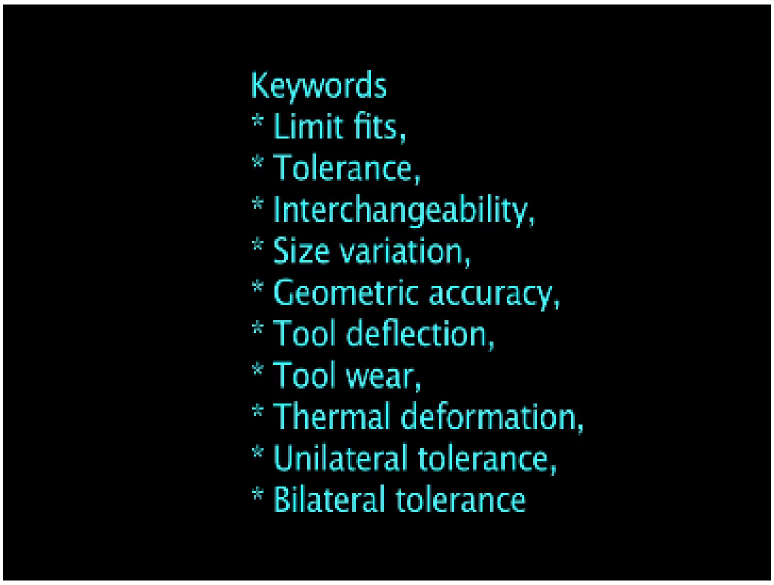
Metrology
Prof. Dr. Kanakuppi Sadashivappa
Department of Industrial and Production Engineering
Bapuji Institute of Engineering and Technology-Davangere

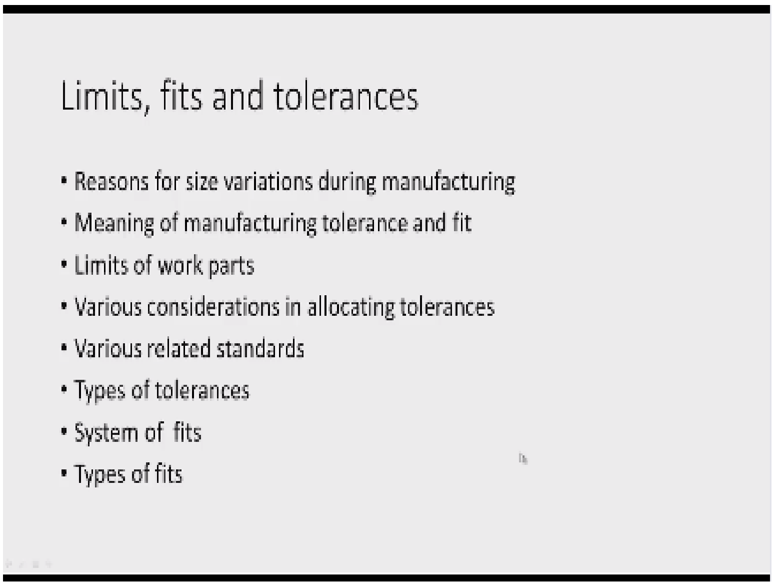
Module-3
Lecture-1
Manufacturing tolerance and fits

(Refer Slide Time: 00:15)



Hi welcome you all for the module 3 lecture 1. I will have overview of the module 3 in which we will be discussing about limits fits and tolerances. Now we will discuss about what are the reasons for size variation during manufacturing and what is the meaning of manufacturing tolerances and fits.

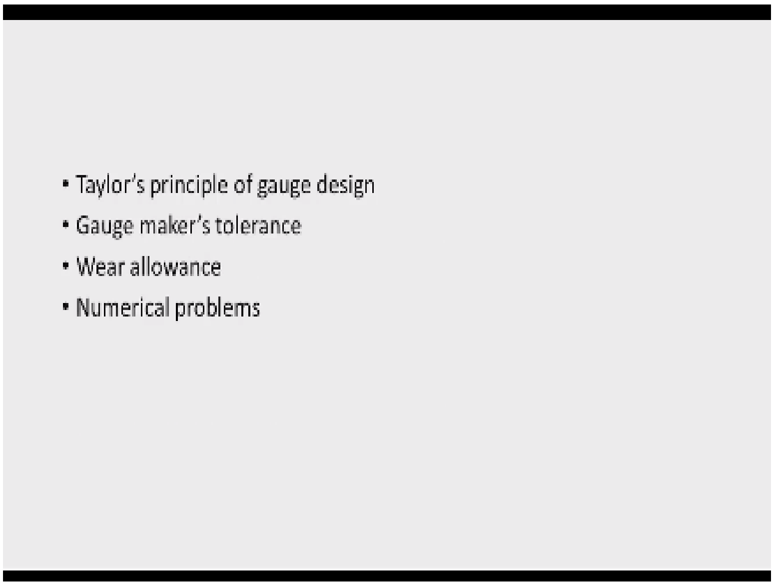
(Refer Slide Time: 00:46)



And what are the elements of our parts like upper limit and lower limit and then what are the various consideration in locating tolerances for the work piece. We will also learn about the rate standards and we will discuss about the various types of tolerance fits and what are the various kinds of types of fits and then what is the meaning of international tolerance grade and what is the necessity of tolerance grade.

And how do we selected the kind of fit that is required depending upon the application and then what are the various geometrical tolerances and then we will also learn about a positional tolerance and then we will move to the limit gauging the part of the lecture wherein may be studying about the meaning of gazing and the necessity of gauging and how the gauges are classified.

(Refer Slide Time: 02:02)



How they are manufactured and what are the various types of gauges that are available for the gauging purpose and then love you learn about Taylor's principle of gauge design and what is the cage makers tolerance and what is wear allowance and then we will have some numerical problems.

(Refer Slide Time: 02:18)

Now will start the first lecture the product design consideration, whenever a new product is developed the various considerations are given for the development design and development of the product. So the engineer will look for the functional aspects of their product, what is the size of the product, what are the various applications of the product, what are the various features a product should have, what is the weight and what is the basic requirement of the customer.

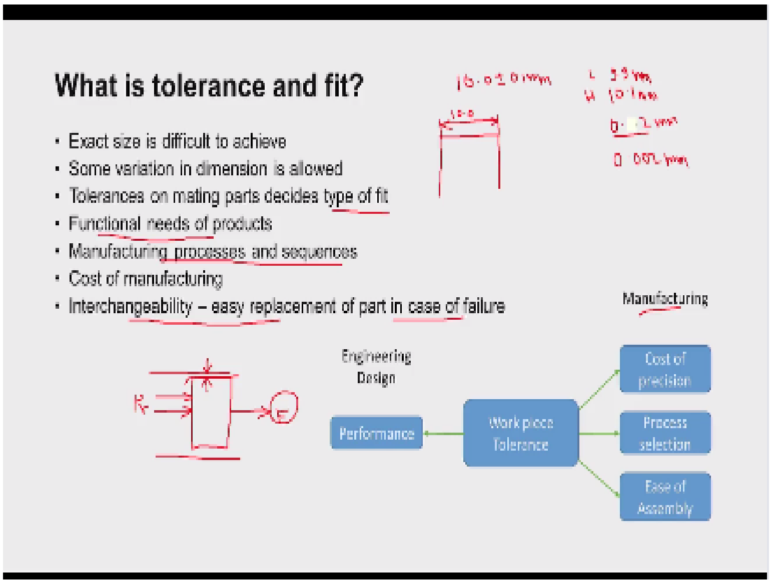
All those things one should study in detail and then the what about the durability and dependability of the product, see the customer purchase product is worried about the reliability product and durability life of the product should exist for a period of time and then economic aspects of product the customer wishes to purchase a good quality material that very economical prices.

So while designing their product design engineers should see how the product can be manufactured more economically and then there is a relationship between designer department and manufacturing department, they will have to sit together and decide about the design and production aspect of the product and then they have the esthetic part of the product the work piece of the product should be pleasing should have the pleasing appearance.

The colour of the product and shape of the product, these are very important aspects which will attract the customers and then how it can be marketed what is a what about the packaging of the product and those things also one should study and it is very important that the product should be always eco friendly should not affect the environment you should not spoil the environment such things also one should consider.

And once the period of the life of the product is over, how it can be discussed in what way can be disposed without affecting the environment. So these are some of the considerations while designing the product.

(Refer Slide Time: 04:51)



Now once the product design is made and specifications are fixed and part of drawings are prepared. All the requirements are built into the drawing of the various parts and it is supplied to the manufacturing department. So in the drawing the sizes of various parts are mentioned, now we know that it is always very difficult to produce the parts to an exact size. So when we try to produce the parts to exact size we have to use more precise manufacturing processes like grinding process or lapping process or super finishing process.

So there by they caused the product will increase, so we should always see what is the functional aspect the product and manufacturing aspect of the product and then we should fix the what is the label deviation. So in order to produce the part more economical some variation in dimension is allowed by design engineer which will satisfy the functional requirement.

So they tolerance some making parts decides the type of fit, for example so the design engineer has made some calculations and we design shows that the size of the shaft is 10 millimeter. Now we know that it is very difficult to produce that shaft exactly to 10 millimetre. So it will allow some variation in the dimension say the shaft and vary within this size 0.1 mm that means the shaft can have the lower limit of 9 point lower limit of 9.9 mm and the upper limit can go up to 10.h higher limit can go up to 10.1 mm.

So these are limits of the work piece, the work piece that is produced is having any size between this 10.1 millimetre and 9.9 millimetre then it will be acceptable. Now by specifying the tolerance we should see what is the functional need of the work piece is very important. For example say we have a hydraulic feeling like this where in there is some pressurized fluid is there and it is acting some there is some pressure.

And now because of this we get a force here can be used to move the move some load, now the design engineer should see what is the functionality of this particular product and accordingly should decide about the tolerance on the shaft as well as a tolerance for the hole. If the clearance between these two is more than the fluid will leak and then the required amount of force we may not get.

If it is too tight then the piston may get jammed in the cylinder, so such suspects we should see before assigning the tolerance values and then the tolerance that is alone will definitely affect the manufacturing processes and sequences we can see this picture this is work piece tolerance, it will have some effects on manufacturing aspects, it will affect the cost of the precision and tolerance is very very tight.

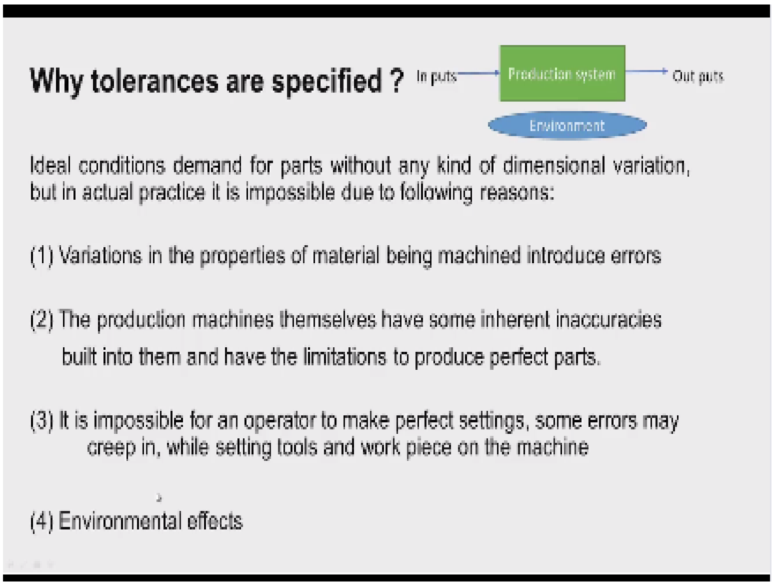
For example instead of 0.2 is specifies 0.002 millimetre then the manufacturing section has to select very good or precise machine tools like printing machine or lapping machine, that by the cost of the product will increase and if it is loose but it satisfy the required functional needs and then we can go for the some processes like find turning or some rough grinding. So it will reduce the cost of the work piece.

And then work piece tolerance will affect what is the process that is to be selected depending upon whether it is tight tolerance or loose tolerance, we have to we can select a whether turning is ok or finding is okay lapping is ok, so like that and it also fees of assembly they are very very tight then assembly time required will be more. If it is loose it can be very easily fitted so that the assembly time will reduce.

And what is the effect on the performance it will definitely affect as we find this particular case, so before the tolerance are assigned it is very very essential that the functional needs as well as the manufacturing aspects should be considered and then if we provider proper tolerance on the work pieces then it will we can have interchangeability concept and it parts replacement of part becomes very easy.

Since they are available or work pieces are available in the market which are produced using this fits and tolerances. So the in case the work parts fails we can easily get the market and then we can fit them and the maintenance of machine becomes very weakness and the parts become very easy.

(Refer Slide Time: 11:39)



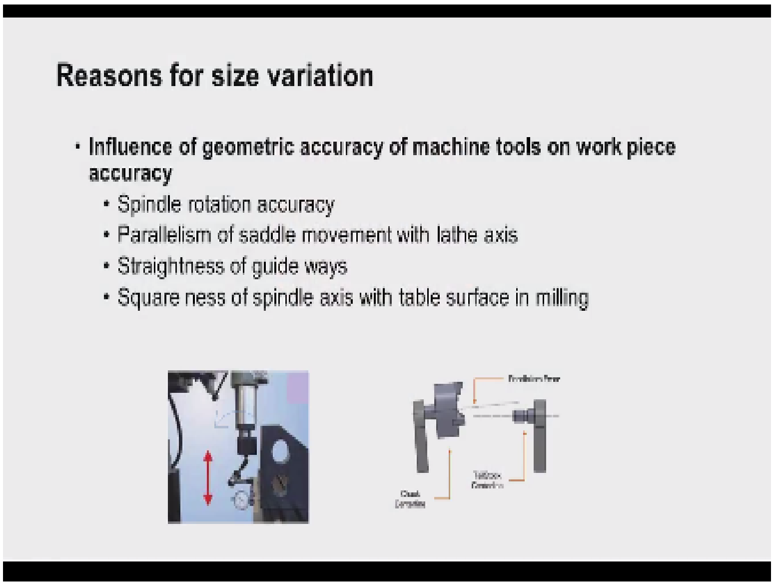
Now we learnt about the tolerances and now we will see why tolerances are specified, now we see this diagram we have the reproduction system where in they have various inputs like the human resource and then machine tools, cutting tools and then various measuring instruments etc. etc. the raw material, so these are the inputs and then finally all the inputs used in a proper way and then we get the output.

And on the production system always will be definitely the environment will affect, the ideal condition demand for parts without any kind of dimensional variation but in actual practice we know that it is a very impossible due to the following reasons. The reasons for variations in the properties of the material being machine for example the metallurgical property may vary from one place to another place in the raw material.

At some places it may be very soft and at some places it may be very hard and the thermal property of the work piece also affects the dimensions and then the production machines themselves will have some inherent inaccuracies built into them. So because of these inaccuracies the work piece is produced the size of work piece produce will vary and then it is impossible for operator to make perfect settings like the setting the cutting tool, setting the machine tools.

And then we make some mistakes while setting the tools because that inaccuracy in the work piece is introduced. Also the environment working environment will definitely affected manufacturing process the temperature, prevailing manufacturing site of the humidity because of this dimensional variations will occur, because of that the size of the work piece will vary.

(Refer Slide Time: 13:48)



Now there are there are some of the reasons why the tolerances are specified and we will discuss about some of these points in detail, now the influence of geometric accuracy of machine tools on work piece accuracy, the manufacturer can see this picture here the machine tool will have some inherent inaccuracy like the movement of the spindle,. It may not be perpendicular to the table .

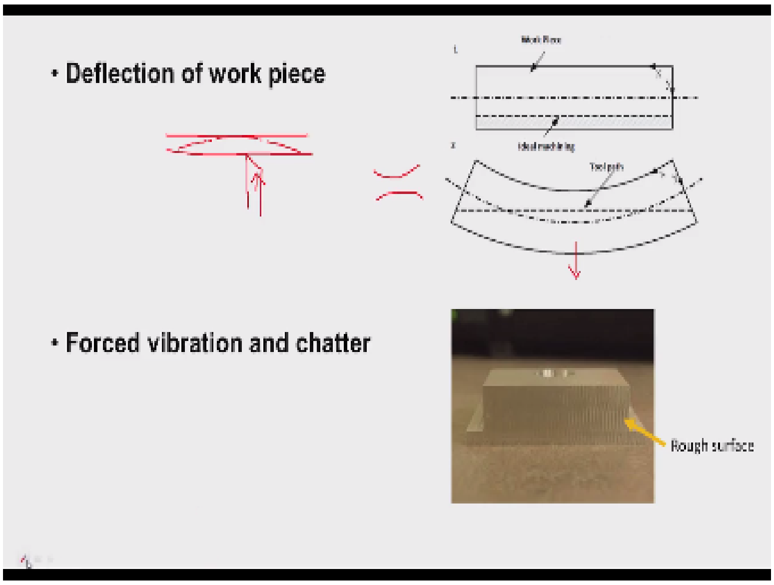
Example say this is the movement of the spindle and the work table machine pool table. Now this may not be perpendicular there will be some error may be few seconds or few degrees. Because of that the geometric of the work piece will be affected, also the moment of the spindle may not be perpendicular to the table, also maybe there may be there are some run out of the spindle is there.

And then the axis I can see in this diagram the axis of the spindle and then the axis of a data structure that not in line, there is some in line for very some inaccuracy because of this there will be some inaccuracy in the work pieces produce, we may get the drum shape of the work piece or may get the taper in work pieces which are unwanted. So spindle rotation accuracy plays a major role and then parallelism of saddle movement with the lathe axis.

For example so we have the saddle and we the guide way and then we have the spindle lathe spindle axis. So the moment of the saddle may not be parallel to the spindle axis because of that may get various kinds of inaccuracy in the work piece and dimensions also will vary and what about the straightness of the guide way, this blade may not be straight, so it may be having some inaccuracies like.

So because of that also the work piece will be affected and then square ness of spindle access to the table surface that is what we discussed in this particular background.

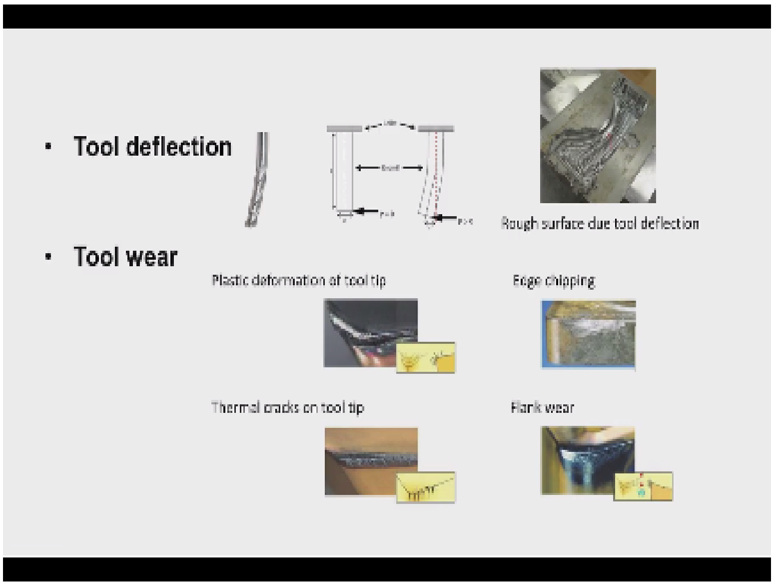
(Refer Slide Time: 16:21)



Then because of deflection the work piece also in accuracy in the work pieces will introduced. Now because of self weight of the work piece now we can see it has some sag is there in the work piece when this is the idle machining when there is no pending the work piece but actually when it bends like this and we take that cut, one piece is removed from the machine tool you may get shape like this.

Also because the cutting force the work piece may deflect so we have the thing work piece like this and when we force the when you press the cutting tool in this direction because of this force it may deflect like this. So then also we get some inaccuracies in work piece and the force vibration and chatter because of the tool chatter full vibration, the surface finish of this work piece is affected as shown here.

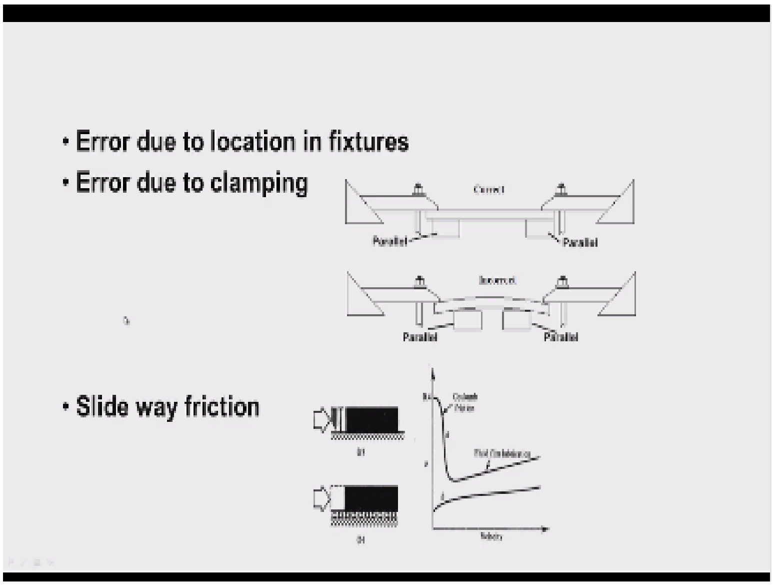
(Refer Slide Time: 17:39)



Now the tool deflection will also cause the geometry geometrical variations of work piece and dimensional variation of work piece. We can see here there is a force acting at the tip of the tool because of that tool is bent like this, so because of this we may get some rough surface dimensional changes as shown in this particular diagram and tool wear also will affect the work piece, the various kinds of tool wear, the flank wear or the edge of the cutting tool.

And then the plastic deformation the tool tip because of the high of gauges develop and then the thermal cracks development on the tool, all these factors will affect the dimensional variations or geometrical variations of a work piece.

(Refer Slide Time: 18:32)

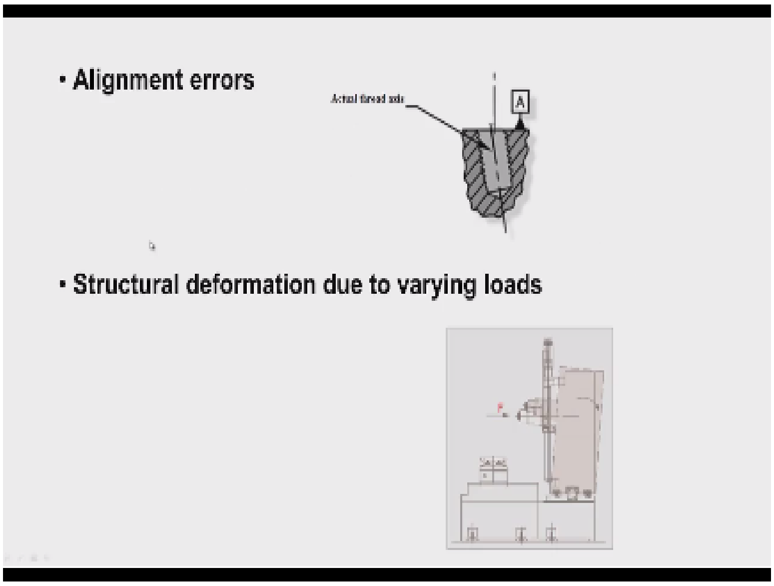


And then due to location in the pictures when the work piece are mounted in the machine sometimes the features are used may be the operator is not selected the proper fixture or may while putting the work piece in the fixture we may not them properly, so because of that the dimensional changes occur and then error due to clamping of the work pieces. Now you can see here we have the table machine tool table on which two parallel plates are placed here.

And then the work piece is placed, this is a work piece, now and then using clamps the work piece is clamped or sometimes the operate may place the parallel plate like this and then even the clamping force is applied the work piece will bend like this and now the cutter will cut the work piece like this and then once we can remove the work piece will bring back and we get inaccuracy of the work piece.

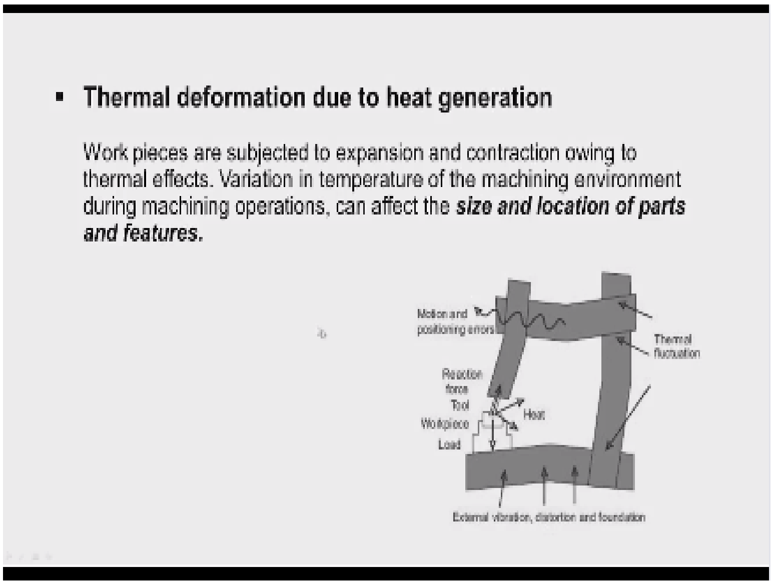
Also because of the slide way friction the movement of the tool will not be proper there will get motion because of that the surface finish may get affected or sometimes geometry is affected.

(Refer Slide Time: 20:03)



Now this is the error due to alignment error access cutting tool is not perpendicular to the surface because of that piece cutting forces acting on the structure the column of the machine tool because of that the column may deflect, so due to this also this work pieces size variation will occur.

(Refer Slide Time: 20:36)

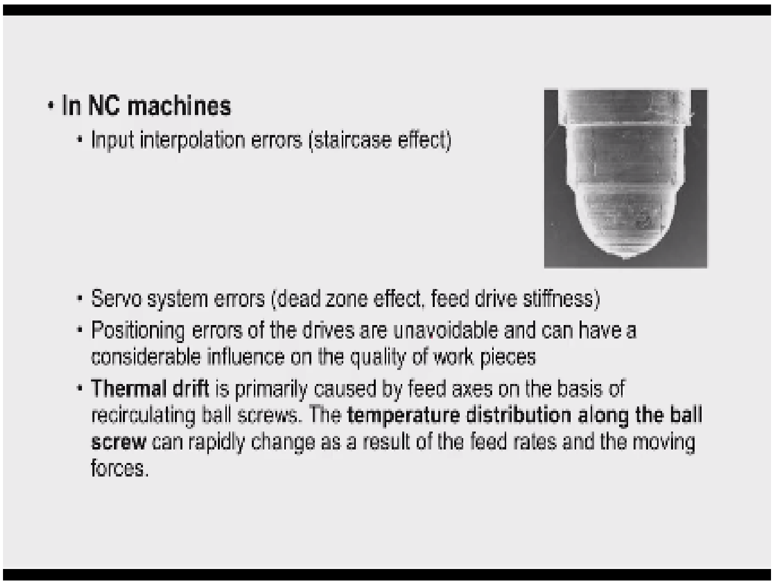


And then we have thermal deformation due to heat generation, now these work pieces there subjected to expansion and contraction or due to heat that is generated during manufacturing process, because variation in temperature this will be subjected it may to expand because the high the higher temperature. Now we can take example of work piece and then some milling operation is performed on the surface.

And then after that immediately the operator goes for drilling of two holes like this, now the because of milling the work piece in the heated condition and immediately goes for joining drilling operation. So I put the drill and there is some tolerance for the central distance. Now once the drilling is over the work piece is removed and then when it cools this because of contraction the centre distance may change.

So like this dimensional changes will occur, now also the vibration from other machine tools or other operators. For example next room may be there is a some pressing operation is going on, so because of that the vibration of tools, so the tool may vibrate and it will affect the geometry of the work piece and also the reaction forces acting on the various thermal of the machine tool or lead to deflection of the machine tool or the lead to deflection of the machine tool and the errors are introduced.

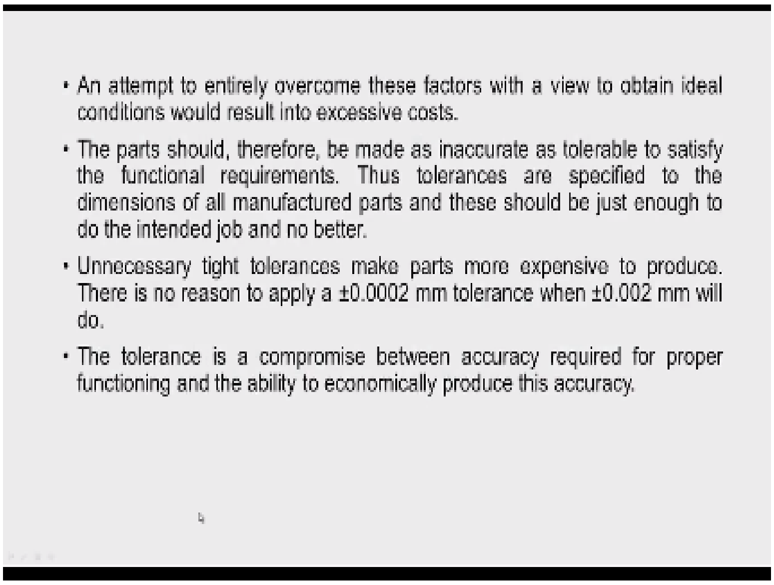
(Refer Slide Time: 22:40)



Now in case of NC machine the input interpolation error will be there because of that there will be staircase effect on the work piece like this, if we observe this picture here we can see there is some status effect ok, so now other reasons for dimensional changes is system errors like the dead zone effect and feed driver stiffness positioning errors of the drives and then that thermal drift is primarily caused by feed axes on the basis of re-circulating ball screw.

Because of the moment of ball screw the heat is generated and because of that the ball screw may expand or it may contract due to thermal changes. So that will because that also they will be changes in the dimensions of the work piece.

(Refer Slide Time: 23:36)

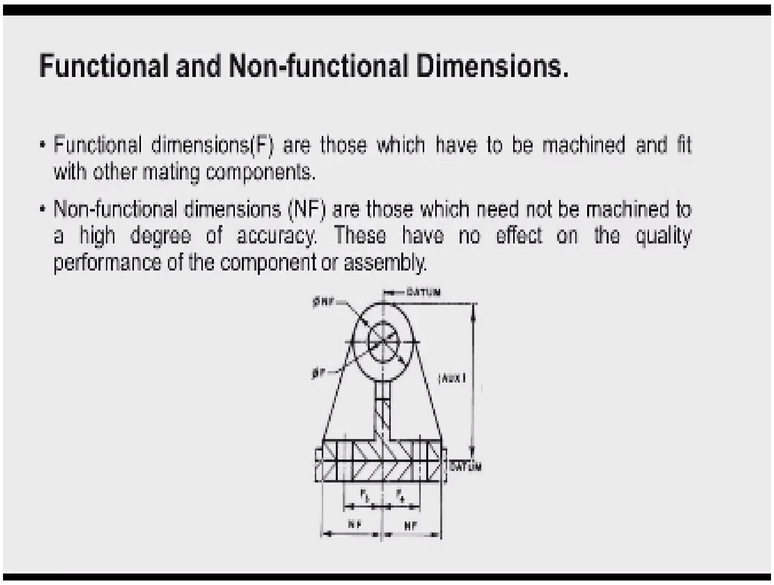


Now any attempt made to overcome the factors discuss or with a view to obtain ideal conditions that is ideal dimensions will lead to very excessive caused, so the parts should therefore be may has inaccurate as tolerable to satisfy the functional requirements. The tolerances are specified to the dimensions of all manufacture parts and these should just enough to do the intended the job.

Unnecessary type tolerances should not be specified say for example +/- 0.002 mm should not be specified when the point +/-0.002 mm will satisfy the need and then tolerance we should understand the tolerance is a compromise between accuracy required for proper functioning and ability to economically reduce this work piece to produce the accuracy.

We should always understand that while specifying the tolerance the need should be satisfied as well as we should consider the manufacturing aspect. So that the product can be produced at economical with economy.

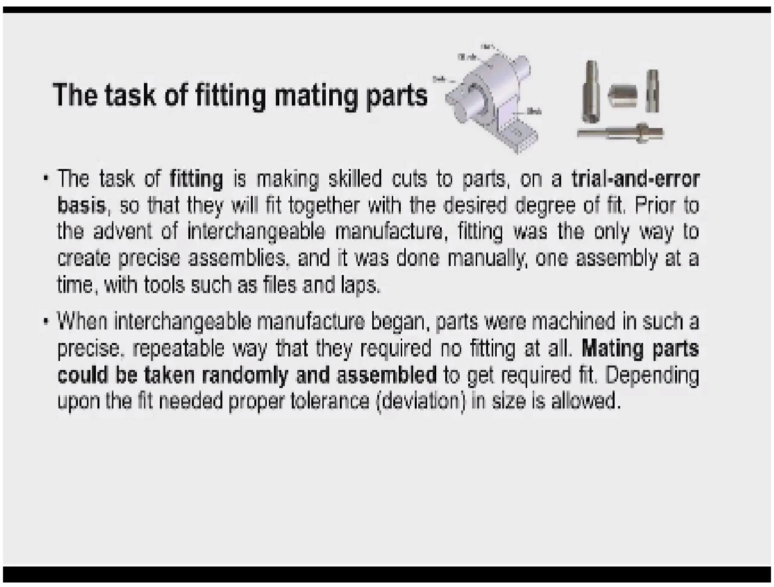
(Refer Slide Time: 25:01)



Now we should understand that when the dimensions of specified there are some functional dimensions and nonfunctional dimensions. If you see this diagram there are some dimensions like NF or non functional dimensions I can see if this is the outer surface of this particular third part where in NM will come in contact this place. So at this place when we not have to specify very stringent tolerance open tolerance will be alright.

In some places like here is functional dimension wherein we have a bore in which some shaft come and it may take a bow out my slide in the boat in such cases we have to specify the tolerances with taking lot of care.

(Refer Slide Time: 26:06)



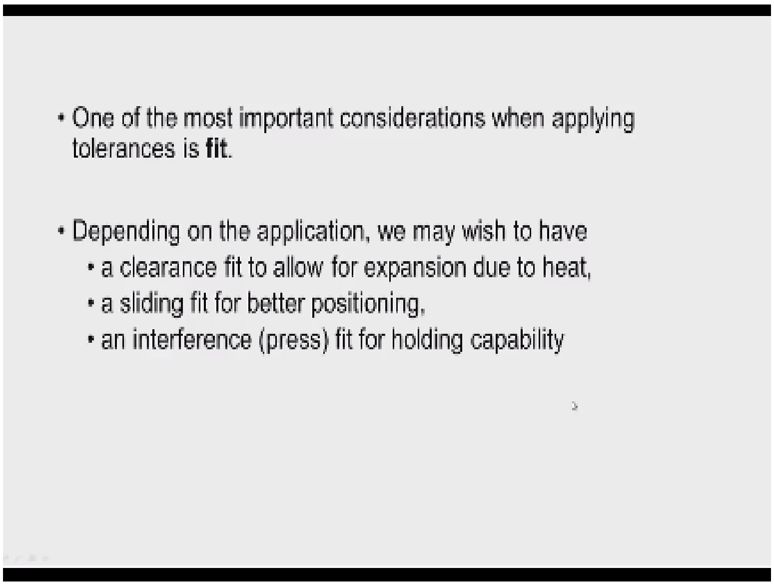
Now let us try to understand what is the meaning of fit and what is the task of fitting the meeting parts. So we can see here we have bore role, we have part varying there is a hole and then we have a bush and then there is a shaft we have to assemble all these three parts. Now we should understand that the bush will be in contact with the body of this particular bearing.

And there should not be any tension between push and the body, so there no motion is reading a rotation is required. So this to push and body should be held at night and then here they have a show of which will be rotating in the bush or maybe sliding in the bush, so there some clearance is required for that it be easily run for it slides. In such cases we carefully we should assign the tolerances the in olden days such kinds of work were done by the operators called fitters.

They use to take the shop and they use to take that is hole and they used to adjust the size by using scraping horror or file operations or some lap tools. So manually the the fitting of the jobs was very slow and it was situations apply for only of situations. Now with advent of the high speed machine tools and the interchangeable manufacture parts are machined in a very precise and repeatable way.

That they required no individual fitting, manual fitting on the parts, mating parts could be taken randomly and assembled to get any required fits, these possible by the use of proper tolerancing and by use of proper fits and tolerances, depending upon if it is needed proper toleration, proper deviation the size is allowed.

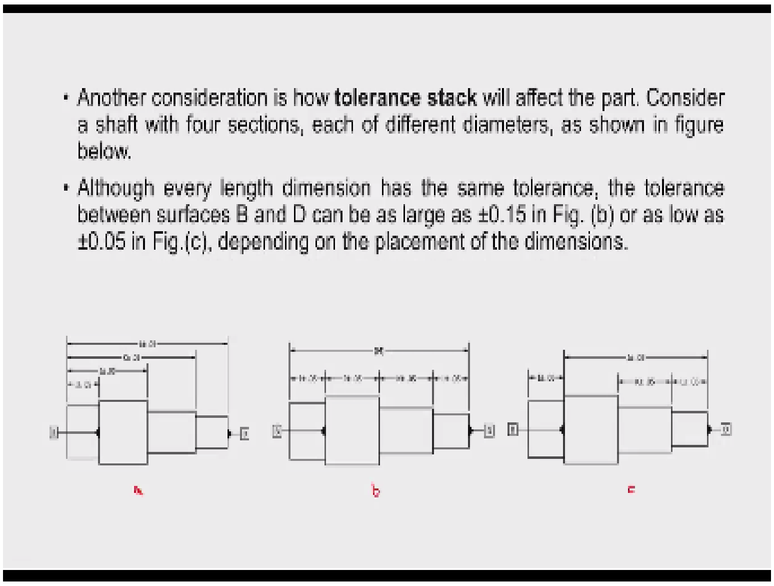
(Refer Slide Time: 28:25)



Now one of the most important consideration when applying tolerance is the fit, what is a kind of fit, I give the example of the bush shaft and the body, so between body and bush no moment is required, so there that if it is required where is a between this and bush some clearances is required. So that shaft moves easily. So the clearance fit is required. Now depending up on the application it sometimes clearance fir is also used to allow expansion due to heat.

Due to continuous usage of the work part, for example sliding shaft running the shop, there will be increasing the temperature because of that shaft may expand, and if sufficient space is not available it may get jammed in the hole. So to allow for expansion also some he is alone and then what is a sliding fit for better positioning is used and sometimes interference fit is used for holding the parts together.

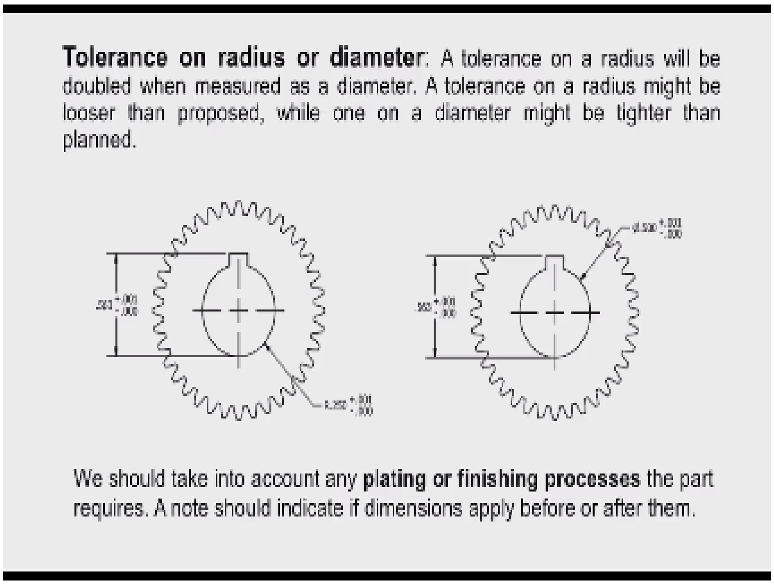
(Refer Slide Time: 29:40)



Now another consideration while allowing the as tolerance case, the tolerance stack now with a design engineer should understand what is the effect of this tolerance stack, we can see here we have three pictures wherein we have 4 circular features are there with varying diameter and they have the varying lens with some tolerances. Now if the tolerance is specified like this as shown in this diagram ok.

Now between the size between the part surface B and surface D okay the tolerance between surfaces B and D can be as large as a +/-0.15 in the specification is like this and the specification of tolerance is like this ok then the tolerance can be as low as a +/-0.05. So this the design engineer should understand and you should carefully assign the tolerances.

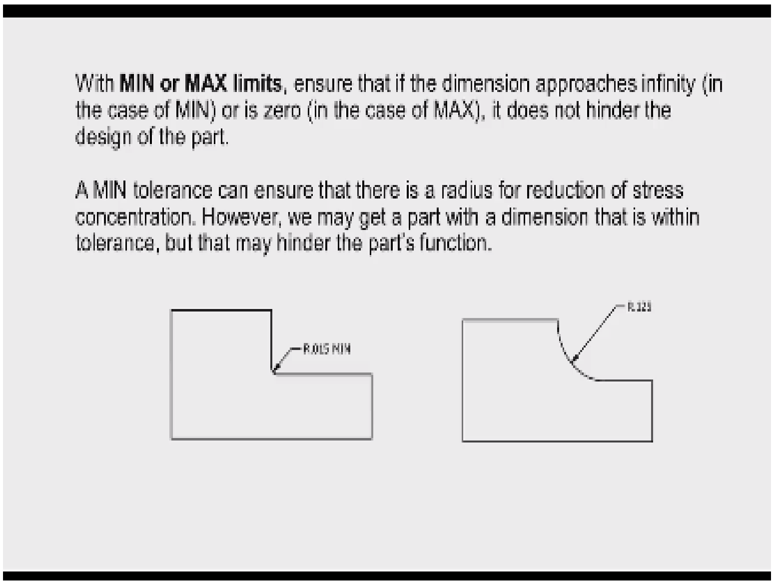
(Refer Slide Time: 30:56)



Now another important points to be considered a specified is the whether the tolerance is specified on radius or diameter. Now in this picture we can see the tolerance is provided on the radius, so 250+/- +0 001mm-0. Now in this diagram the tolerance is provided on the diameter that is 500+0.001 mm and the lower limit is 500 mm. So depending upon whether it is placed on radius or diameter the diameter will be vary.

The tolerance on a radius might be loose than propose while the tolerance on the diameter may be tighter than plan should be considered carefully and then whether any platting finishing operation is a specified on the surface. So whether plating is required what is the thickness of the plating that also should be specified weather the dimension is specified in the drawing is before plating or finishing operation or after finishing and plates.

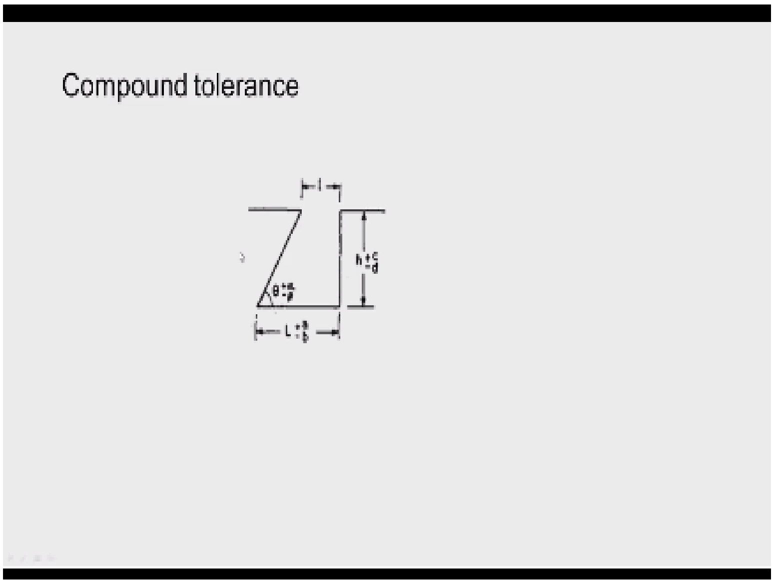
(Refer Slide Time: 32:13)



So such things one should consider and sometimes the minimum or maximum limit specified drawings we can see here we have a work part like this we have a flat here and the radius of the reflect 0, 0.015 minimum, now if we follow this the operator may produce a part with a radius of 0.125 millimetre. As per the specification this work this size is alone but whether it affects the functionality that we should consider.

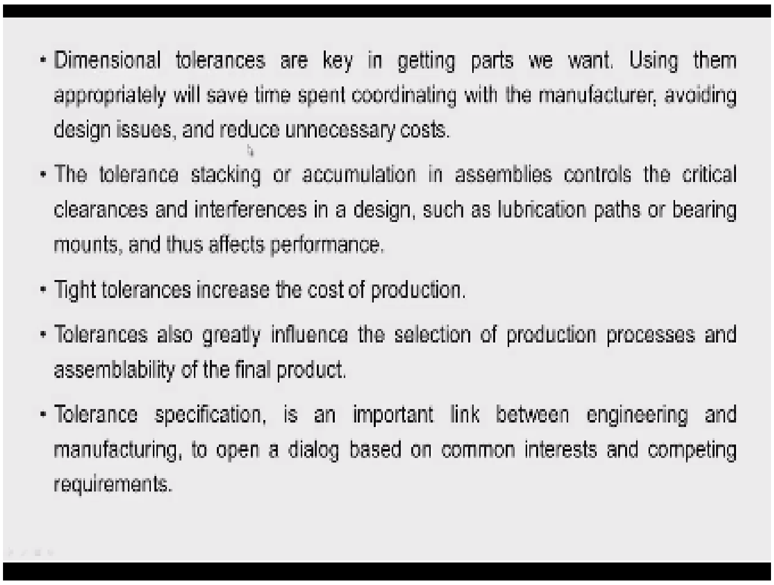
So we should be careful by specifying minimum or maximum limits, the functionality we should consider.

(Refer Slide Time: 33:06)



Now sometimes compound tolerances are used we can see here we have a work part with this particular shape wherein there is a tolerance for length at this particular place and there is tolerance for height and then there is tolerance for the angle also. Depend upon the actual size of this theta length and height is L will vary, so by giving always tolerance value should see the functional aspect of this length.

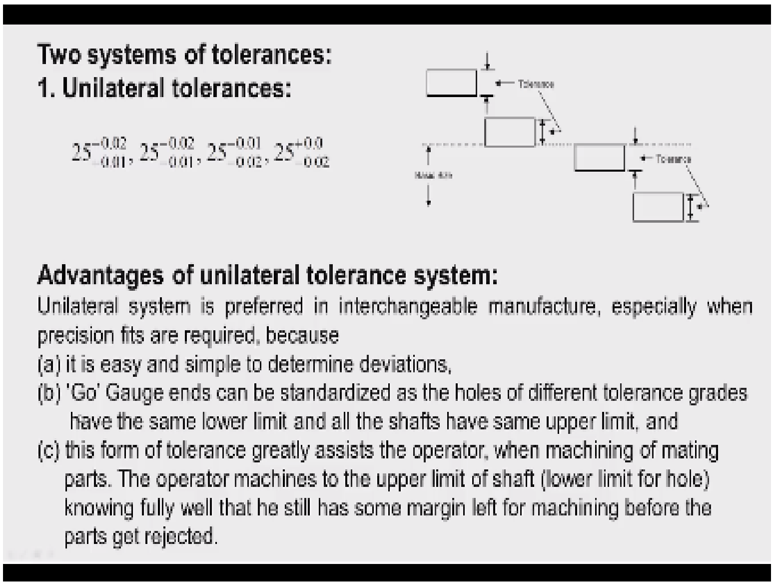
(Refer Slide Time: 33:43)



So the dimensional tolerances are key in getting parts what we required and using them appropriately will save time spent coordinating with manufacturer and avoiding the designe issues and reducing unnecessary costs and tolerance stacking or recommendation assemblies they control the critical clearance and the interference in a design such as lubrication pass or bearing mounts and does the performance is affected.

And we should understand that tight tolerances will always increase the cost of the production and they will also influence the selection of production processes and the assemblability of the final product will also be affected. So the all tolerance specification which important link between the engineering and manufacturing departments and to open a dialogue based on common interest and competing requirements.

(Refer Slide Time: 34:46)



So far the studied about what is the tolerance and what is the fit and why tolerances are specified and what is the need of fit and those things now let us try to understand what are the eight systems of tolerance there are two systems of tolerances one is called unilateral tolerance and the other one is called bilateral tolerance. Now we can study this picture wherein we have this is the basic size of the component, basics size of the hole or basic size of the shaft.

And in the case of unilateral tolerance it is specified on one side of the basic size, for example 25+0.02 mm 25+0.01 mm, now we can see here this is tolerance band, so this is a tolerance band or it is also known as tolerance zone ok. So on one side of the basic size the tolerance is specified. This is another example now here there is no gap between basic size and the lower size of the part.

Here there is some gap between the basic size and the lower size, say we have a hold like this ok, and this is the hole and then this is the layer limit of the hole and this becomes the upper limit of the hole and this is the tolerance that is alone. So the size of the whole can be anywhere between this level and this level so it can any size we can have. Now similarly we can allocate the tolerance.

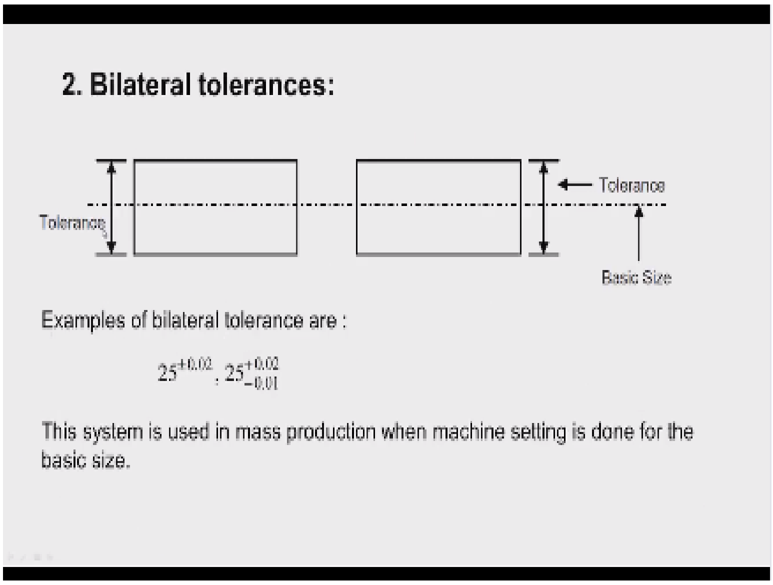
And on the other side the basics size here it is plus side of the basic size and here it is + side of the basic size and here it is – side of the basic size. For example 25-0.02 to 25-0.0. This is the limit of the work part, ok so only one side of the basic size the tolerance is allowed, our example is 25, so this is lower limit is 25, so this is the lower or upper limit of the if we take this as a hole the upper size of the hole is 25.

And then the tolerance is lower limit will become 25-0.02 mm, so this is the lower size of the hole ok. Now what are the advantages of unilateral tolerance system, it is easy and simple to determine the deviations and go gate ends can be standardized as the holes of different tolerance grace are the same lower limit and all the shafts have same upper limit. So what is this tolerance grade will understand after sometime.

And now this type of tolerance are greatly assist the operator when machining of mating parts, the operator machines to the upper limit of the shaft or lower limit of the hole knowing fully well that he still has some margin left for mentioning before the parts get rejected. So for example we have a hole that to be drilled, a hole is to be drilled like this. This is the lower limit of the hole and upper limit of the hole ok.

They are the basic size say it is this the basic size is coinciding with the lower limit basic size, now it takes a drill tool which is corresponding to this basic size or lower limit and start drilling and then there is lot of material left, so it size may increase because vibration with their size increase but still the work piece can be accepted, the reason is the upper limit is up to this upper limit of hole is up to this.

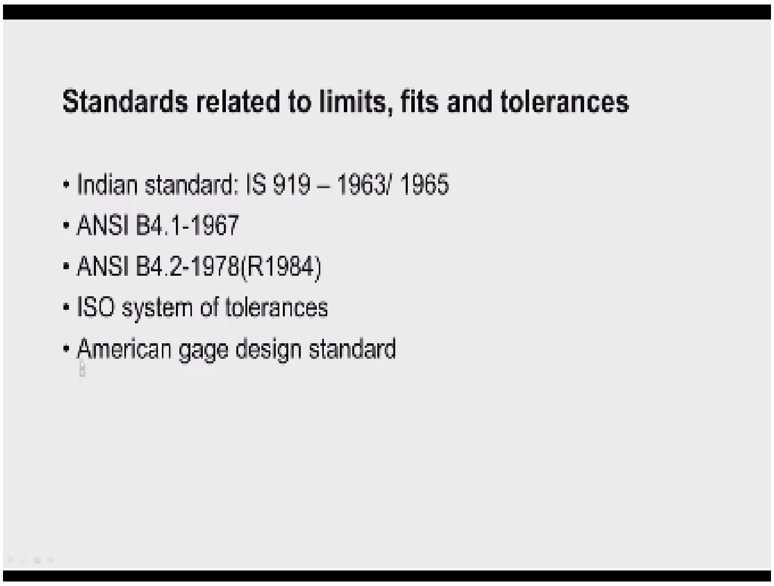
(Refer Slide Time: 39:48)



Now the second type of tolerancing is bilateral tolerance where in we have this basic size and tolerances is tolerance is provided on both side with the basic, so some example we can see here 25 is a basic size and the upper limit of the shaft is 25.02 mm and lower limit of the shaft is 24.98. Now the basic size is in between these two and deviation is allowed basic size is 25 millimeter, so the deviation is allowed on both sides of the basic size.

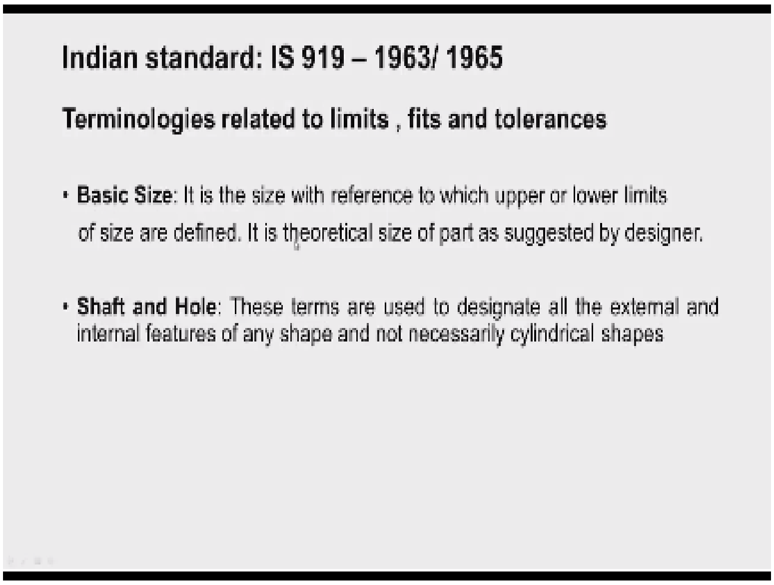
So that is bilateral tolerance, now other example is 25+0.02 mm and 25-0.01 mm, so 25 mm is a basic size and then this side you can go up to 0.02 mm and on the negative side can work 0.01 mm. So this system is used in the mass production when the machine setting is done for the basic science.

(Refer Slide Time: 41:42)



Now let us learn about the various standards available which are related to limits fits and tolerances. Now we have Indian standard IS 919 and American system ANSI B4.1 and ANSI B4.2 which deals with metric units and then we have a ISO system of tolerances all these standard they are followed ISO system of tolerances and also we have American gauge design standard.

(Refer Slide Time: 42:23)

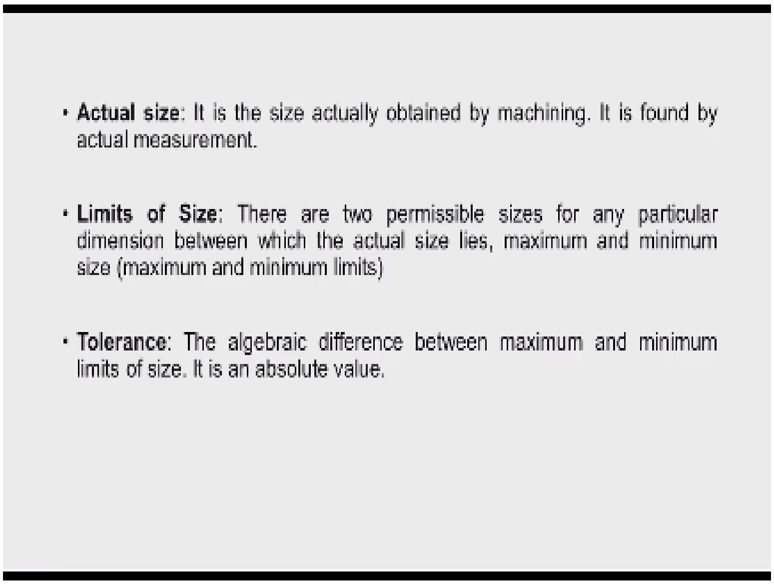


Now let us learn about some terminologies are related to limits fits and tolerances as per Indian standard 919. Now we should understand what is the meaning of basic size, it is the size with reference to which upper and lower limits of the size are define. It is theoretical size of the part as suggested by the designer. So the design engineer will make some calculations and finally he will arrive at some size for the shops say 25 millimeter.

So this is the basic size as designed by the engineer. Similarly for the hole the engineer may arrive some theoretical size, so the designed size is known as basic size and then we have terms shaft and hole are these terms are used to designate all the external and internal features of any shape and not necessarily a cylinder shapes. For example the hole means they internal features.

For example we have a hole here, so this feature this is called hole need not be a cylindrical shape, if you have a shape like this square hole or rectangular hole or a triangular hole. So that is also called a hole irrespective of the shape. It basically represent the internal features. Similarly external features example we have cylinder like this, this external feature diameter, outside feature is known as shaft, it can be round shaft or it can be a square shaft, or rectangle shaft or triangle shaft.

(Refer Slide Time: 44:49)

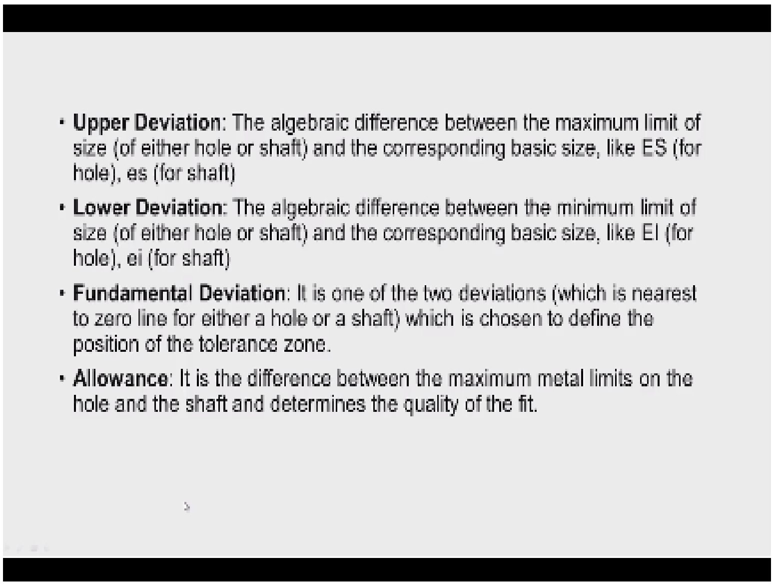


All the external features are known as shafts, now what is the meaning of actual size, it is the size actually of obtain after machining, it is found by actual measurement. For example so we have produced some hole by using drilling operation, see the design size are basic size maybe 25 but after producing what is the actual size, so which is measured by using some instrument like maybe the micrometre internal micrometre or Vernier caliper.

So the actual size maybe 25.02 mm, so this is the actual size and this is the basic size and then we have a limit of the size, there are two permissible sizes for any particular dimension between which the actual size lays, that is maximum and minimum size of the maximum limits and minimum limit, so if we take the example of the shaft, so the shaft, now this shaft the basic size is 25 mm as listed by the design engineer.

And also he gives some tolerance it maybe 25.01 mm or it can go below 25 by 24.99 mm. So this is called the upper size or maximum size of a shaft and this is the minimum size of a shaft and then we have the tolerance the algebraic difference between maximum and minimum limits of size is known as the tolerance and it is an absolute value. For example in this case the difference between the maximum limit of the maximum size and minimum size is 0.02 mm.

(Refer Slide Time: 47:44)



So this is the tolerance for this particular case, now let us conclude this session, in this session we studied about the basics of limits fits and tolerances, what is the meaning of tolerance, what is the meaning of fit, why the fit and tolerance are required and what are the various types of assigning the tolerances like bilateral tolerance, unilateral tolerance such things we study. We will stop at this stage, in the next class we will continue the discussion, thank you.