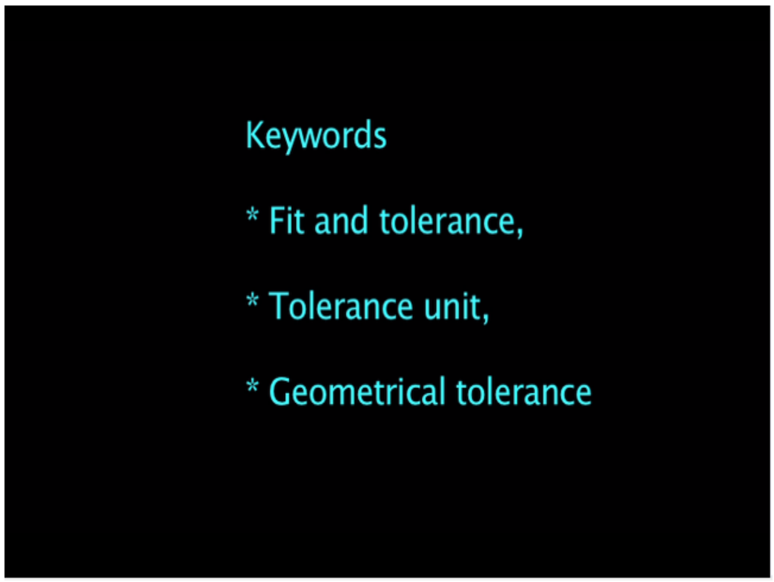
Metrology
Prof. Dr. Kanakuppi Sadashivappa
Department of Industrial and Production Engineering
Bapuji Institute of Engineering and Technology-Davangere

Module-3
Lecture-4
Selection of fits, Geometrical tolerances

(Refer Slide Time: 00:13)



I, welcome you all for module 3 lecture 4. In this lecture we will be discussing about the various factors favouring the loose tolerances and tight tolerances. We will have the discussion on the selection of different kinds of fits and the assessment of the fit and tolerances. And how we can calculate the tolerance unit value also we will discuss about the geometrical tolerances.

(Refer Slide Time: 00:55)

Now I am showing cube engineering parts, the bearings which require very fine finishes and very tight tolerances. We can see here we have some bearings, ball bearings with very fine finish. So, it is very essential that we have to control the dimensions. And it also necessary that we have to control the features, geometric features . For example the roundness or flatness, straightness etc., etc.

So, that the function in a proper way. Here we have fluid film bearings used in hard disc. We can see with have very fine finishes with very tight tolerances. So, that the function properly without much run out a problem so, they require very fine IT grades like IT4, IT5 etc. So, that here functioning will be proper.

(Refer Slide Time: 02:11)

Table 9: Factors in favor of wide and narrow tolerances

Factors in favor of wide (Loose) tolerances	Factors in favor of narrow (Tight) tolerances
• Yield in manufacturing is increased. Fewer defects are produced.	• Parts interchangeability is increased in assembly
• Fabrication of special tooling (Dies, jigs, molds, etc.) is easier. Tools are therefore less costly	• Fit and finish of the assembled product is better, for greater aesthetic appeal
• Setup and tooling adjustment is easier.	• Production functionality and performance are likely be improved.
• Fewer production operations may be needed.	• Durability and reliability of the product may be increased.
• Less skilled, lower cost labor can be used.	• Serviceability of the product in the field is likely to improve due to increased parts interchangeability.
• Machine maintenance may be reduced.	• Product may be safer in use.
he need for inspection may be reduced.	
verall manufacturing cost is reduced	

Now this table shows the factors in favour of wide and narrow tolerances. When, we should go for narrow tolerances, when we should go for wider or loose tolerances the factors in favor of loose tolerances or given below. They increase the yield in manufacturing and there will be fewer defects are produced. And fabrication of tooling like dies, jigs etc., will be very easy.

So, tools will be there, will not be very costly. Because of the loose tolerances, we need not have use very precise machining, processors. So, the production cost of the parts will be less. And setup and tooling adjustment will be very easy. If we have the loose tolerances and then fewer production operations may be needed. We need not have to go for very finer or finishing processors like lapping etc., etc.

Simple fine turning rough grinding maybe enough. And the less scaled and lower cost labour can be used. If you select very tight tolerances, what happens says we have to go for the lapping processes, finishing processes, dime and turning processes like that. So, if you so and such cases we have to used very skilled operators. If the tolerances are wide we can use semi-scaled operators.

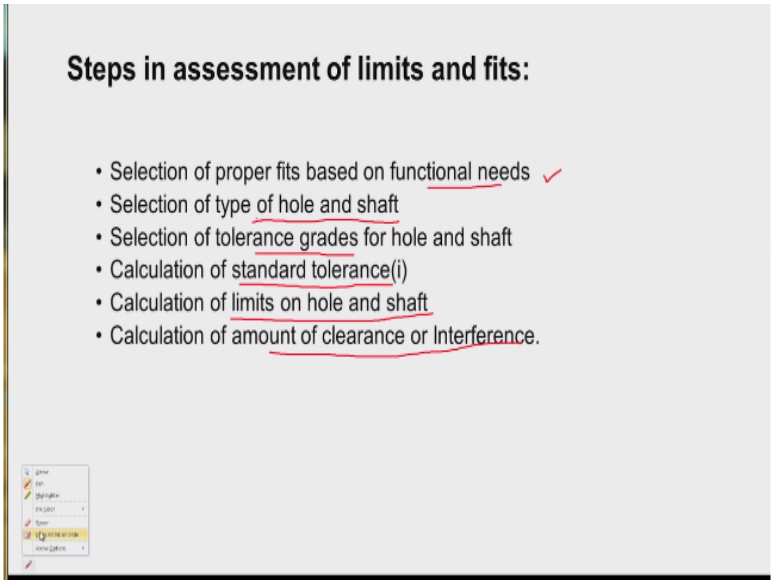
So, that the labour cost will be less. And also the machine maintenance get reduced and they made for inspection will also be cost of inspection will be reduced reason is we can use normal measuring instruments . And if we provide very tight tolerances, then we have to go for very fine precise measuring tools. So, investment on inspection cost will increase if you have tight tolerances.

And then overall manufacturing cost is reduced, reason is the not so precise machines can be used and semi-scaled operators can be used and the inspection cost will reduced. So, due to these the overall manufacturing cost also will reduce. And what are the conditions favouring tight tolerances. So, when we provide very appropriate tight tolerances, the parts inter-changeability is increased in assembly.

That means we can randomly select the mating parts and we can easily assemble. So, which may not be possible, if the tolerances are very very loose and then fit and finish with their assembled product will be better and the aesthetic appeal will also be better if you have tight tolerances. And then production functionality and performance will be improved. And the products produced will be durable and they will be reliable in service.

And the serviceability of the product in the field is likely to improve due to increased parts interchangeable. That means the components to be replaced there readily available in the market. Like bearings, belts etc., etc., tapper pins different kinds of fastener they are readily available. So, we can purchase them and we can replace them. So, inter-changeability aspect will improve and product will be safer to use.

(Refer Slide Time: 06:08)



Now we will move to steps in assessment of limits and fits. That means how we can assign the proper tolerances and how do we get the proper fit and how to select the proper fit. So, these are the steps followed for assessing the limits and fits. The selection of proper fits based on functional needs. That means what is the basic intention of the product where it is used, whether running clearance is required or very tight clearance tight fit is required or very wide tolerance.

And very wide clearance is required. So, depending upon the functionality we have to select proper fits whether we want clearance fit or transition fit or interference fit like that. so, it basically depends upon the functional needs what is the application of the making parts and then once we select the proper fits based on functional needs, within that we have to select what type of hole is required or what type of shaft is required.

For example if you select clearance fit within that what is the hole that together. We require h hole or b hole or c hole. Similarly when they know come to the shaft what type of shaft is required h shaft is required or m shaft is required or b shaft is required. So, that we have to select and then the selection of tolerance grades for hole as well as for shaft also we have to properly select.

Whether we require h7 or h8 or if it is the very precise operation is required, very proper location is required. Then we may have to go for IT4, IT5 like that and if it is the for the manufacturing of gauges used for inspection purpose. We may have to go for IT2, IT3 like that. So depending upon the again based upon the application we have to select proper tolerance grades for hole as well as for shaft which will decide the amount of the tolerance on the hole and shaft.

And then we have to calculate the standard tolerance (i) for a particular combination. And then calculation of limits on hole and shaft, that means once we know what is the type of hole and what is the type of shaft. And then what is the tolerance grade we are using and after finding the **ta** standard tolerance value. Then we can fix out what is the upper limit for the hole, what is the lower limit for the hole.

And similarly what is the upper limit for shaft and what is the lower limit for the shafts. Those things can be calculated and then after once we find the limits for hole and shaft, we can calculate what is the amount of clearance what is the maximum amount of clearance what is the minimum amount of clearance. If it is clearance fit and then we can if it is interference fit what is the maximum interference, minimum interference.

And if it is transition fit what is the maximum interference what is the minimum clearance. Such things can be calculated. And once we calculate all these things, so we can prepare the drawing we can measure al the tolerance etc., etc., and that drawing can be supplied to the manufacturing unit for the making of the products.

(Refer Slide Time: 09:52)

Selection fits

Table 10: Clearance Fits (Hole Basis System):

Shafts	Grades	Description of fit	Application
a, b, c	11	Very large clearance	Generally not used
d	8, 9, 10	Loose running	Loose pulleys
e	7, 8, 9	Loose clearance	Electric motor bearings, heavily loaded bearing
f	6, 7, 8	Normal running	Lubricated bearings (with oil or grease), pumps and smaller motors, gear boxes
g	5, 6	Precision running	Lightly loaded shafts, sliding spools, accurate bearings
h	5 to 11	Extreme clearance (preferably for non-running parts)	Sockets and spigots of joints

Preferred Clearance fits : H11/c11, H9/d9, H8/f7, H7/g6, H7/h6, C11/h11, D9/h9, F8/h7, G7/h6

Now let us discuss about the selection of fits. Now it again it basically these fit depends upon what is the application whether the fit we have to select for the pulleys and then motor bearings and then heavily loaded bearings. And whether there assigning the fit and tolerance for lubricated bearings or gear boxes, loaded shaft etc., etc.

Based upon the application we can select the what is the type required whether we can go for latch clearance, loose clearance and then normal running clearance, precision running clearance. So, once we fix up the type of fit that is needed, then we can go for what is the shaft. So, this is basically old basis system, so h capital H sole is required.

Now we have to select the shaft what type of shaft is required whether a shaft is required, b shaft see when we the clearances are very large clearances are needed for general usage. Then we go for a, b, c shaft which will provide very large clearances. And then for accurate bearings may have to go for g shaft. And then also depending upon the application we have properly select what is the tolerance grade, whether 4 is required, 5 is required, 6 is required, 8 is required.

So, like this such a tables we can use for selection of fits. And prefer the clearance fits are mentioned here H11/c11, H9/d9. So, and then H7/h6 like this these are preferred. Once we fix up what is the type of shaft and grade. Then we can select appropriate preferred nearest preferred clearance fit.

(Refer Slide Time: 12:07)

Table 11: Transition Fits (Hole Basis System):

Shafts	Grades	Description of fit	Application
js	5, 6, 7	Slight clearance to slight interference	Very accurate location, couplings, spigots, gears,
k	5, 6, 7	No clearance to little clearance	Precision joints likely to be subjected to vibrations
m	5, 6, 7	Slight interference (on average)	Forced assembly is required
n	5, 6, 7	Slight interference and very little clearance	Semi-permanent or tight fit assemblies

Preferred Transition fits : H7/k6, H7/n6, K7/h6, N7/h6

Similarly we have tables available for selection of transition fits again based on the hole basis system. Again we can see there are many application are mentioned here very accurate location, couplings, gears and then precision joints forced assembly sometime may be required and semi-permanent or tight fit assembly we may have to design. In such cases we may have to appropriately select the fit slight clearance required or very little clearance is required.

That we have to find and then we have to appropriately select the shaft whether case required m is required and n is required. And then we can also select the IT great and then we have the preferred transition fit. So, we can select the preferred fit also.

(Refer Slide Time: 13:09)

Table 12: Interference Fits (Hole Basis System):

Shafts	Grades	Description of fit	Application
p	6, 7, 8,	True interference (light)	Fixing bushes, standard press fit
r	5, 6, 7	Interference (but can be dismantled)	Tight press fit. Keys in key ways
s	5, 6, 7	Semi permanent/ permanent fit	Valve seating, collars on shafts
t, u	----	High degree of interference	Permanent assemblies

Preferred Interference fits : H7/p6, H7/s6, H7/u6, P7/h6, S7/h6, U7/h6

Similarly we have interference fit again some applications are mention here for fixing bushes. We may go for interference fit for tight press fit and then valve seating and then permanent assemblies. So, in such cases we go for interference fit for example for permanent assembly with H capital H hole we can use the T shaft and U shaft. And for valve seatings collars etc... We can go for S shaft with tolerance creates 5, 6 or 7.

And then again we have the preferred interference fits we can select the type of fit out of these preferred interference fits.

(Refer Slide Time: 14:06)

EXAMPLE: The diameter of the shaft is 70 mm, and Tolerance grade H8 f 7. Calculate the Fundamental deviation, Tolerances and Limit of size for Hole and, Shaft. Mention the type of fit, also calculate maximum and minimum interference/clearance.

Solution:- Calculation of the Standard tolerance unit;

$$i = 0.45 \times \sqrt[3]{D} + 0.001 \times D$$

The size range for 70mm Diameter would be 50-80 mm as per ISO 286 (table 7)

D=√ 50 x80 mm=63.25mm

i= 0.45 x cuberoot(63.25)+0.001X 63.25 = 1.865micro meter or i = 0.001865mm

For Shaft of f7, tolerance value = 16i from the table 5;

Tolerance=16i=16x0.001865=0.030mm

For Hole of H8, tolerance value = 25i from the table 5;

Tolerance=25i=25x0.001865=0.046mm

Now after studying all these basics we will some numerical examples. I have taken an example here we follow all the steps the for example selection of fit and calculation of tolerance unit etc., etc., so, that the basics can be understood clearly. Now the problem is the diameter of the shaft is 70 millimetre that is the basic size or the design size of the shaft is 70 millimetre. And then tolerance grade is H8 for hole and f7 for shaft.

So, H hole with 80 8 grade is used and for shaft f shaft is used with 87 so, tolerance grades and type of hole. They are already mention in the example now what we have to do is we have to calculate the standard tolerance unit. And then whether there is any fundamental deviation and then what is the upper limit and lower limit for shaft. Similarly what is the upper limit and lower limit for hole that can be calculated and finally.

We can say whether the type of fit obtained is clearance fit or interference fit. Now we have to find fundamental deviation tolerances for hole and shaft and limits of size for hole and shaft. Also we have to mention what is the type of fit and also we have to calculate maximum and minimum interference or maximum and minimum clearance depending upon fit obtain. We have to mention what is maximum interference are clearance.

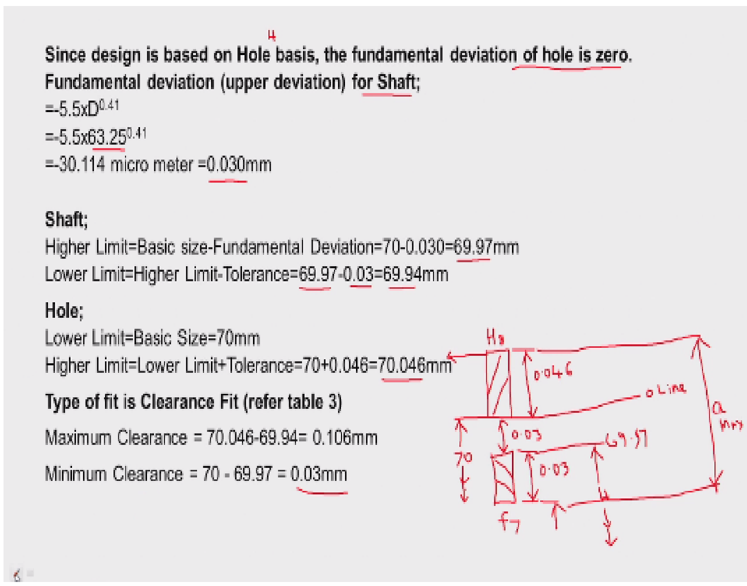
And what is maximum type maximum amount of these interferences and clearances, now we can the solution is given here calculation of standard tolerance unit. So, we can use this equation i is equal to 0.45 times cube root of D+0.001 times D so, where D is diameter mean diameter that means we know that basic size is 70 millimetre. And then where it falls in what step it falls that we have to see that we can do using table7 70mm.

It falls in the range of 52 to 80 now, we can find the mean D that is cube root of I am sorry square root of 50 times 80. So, this will be equal to 63.25millimetre this is the value of mean diameter now we have to insert this D in this equation to find the tolerance unit. So, the value of D is inserted here 0.45 times cube root of 63.25+0.001 times 63.25. So this will give us 1.865 micrometre or 0.0018millimetre.

Now after finding this i we can find the tolerance values for shaft and hole now, we are using f7 shaft so, tolerance value for height is greater 7 is equal to 16i so, this we can get from table number5. So, now tolerance value for the shaft is equal to 16 times i where i is equal to 0.0018millimetre so, this will be equal to 0.03millimetre. Similarly we can find the tolerance value for hole, so where using H hole with it grade 8.

So, tolerance value for IT 8 will be 25i and then i value we have already calculated that is 0.0018 millimetre. So, tolerance for shaft I am sorry for tolerance for hole will be equal to 25i that will be equal to 0.046millimetre.

(Refer Slide Time: 18:48)



Now after finding tolerance unit and tolerance values for hole and shaft we can find the fundamental deviation since we are using hole basis system. The fundamental that is H hole we are using so, fundamental deviation for hole is 0, now we have to find the fundamental deviation for the shaft f. Now from the table number4 we can get this equation for getting the fundamental deviation for f shaft.

So, for f shaft fundamental deviation that is upper deviation is equal to -5.5 times D to the power of 0.41. So, we have to feed the value of D that is 63.25 then we get fundamental deviation which is nothing but upper deviation as 0.030millimetre. So, once we find the fundamental deviation tolerance value, we can fix up the limits for shaft and hole. Now higher limit for shaft is equal to the basic size of the shaft that is 70millimetre –fundamental deviation.

So, that is 0.03 millimetres so, we get higher limit as 69.97millimetre similarly for lower limit of the shaft higher limit of shaft–tolerance that is higher limit is 69.97. And the tolerance value for the shaft is 0.03, so this will give as lower limit of 69. 94millimetre. Similarly for hole we can find lower limit and upper limit, so in this case lower limit for hole is 70millimetre reason is fundamental deviation is 0.

That means lower limit half the hole will be equal to basic size that is 70millimetre and higher limit can be calculated by adding tolerance to the lower limit. So, we get 70.046millimetre. Now we can show this graphically like this we have 0 line this is 0 line from this 0 line we show all the other dimensions like tolerance value, basic size etc, etc... Now we are using H8 hole. So, this is H8 and for H hole H8 hole, we have calculated the tolerance value.

So, tolerance for H hole is 0.046 millimetre that is this tolerance value is 0.046, 0.046 millimetres and then coming to the this is the lower limit for the hole which is equal to the basic size 70millimetre. Since the deviation is 0 the lower limit for the hole is equal to the basic size and we have to add this tolerance value to this basic size to get the upper limit of the hole then upper limit of the hole is 70.046millimetre this is the upper limit.

Now we have f7 shaft so, this is f7 shaft with the tolerance value of tolerance value for f7 shaft is 30 microns. So, this is 0.03millimetre and also the fundamental deviation that is upper deviation. So, upper limit of the shaft is nearer to the 0 lines so, this is upper deviation and we have calculated the upper deviation for the shaft that is 0.03millimetre. Now using the tolerance value and this upper deviation we can fix up the limits for the shaft.

So, that we have already found here year limit of the shaft, this is the higher limit h limit for shaft. So, higher limit is 69.97 and then lower limit we get at this point. So, this gives us the lower limit that means from the upper limit they have to detect this tolerance value. Then we obtain the lower limit and so, this is the maximum clearance so this guess as the maximum clearance that means to get the maximum clearance.

When the shaft size is equal to the lower limit, and the hole size is equal to the maximum, then we get the maximum clearance maximum. So, what we have to do is we have to add this tolerance for shaft and then we have to add this fundamental deviation. And then we have to add this tolerance for hole then we get the maximum clearance, and this difference between the minimum size of the hole. And maximum size of the shaft will give as the minimum clearance that is equal to 0.03mm.

(Refer Slide Time: 26:15)

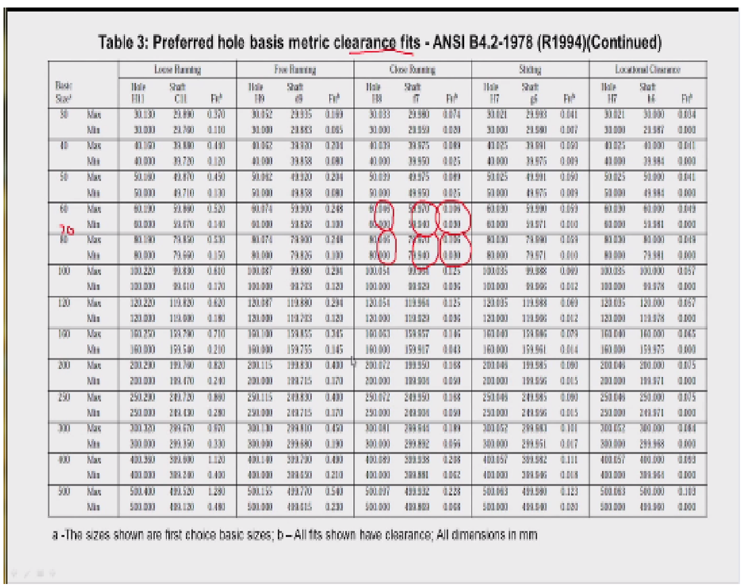


Table 3: Preferred hole basis metric clearance fits - ANSI B4.2-1978 (R1994)(Continued)

Basic Size[a]		Loose Running Hole H11	Loose Running Shaft C11	Loose Running Fit[b]	Free Running Hole H9	Free Running Shaft d9	Free Running Fit[b]	Close Running Hole H8	Close Running Shaft f7	Close Running Fit[b]	Sliding Hole H7	Sliding Shaft g6	Sliding Fit[b]	Locational Clearance Hole H7	Locational Clearance Shaft h6	Locational Clearance Fit[b]
30	Max	30.130	29.890	0.370	30.052	29.935	0.169	30.033	29.980	0.074	30.021	29.993	0.041	30.021	30.000	0.034
	Min	30.000	29.760	0.110	30.000	29.883	0.065	30.000	29.959	0.020	30.000	29.980	0.007	30.000	29.987	0.000
40	Max	40.160	39.880	0.440	40.062	39.920	0.204	40.039	39.975	0.089	40.025	39.991	0.050	40.025	40.000	0.041
	Min	40.000	39.720	0.120	40.000	39.858	0.080	40.000	39.950	0.025	40.000	39.975	0.009	40.000	39.984	0.000
50	Max	50.160	49.870	0.450	50.062	49.920	0.204	50.039	49.975	0.089	50.025	49.991	0.050	50.025	50.000	0.041
	Min	50.000	49.710	0.130	50.000	49.858	0.080	50.000	49.950	0.025	50.000	49.975	0.009	50.000	49.984	0.000
60	Max	60.190	59.860	0.520	60.074	59.900	0.248	60.046	59.970	0.106	60.030	59.990	0.059	60.030	60.000	0.049
	Min	60.000	59.670	0.140	60.000	59.826	0.100	60.000	59.940	0.030	60.000	59.971	0.010	60.000	59.981	0.000
80	Max	80.190	79.850	0.530	80.074	79.900	0.248	80.046	79.970	0.106	80.030	79.990	0.059	80.030	80.000	0.049
	Min	80.000	79.660	0.150	80.000	79.826	0.100	80.000	79.940	0.030	80.000	79.971	0.010	80.000	79.981	0.000
100	Max	100.220	99.830	0.610	100.087	99.880	0.294	100.054	99.964	0.125	100.035	99.988	0.069	100.035	100.000	0.057
	Min	100.000	99.610	0.170	100.000	99.793	0.120	100.000	99.929	0.036	100.000	99.966	0.012	100.000	99.978	0.000
120	Max	120.220	119.820	0.620	120.087	119.880	0.294	120.054	119.964	0.125	120.035	119.988	0.069	120.035	120.000	0.057
	Min	120.000	119.600	0.180	120.000	119.793	0.120	120.000	119.929	0.036	120.000	119.966	0.012	120.000	119.978	0.000
160	Max	160.250	159.790	0.710	160.100	159.855	0.345	160.063	159.957	0.146	160.040	159.986	0.079	160.040	160.000	0.065
	Min	160.000	159.540	0.210	160.000	159.755	0.145	160.000	159.917	0.043	160.000	159.961	0.014	160.000	159.975	0.000
200	Max	200.290	199.760	0.820	200.115	199.830	0.400	200.072	199.950	0.168	200.046	199.985	0.090	200.046	200.000	0.075
	Min	200.000	199.470	0.240	200.000	199.715	0.170	200.000	199.904	0.050	200.000	199.956	0.015	200.000	199.971	0.000
250	Max	250.290	249.720	0.860	250.115	249.830	0.400	250.072	249.950	0.168	250.046	249.985	0.090	250.046	250.000	0.075
	Min	250.000	249.430	0.280	250.000	249.715	0.170	250.000	249.904	0.050	250.000	249.956	0.015	250.000	249.971	0.000
300	Max	300.320	299.670	0.970	300.130	299.810	0.450	300.081	299.944	0.189	300.052	299.983	0.101	300.052	300.000	0.084
	Min	300.000	299.350	0.330	300.000	299.680	0.190	300.000	299.892	0.056	300.000	299.951	0.017	300.000	299.968	0.000
400	Max	400.360	399.600	1.120	400.140	399.790	0.490	400.089	399.938	0.208	400.057	399.982	0.111	400.057	400.000	0.093
	Min	400.000	399.240	0.400	400.000	399.650	0.210	400.000	399.881	0.062	400.000	399.946	0.018	400.000	399.964	0.000
500	Max	500.400	499.520	1.280	500.155	499.770	0.540	500.097	499.932	0.228	500.063	499.980	0.123	500.063	500.000	0.103
	Min	500.000	499.120	0.480	500.000	499.615	0.230	500.000	499.869	0.068	500.000	499.940	0.020	500.000	499.960	0.000

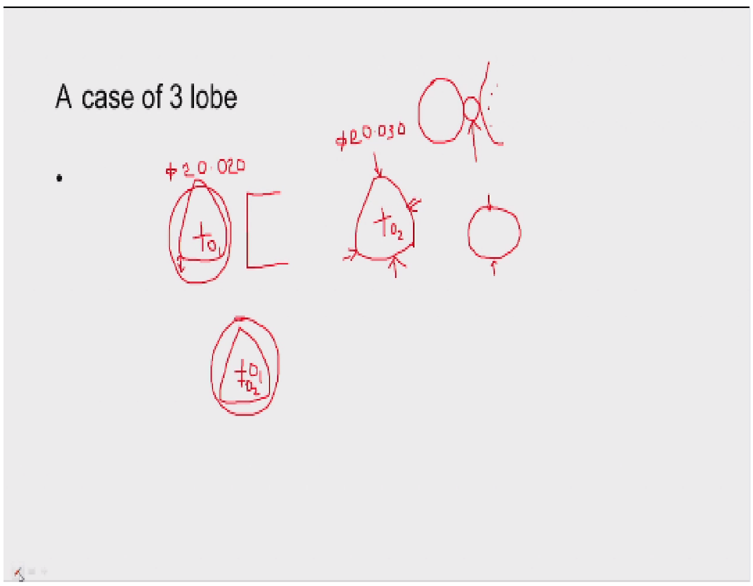
a - The sizes shown are first choice basic sizes; b – All fits shown have clearance; All dimensions in mm

Now we can compare these values the shaft limits and maximum clearance minimum clearance with the table3 the basic size is we are using 70millimetre. So, we get 70millimetre between the 60 and 80 and then we have calculated the tolerance for hole is 46millimetre. So we are using H8 hole so, we have H8 hole here So, now we can see here the this difference is 46 microns for 60millimetre basic size.

Similarly we have same tolerance value for 80millimetre basic size for 70millimetre basics also the tolerance value will be equal to 0.46millimetre 0.046millimetre. And similarly for shaft the tolerance value we can see here is the difference is 30microns, similarly for this 80 millimetre size that is tolerance value is 30micrometre 0.03mm. So, for 70 also same value will be there and then we can also see the maximum clearance and minimum clearance**.**

All these are clearance fits you can see minimum clearance is 0.03 millimetre and maximum clearance is 0.106 millimetre that we can observe here minimum clearance is 0.03 millimetre and maximum clearance is 0.106 millimetre like this we can calculate the values or if ready tables are available, we can use the ready table to get the to fix up the limits for hole and shaft and to get the type of fit.

(Refer Slide Time: 28:28)



Now we have a special case of 3 lobe now in all the previous examples, what we studied is based upon the hole size and the shaft size. We get particular type of fit it means if shaft is smaller than the hole we get clearance fit, if the shaft is greater than the hole. Then we get the interference fit and sometimes we get transition fit depending upon the actual size of the hole and shaft if the tolerance zones are overlapping.

Now this fit can we calculated by measuring the actual sizes of hole and shaft or if we know the tolerance zone for hole and shaft. We can we come to know whether we get it clearance fit or interference fit wet sometimes what happens is the shaft will not be circular that means we may get some lobes like this. So, this happens normally in centralise grinding varying we have supporting role.

And then we have the work piece and then we have rest blade2 keep the work pieces and then we have the grinding wheel. So, if the work piece setting is not proper or the blade height is not set properly. Then there are chances let we may get the some errors like this you may get lobed shafts now if may measure this diameter using a micrometre that means 2 point contact method.

Everywhere it gives same value it so micrometre measurement will not give us whether there is any lobbing or not. If it is elliptical shape it gives the difference we can find fine the maxi this is minimum diameter and maximum diameter. Then we can find at there is any different we can find that there is lobe ovalities there. But for 3 lobe the micrometre measurement will not indicate, whether there is any 3 lobe or not.

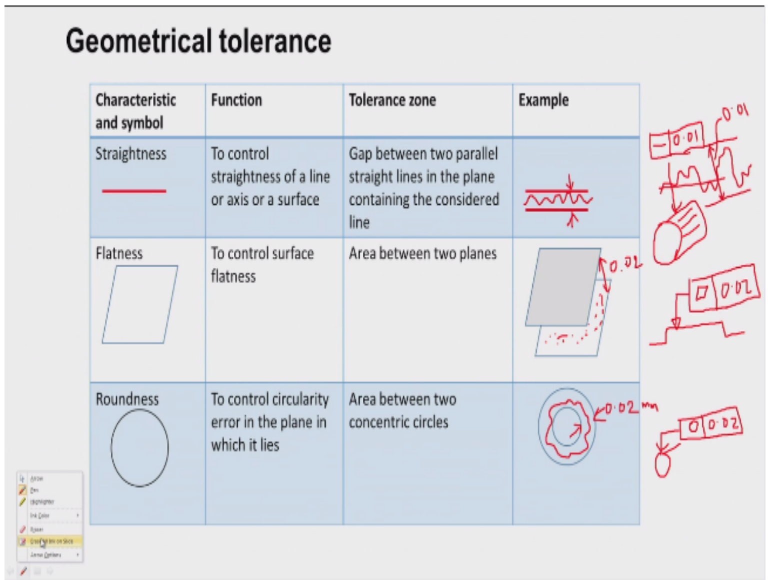
So, we just measured the diameter and we measure the diameter the hole and then we try to we calculate the type of fit what happens is say we have a diameter of hole is a 20.02millimetre. This is the diameter of the hole and then we have micrometre measurement gives that the diameter of shaft is say 20.030millimetre. So, this is the diameter of the shaft obtain by micrometre then what we can conclude is the diameter the shaft is greater than the diameter of the holes.

So, immediately we say that the fit type of fit what we are going to get is interference fit. But in actual practice when we try to insert this 3 lobed shaft into the hole it will entered. So, how this is possible see now we try to insert this so, this is the centre of the hole oven and then this is centre of the shaft o2. Now when both the centres coincide then the situation will be like this that means o2 is coinciding with oven then there will be interference.

But in actual practice what happens is we have this circle with oven centre, and then we try to insert this is now you can see re is some clearance. So what happens is the shaft will move down and then it will enter into the hole so situation will be like this. And o2 will be somewhere here. So actually we get instead of getting interference fit as per the measurement. We get the clearance fit in actual practice.

So, we should be careful in getting the fits that means near dimensional measurement or the specification of dimensional tolerance for hole and shaft is not enough. We have to specify the geometrical tolerance also. So what is the amount of error that can be allowed on the straightness or flatness or roundness or cylindricity. So, geometrical tolerance is also very very important to get proper type of fit and for proper functioning.

(Refer Slide Time: 34:09)



See now we will have discussion on geometrical tolerance now you can see here the straightness of an edge say we have line like this. So we have line like this but in actual practice we may measure it may be like this. So, it will vary the points will vary of course the gap will be very small it will be in terms of few microns. So, now it is very difficult to machine a perfect straight edge.

So, we have to allow some tolerance for the straightness and then that we can do by specifying what is the gap between these two lines say it may be some like 0.01millimetre or 0.02millimetre like the depending upon the application. So, straightness is also specify on the drawings, so it is to control straightness of a line or axis or a surface. So, basically it is gap between two parallel straight lines in a plain containing the consider line.

That means the line may be like this and what is the maximum deviation on straightness that is allow. So, that is mention like 0.01, 0.02 etc. So, now the single used for specifying the so, we have cylinder like this and will be having so many generators. And we want the straightness of this should not exceed say 0.03 so, we have to specify like this. So, straightness symbol is a short line and then we have to specify, what is the value.

Whether it is 0.01millimetre or 0.02millimetre so, that value this is the tolerance value and this is the feature straightness this for straightness this like this we specify the straightness. Similarly we can specify the flatness. So we have a surface like this for example the guide way of later any machine tool. And you should be flat it is necessary that you should be straight as well as you should be flat. So, that we can specify using this symbol flatness symbol like this

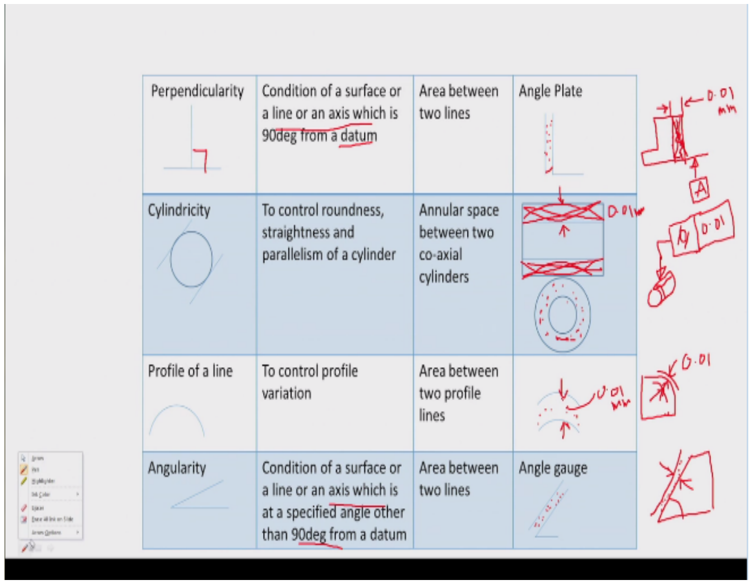
So, we have to specify the, what is the feature flatness in this case. And then we have to give what is the value whether it is 0.02millimetre or 0.01millimetre like that. So, this is how we specify the flatness it is basically to control the surface flatness and it is area between two plains so, we have two plains here and the surfacing question say this is the surfacing question all there are many high points and low points on the surface.

All the high points and low points should be contain within these two plains which are separated by the given tolerance value. In this case 0 02millimetre now similarly we can we specify the roundness what is the roundness that is allow it is basically to control secularity error in the plain in which it place. And then how it is specify we use two concentric circles so, that area between two concentric circles gives us the roundness.

That means so we have shaft which is necessary that it should be round but in practice there will be some variations low points and high points will be there. All the low points and high points on that particular round part should be should lay within these two circles. So, this value is mention what is the amount of tolerance that is allowed whether it is 0.01 or 0.02 like that.

That value is mention and we specify the roundness like this. We use that symbol and then we give the value, so this is how we specify the roundness.

(Refer Slide Time: 39:05)



Perpendicularity	Condition of a surface or a line or an axis which is 90deg from a datum	Area between two lines	Angle Plate
Cylindricity	To control roundness, straightness and parallelism of a cylinder	Annular space between two co-axial cylinders	
Profile of a line	To control profile variation	Area between two profile lines	
Angularity	Condition of a surface or a line or an axis which is at a specified angle other than 90deg from a datum	Area between two lines	Angle gauge

Now we will move to the perpendicularity or sometimes it is necessary that the two lines or two axis or two surfaces should be perpendicular to each other okay. There will be a datum surface and then there will be working surface. So, that angle between this working surface and this datum surface should be 90 degree. So that is known as perpendicularity, now how it i specify, they say we have an angle plate lie in this.

And this is the datum surface so, it is shown like this say a surface a so, this is the datum and then we have to specify the tolerance for the perpendicularity. So, we what we do is we draw a line parallel to the surface and then we mention what is the amount of perpendicularity error that is allowed. So for example 0.01millimetre that means any point on this particular surface should lay within this two plains.

The surface can be like this or it can be like this or it can be some rough surface like this. All point should lay within these two parallel lines. So, the high points low points it can be like this with reference to this a. And then cylindricity is also important now we say have the symbol for cylindricity circle with two parallel lines. So, this cylindricity is to be controlled in a many engineering applications.

So, this symbol is used to control roundness, straightness, and parallelism of the cylinder. It is the annular space between two co-axial cylinders. So, annular space between two co-axial cylinder we have two cylinders separated by some distance. This is the tolerance value may be 0.01millimetre or 0.02millimetre something like that. Now all the points high points and low points on the complete surface of the cylinder, should lay within these two circles.

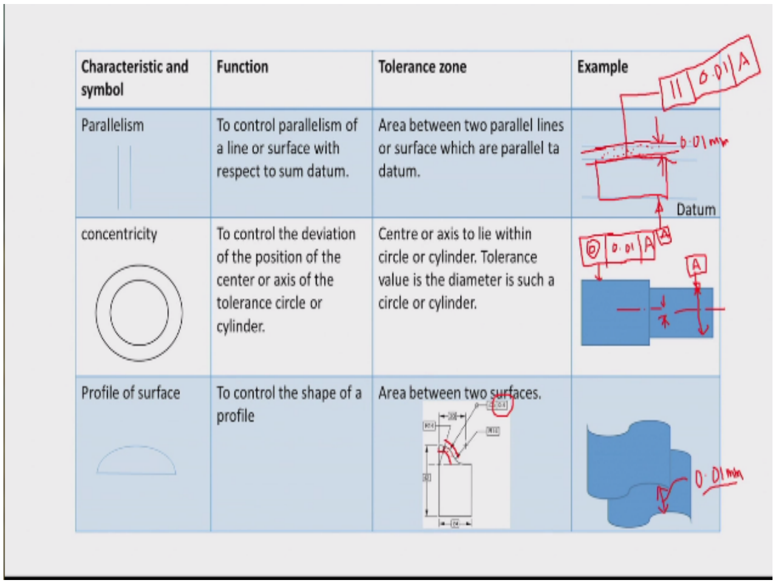
And the shape can be like this may be a tapper or it may be tapper in the other direction or it can be some bell shaped double bell shape, or it can be drum shape any shape within the all the variation should be within this global tolerance. Now how this is specified we say we have a cylinder like this, and then we mention, what is the feature in this case it is cylindricity. And then we may mention what is the value in millimetre.

So this is how we specify the cylindricity that means you take any generator any generator or any circle all the values all the high points and low points should lay within this range. And then we have profile of a line say we have a work piece like this okay and then we have a profile like this some radius is there. So, now we have to control this profile variations for that we have to draw, we use this symbol.

The meaning is will be two arcs parallel arcs separated by some distance, and then what is the alert tolerance is given that means on the lines on the profile on the high points and low points on the profile should lay within these two arcs separated by the tolerance amount in mm so, that is profile of a line. Similarly the angularities sometimes we have to maintain the angle between two surfaces or two lines or two axis and angle is other than 90 degree.

If it 90 degree then we say it is perpendicularity other than 90 we say angularity so, we say have a part like this. And then this is the working surface this angle is to be control with reference to this datum surface. So, what we do we draw other line parallel line parallel to this. And all the high points and low points on this particular surface should lay within this tolerance that is allowed. It can be 0.01 at the point depending upon the application we can specify what is the deviation from angularity.

(Refer Slide Time: 44:36)



Characteristic and symbol	Function	Tolerance zone	Example
Parallelism	To control parallelism of a line or surface with respect to sum datum.	Area between two parallel lines or surface which are parallel ta datum.	
concentricity	To control the deviation of the position of the center or axis of the tolerance circle or cylinder.	Centre or axis to lie within circle or cylinder. Tolerance value is the diameter is such a circle or cylinder.	
Profile of surface	To control the shape of a profile	Area between two surfaces.	

Similarly we have the symbol for parallelism say we have work piece like this. And then this top surface should be parallel to this datum. Datum is shown like this so, this is the datum say a, a surface is datum. And all the points on this particular surface or line should lay within these two lines which are separated arcs surfaces separated by the given amount of tolerance. For example 0.01mm high points and low points.

All should lay within these two lines or two planes now how this is designated. So, it is shown like this we have to use the parallelism symbol and then we have to give the value what is the tolerance that is alone. And then we have to specify datum with reference to which plane they should be parallel. So, that reference or datum surface also we have to specify and then we have another feature concentricity say we have two steps damper steps like as shown here.

And then the axis of this particular part should be concentric with the axis of the other part. So, and if there is any deviation then what is the amount of deviation okay that is indicated by using two circles concentric circles. And that is specified like this so, we have this is the reference datum say this is A, surface A okay and then this axis should be parallel to this or concentric with this within the certain tolerance.

So, we specify like this, this is the symbol feature symbol for concentricity, and then we have to specify what is the tolerance. And then we have to specify with reference to which reference or datum like this we specify the concentricity. And then similarly profile of a surface sometimes we need to control the profile of a surface. So, in that case what we do is we use this particular symbol the meaning of this is the we have to surfaces two surfaces like this separated by the given tolerance amount.

For example 0.01millimetre okay all the points on the profile surface should lay within these two surfaces separated by the given amount of tolerance. So now how to specify the profile tolerance is shown here, we have to use this symbol and then we have to mention what is the deviation that is allowed. Now in this lecture we discussed about the various aspects like selection of fits calculation of tolerance unit value.

And then what are the factors favouring loose tolerances and tight tolerances, we also discussed about the various geometrical tolerances for geometrical features. In the next class we will be discussing about the various positional tolerances, how to specify positional tolerance and then we will also say some numerical problems.