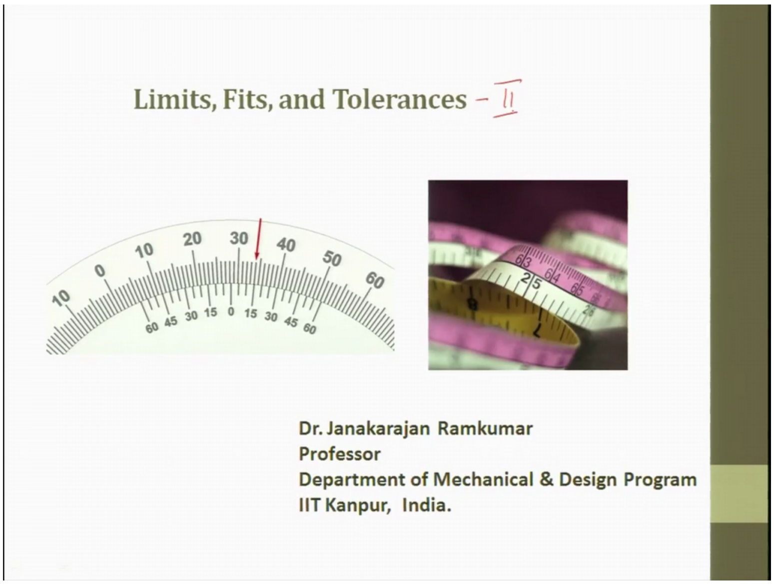
Engineering Metrology

Prof. J. Ramkumar
Dr. Amandeep Singh Oberoi
Department of Mechanical Engineering and Design Programme
Department of Industrial & Production Engineering
Indian Institute of Technology, Kanpur
Institute of Technology, Jalandhar

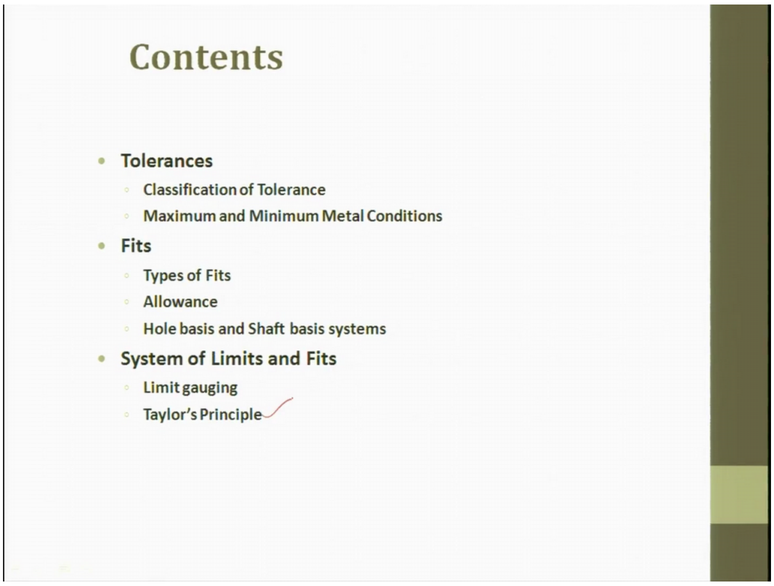
Lecture – 06
Limits, Fits, and Tolerances (Part 2 of 4)

Welcome to the lecture on Limits, Fits and Tolerance this is 2, part 2 ok.

(Refer Slide Time: 00:18)



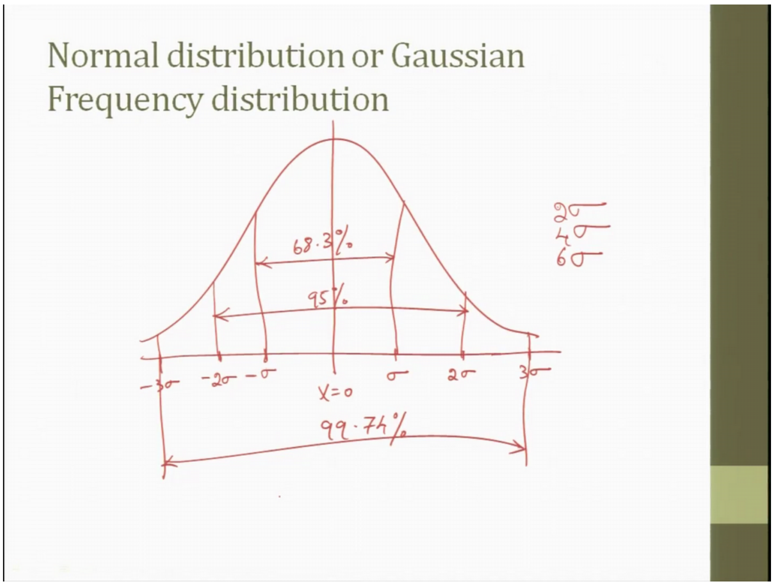
(Refer Slide Time: 00:23)



So, the content which will be covered in this lecture is going to be classification of tolerance. Then we will see minimum metal condition I started in the last lecture that for a shaft what will be the maximum metal condition? And for a hole what will be the material maximum material condition? Then we will discuss about fits because all these things I had talked about single component now when we try to assemble these 2 they become fits.

So, we will see the fit type of fits then we will see allowance, then there are 2 systems which are most commonly used one is hole based system and other one is shaft based system we will see. In detail hole based system shaft based system is not much used for technical reasons, we will see that and last we will see the system on limits and fits. Then we will see about limiting gauging and Taylor's principle this is very very important as far as gauge design is concerned this is very very important ok.

(Refer Slide Time: 01:22)



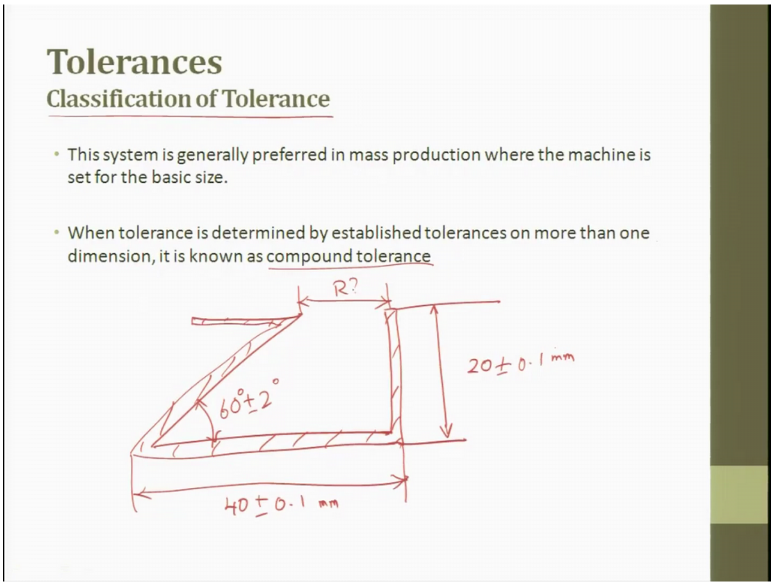
So, we have been talking in the last lecture more about the interchangeability concept and you might have also listened to lot of companies which are giving add saying that we follow 6 sigma, we follow 12 sigma. So, what is this 6 sigma and 12 sigma? And second thing is for any event to happen we always keep saying any event to happen it always follow the binomial distribution. So, what is that? So, this is how it looks like this, is X equal to 0.

So, I will make. So, this is minus sigma, this is minus 2 sigma, and this is minus 3 sigma, this is sigma 2 sigma, and 3 sigma. So, if you look at this is a typical binomial distribution, or a Gaussian frequency distribution curve. So, in this curve you can see this the when we somebody talks about this is the mean, and these are sigma, sigma is nothing, but a standard deviation. I take one sigma here minus 1 sigma, the this is called as 2 sigma, the gap is 2 sigma 68 percent 68.3 percent of the even to happen or the material which you produce or a shaft which you produce falls in this category.

When we talk about 4 sigma level it is minus 2 1 plus 2 here 95 percent of the occurrence happen and when we talk about 6 sigma you can see that 99.74 percentage of the part which are produced by a machine or a event to happen falls under plus sigma and a minus sigma and plus sigma. So, it can be for the 2 sigma, 4 sigma and 6 sigma. Many a time’s people use to talk about 6 sigma and they say the process followed in my company follow 6 sigma.

So, when somebody says 6 sigma either is 99.74 percent of the part which are produced are within the expectation limits or are within the limits of the drawings so, it is accepted. So, this is the brain child or this is one of the lead toward interchangeability concept.

(Refer Slide Time: 04:29)



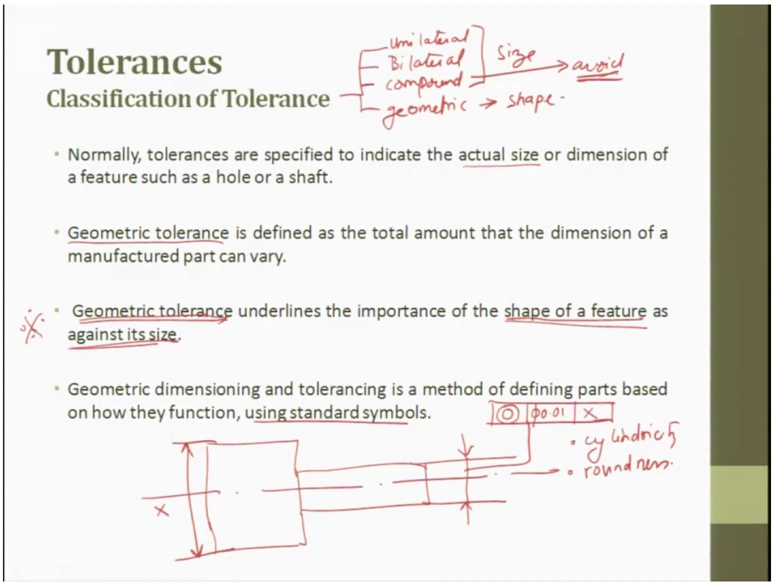
Going back to the lecture like classification of tolerances we saw unilateral tolerance, then we saw bilateral tolerance. So, now, we will see a a compound tolerance compound tolerance is nothing, but this system is generally preferred in mass production where the machine is set for a basic size we follow compound tolerance. So, when compound tolerance are determined by established tolerance on more than one dimension, it is called as compound tolerance.

So, let us take a simple figure or a part for compound tolerance ok. So, here is a part where there is let us take it is a sheet metal part assume it is a sheet metal part. So, sheet metal part, so this is 20 plus or minus 0.1 limit of and here it is theta plus or minus 2 degree, this length is 40 plus or minus 0.1 millimeter and now this is R ok.

So, we have suppose to find out this R. So, you can see if you go back this is a curve this is a part where there it follows compound tolerance, but to a large extent we should try to avoid compound tolerance because it is very difficult to first of all give the tolerance and second thing also to make sure that the compound produced is co is falling under this tolerance.

So, when tolerance is determined by established tolerances on more than one dimension it is known as compound tolerance. So, here there is a dimension of 40, here there is a dimension of 20, and here there is a height which has to be maintained with a 20 plus or minus 2 millimeters. So, we have to find out R. So, this is a compound tolerance.

(Refer Slide Time: 06:54)



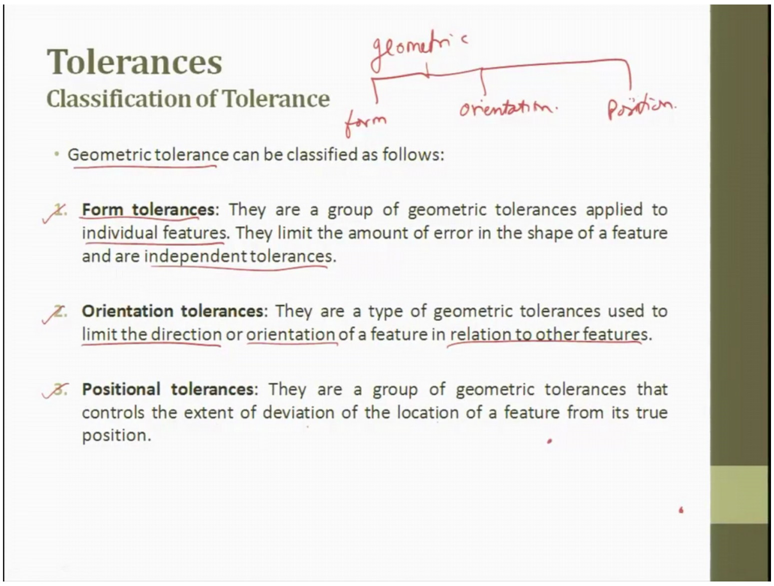
So, normally the tolerance are specified to indicate the actual size or dimensions of a feature such as a hole or a shaft. So, the next tolerance what you are going to see is a geometric tolerance. So, what is geometric tolerance? So, geometric tolerance are basically is this basically defined as the total amount that the dimension of a manufacture part can vary. The geometric tolerance underlines the importance of shape of a feature and not the size as against the size; this is a very important point to be noted.

So, geometric tolerance underlines the importance of shape for example, if I try to take a shaft, I try to draw the this is how we try to give the geometric tolerance. So, geometric tolerance is generally given it talks about cylindricity, it talks about roundness; roundness of this shaft with respect to with respect to the diameter of something that.

So, so here geometric tolerance is more talked about in terms of shape of a feature as against the size. The geometric dimensioning and the tolerance is a method of defining part based on how they function using standard symbols so this is a standard symbol.

So, till now we saw 4 different unilateral, then we saw bilateral. We saw compound and now I have introduced you geometric. Geometric is more of shape, these three are more of size and to a large extent we should avoid because it is very difficult from the manufacturing and from the calculation point of view.

(Refer Slide Time: 09:51)



So, the geometric tolerance can be further classified into 3 parts. So, they are one it is called as form, another one is called as orientation, and the third one is called as positional. Form they are a group of geometric tolerance applied to individual feature, these are 2 features right. They limit the amount of error in shape of the feature and are independent of tolerance that is a dimension tolerance right.

So, here form tolerance is more talked about in terms of individual feature, orientation tolerance they are a type of geometric tolerance used to limit the direction; the direction or the orientation of a feature in relation to the other feature. So, here orientation tolerance is always you try to talk about the direction with which it has to be inserted or it may it has to be manufacture with reference to another feature. This is a very important point with respect to another feature this is called as orientation feature tolerance.

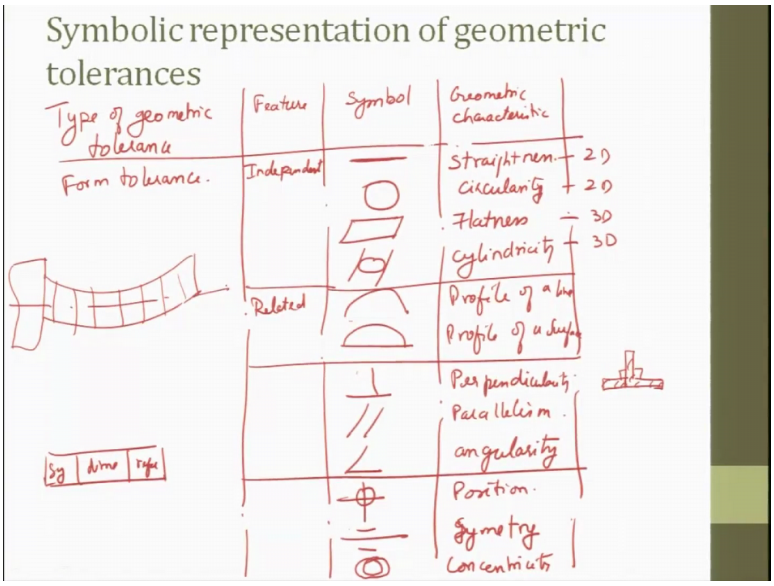
And the last one is called as positional tolerance, positional tolerance they are a group of tolerance of geometric. A group of geometric tolerance that controls the extend of deviation of the location of a feature from it is true position. See position tolerance is something like this I you use GPS, and then you try to say I am starting from a this is a start destination and I am travelling here right.

So, what you say you say? This is my start destination what is your end destination. So, you say the start destination with end destination with respect to start destination. So, every time when you try to talk about how much distance to travel with respect to start with respect to that reference we are saying. So, that is positional tolerance.

So, if you want to talk about this orientation tolerance and position tolerance these two along with form, these three play a very very important role when you do assembly. When you try to produce a part for interchangeability point of view we talk about unilateral, we talk about bilateral, we talk about compound, but moment geometry comes into existence it is more focused toward assembly.

So, under geometry under geometric tolerance you have 3 classifications, one is form; form in terms of shape, then it is orientation, then the last one is going to be position with respect to some datum, with respect to some datum you are trying to make a feature. So, that is position.

(Refer Slide Time: 12:44)



So, next we will be symbolic representation of geometric tolerance. So, let me write down the type of tolerance, type of geometric tolerance. The next one is going to be a feature the next one is going to be the symbol ok. So, let us talk about form tolerance.

So, here we are going to put the symbol so, this is so feature. So, feature whether it is independent or dependent on the other. So, it is independent features.

The geometric characteristics are it is called it is called straight necessity raight straightness right. So, the next one is going to be this is going to be circular. So, it is called as circularity. So, these are the symbols this you can see the drawings you will see those, such types of symbols and you see such types of and these are the meaning for it, so then it is flatness.

So, this is nothing, but flatness symbol then you will see cylindricity; cylindricity is nothing but this; this is cylindricity. Circularity and straightness is 2 D parameter, flatness is 3 D parameter, cylindricity is 3 D parameter. Why is that 2 D and 3 D? For example, if you want to construct a cylinder I try to take several sections of circles, several slices of circles I reconstruct I get it straightness is also the same.

If I try to take along x y plane several lines and if I patch up together I reconstruct them I get a flatness. Then the next one is going to be which is related, so which is related you will see that. So, here it is going to be this is profile which is profile, then it is profile of a surface, this is profile of a line, this is profile of a surface. You see you can use this symbol and you can start correlating with 2 D and 3 D parameters ok.

So, now I will next is orientation, when we were talking about. So, this is perpendicularity. Can you think of an example where this is very important wherever you have something like an L angle, you wanted to measure or if there is a channel like this you want to measure its 90 degree there might be some requirement.

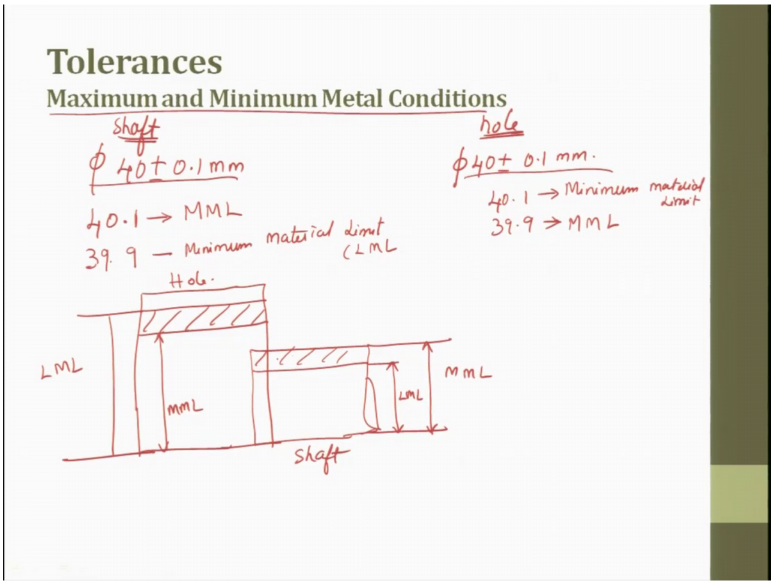
So, then we talk about what is the perpendicularity of it? So, then we also talk about parallelism, then angularity, then we talk about position. So, if you go back and see we drew this right. So, here we talk about some we talk about a symbol here then we talk about some dimensions here whatever it is and then we talked about this is with respect to some reference or something like that.

So, this 3 row we will have 3 columns. So, this we will try to tell they will put the symbol here then they right to say what is the dimension or deviation and then they also talk about this. So, you can have position, you can have also symmetricity this is nothing, but symmetric. Then you can have concentricity 2 circles this is concentricity. So, these are some of the symbols which are used these are some of the symbols which are used and all the symbols are available readymade in several of the hand books.

So, what you have to understand is looking at these symbols. Why are these symbols important? Because you are talking about geometric tolerance why is this important? Geometric tolerance is important because till now what we are talking about is only dimension, but geometry also plays a role.

For example, if I try to take a chart and if I try to measure the diameter of the shaft, at every diameter of a shaft at every section plane you will see it will try to fall under it will it will fall under the dimensions whatever it is. But if the shaft is something like this, all the dimensions whatever you measure at several planes yes perfect, but with respect to his axis it is gone. So, when you have to make sure such assembly such components have to go for assembly then comes the importance of the geometric tolerance. Under geometric tolerance we had three; one if form, another one is orientation, and position. So, all the 3 are important. So, position we will always try to talk with respect to a datum.

(Refer Slide Time: 19:26)

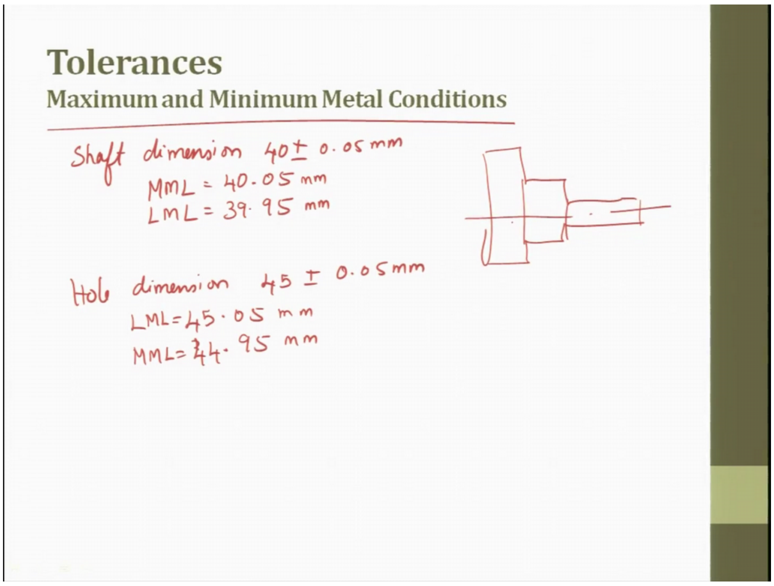


Now, let us move to the next topic which is maximum and minimum material condition ok. So, let us take a diameter. Let us take a shaft, diameter 40 plus or minus 0.1 millimeter, let us take a hole diameter 40 plus or minus 0.1 millimeter. So, what are the two possibilities? It can be 40.1 and this can be 39.9. So, this is called the maximum material limit for a shaft and this is called as a minimum material limit ok.

So, let us take a hole, so hole I get 40.1 and this is 39.9. So, when I say the hole size is 39.9, it is called maximum material condition and when it is 40.1 it is called as minimum material limit ok. So, here let me try to drop us this is the hole, and this is a shaft this is a shaft ok, so the hole ok. So, this is the, I will now change and represent this minimum as L M L right L M L condition and this is maximum material limit for a hole right and when I talk about the shaft this is called the M M L and this is called the L M L ok.

So, now, you see that the shaft and the hole maximum material conditions are different. So, why is this very important? This is very important because you are trying to fit a shaft inside a hole. So, now, when you talk about assembly it is all about fits. So, till now what we were talking about is limits. So, we were talking about tolerances and now we are talking a new terminology which is fit and by the way what here we are talking about is only dimensional here you will also have form tolerances all these things coming. So, that makes life little more complex. So, this is maximum material condition and this is minimum material condition.

(Refer Slide Time: 23:17)

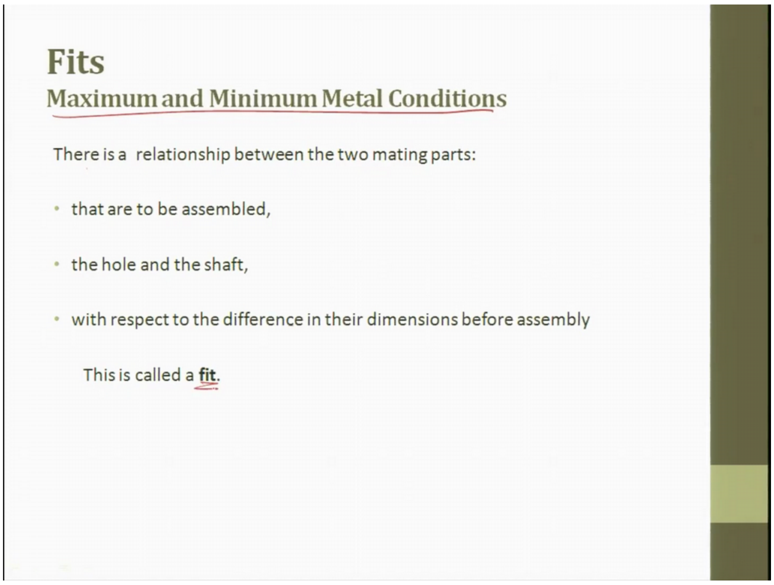


So, from this knowledge what we will try to go? We will try to go towards fits, but before going into fits let us try to do a simple problem and see what is how do we talk about this maximum material condition and minimum material condition. So, I will try to take a once a one more small example a shaft dimension is nothing, but 40 plus or minus 0.05 millimeter ok.

So, and the hole dimensions is going to be 45 plus or minus 0.05 millimeter. So, what will be the maximum material limit and least material limit for a shaft? So, it will be 40.05, and this will be 39.95 all these things are millimeters. Say when we take about the hole it will be 45.05 and it will be 44.95. So, here this will be the L M L and this will be M M L.

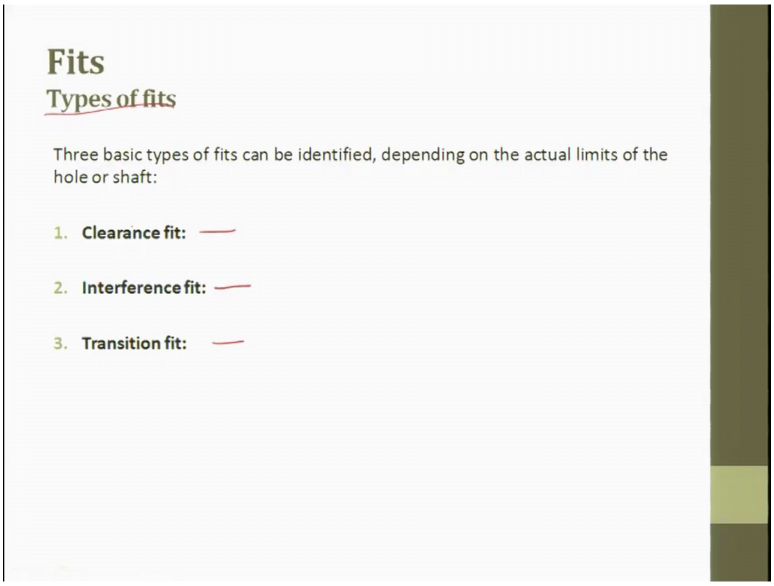
So, you should understand the difference between these two this is very important because later when we start working in the fixed domain the problem whatever we will see or the problem solving questions will be what is the difference between this fellow and this fellow such that you will decide what is the fit? So, only for that we are more always focused about these things and here it is very simple, but in real time application you can also have a step shaft; you can also have a step shaft. So, you will have 2 dimension and you will have a corresponding hole. So, all these things will make little more problem a complex.

(Refer Slide Time: 25:50)



So, in order to understand fits we have to we have already learnt what is the maximum and minimum material condition? So, now, let us move towards the next topic which is called as fits. So, there is a relationship between two mating parts that are to be assembled it. For example, you take a hole and a shaft with respect to the difference in the dimension before assembly we always try to call this a fits.

(Refer Slide Time: 26:20)

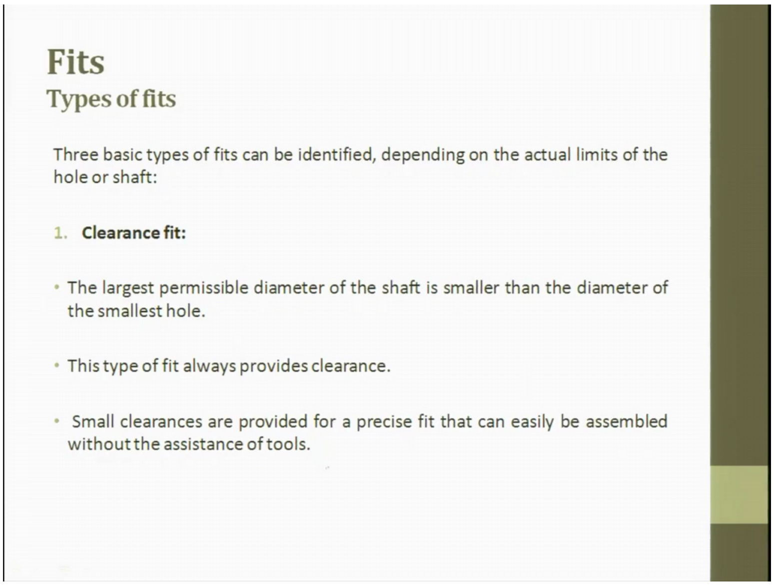


So, fits there are 3 different types of fits, one is called as clearance fit, the other one is called as interference fit, the last one is called as transition fit. Just go by English meaning clearance; clearance means the hole should be large, the shaft should be small the shaft should be easily sliding inside the hole. So, naturally the hole will be larger than the shaft, so the hole minimum size should be larger than the maximum size of the shaft.

So, where do we do? Wherever you want to have loose fit assembly or wherever you wanted to have just a sliding pin to just locate. So, then we always go for this clearance fit interference. Interference means where the shaft size is larger and the hole size is smaller, the shaft is interference or it is hindered before pushing it into the hole. So, that is called as interference fit I am just talking only about the English meaning.

And then transition is I have both the sizes equal just by a small push the pin gets locked into the hole. Transition means it gets shifted from clearance to interference or from interference to clearance. So, it is transition stage a middle stage, so this is the third one which are there. So, fits are very important these fits have a link to the previous condition maximum material and minimum material what we saw ok. And before that what is our tolerances? The tolerance maximum material minimum material this leads to fit. Now, I think you will be able to appreciate why are these fits more talked about when in manufacturing and also in metrology?

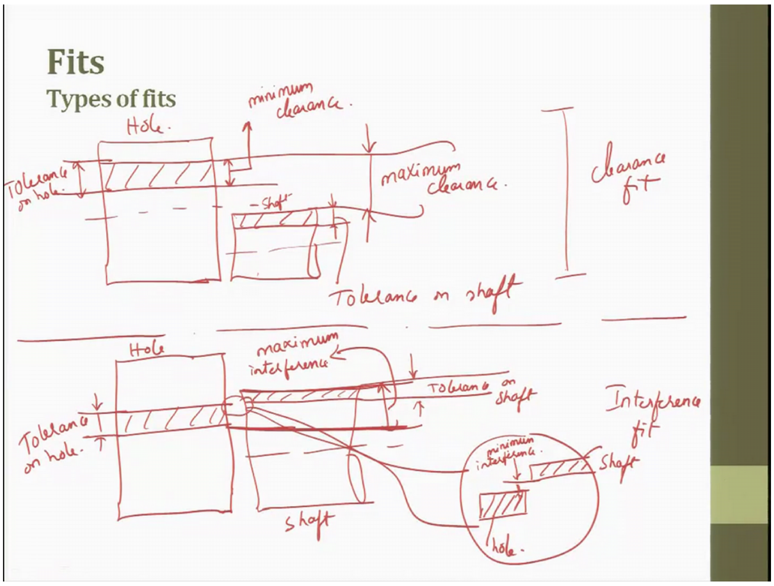
(Refer Slide Time: 28:22)



So, what is a clearance fit? So, the three basic fits can be identified depending upon the actual limit of the hole or the shaft. Clearance fit, the largest permissible diameter of the shaft is smaller than the diameter of the smallest hole which I told you earlier it is very clear that the hole size is larger and the shaft size is smaller ok.

So, this type of fit always provides clearance. So, wherever you want to do reassembly we always go assemble, disassemble, frequently we always go for this. Small clearance are provided for a précised fit that can be easily assembled without the assistance of any tool.

(Refer Slide Time: 29:20)



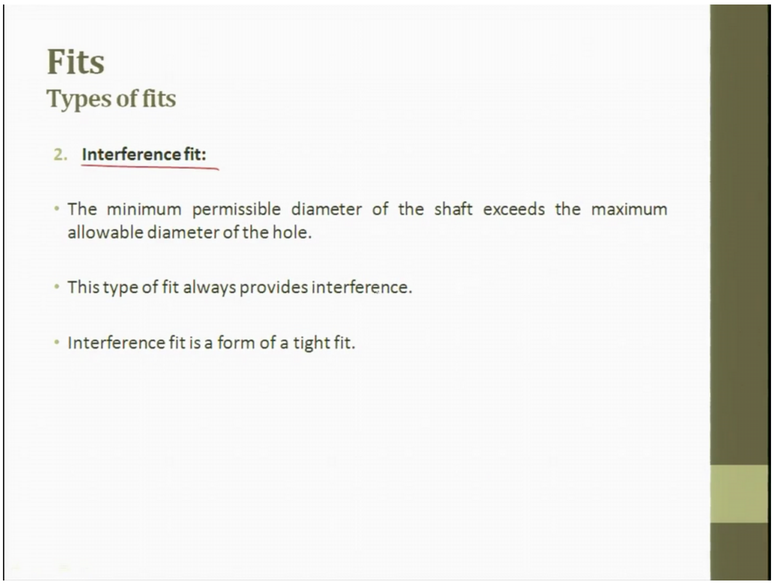
So, let us try to draw it in a schematic diagram this is a hole, this is a shaft. So, this is called as the minimum clearance ok. This is called as the maximum clearance right and this is a tolerance on hole, this is a tolerance on shaft ok. This is for, this entire thing is for clearance fit right; this is for clearance fit. The next fit what we were discussing is about interference fit. So, let me draw a line, you will study about interference fit I said interference something is hindering interfering. So, here let us try to take a dimension this is the hole, this is a tolerance ok, this is a shaft ok.

So, this is the tolerance or hole, this is a tolerance on shaft ok. So, if you see here so let me make it little bit so I will try to zoom this portion, so this is a zoomed here ok. So, I am trying to zoom it here. So, assume this is the hole this is a shaft this is hole tolerance and this is shaft tolerance. So, the distance between these two, the distance between these two is called as a minimum interference clear.

So, this is a minimum clearance and this is a minimum interference and this is a minimum interference and what is a maximum interference is between this and this; between this and this whatever it is. So, this will be the maximum interference sorry in the textual matter which is put in the course you will try to have a better picture for in my sketch I do admit my sketch is poor, so please pardon me.

So, but I think the concept is very clear for you. So, this is clearance width this is interference width you see whole shaft lose. Now you see shaft is tightly push. So, you have to apply lot of pressure to push the hole inside the shaft inside the hole.

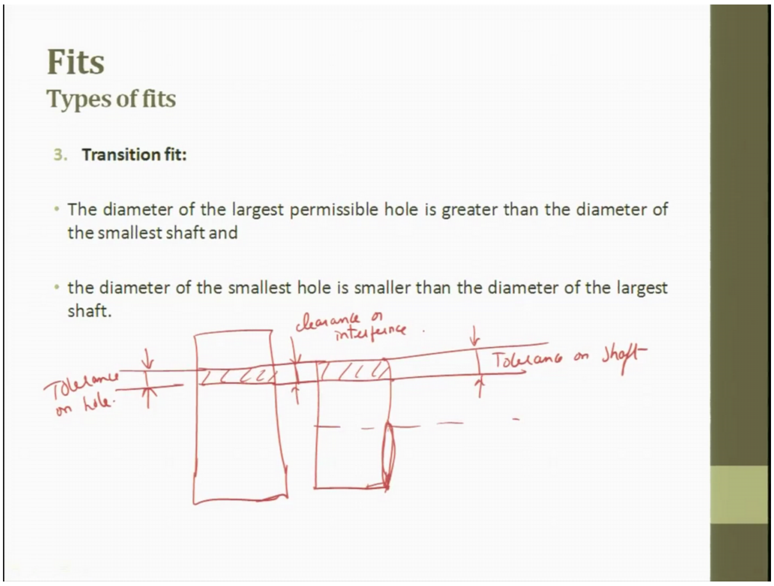
(Refer Slide Time: 34:38)



So, when we talk about interference the minimum permissible diameter of the shaft exceeds the maximum allowable diameter of the hole. The shaft is larger the hole is smaller; this type of fit always provides interference this is other way called as type fit. If you want to have a permanent assembly for example, a shaft is pressed inside a hole for example, a bearing is pressed inside the housing or a cover when you fix push the bearing and the bearing will not be further dismantled it will be tightly pressed and kept it is interference fit it is interference fit ok.

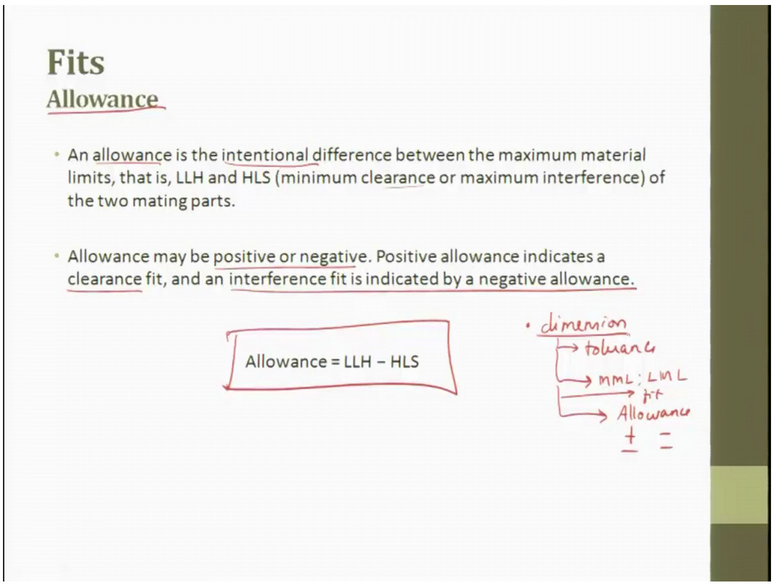
Interference fit has lot of applications, interference fit will not allow you to dismantle frequently clearance yes you can do it clearance fit whenever you want you dismantle it. But here once pressed tightly held it will be held so that is interference fit. So, we have seen the drawing for interference fit.

(Refer Slide Time: 35:47)



So, let us move to transition fit so transition fit it is it I told you very clearly transition between clearance and interference. So, in between the diameter of the largest permissible hole is greater than the diameter of the smallest shaft and the diameter of the smallest hole is smaller than the diameter of the largest shaft. So, here what happens? If I try to take a hole and I try to take a shaft. So, hole shaft this is a tolerance on hole, this is a tolerance on shaft ok. And this is the called as this is called as clearance or interference between them. So, this is nothing, but the transition fit ok.

(Refer Slide Time: 37:29)



So, now let us try to understand a one more concept called allowance. So, what is allowance? So, allowance now has to be linked with maximum material condition, minimum material condition clearance all these things right. So, what is allowance? An allowance is the intentional difference. Please understand the intentional difference between the maximum material limit that is LLH and HLH minimum clearance or maximum interference of the two mating part is nothing, but allowance is it clear.

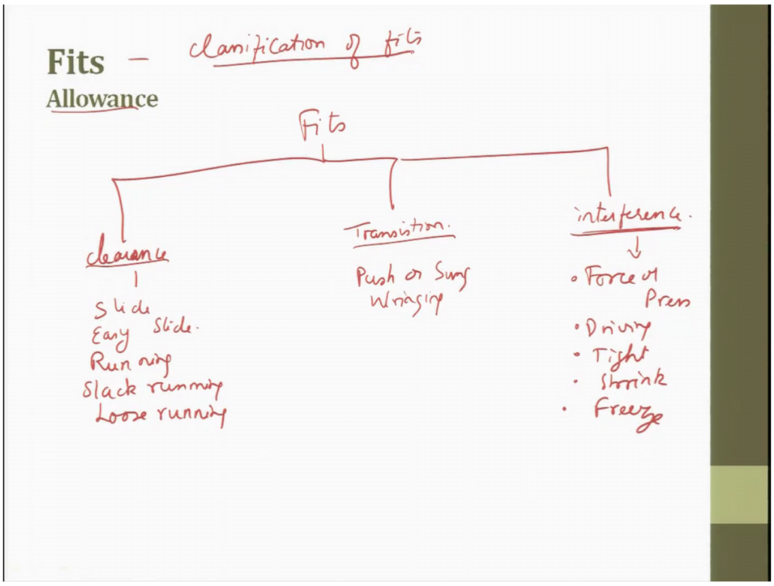
Allowance is the intentional difference between the maximum material limit; maximum material limit that is L L H or whatever M M H whatever you want to do you can write and H L S; that means, to say minimum clearance or maximum interference if you see I have written it in this way clearance minimum clearance. So, if you go back and see this figure I have said maximum interference minimum clearance ok

So, this is what is said in this minimum clearance or maximum clearance between the two mating parts allowance may be positive or negative please make a note. Allowance need not be positive alone, allowance can be also negative positive allowance indicate a clearance fit and negative allowance indicates a interference fit this formula is very important. So, in the examination there can be a problem asked in this topic.

So, what we do is? We start giving the dimensions we start giving the dimensions for the part. So, you are suppose to find out what is the tolerance? Then we have to we are suppose to find out what is the maximum material condition and the lower or limited or limited material condition. Then from here you will have to find out which fit it is and then you will try to find out what is the allowance whether it is positive or negative and try to solve.

So, these are you can have a problem in this in this type in the examination you can expect a problem in this type. So, first we saw about tolerances. So, then tolerances are the maximum and minimum. So, what is the use of maximum minimum? We are trying to find out the fit, and once you find out the fit now we are trying to find out the allowances.

(Refer Slide Time: 40:22)

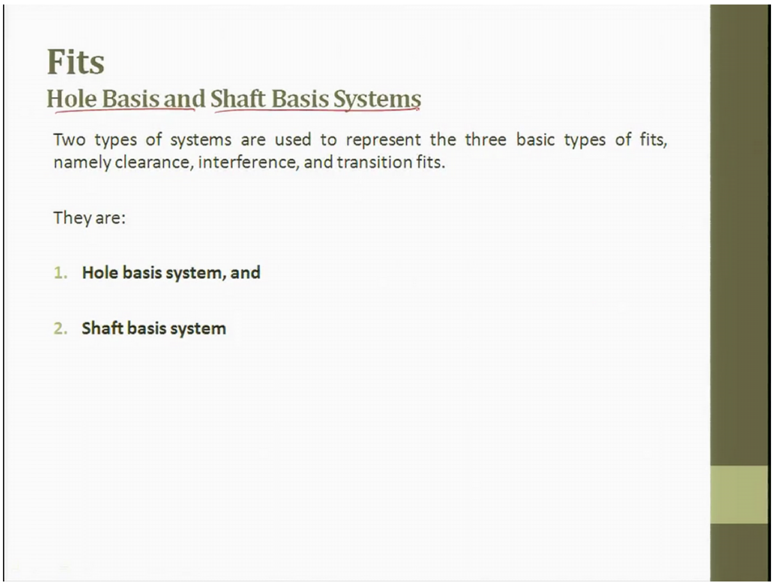


So, before that I would like to just put a small schematic diagram. So, clearance, you have interference, and you have transition. So, in clearance you will have slide easy slide running in, these are some of the slack running. So, these are some of the applications, or this is what the loose running. So, these are some of the fits like under which falls under clearance fit.

Then you talk about transition it is always push or not or it is ringing when you do this ringing it is more of slip gauge. What we use is running and interference we always try to have force or press fits ok. So, here we need driving pushing right, pushing we need it is it is tight and we sometimes do a shrink fit we also do a freeze fit.

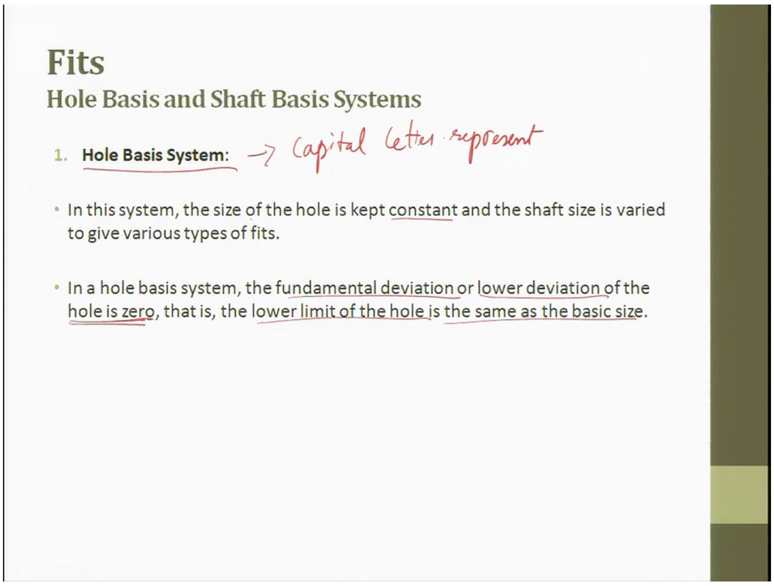
So, all these things are possible fits of interference fits ok. So, these are the allowances and here are the clearance. So, this figure basically talks about this figure talks about a classification of the this cycle say it is classification of fits various this thing, so we talk about.

(Refer Slide Time: 42:26)



Next let us get into understanding fits you have hole based system and shaft based system. Two types of systems are there are used to represent three basic types of fit, namely clearance, interference, and transition. There are two types of two types of system one is called as hole based system, the other one is called as shaft based system.

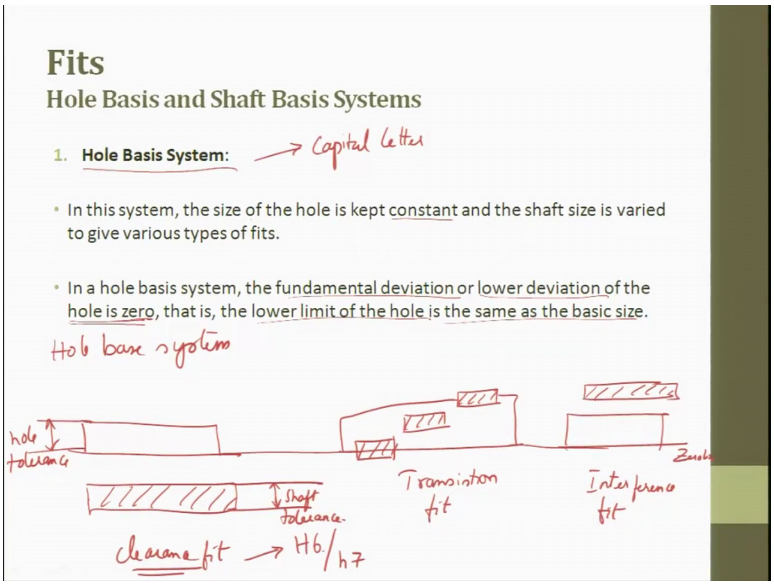
(Refer Slide Time: 42:49)



What is hole based systems? Hole base system is in this system the size of the hole is kept constant and the shaft size is varied to give various types of fits ok. So, this is what is hole base system; hole base system you we can see here we always use a capital letter for a presentation for capital letter is used to represent a hole based system.

So, in this system the size of the hole is kept constant and the shaft size is varied to give various types of fit. In a hole based system the fundamental deviation or lower deviation of the hole is zero, so please make a note, the fundamental deviation or the lower deviation is always zero, that is, the lower limit of the hole is the same as the basic size.

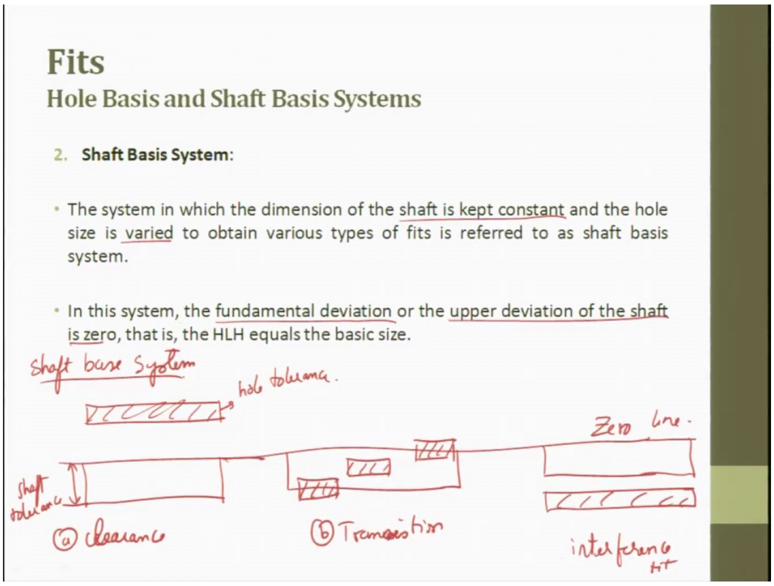
(Refer Slide Time: 43:59)



So, in a hole base system; in a hole based system. So, let us try to draw this is the shaft, this is a hole. So, this is a hole tolerance and this is a shaft tolerance. So, this is a clearance fit in a hole based system this is a transition fit and this is a zero line, so, this is a zero line. So, this is a hole and here is your shaft. So, this is a zero line and this fit is called as interference fit.

And in a hole base system we will always use a capital letter to represent a hole based system. For example if I have a shaft if I have to express a clearance fit between two between a shaft and the hole I write it as H 6 and H 7. So, when I do that. So, this is H 6 is a hole based system, and this is a shaft based system may be this is used for sealing, oaring, whatever it is whatever conditions it is.

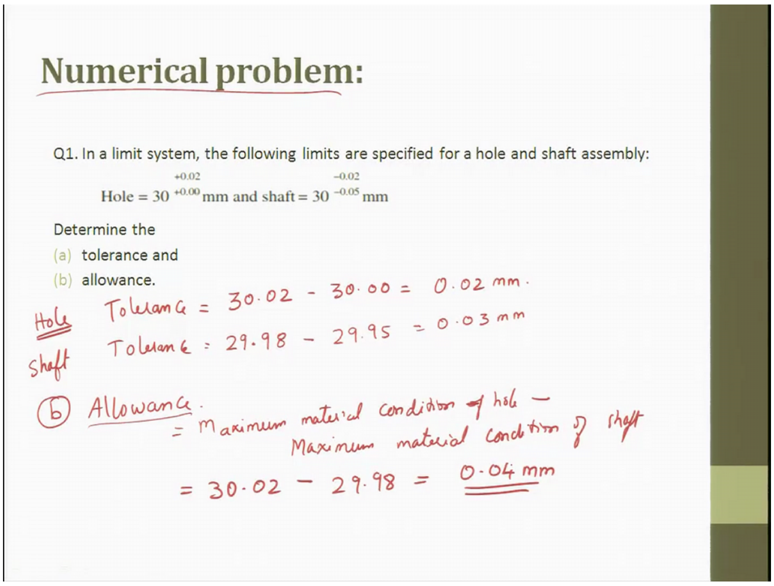
(Refer Slide Time: 46:39)



So, when we try to go for shaft based system it is a same. So, for a shaft based system shaft based system the shaft is kept constant and the hole is varied ok. So, here the fundamental deviation or the upper deviation of the shaft is kept as zero. So, when we try to draw a shaft based system. So, we have this is a shaft, and here is your hole, this is hole tolerance, and this is shaft tolerance ok. So, here this is my zeroth line, zero line not zeroth line, zero line ok, so here.

So, here this is your clearance fit for a shaft base this is for a, this is nothing, but a transition fit, transition fit and the last one is a interference fit this is interference fit. So, hole based system, shaft based system this is how the representation is given. And in a hole based system the hole is kept constant, in a shaft based system the shaft is kept constant ok.

(Refer Slide Time: 48:42)

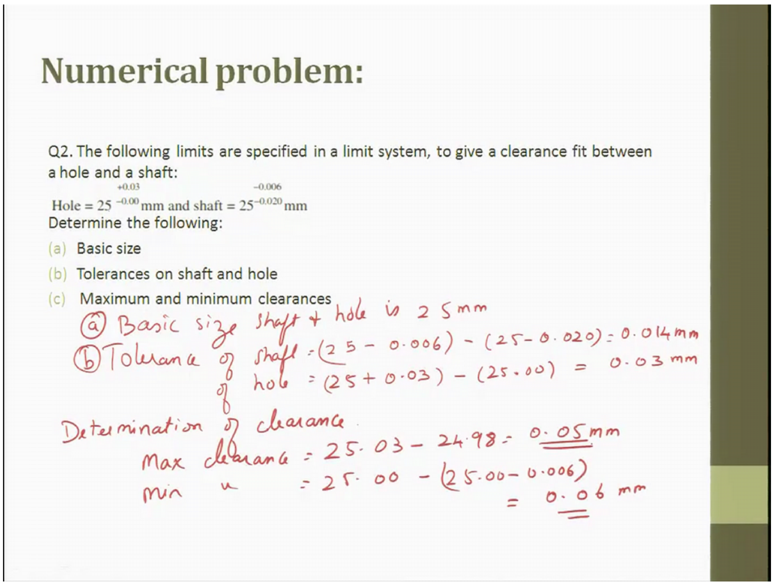


So, now we will see some of the problem to be solved. So, let us try a numerical problem till now whatever we have covered. So, in a limit system the following limits are specified for a hole and a shaft, the hole it is given and for a shaft it is given. So, what are they ask they have asked to find out what is the tolerance? And what is the allowance?

So, let us try to figure out the tolerance. Tolerance if you want to figure out it is nothing, but so for a hole tolerance is 30.02 minus 30.00 so which is nothing but 0.02 millimeter. Now, let us take for a shaft the tolerance it is 20, or I can easily solve it like this it is minus ok. So, it is 29.98 minus 29.95 ok, so it is 0.03 millimeter. So, now what is the allowance? B, so this is a this is allowance. How do you find out? Allowance is nothing but maximum material condition minus condition of hole minus maximum material condition of shaft ok. So, this is nothing, but 30.02 minus 29.98 so which is nothing, but 0.04 millimeter this is a allowance, so now, you are able to understand.

So, and here what is the tolerance? It is unilateral this is also unilateral these are all in one direction do you use that that is 0 plus it is both are in minus it is not bidirectional, or it is not a bilateral tolerance right. So, you so here for a hole it is 30.02, and 30.00 it is 0.02 millimeter. Tolerance for a shaft is going to be 0.03 millimeter. So, the allowance maximum material condition of the hole, minus maximum material condition of the shaft is nothing, but 0.04 millimeter.

(Refer Slide Time: 52:00)



So, let us try one more problem. So, this problem the following limits are specified in the limiting system to give a clearance fit between the hole and a shaft. The hole is 25, the dimensions are given shaft it is given. So, determine the following basic size, so determine the following basic size and next is tolerance of the shaft and the hole next the last one is going to be maximum and minimum clearance.

So, let us try to take the basic size the basic size, or the shaft and the hole shaft and hole is 25 millimeter, next is tolerance of shaft is 25 minus 0.006 minus 25 minus 0.02. So, which is nothing, but 0.01 4 millimeter, tolerance of a hole is 25 plus 0.03 minus 25 0 to 25.00. So that is nothing, but 0.03 millimeter ok. So, determination of clearance which is nothing, but maximum clearance is going to be 25.03 minus 24.98 which is nothing, but 0.05 millimeter and minimum clearance is going to be 25.00 minus 25.00 minus 0.006, so this is going to be 0.06 millimeter. So, this is the answer. So, we have covered 2 problems.

So, we have tried to solve basic size we have tried to take an example of tolerance and then we have also tried to find out the maximum material. We have also found out the allowance in the previous sum and here we have tried to figure out the clearance, maximum clearance and the minimum clearance for a hole and a shaft system. We will continue in the next lecture little bit problems and then followed by the limit fits and tolerance topic itself.

Thank you.