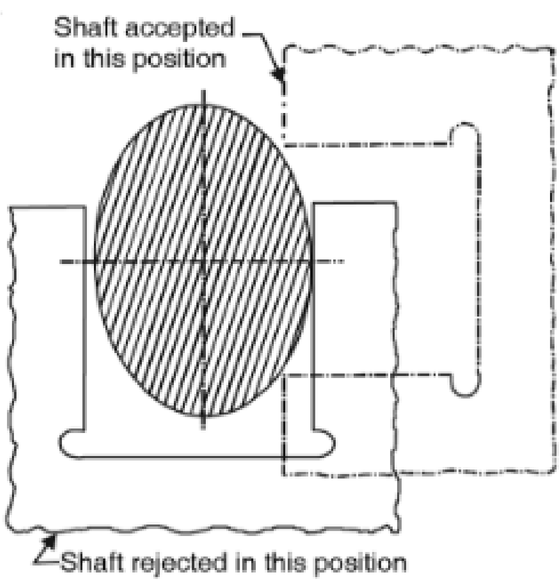


Fig. 10.16

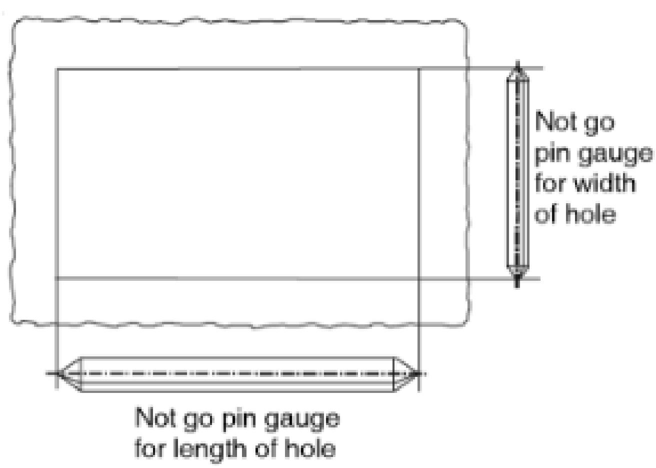


Fig. 10.17

(*iii*) **Non-circular Holes and Shafts.** For checking non-circular holes and shafts, the above principles will apply, but for each dimension there will be a separate 'Not Go Gauge', which will correspond to the minimum metal condition of the component, Fig. 10.17. The 'Go Gauge' would of course be of full form and would correspond to the maximum metal condition of the component.

10.7. ADVANTAGES OF LIMIT GAUGES

1. These are conveniently used in mass production for controlling various dimensions and thus ensure interchangeability.

2. These can easily be used by semi-skilled labour.

3. These are economical in their own cost as well as engaging cost.

10.8. LIMITATIONS OF LIMIT GAUGES

1. It is generally uneconomical even with excellent equipment and cost control to manufacture limit plug gauge to a tolerance on the gauge finer than 0-.0013 mm corresponding to a work tolerance of about 0.013 mm. There is also difficulty in the use of such fine limit gauges in the production shops. Finer tolerances than those specified above should preferably be measured directly with instrumental.

2. Limit gauges only indicate whether the component is within the tolerance limit or not. They do not indicate the exact size of the component.

4.10 Need for Limit Gauges

As discussed previously to follow the principle of interchangeability, the components should be manufactured to the correct sizes. But we also know that due to practical difficulties, all components cannot be manufactured to exact sizes but there exists permissible variation in the sizes called tolerance. The actual or absolute size of a component, provided that is within the limits specified, is not very much important because these limit will have been selected & correlated in such a way that satisfactory functioning is assured if the sizes of the components lie within them. It is this permitted variation in the size which results in economy but, on the other hand, unless some effective system of control of size is employed the advantages of manufacturing on an interchangeable basis will be lost.

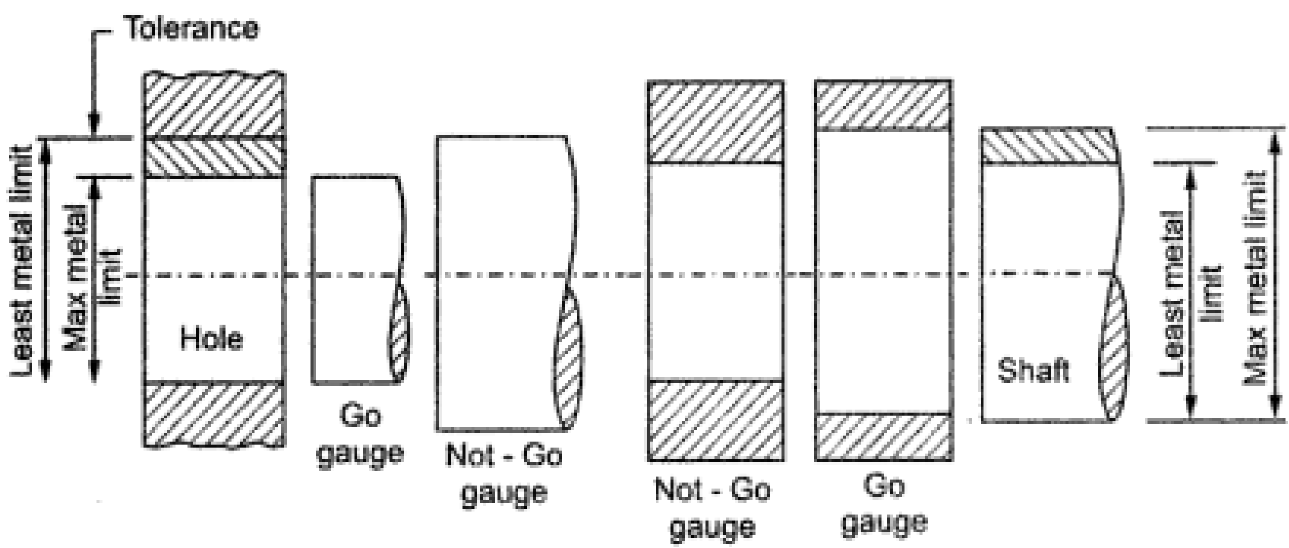


Fig. 4.15 Relation between gauge size and metals limits

A good solution for this is to use fixed or non-recording limit gauges. Limit gauges simply ensures that the size of the component being inspected lies within its specified limits, they however, do not determine the actual size. To check a given dimension on a component two gauges are required having basic sizes corresponding to the two limits of size for the component. They are known as the Go gauge & Not Go gauges. The Go gauge checks the maximum metal limit (lower limit) of the work & the Not Go gauge checks the least material limit. The difference between the basic sizes of two gauges is equal to the tolerance on the components. Each of these two gauge is offered in turn to work. Clearly if the size of the component is within the prescribed limits. The gauge made to the maximum metal limit will assemble with it, whereas the other will not. It is for the reason, the gauge made to max. metal limit is called the 'Go' gauge & that made to least metal limit, the Not Go gauge.

4.12 Gauge Tolerance

Gauges like any other product requires standard manufacturing procedure like any other products, it is very difficult to manufacture the gauges to the exact required dimensions. The theoretical gauge size as determined from the maximum & minimum limits of the component need some modification to allow for reasonable imperfections in the workmanship of the gauge-maker. Thus the tolerance on gauge allowed to cater for workmanship of a gauge maker is known as Gauge makers allowance or tolerance or simply Gauge tolerance. Logically the gauge tolerance should be kept as small as possible.

There is unfortunately no universally accepted policy or some standard mathematical formulae which will give the value for gauge tolerance. Limit gauges are made 10 times more accurate than the tolerances they are supposed to control. This means the value of gauge tolerance should be $1/10^{th}$ of the work tolerance. Disposition of the gauge tolerance relative to the Nominal gauge size requires a policy decision. For instance, if the gauge tolerance increases the size of a Go plug gauge from 50.0 to 50.1 mm, & decrease the size of the Not Go gauge from 51.0 to 50.9 mm shown in Fig., them depending upon the actual size of these two gauges, some of the acceptable work (From 50.0 mm onwards to the actual size of the Go gauge) will be rejected by the Go gauge. This rejection will reach its maximum value when the Go gauge size is 50.1 mm. Similarly when the Not-Go gauge size is 50.9 mm the acceptable work (50.9 to 51mm) will be rejected. The % of rejection depends upon the machine setting. If the machine has been set to midway between the two limits (i.e. at 50.5 mm) the percentage rejects will be comparatively less. If the machine has been set of 50.5, many of the pieces will be between 50.0 to 50.1mm & the percentage of the rejects touches a higher figure. However it is obvious that by disposing the gauge tolerance in the above manner, certain percentage of acceptable work will be rejected by these gauges.

Similarly if the gauge tolerance increases the size of the Not Go gauge & decreases the size of the Go gauge, then the gauge will tend to accept work which is just outside the specification. This can be clear from fig.4.18 above.

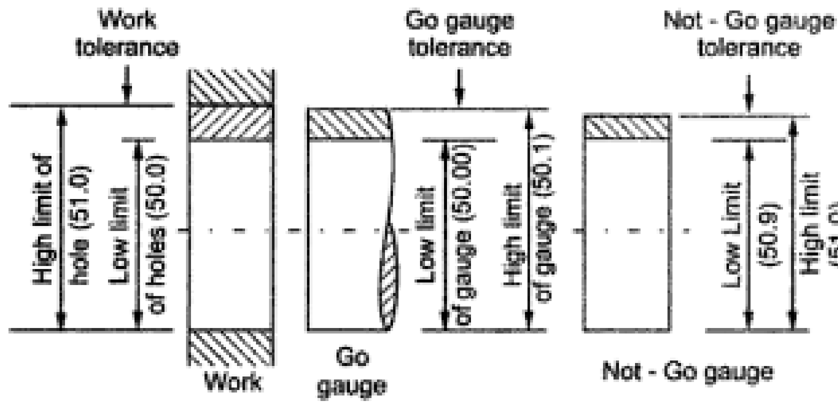


Fig. 4.18 (a) Effect of disposition of gauge tolerance on acceptance / rejection of work

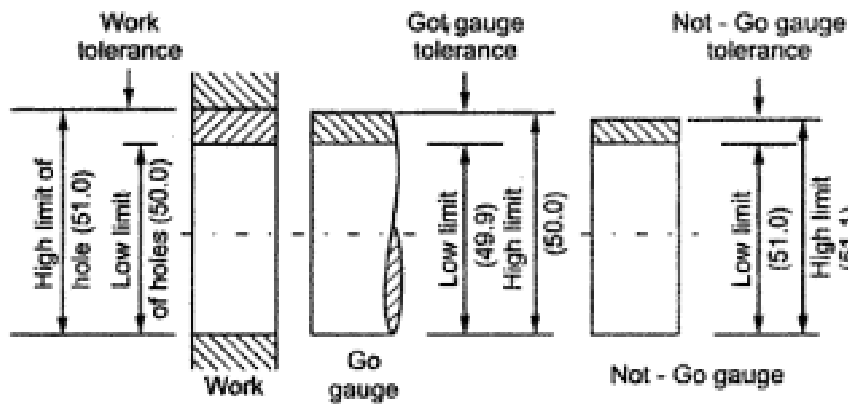


Fig. 4.18 (b) Effect on disposition of tolerance on acceptance / rejection of work

4.13 Wear Allowance

Mostly the measuring surface of Go gauges which frequently assemble with the work, rub constantly against the surface of the work. This results in wearing of the surfaces of the gauges & as consequence of this they loose their initial dimension. Thus Go gauge should be provided with additional allowance. This allowance is provided in direction opposite to the wear direction. Wear allowance in applied to a nominal Go gauge diameter before gauge tolerance is applied. It may be seen that the gauge reject more number of good pieces as compared to the gauge having only gauge tolerance. Hence it is once again emphasizes that a policy decision with regard to the disposition of the gauge tolerance is of vital importance.

There are several empirical rules for the amount of wear allowance. Some companies take it 5% of work tolerance or somemay prefer 20% of the gauge tolerance. A widely accepted industrial practice is to consider it to be 10% of the gauge tolerance. So it is obvious that a wear tolerance 10% of gauge tolerance is equivalent to 1% of work tolerance & because one micron is the smallest unit to be used in expressing the dimensions in gauge work, it is clear if the work tolerance is less than 0.1 mm, the wear allowance will be less than 0.001 mm & will have no practical effect on the gauge. Hence no wear allowance is given on gauges for these components whose work tolerance is less than 0.1 mm.

4.14 Types of Gauges

The gauges depend on the use as

i) Plain Gauges

ii) Gauges for Threaded holes & shafts – special Gauges.

A large amount of work in industry today is being gauged by plain gauges. The plain gauges are further classified on various basis as follows.

1. According to type
2. According to purpose
3. According to form of substance tested.
4. According to design.

1. According to type:

According to the type the limit gauges are classified as:

A) Standard gauges

Every Go gauge is almost replica or a model of the part to be checked. For example if a bushing is manufactured to mate or assemble with shaft, then shaft is mating part. The bush is checked by a Go gauge which in so as the form of its surface, size etc. is concerned is a copy of the mating part, that is the shaft.

If the Go gauge is according to the exact size of the part being checked then this type of gauges is called standard gauges. We know, we have provided to some tolerance on the part as well as gauge, so in practice the standard gauge is of no use. The standard gauge is also cannot be used to check the parts having interference fit.

B) Limit gauges:

These are the commonly used gauges in all types of industries. Every part has two limits of dimensions of a part, high & low. The part is checked by successively assembling each of the gauges with it. These are of two types of Go & Not Go gauges.

Go gauges are designed to maximum material condition & they should pass the part whereas the Not Go gauges are designed to minimum material condition & they must not enter the part to be checked.

2. **According to purpose:**

According to the purpose the Gauges may be classified as

A) Workshop or Shop Gauges

It is already discussed that the disposition of gauge tolerance requires a policy decision. Well defined zones of tolerance must be established for the gauges if their use is not be accompanied by much uncertainty. For example if it is desired that no accepted product should be outside the specifications, then it is essential to have gauge tolerance both for Go & Not Go gauge within the work tolerance. Experience has shown that two basic & to certain extent conflicting principles which govern the use of limit gauges are the following:

1. No work should be produced by the workshops are accepted by inspection department which lies outside the prescribed limit of size.
2. No work should be rejected which lies with in the prescribed limits of size.

Both principles are important, concern with quality, conscious would concentrate more on the former principle, while on organization manufacturing for a highly competitive market would naturally tend to stress the latter principle. One solution, commonly adopted in the past is to use two separate set for the application of above two principles. The gauge set used during the manufacturing is known as workshop or simply shop gauges. In order that the shop gauge should confirm to the first principle, the zones of tolerance on these gauges were placed entirely within the corresponding work limit. Wear allowance shown applied only when the work tolerance is above 0.1 mm. Wear was permitted to take place until the maximum metal limit of the workpiece had been reached. Under these circumstances all work accepted by such gauges must be within specified limit.

There was possibility, however that if such gauges were used for final inspection, some pieces being within specified limits might be rejected, particularly, if the gauges were new & therefore near their own maximum metal limit.

B) Inspection gauges

To overcome the above limitation of the shop gauges, a separate set of gauges is used during the inspection of the parts. In these gauges the tolerance zone is placed outside the work limits. Because the fit on a Go inspection gauge should be fairly slack, wear should be small & therefore no wear allowance was made.

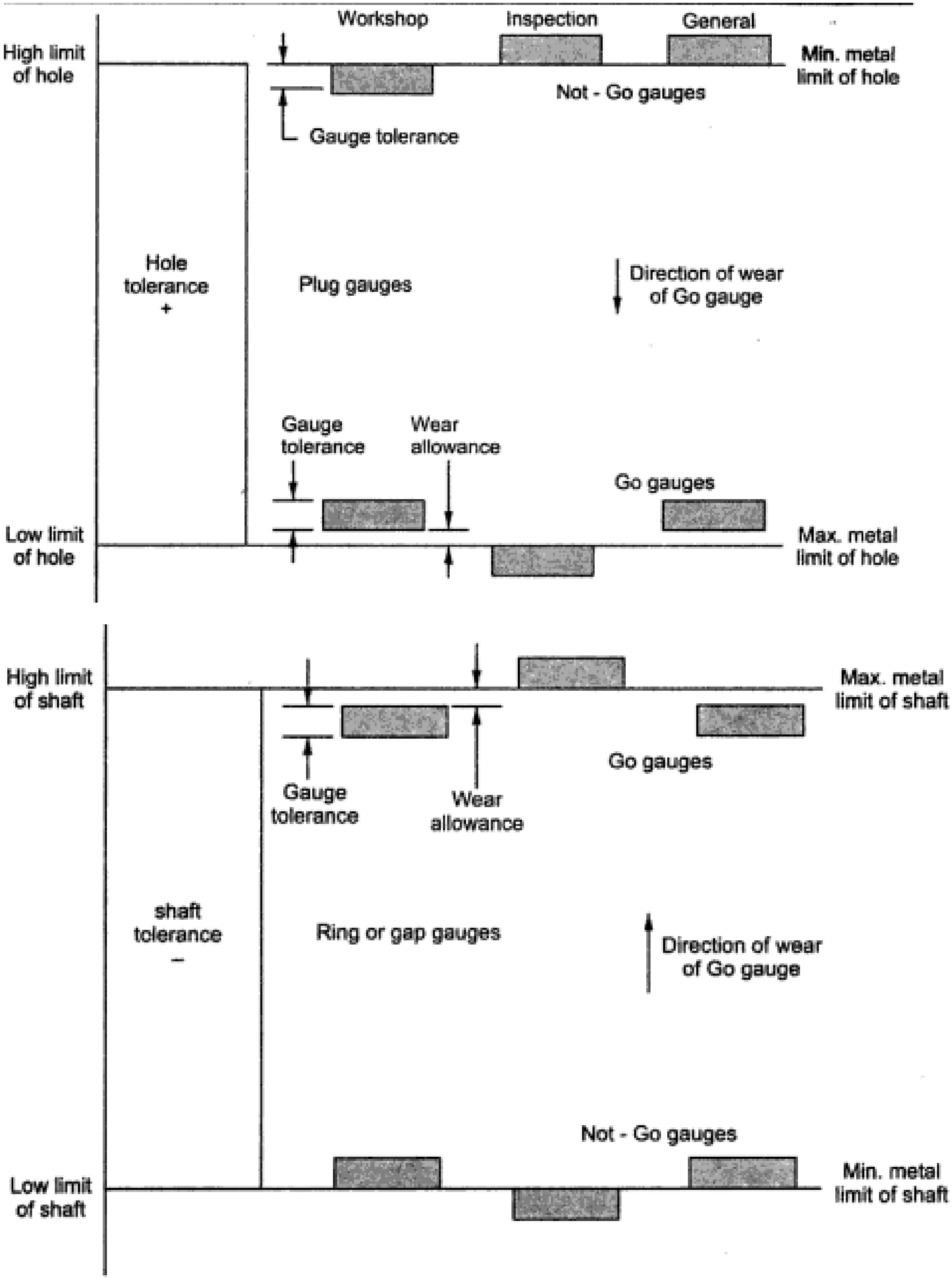


Fig. 4.19 Gauge Tolerance & Wear Allowance

C) General gauges

Although all work lying within the prescribed limit would be accepted by inspection gauges there was always the danger that some part outside the work limits, & particularity maximum metal limit might be accepted. Fig. shows that such work is infringing on the allowance or minimum clearance zone with the result that either the parts might not mate, or if they did, seizure might takes place. For this reason, to approach the first principle of limit gauging the General gauge has been recommended. In this gauge the tolerance zone for Go gauge is placed inside the work limits & tolerance zone of the Not – Go gauge is placed as in inspection Not Go gauge.

D) Purchase inspection gauge

The need for such a gauge arises when the products of other plants are to be accepted. The purchaser must remember that the parts may have been gauged by working gauges work to maximum possible degree. So the Go gauge should be designed carefully.

E) Master gauge

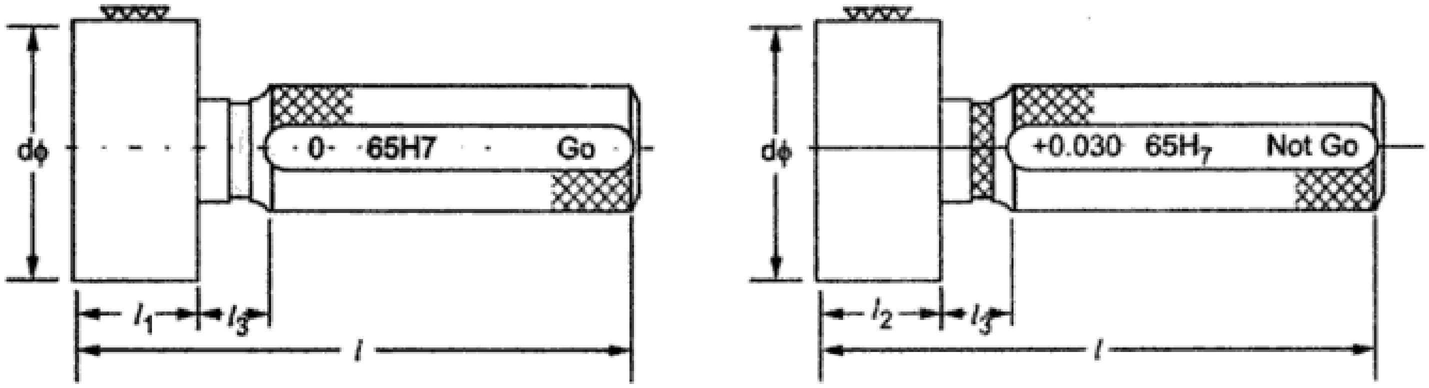
These gauges are used for checking the size and condition of other gauges. Master gauges are the reverse or opposite form to working or inspection gauges. These gauges are checked by universal measuring machine.

3. According to form of tested surface

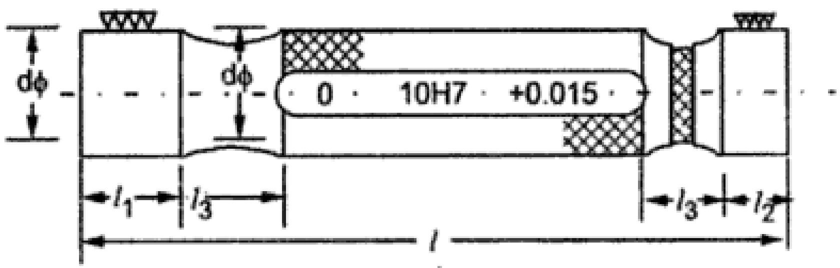
According to the form to be tested, the gauges are classified as:

a) Plug Gauges

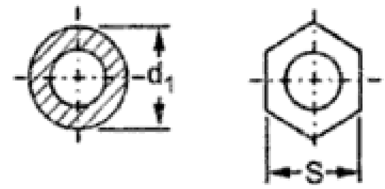
These gauges are made for checking hole sizes. The gauges are made from C85 wear resistant steel & hardened not less than 750 HV. The gauges surface of plain gauges should be stabilized & ground & lapped. Handles for plain plug gauges should be made on any suitable steel. Plug gauges may be single end or double ended. Also these are may be made integrated or may be fastened as shown in fig.4.20(b)



(a) Go and not Go plug gauges with single end



(b) Go and not Go plug gauges with double end



(c) Section of handles

Fig. 4.20 Go-Not Go type plug Gauges

b) Ring or Snap gauges:

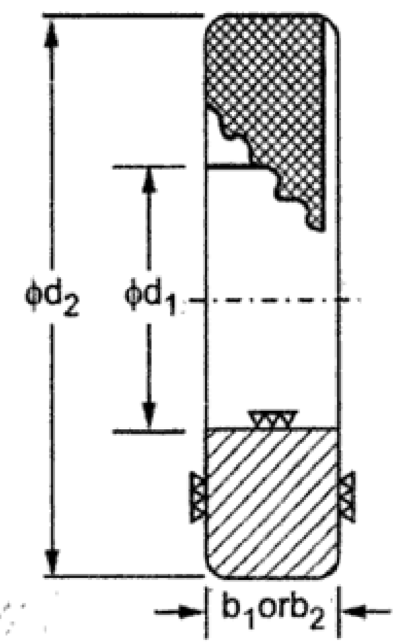


Fig. 4.21 Plain Ring Gauges

These gauges are used to check shafts. All aspects such as material, the quality of gauging surfaces, designation of gauges, marking on gauges are as for plain plug gauges.

Thus, a Go plain ring gauge for gauging a shaft of 25 mm diameter with a tolerance h_8 will be designated as Go plain Ring gauge 25h_8.

Snap gauges can be used for both cylindrical as well as non-cylindrical work as compared to ring gauges which are used only for cylindrical work. Material & other specifications of snap gauges are same as ring gauges. Double ended snap gauges are shown in Fig.4.22

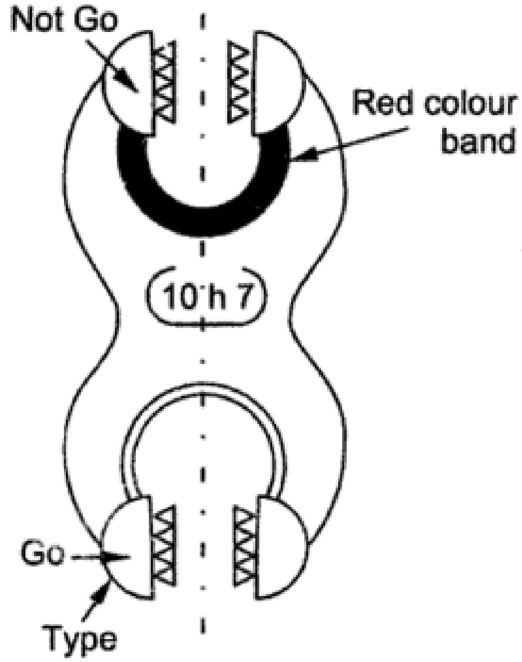


Fig. 4.22 Snap gauge with double end

The snap Gauge can be made in progressive form or adjustable as shown in Fig.4.23 Adjustable snap gauges are quite expensive but they can be used to small range.

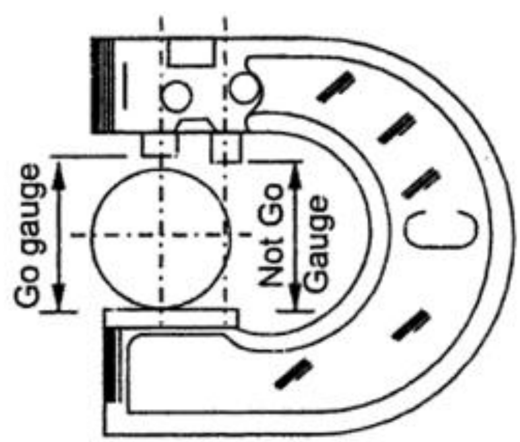


Fig. 4.23 Adjustable gap gauge

Types of Assemblies

There are three ways by which the mating parts can be made to fit together in the desired manner. These are :

(1) Trial and Error (2) Interchangeable Assembly

(3) Selective Assembly

(1) **Trial and Error**. When a small number of similar assemblies are to be made by the same operator the necessary fit can be obtained by trial and error. This technique simply requires one part to be made to its nominal size as accurately as possible, the other part is then machined with a small amount at a time by trial and error until they fit in the required manner. This method may be used for "one off jobs", tool room work etc. where both parts will be replaced at once.

(2) **Interchangeable Assembly.** When a large number of components are to be produced then it will not be economical to produce both the mating components by the same operator. In addition to economy it is also essential to produce the components within the minimum possible time. This is only possible by mass production system. In mass production system there is a division of labour. The components are produced in one or more batches by

different operators on different machines. Under such conditions in order to assemble the mating components with a desired fit, a strict control is exercised and the parts are manufactured with specified tolerance limits.

When a system of this kind is used any one component selected at random will assemble correctly with any other mating component that too, selected at random, the system is called interchangeable assembly. The manufacture of components under such conditions is called interchangeable manufacture. Production on an interchangeable basis results in increased output with a corresponding reduction in manufacturing cost.

Example : Suppose a clearance fit is required between the mating parts with hole, specified as $25^{+0.04}_{-0.00}$ mm

and shaft $25^{-0.02}_{-0.04}$ mm

In this case the maximum permissible size of the hole will be = 25.04 mm and the minimum permissible size = 25.00 mm. The dimensions of the number of holes produced will lie between these two limits. Similarly, the maximum permissible shaft size = 24.98 mm and the minimum permissible size of shaft = 24.96. The dimensions of all the shafts produced will lie between these two limits. Therefore, even if we select any hole at random and similarly any shaft at random with these permissible tolerances they will assemble with each other and give the desired clearance fit.

Interchangeable assembly requires precise machines or processes whose process capability is equal to or less than the manufacturing tolerance allowed for that part. Only then every component produced will be within desired tolerance and capable of mating with any other mating component to give the required fit.

Advantages of Interchangeability

1. The operator is not required to waste his skill in fitting the mating components by trial and error and thus assembly time is reduced considerably.
2. There is an increased output with reduced production cost.
3. There is a division of labour, the operator has to perform same limited operations again and again thus he becomes specialized in that particular work, which helps to improve quality and reduce the time for operations.
4. It facilitates production of mating components at different places, by different operators.
5. The replacement of worn out and defective

(3) Selective Assembly. It is sometimes found that it is not economical to manufacture parts to the required high degree of accuracy so as to make them interchangeable. The consumer not only wants quality and precision trouble-free products but also he wants them at economical prices. Often special cases of accuracy and uniformity arise which might not be satisfied by certainty at the fits given under fully interchangeable system. For example, if a part of its low limit is assembled with the mating part at high limit, the fit so obtained may not fully satisfy the functional requirements of the assembly. Complete interchangeability in the above cases can be obtained at some extra cost in inspection and material handling by using selective assembly whereby parts are manufactured to rather wider tolerances.

In selective assembly the components produced are classified into groups according to their sizes by automatic gauging. This is done for both mating parts, holes and shafts, and only matched groups of mating parts are assembled. It results in complete protection against defective assemblies and reduces matching costs since the parts may be produced with wider tolerances.

A practical example of this system is the assembly of pistons with cylinder bores. Let the bore size be 50 mm and the clearance required for the assembly is 0.12 mm on the diameter. Let the tolerance on bore and the piston each = 0.04 mm. Then,

Dimension of bore diameter is $50^{\pm 0.02}$ mm.

Dimension of piston shaft is $49.88^{\pm 0.02}$ mm

By grading and marking the bores and the pistons they may be selectively assembled to give the clearance of 0.12 mm as given below :

Cylinder bore	49.98	50.00	50.02
Piston	49.86	49.88	49.90

Process Planning Sheet

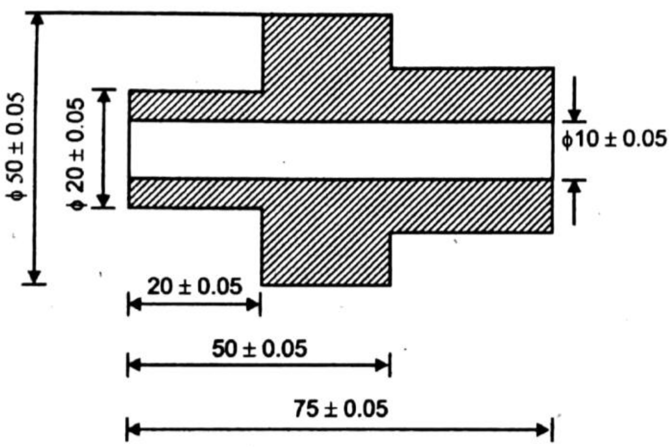
Figure below shows the drawing of machine component. Prepare the plan of manufacturing of this part in terms of :

(a) **Process selection.**

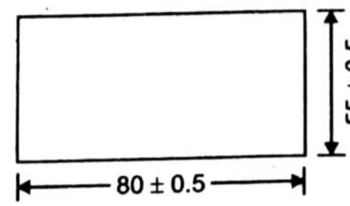
(b) **Process planning sheet.**

(c) **Tolerance chart for longitudinal dimensions.**

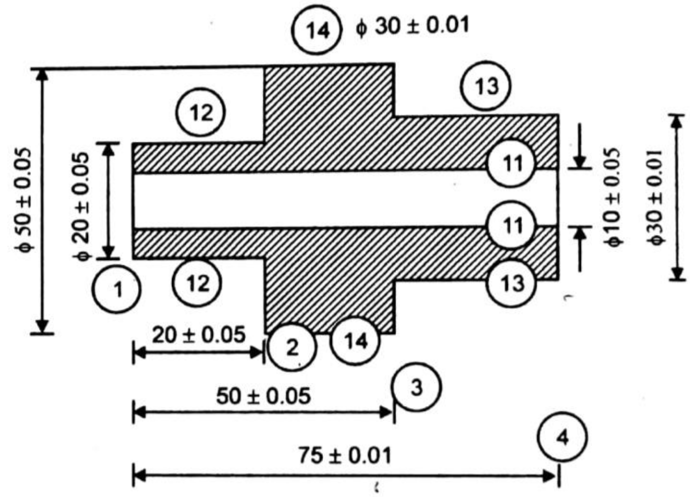
(d) **Raw material size.**



Ans. (i) Selection of raw material :



(ii) Index table (Process selection) or operation selection :



Index table :

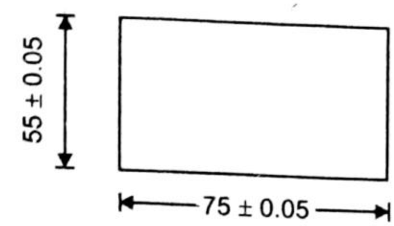
Sr.No.	Index	Dimension	Tolerance	I.T. grade	Operation
1	1-2	20	± 0.05	IT10	Turning
2	1-3	50	± 0.05	IT10	Turning
3	1-4	75	± 0.01	IT6	Grinding
4	11-11	10	± 0.05	IT10	Drilling
5	12-12	20	± 0.05	IT10	Turning
6	13-13	30	± 0.01	IT6	Grinding
7	14-14	50	± 0.05	IT10	Turning

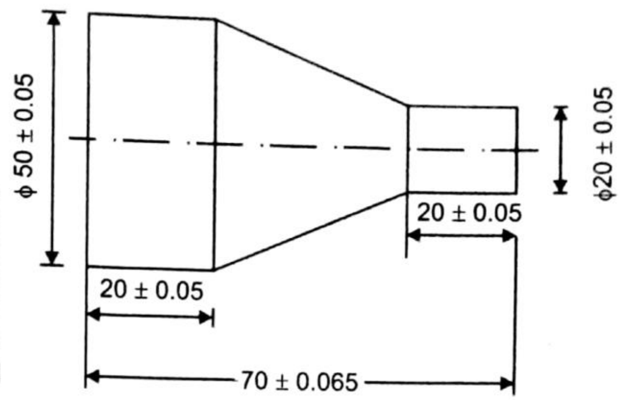
Operation (1)	Description (2)	Index (2)	Sketch (4)	M/c Tool (5)	Working Device (6)	Cutting Tool (7)	Remark (8)
10(a)	Face (1)	4 – 1	55 77.7	Lathe	FJC	SPCT	Locate at (4) Depth of cut = 2.3 mm Qualifying cut
10(b)	Drill ϕ10 Length 77.7	11 – 11	55 10 77.7	Lathe	FJC/T.S.	Drill bit	Secondary operation
10(c)	Turn ϕ50 Length 50.2	14 – 14 1 – 3	55 ϕ50 50.2	Lathe	FJC	SPCT	Depth of cut = 2.4 mm Secondary operation
10(d)	Turn ϕ20 Length 20.2		ϕ20 20.2 50.2	Lathe	FJC	SPCT	Depth of cut = 15 4 + 4 + 4 + 3
20(a)	Face (4)	12 – 12 1 – 2	20.2 50.2 73.4	Lathe	FJC	SPCT	Locate at 1 Depth of cut = 2.3 mm Process critical
20(b)	Turn ϕ30.4 Length 25.2	13 – 13	30.4	Lathe	FJC		Depth of cut = 12.3 mm 4 + 4 + 4 + 0.3 Process critical
30(a)	Grind ϕ30	13 – 13	ϕ30	Cylindrical Grinding m/c	Vice	Grinding wheel	Depth of cut = 0.2 mm Product critical
40(a)	Grind 4	1 – 4	75.2	Surface Grinder	Vice	Grinding wheel	Depth of cut = 0.2 mm Product critical
50(a)	Grind 1	4 – 1	75.2	Surface Grinder	Vice	Grinding wheel	Depth of cut = 0.2 mm Product critical

v) **Longitudinal tolerance chart :**

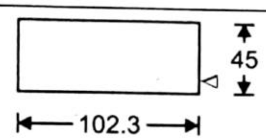
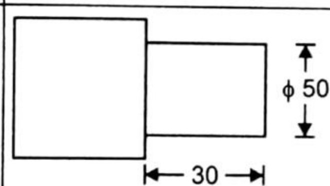
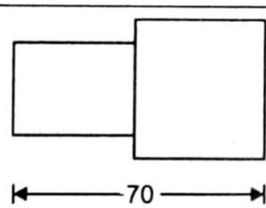
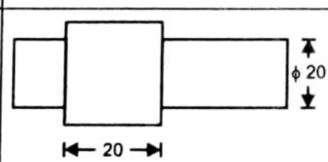
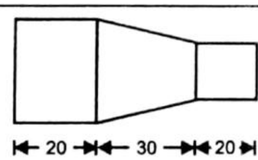
Op. No.	Index	Working Dimension		1 2 3 4	Resultant		Stock removal	
		Dim.	Tol.		Dim.	Tol.	Dim.	Tol.
10a	4-1	77.7	± 0.05		-	-	2.3	± 0.55
10d	1-2	20.2	± 0.05		-	-	-	-
10c& 10d	1-3	50.2	± 0.05		-	-	-	-
20a	1-4	75.4	± 0.05		-	-	2.3	± 0.10
40a	1-4	75.2	± 0.01		-	-	0.2	± 0.06
50a	4-1	75	± 0.01		75	± 0.01	0.2	± 0.02
10d	1-2	20	± 0.05		20	± 0.05	-	-
10c & 10d	1-3	50	± 0.05		50	± 005	-	-

Raw material size is 55 ± 0.5 × 75 ± 0.5 mm .





(ii) **Process planning sheet :**

Operation (1)	**Description** (2)	**Index** (3)	**Sketch** (4)	**M/c tool** (5)	**Working Device** (6)	**Cutting tool** (7)	**Remarks** (8)
10(a)	Face (1)	4 – 1		Lathe	FJC		Locate at Face 4 Depth of cut = 2.5 mm
10 (b)	Turn ϕ50 Length 50	11 – 11 1 – 2		Lathe	FJC	SPCT	Locate at Face 1 Depth of cut = 2.5 mm
20 (c)	Face (4)	1 – 4		Lathe	FJC	SPCT	Depth of cut = 4 + 4 + 4 + 3
20(b)	Turn ϕ20 Length 20	12 – 12 4 – 3		Lathe	FJC	SPCT	
20(c)	Turn Taper from ϕ20 to ϕ50 Length = $\sqrt{30^2 + 15^2}$	14 – 14 5 – 4		Lathe	FJC	SPCT	