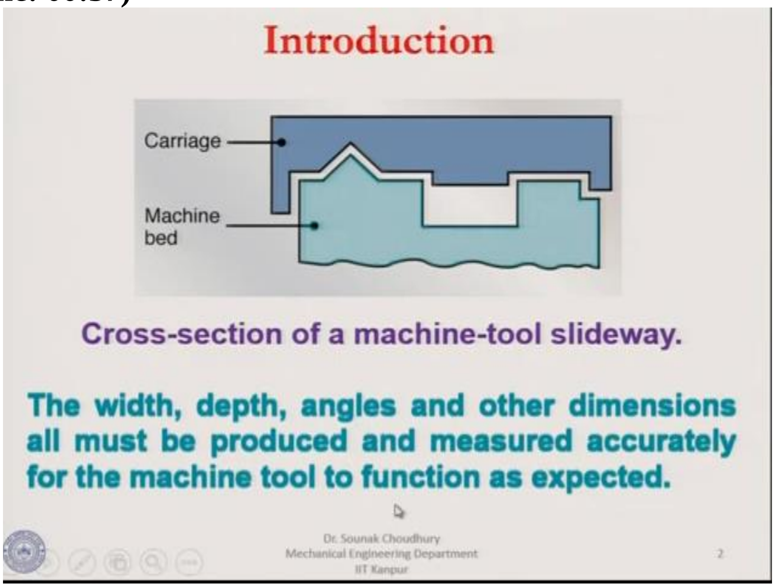
Production Technology: Theory and Practice
Dr. Sounak Kumar Choudhury
Department of Mechanical Engineering
Indian Institute of TechnologyKanpur

Lecture - 23
Engineering Metrology

Hello and welcome to this series of discussions in the production technology theory and practice. The last topic of this course is the engineering metrology that we would like to discuss today.

(Refer Slide Time: 00:37)

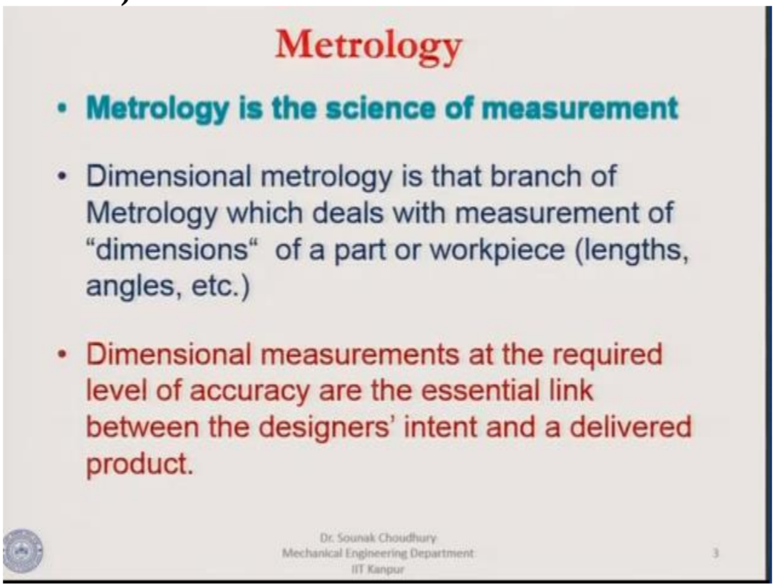


If you see the slides here, engineering metrology, if we have to define, before that that this is a machine tool carriage and this is the machine bed. The carriage moves over the bed on this length of the bed to and fro. This is the machine tool slide way. The width, depth, angles and other dimensions all must be produced and measured accurately for the machine tool to function as expected. Because it is a very lengthy machine bed and all along this the carriage has to move to and fro freely.

If this dimension and the dimension here are not adequate in that case, the carriage may not move and get stuck somewhere depending on how much is the deviation in the dimensions we have. If this dimension is improper with respect to this dimension in that case also the carriage may not be able to move along the machine bed. This surface has to be perpendicular to this. This dimension has to be exactly maintained within certain tolerance level.

The width, depth, angles and all other dimensions must be produced and measured accurately for the machine tool to function.

(Refer Slide Time: 02:26)



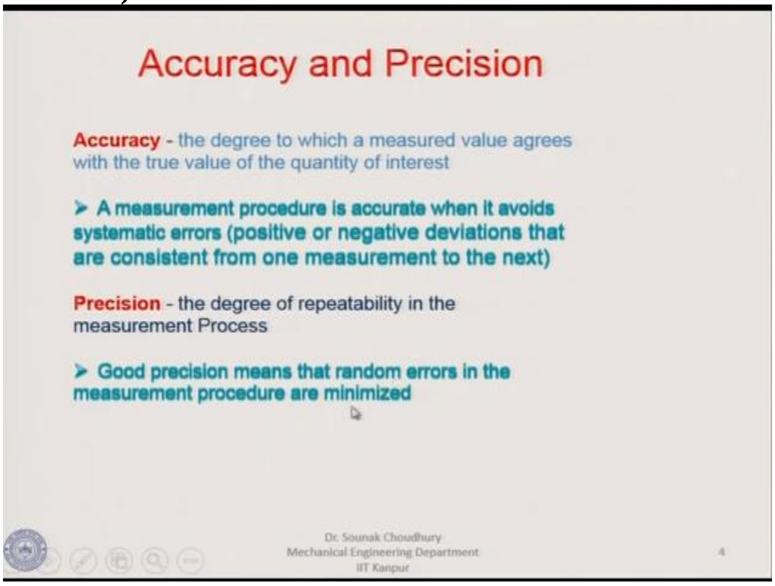
Metrology is the science of measurement, dimensional metrology is that branch of metrology which deals with measurement of dimensions, it can be length, it can be angles, it can be diameter. The dimensional measurements at the required level of accuracy are the essential link between the designer's intent and a delivered product.

Earlier I discussed the concept of design for manufacturability. The designer is designing a part and putting that in the form of an engineering drawing, on that drawing the designer is specifying all the dimensions and what else to be done regarding the surface finish. Regarding the taperness, how fine the taperness should be.

How much the cylindricity is to be given and all details have to be on the drawing then only the product will be manufactured properly. Therefore, what is said here is that the measurement is the essential link between the designers intent what he wants to do and the delivered product because delivered product has to exactly match with the intention of the designer and the intention of the designer is laid down on the drawing through the dimensions through, the tolerances, through the fits, all those things we will be discussing now.

Overall, the metrology deals with the science of measurement, be it tolerance, measurement, be it roundness measurement or the length measurement, angle measurement, any measurement and to maintain them is the job of the metrology which is a science of measurement in one word.

(Refer Slide Time: 04:53)



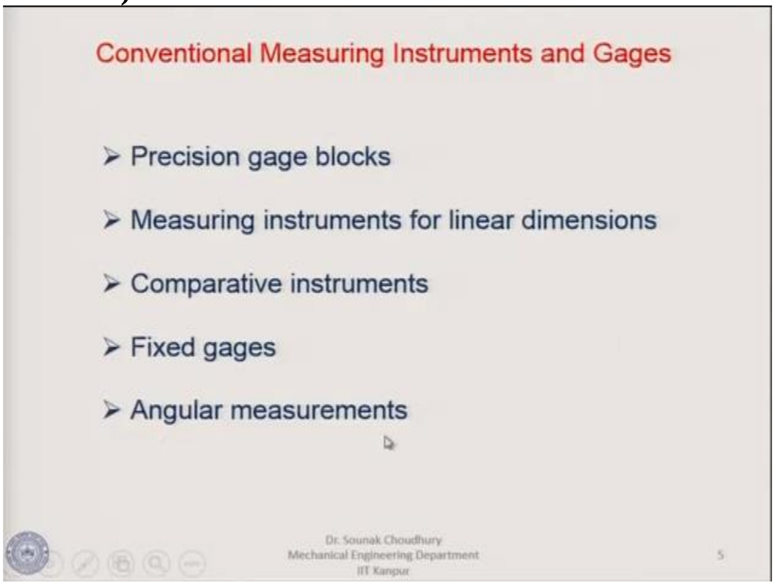
Let us see before we go into those details, let us talk about the accuracy of measurement. While using the concept of precision, we very often confuse between the concept of accuracy and the precision. Accuracy is the degree to which the measured value agrees with the true value of the quantity of interest; we have measured a value which may not be the true value of the quantity.

We say that a measurement procedure is accurate. This accuracy we are talking about when it avoids systematic errors, positive or negative deviations that are consistent from one measurement to the next. I will give you an example. Let us say that you are cutting certain length and you are measuring the force. With the same parameters and with the same given input, you repeat that 3 times.

And see what is the discrepancy each time that you are getting with respect to the other measurements. This way you can find out the accuracy of the measurement. And always in the experiments we ask that the experiment should be repeated so that accuracy is maintained. Now, once again, the measurement procedure is accurate when it avoids systematic error.

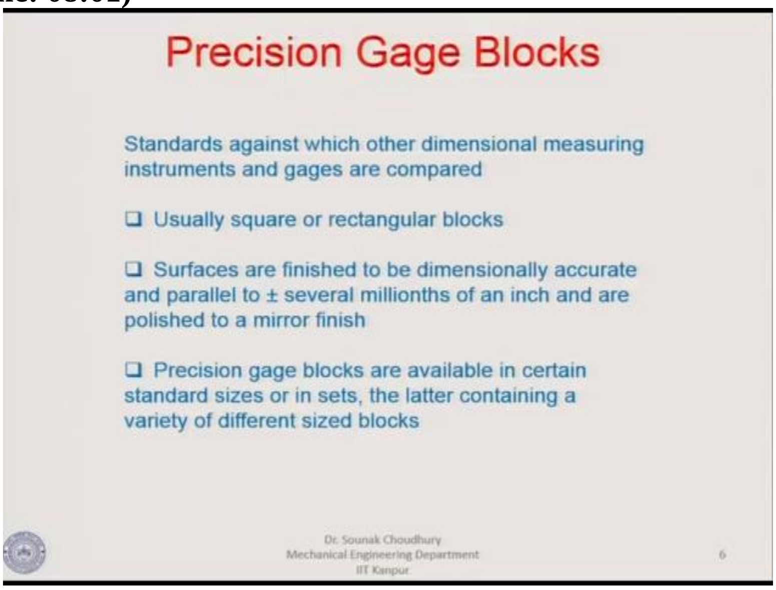
If there are systematic errors then of course, it is not accurate positive or negative deviations that are consistent from one measurement to the next either it can be one deviation or another, but it will be consistent. Now, the precision is the degree of repeatability in the measurement process, how well it is being repeated; good precision means that random errors in the measurement procedure are minimized.

(Refer Slide Time: 07:16)



Conventional measuring instruments and gages are the following. Precision gage blocks are the measuring instruments for linear dimensions. Then we have the comparative instruments, we have the fixed gages or the angular measurements. Most of these measuring tools we will see in the laboratory when I will show you inside the laboratory. Before that I will give you a presentation also with all the tools and I will show the actual tools used in the laboratory.

(Refer Slide Time: 08:01)

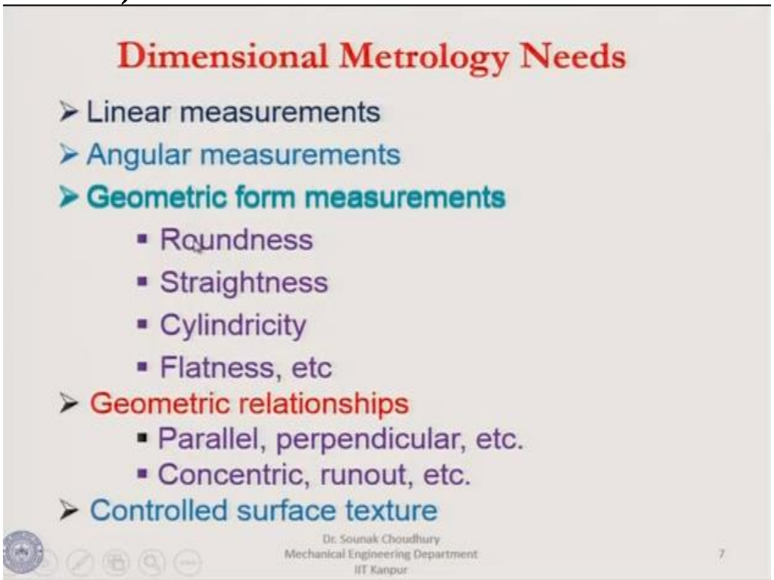


Precision gage blocks are standards against which other dimensional measuring instruments and gages are compared. Usually square or rectangular blocks since they are very precision then their sizes are very accurate. Surfaces are finished to be dimensionally accurate and parallel to plus minus several millionth of an inch and are polished to a mirror finish you may remember that these are the small blocks with absolutely smooth surface.

They are given with the different thickness those are in a box and they are so smooth that if you put one on top other, it is very difficult to separate because the undulations on their surfaces are so small that they stick to each other. And it is difficult to segregate them because they are very smooth; usually they are normally rectangular blocks with finished surfaces. Precision gage blocks are available in certain standard sizes in sets, the later containing a variety of different size blocks.

When there is a set, as I said that they are in a box, so various sets of those gages are there with different thicknesses. On each of these blocks it will be written what is the thickness of that so that that can be that particular block with the written thickness on it can be used for measuring the or comparing the thickness.

(Refer Slide Time: 09:41)



Dimensional metrology needs linear measurements, angular measurements, geometric form measurements, which include the roundness, straightness, cylindricity, flatness etcetera. And geometric relationship that is the parallel, perpendicular etcetera, concentric runout etcetera. These are the geometric relationship and for measurement will be this flatness, cylindricity, straightness, roundness and the control surface texture that I will come a little later.

(Refer Slide Time: 10:19)

Types of Measurement and Instruments Used

Measurement	Instrument	Sensitivity	
		μm	μin.
Linear	Steel rule	0.5 mm	1/64 in.
	Vernier caliper	25	1000
	Micrometer, with vernier	2.5	100
	Diffraction grating	1	40
Angle	Bevel protractor, with vernier	5 min	
	Sine bar		
Comparative length	Dial indicator	1	40
	Electronic gage	0.1	4
	Gage blocks	0.05	2
Straightness	Autocollimator	2.5	100
	Transit	0.2 mm/m	0.002 in./ft
	Laser beam	2.5	100
Flatness	Interferometry	0.03	1
Roundness	Dial indicator Circular tracing	0.03	1
Profile	Radius or fillet gage		
	Dial indicator	1	40
	Optical comparator	125	5000
	Coordinate measuring machines	0.25	10
GO-NOT GO	Plug gage		
	Ring gage		
	Snap gage		
Microscopes	Toolmaker's	2.5	100
	Light section	1	40
	Scanning electron	0.001	0.04
	Laser scan	0.1	5

Now, the types of measurement and instruments used are the following. Linear, you all know these are the steel rules, then we have the vernier caliper in the linear measurement, micrometre with vernier, diffraction grating. Here, in case of steel rule, the sensitivity is about 0.5 millimetres, whereas in case of vernier caliper it is about 25 micron. It is more accurate than this steel rule. Micrometre with vernier may have accurate sensitivity of up to 2.5 micron.

For measurement of angle, we have the bevel protractors. Many of you might have seen and I will show you in the lab alsowe have some picture on the bevel protector, I will show you they are also with vernier. They are sine bars for measuring the angles. Both the bevel protector and the sign bar are very popularly used for measuring angles.

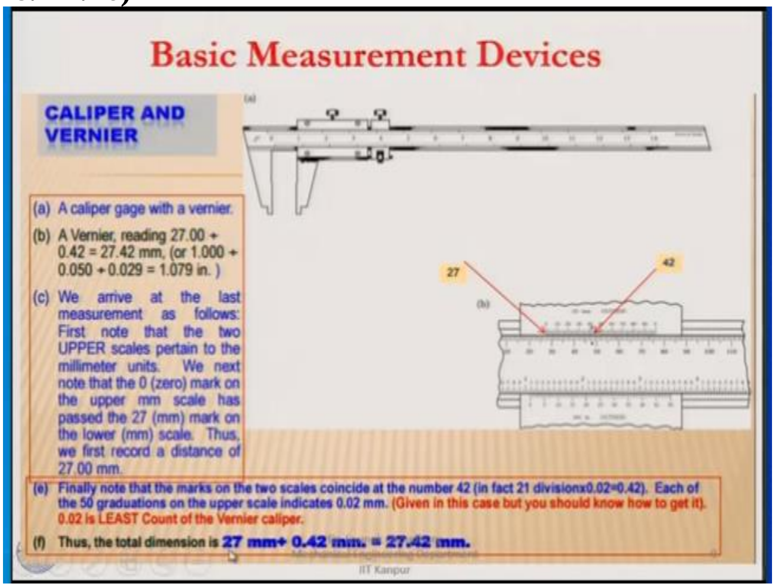
For measuring the comparative length dial indicators, electronic gages, gage blocks are used. Dial indicator is not very accurate neither it is so sensible in comparison to electronic gages or gage blocks; here you can see that they are not very accurate nor sensitivity is very high. The straightness can be measured by autocollimator which is an instrument and this you might have already used in the lab. And here the sensitivity is reasonably high because here the straightness is being measured.

Whereas for the transit, laser beam the sensitivity is even better. The flatness is measured by interferometry and here are the values of the sensitivity in micron, where it is not written it is micron and otherwise it will be written in millimetre per meter. We are talking about the transit per meter of the length here.

The roundness is measured by the dial indicator, circular tracing and the profile can be measured by the radius or fillet gage, dial indicator, optical comparator, and coordinate measuring machine. Coordinate measuring machine is a very specific one, this is very expensive and very accurate, this kind of machines are being used to get the points or the coordinates for a 3 dimensional body or a 3 dimensional figure.

Now, GO-NOT-GO gages are the plug gage, ring gage, snap gage these are the GO-NOT-GO that means, you will have the gage it will GO-NOT-GO accordingly. this is a qualitative measurement not quantitative. Microscopes: there are toolmakers' microscope, light section microscopes, scanning electron microscope and the laser scanning. Here also you can see that their sensitivity in microns will be different when the laser scanning sensitivity is that high. But scanning electron microscopes are the best in terms of the sensitivity.

(Refer Slide Time: 14:26)



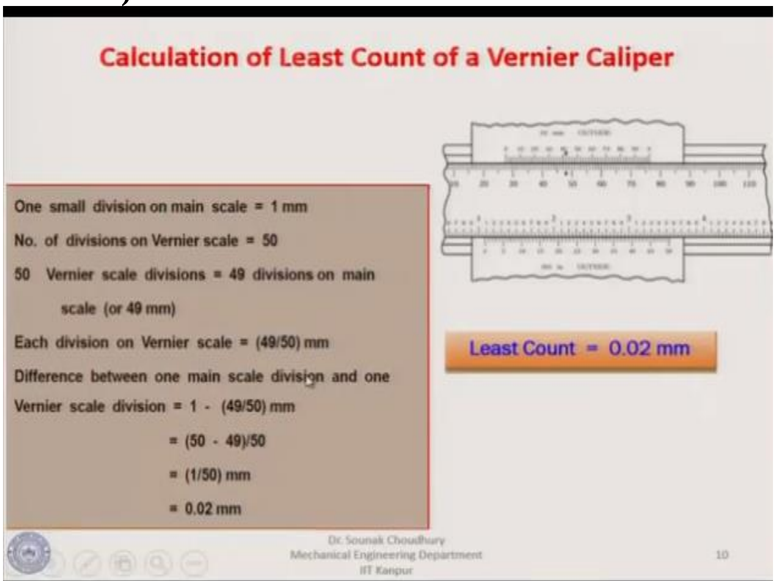
Here there is a normal vernier caliper what is shown here. So, here you can see that there is a main scale and there is a vernier scale; here there is a vernier scale as well. This can move along the length of the main scale and this is the jaw. Between these 2 jaws, the internal or the external diameter can be measured. Or the same vernier caliper can be used for measuring the diameter of the internal hole.

In that case the vernier caliper jaws are the fixed jaws, this is the movable jaw, this jaw between these 2 jaws, the diameter of the internal hole can be measured. Here if you see that this is exaggerated, this is just to show how to read the main scale and the vernier scale and overall how to read the diameter when you are measuring the diameter how to read this, that is the main scale whatever the main scale is showing that is the basic which is showing as 27.

Then we will see which line of the vernier scale is matching with the main scale. This is 42 if you see in this that is 42 number line is matching with this main scale. Now, this will be the 0.42 because it is multiplied here it is written the note that the marks on the 2 scales coincide at the number 42 that means 21 divisions into 0.02, this is 0.42, each of the 50 graduations on the upper scale indicates 0.02 here and so this is given as the least count.

This 0.02 is the least count and whichever is matching with this is 21 that would be multiplied by the least count. Therefore, the dimension what is shown here is a 27 + 0.42 which will be 27.4.

(Refer Slide Time: 17:06)



How to calculate this? We will go to this point of 0.02; how to find out? One small division on the main scale here you can see that this is one millimetre, it is specified as 1 millimetre. The number of divisions in the vernier scale are the 50 number of divisions and 50 vernier scale divisions therefore is equal to 49 divisions on the main scale here. If it is that then all 50 in the vernier will be covering, enveloping 49 on the main scale.

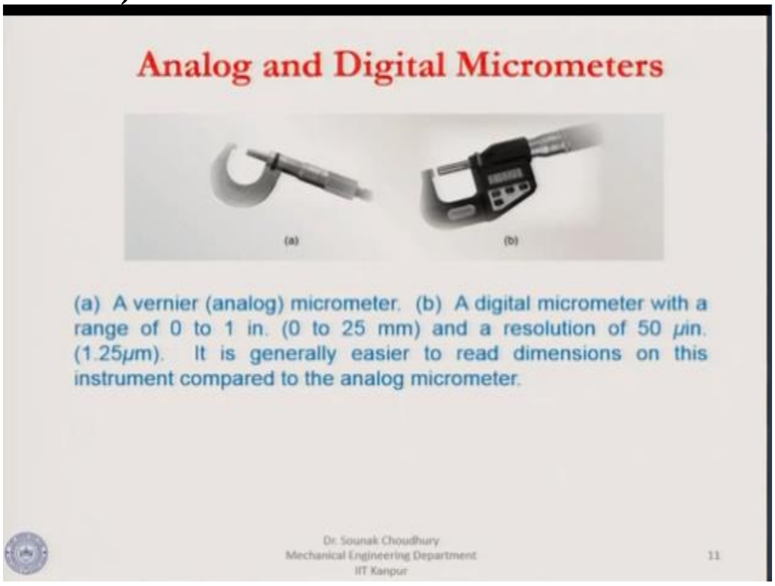
So, each division on the vernier scale therefore will be 49 divided by 50 in millimetre and the difference between one main scale division and one vernier scale division will be 1 minus this because one small division on the main scale is 1 millimetre and on the vernier it is 49 /

50 because the vernier scale division equal to 49 division in the main scale, if this is 50 here that will envelop 49 of the main scale.

Therefore, it is 49 / 50 and 1 minus that will be the vernier scale division. This will be equal to 0.02 which is considered to be the least count of the vernier caliper. In this way the least count has to be found out and less value of the least count it is better because in that case the accuracy is more; accuracy of measurement for example, here it is 0.02. Therefore, we can have only 2 decimal levels, because that is how we found out that the value is 27.42.

So you can see that there are 2 decimal levels but suppose if we had the least count as 0.002, in that case, up to 3 decimal levels we could have the measurement, and that means the measurement is more accurate. In that case, accordingly will have finer scale in the vernier and that will give you more accurate reading of the diameter.

(Refer Slide Time: 19:48)



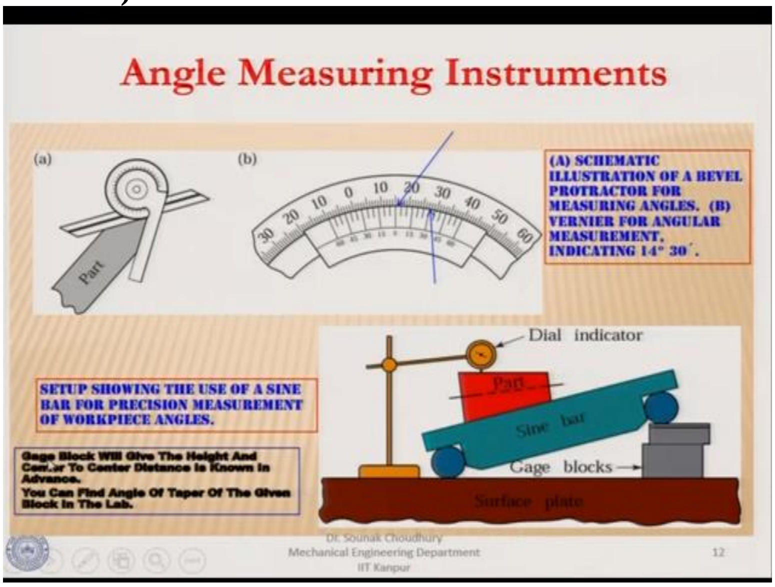
Now, these are the analog and the digital micrometres, analog micrometre has the main scale and round scale and rotating scale. Here it is round one and here it is the main scale. This is the vernier and then again depending on the dimension that is showing here, you can read from the main scale and the new round scale what is the actual reading up to the decimal level depending on the least count of the micrometre.

This is an analog micrometer because you have to read from the scale and this is the digital one, that is, whatever you are reading this will be displayed on the display. That is the difference. A vernier which is analog micrometre here and the digital micrometre with a range of 0 to 25 millimetre. Within that you can find out the diameter.

So, you can measure the diameter in this range and of course the resolution is 1.25 micron. It is generally easier to read dimensions on this instrument compared to the analog micrometre it is because displayed. The vernier scale is very fine, so, to find out which graduation is matching with the main scale is difficult.

This can give rise to certain error. That will not happen if it is a digital micrometre, the digital micrometres will be more expensive, but here it is easy to find out the dimension because it will be displayed.

(Refer Slide Time: 21:50)



We will see the digital micrometres and analog micrometres in the lab. Here is what I was talking about, that this is the protectors; suppose we have a part as shown in the gray; this part can be placed in this or this is rotated about this axis. It can be adjusted according to the angle and then the angle will be displayed here that what is the angle that is shown here.

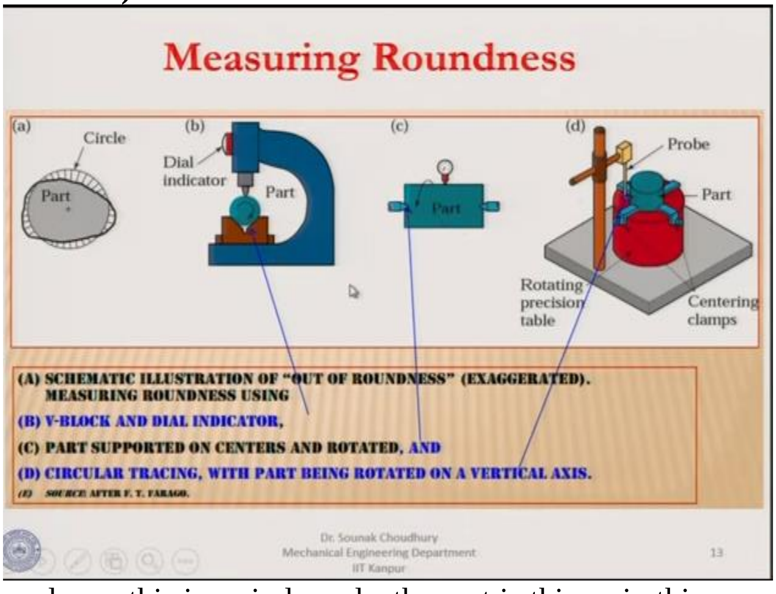
This is here it is shown in more details that there is a vernier scale and there is a main scale and you can more accurately find out what is the dimension depending on the least count of the vernier scale. That means that up till which decimal level you can find out that depends on that. This schematic illustration of a bevel protector here shows the measuring of the angle and this is the vernier.

This is more details of the vernier scale here for angular measurement and in this as it is shown it is indicating that here it is showing 14 and the minute and in the second it is showing the 30. Here is the setup that shows the use of a sine bar; this is the sine bar, in green it is shown here, for the precision measurement of the workpiece angles. This is the part; what should be the taper angle that can be found out from this sine bar, this is a flat and this is the taper or both are tapered.

In this case this taper angle has to be measured. The sine bar is attached to this and adjusted to this surface. With the help of the gauge blocks and the dial indicator will show you what is the angle with respect to this. Gauge blocks measure the height, and the centre to centre distance is known from here to here.

In that case geometrically we can find out the angle of taper of the given block in the lab that you normally do and we also ask our students to do that to find out this because here the this distance from here to here is known and what is the height that will be known here by these gauges?

(Refer Slide Time: 24:37)



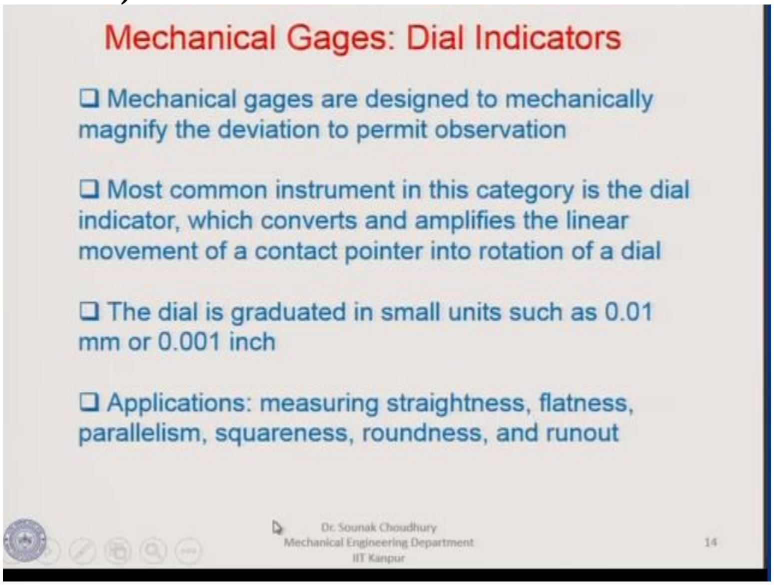
For measuring roundness, this is a circle and s the part is this so in this case, you can see that part is not really circular, this is highly exaggerated. It does not happen so much. This can be measured by the dial indicator while putting the cylindrical job on a V block this is the V block and this is the cylindrical job which is rotated manually after putting that on the centres, as it is shown here.

These are the centres. Job is rotated about these centres and then dial indicator shows you the out of roundness in this cross section, then it can be put in this here in this cross section, next in this cross section and so on. And the average can be taken along the length or it can be put on a V block as I said it can be rotated manually and then there is a dial indicator that is used to measure the out of roundness.

This is the schematic illustration of out of roundness. This is the measuring roundness using the V block and the dial indicator. Parts are supported on centres and rotated as shown in the slide.

The probe is a stylus and that stylus is like the stylus of the dial indicator. This is the dial indicator and the probe and depending on the out of roundness of this when it is rotating, it will show what will be the out of roundness.

(Refer Slide Time: 27:21)

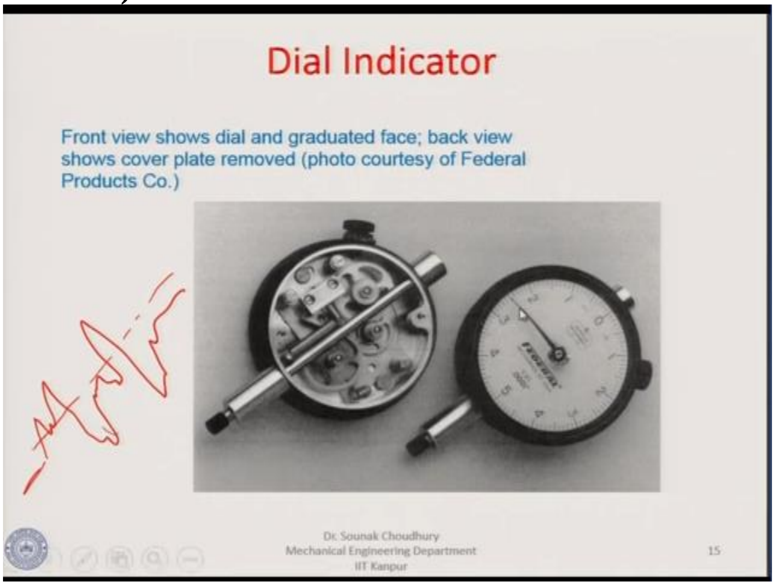


The mechanical gages, dial indicators are designed to mechanically magnify the deviation to permit observation. Most common instrument in this category is the dial indicator which converts and amplifies the linear movement of contact pointer into rotation of a dial. I will show you later what does it mean to convert and amplify the linear movement of a contact pointer into the rotation of a dial.

The dial is graduated in small units such as 0.01 millimetres or even it can be 0.001 millimetre depending on the accuracy of the dial indicated that I already told you.

Applications are for measuring straightness, flatness, parallelism, squareness, roundness and runout. These are the purposes for which the dial indicator can be used.

(Refer Slide Time: 28:30)

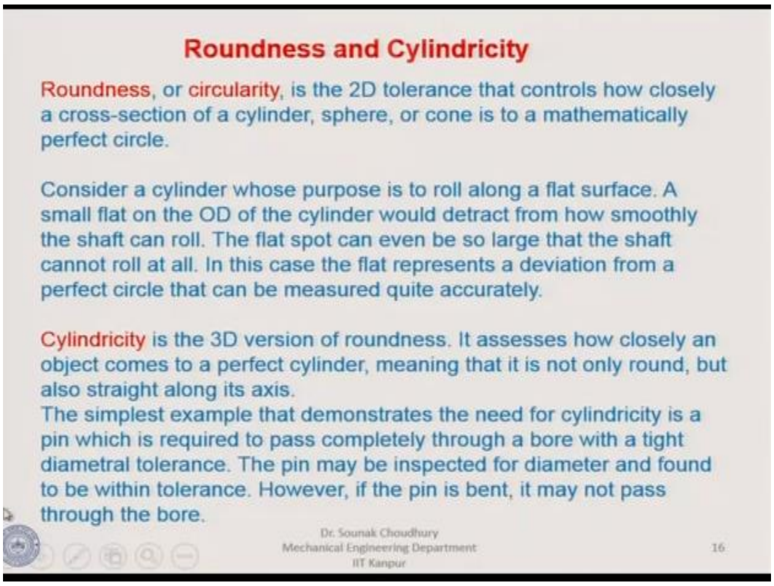


Here is the dial indicator, this is the probe and this probe is connected to the gear train and those gear trains are extremely sensitive. Therefore, as the probe goes up and down depending on that on the undulations in that case, this will be highly magnified by the gear train which is here and then accordingly the indicator will show you the deviation or the indicator will show you what are the discrepancies.

For example, suppose if it is the undulations you want to find out on the surface roughness, this is the surface highly exaggerated. When the dial indicator will move on this the dial indicator will go up when it will come here if this is the main line on which the dial indicator will go, dial indicator pointer is here which is 0 then when the dial indicator is moving, it will come at this point here. Then the dial indicator will go up then again it will go down and this will be indicated by the pointer here.

Now, accordingly you will find out what will be the roughness values. Similarly, we can also find out what is the value of the out of roundness because we said that for straightness we can find out for the flatness, we find out parallelism, squareness, roundness runout, all these defects we can measure using the dial indicators. And this is the design of the dial indicator.

(Refer Slide Time: 30:23)



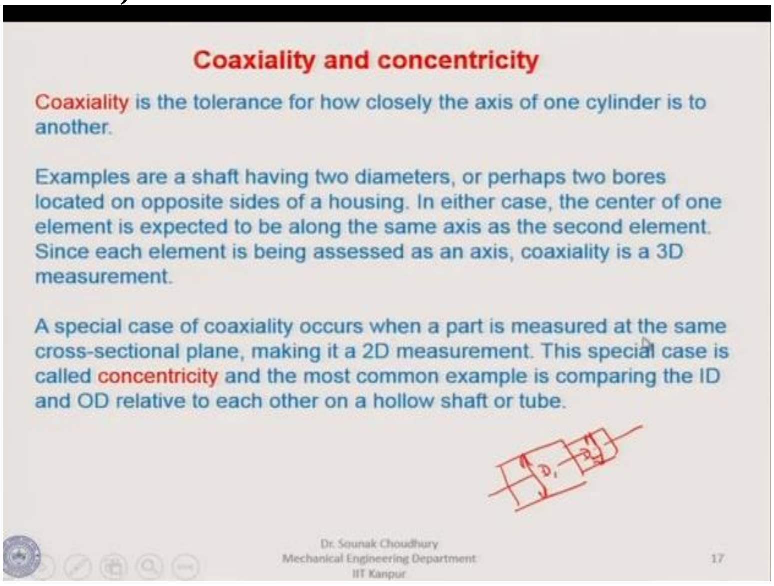
Roundness and cylindricity: let me give you a basic concept of the definition. Roundness or circularity is the 2 dimensional tolerance that controls how closely a cross section of a cylinder, sphere or cone is to a mathematically perfect circle. Roundness is a mathematically perfect circle where on the periphery all points will have a same radius from the centre.

Now, here we are saying the circularity or roundness is the 2 dimensional tolerance that says how closely a cross section of a cylinder. Let us see if this is a cylindrical part. Then at any point here we have to see how much it is deviating from the perfect mathematically given circle, that is what is the roundness. Now, consider a cylinder whose purpose is to roll along a flat surface. A small flat on the outer diameter of the cylinder would detract from how smoothly the shaft can roll.

The flat spot can even be so large that the shaft cannot roll at all, in this case the flat represents a deviation from a perfect circle that can be measured quite accurately. This is just an example. We are exaggerating by saying a flat, what we mean to say is that it is the cylinder which is capable to roll along a flat surface and if there is a flat on the cylinder it cannot roll. That is out of cylindricity. By saying that there is a flat on a shaft, we mean that it is out of cylindricity.

Cylindricity is the out of roundness; the cylindricity is the 3 dimensional version of a roundness, it assesses how closely an object comes to a perfect cylinder, meaning that it is not only round, but also straight along its axis. The simplest example that demonstrates the need for cylindricity is a pin this is required to pass completely through a bore with a tight dimensional tolerance. If the pin is bent, it may not pass through the bore, so, that is what we are saying that then it is not the cylindricity.

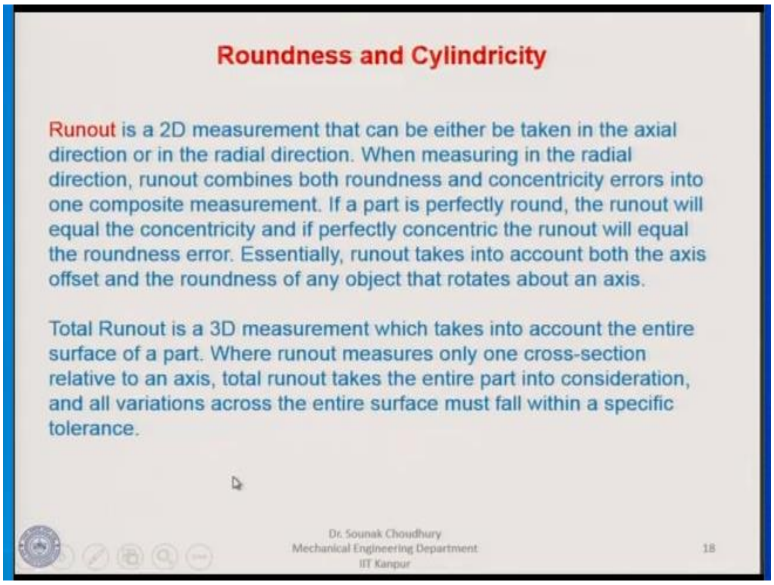
(Refer Slide Time: 33:08)



Coaxiality is the tolerance for how closely the axis of one cylinder is to another one, whether they are coaxial or not. Example of coaxiality is the shaft having two diameters. This is a shaft and it may be the same axis and diameter D 1 and diameter D 2 both have the same axis, but has 2 bores located on opposite sides of a housing they also have the same axis in either case the center of one element is expected to be along the same axis as the second element.

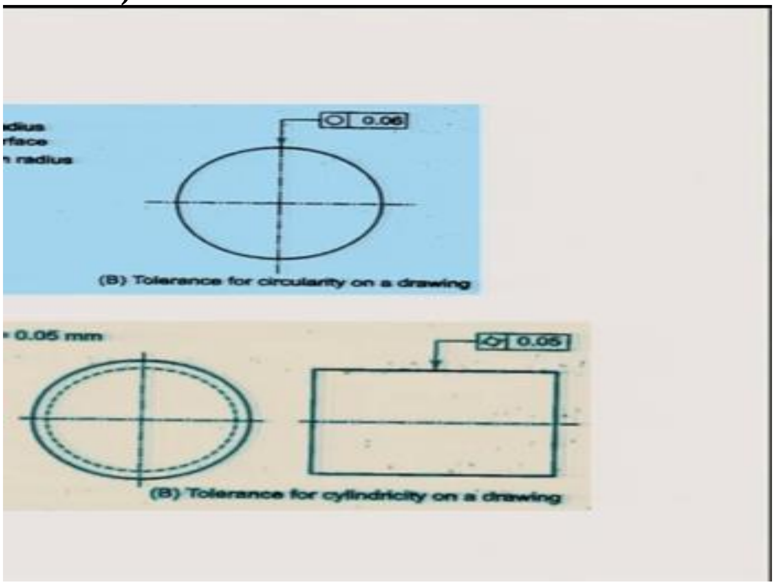
Now, since each element is being assessed as an axis, coaxiality is the 3 dimensional measurement. A special case of coaxiality is the concentricity. Here concentricity occurs when a part is measured at the same cross sectional plane making a 2 dimensional measurement. The most common example is comparing the internal diameter and outer diameter relate to each other on a hollow shaft on a tube, when the tube is hollow in that case when you are considering the internal diameter or external diameter, here the concentricity will come into picture.

(Refer Slide Time: 34:48)



Runout is a 2 dimensional measurement that can either be taken in the axial direction or in the radial direction. Now, here this is the explanation given; runout takes into account both the axis offset and the roundness of any object that rotates about an axis, this is important. Now, here when measuring the radial direction run out combines both roundness and concentricity errors; this is also important here.

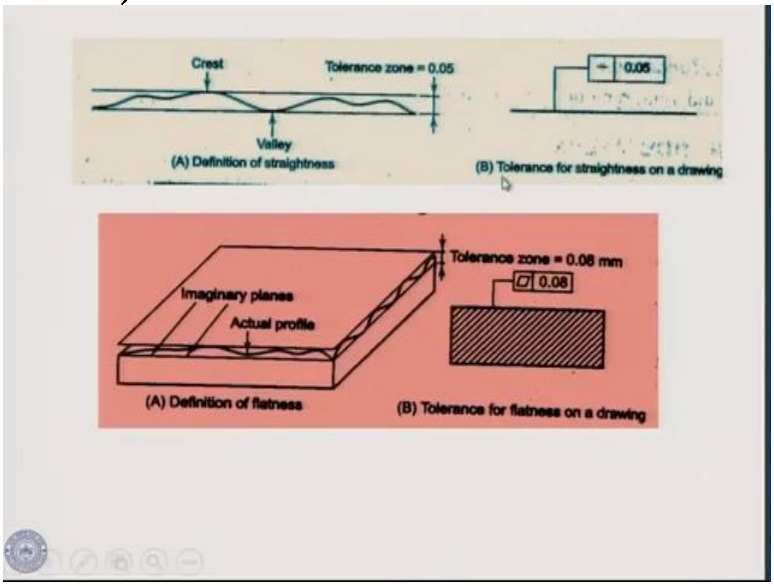
(Refer Slide Time: 35:29)



Here these are the diagrams through which we are showing what is the circularity, actual surface maximum radius, minimum radius and how this can be designated. The tolerance of circularity on a drawing can be given as shown in the slide. We will see in more details in a table. This is the tolerance for the cylindricity on a drawing. All these symbols are tabulated. This is the cylindricity definition.

That means, here we have the smallest cylinder and the largest cylinder and within that it can fluctuate like this from point to point and this is how it is given; it cannot be more than 0.05.

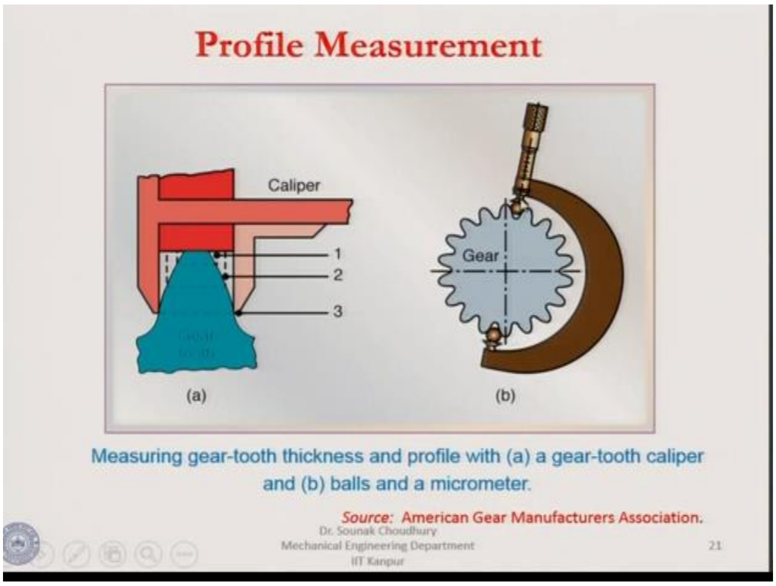
(Refer Slide Time: 36:30)



These are the definitions of the straightness and the tolerance for straightness or a drawing can be given in this way then when you have the surface, you are showing it in this way that this is the straightness which is within 0.05 it cannot be more than 0.05, it should lie within this value, so this is called the tolerance zone. Flatness here we have the imaginary planes this is the actual profile, but absolutely flat profile if you are putting it with the lower line, with the valley and the peaks.

This constitutes difference between these two lines or these two imaginary planes which is the tolerance zone. So, this is how the flatness on a drawing can be shown. This is a drawing this is a surface and the flatness of this surface should not be more than 0.08 millimetres; when nothing is mentioned here in engineering drawing it always means that this is in millimetre.

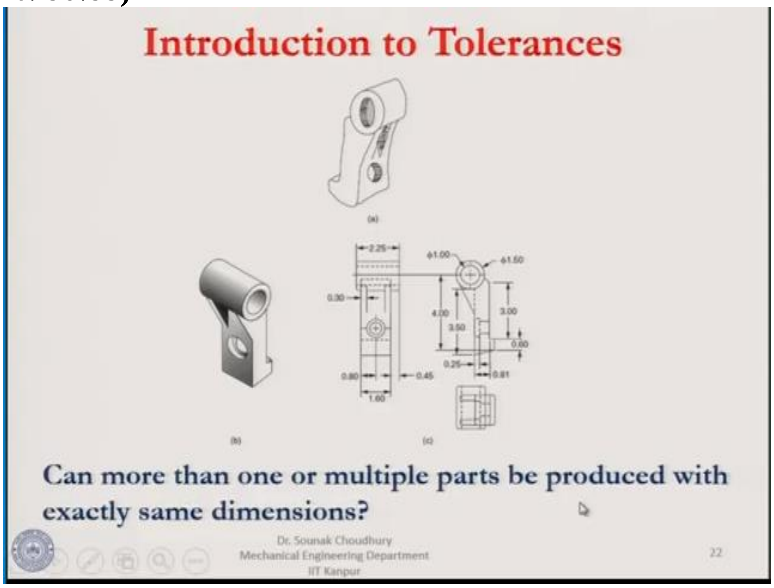
(Refer Slide Time: 37:52)



Next that we see is the profile measurement, so profile measurement is important particularly in the lab when we measured the gear profile it can be the bevel gear, it can be the spur gear it can be the worm and worm gear. This is important to exactly find out what is the profile. Here a gear tooth is shown. And this is how the gear tooth thickness can be measured with the help of vernier caliper.

And here is the micrometer ball and micrometer ball is made so that it sits on the pitch circle diameter of the gear. This will be accurately measuring the thickness and the profile.

(Refer Slide Time: 38:53)



Introduction to tolerance: what is tolerance? We have a part and this is the pictorial view of the part and this is the drawing. In this drawing there are 3 views shown and all the dimensions are visible, these are given according to the designers intent and a designer is telling that it has to be this hole of this diameter and this has to be 1.5. Now, when the manufacturer makes this part, the manufacturer may not be able to maintain exactly that requirement.

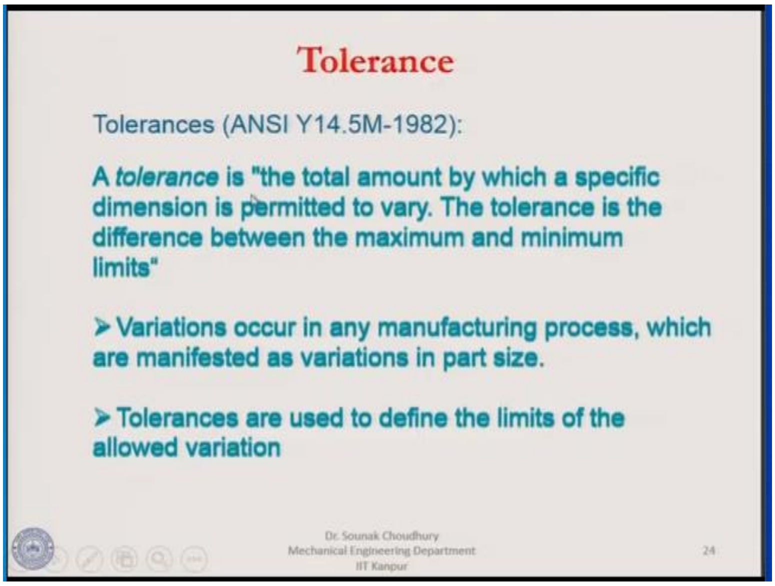
Therefore, it is not possible to maintain exactly 1.5 unit of something, let say 35 millimetres of length or 25 millimetre of a diameter. It is very difficult to maintain that length exactly. Therefore, there should be some kind of variation in that length that is accepted by the designer which means that if it is 35.01 millimetre, the assembly can still be done or if it is 34.995 still it is fine because it can be assembled.

Similarly on the diameter also you can have some kind of tolerance because the exact diameter is very difficult to machine. This is the concept of tolerance. Can more than one or multiple parts be produced with exactly the same dimension? We said that it cannot be.

(Refer Slide Time: 40:41)

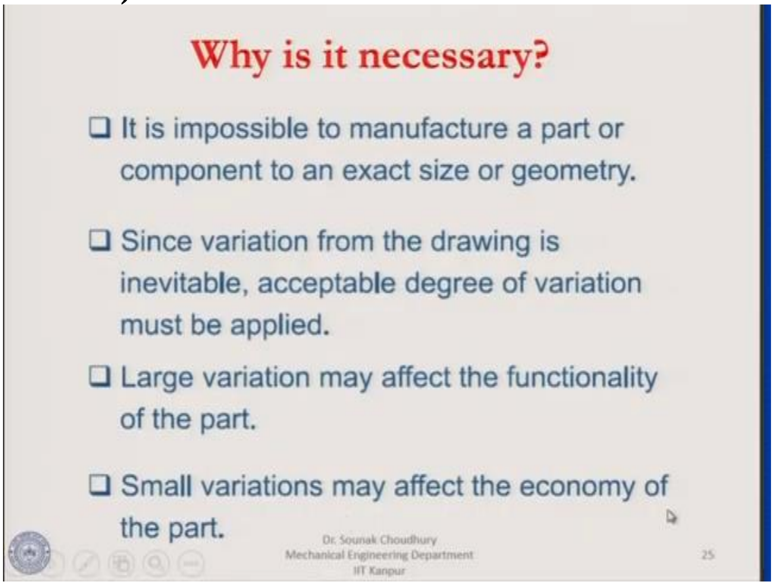
The allowable deviation from a standard for example, the range of variation permitted in maintaining a specified dimension in a machined piece is the tolerance. This is simply taken from the webster dictionary how the tolerance is defined. This is the allowable deviation I gave you example, this means that if we have this shaft, it can have the maximum diameter this and it can have the minimum diameter like this. The tolerance will be this much from here to here, this is the concept of the tolerance.

(Refer Slide Time: 41:22)



Therefore, a tolerance is the total amount by which specific dimension is permitted to vary. The tolerance is the difference between the maximum and the minimum limits. Variations occur in any manufacturing process, which are manifested as variation in the part size. Tolerances are used to define the limits of the allowed variation, how much we can allow; that is why it is called the tolerance - how much you can tolerate. This is the American society definition of the tolerance.

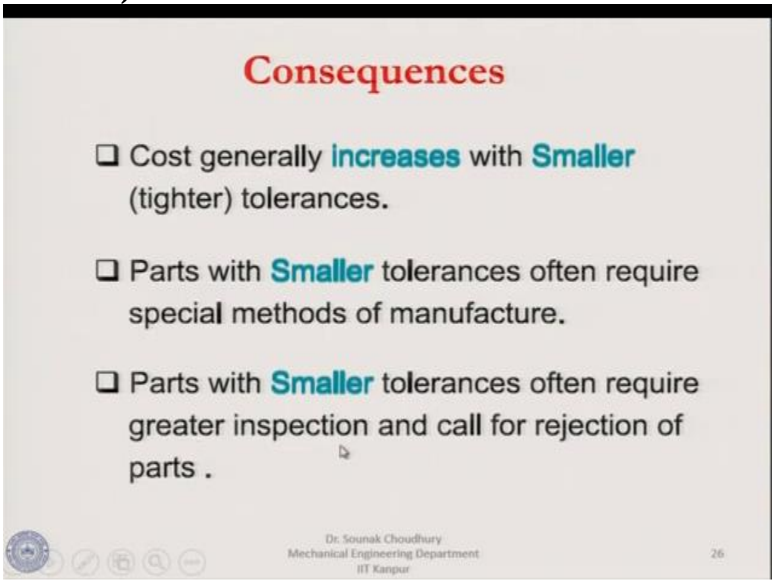
(Refer Slide Time: 42:01)



Why it is necessary because it is impossible to manufacture a part or a component as an exact size of geometry. Since the variation from the drawing is inevitable, acceptable degrees of variation must be applied. Large variation may affect the functionality of the part. Suppose if it is to be assembled, let us say that a shaft goes into the inner bore of the bearing.

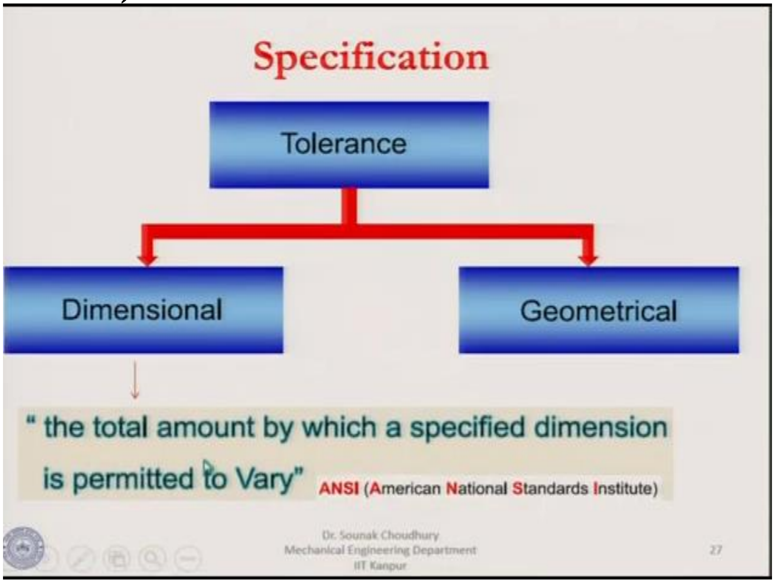
If it is not within the tolerance then the shaft may rotate with respect to the bearing that we do not want; suppose we want the shaft to be rotated along with the inner races of the bearing. Therefore, if the diameter is bigger than what is within the tolerance zone, then it cannot be assembled, that is another problem. So, what I mean is that the large variation may affect the functionality of the part. Small variations may affect the economy of the part.

(Refer Slide Time: 43:12)



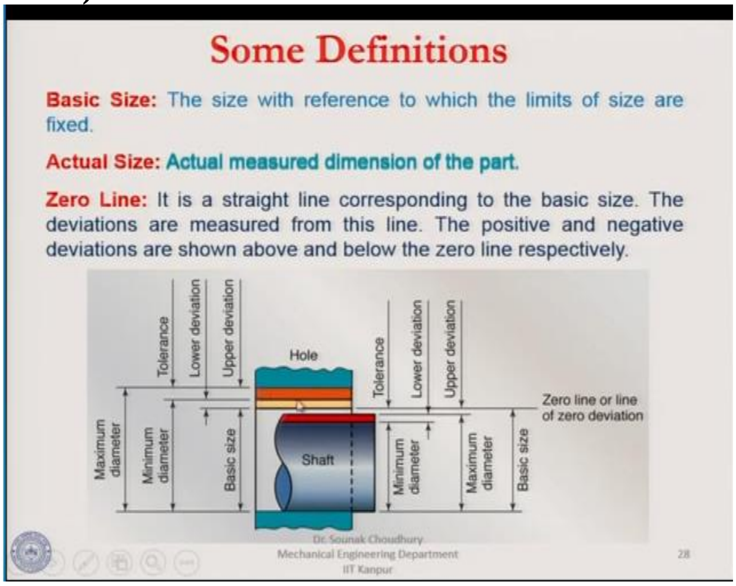
Cost generally increases with the smaller tolerances because if it is very tight tolerance, let us say 0.0001, you cannot really get it within that fine tolerance and if you have to get it then the machine has to be very accurate. Then the cost of the product will be very high. Parts with smaller tolerance also is called a tight tolerance often required special methods of manufacture. Parts with smaller tolerance often require greater inspection and call for rejection of the parts.

(Refer Slide Time: 43:51)



Now, it can be dimensional or geometrical, the total amount by which a specified dimension is permitted to vary.

(Refer Slide Time: 44:00)

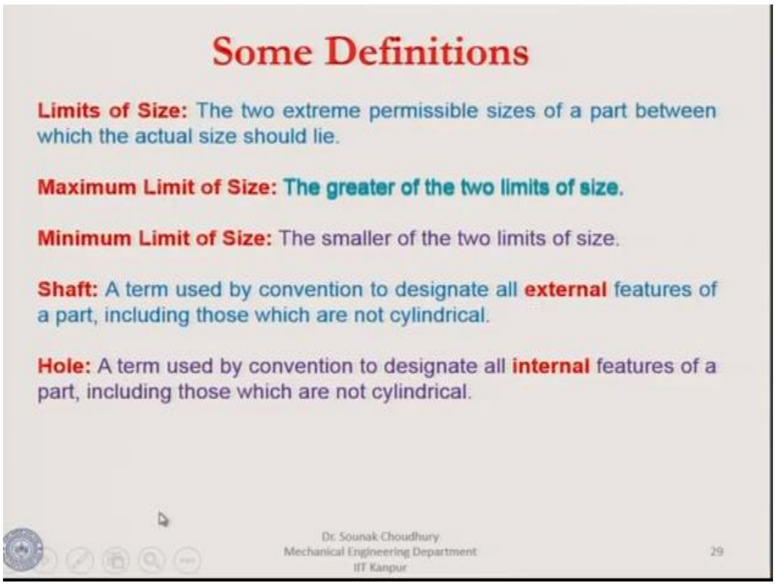


Let us see here. Here we have the hole, this is the hole and here is the shaft. For the shaft, this is the minimum diameter and this is the maximum diameter. So, this is the tolerance for the hole, the basic size of the hole is from here to here. This line we are calling as a zero line or line of zero deviation. Therefore, from this zero line how much we can have as the deviation we are saying that we can have up to this. Now, it is exaggerated .

We are saying that the hole diameter can be from here to here so this will be the upper deviation. In case of shaft, this will be the upper deviation from here because upper deviation for this will be from here and this is lower deviation. This means that the shaft cannot have more than that. This is the upper deviation and the shaft cannot have less than that that is the deviation. Basic size I already told you that this is that the minimum diameter is here and the maximum diameter is here.

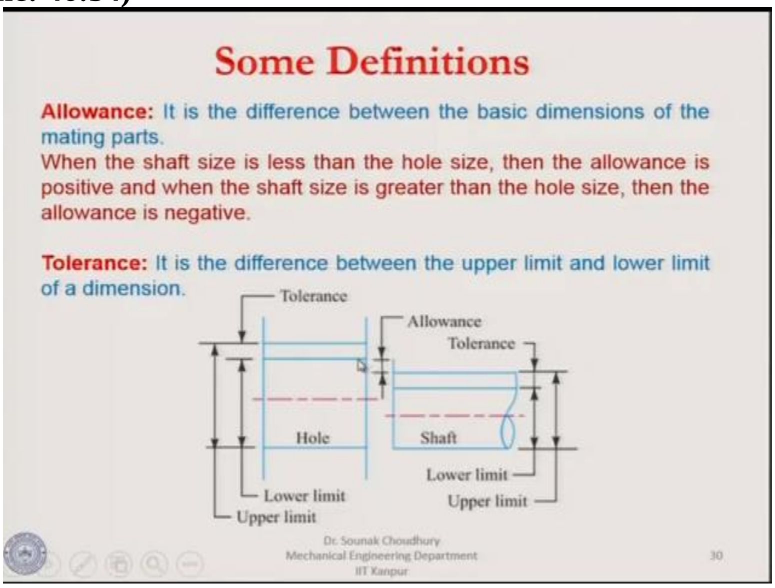
So, the tolerance will be from here to this and here the tolerance for this will be this is for the shaft and this will be from here to here, this is the tolerance from here to here this will be for the hole. I will show you in more details a little later.

(Refer Slide Time: 46:12)



In the next diagrams, limit of size, maximum limit of size, minimum limit of size shaft and hole. These are some definitions. Limit of size is the two extreme permissible sizes of a part between which the actual size lies. Maximum limit, minimum limit these are variation of the limit of size. Shaft and hole terms are used by convention to designate all external features of a part including those which are not cylindrical and internal when they are mentioned through the hole term.

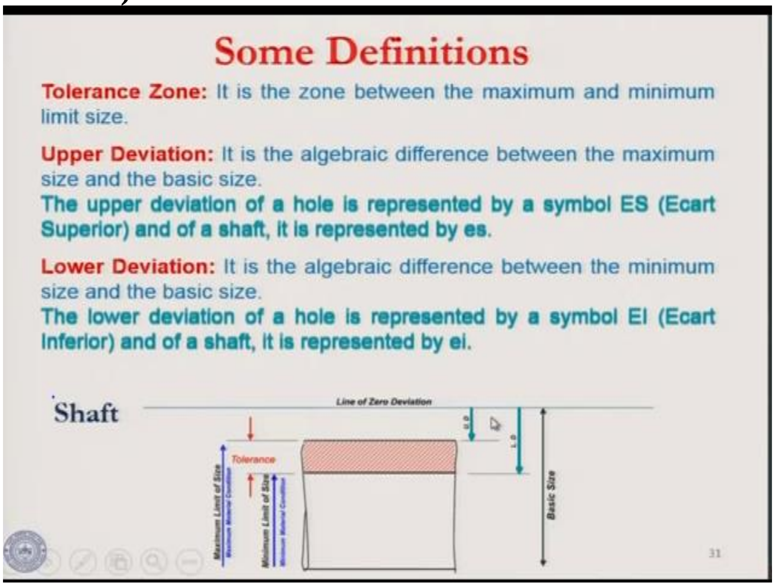
(Refer Slide Time: 46:54)



Now, here you can see what is allowance and what is tolerance? Allowance and the tolerance are given in this diagram. Let us say that this shaft goes into the hole. These 2 lines coincide, in that case we have a tolerance in the shaft as this much and tolerance in the hole as this much meaning the maximum diameter of the hole can be this and the minimum diameter of the hole can be this that this shaft can go in here; these are all highly exaggerated.

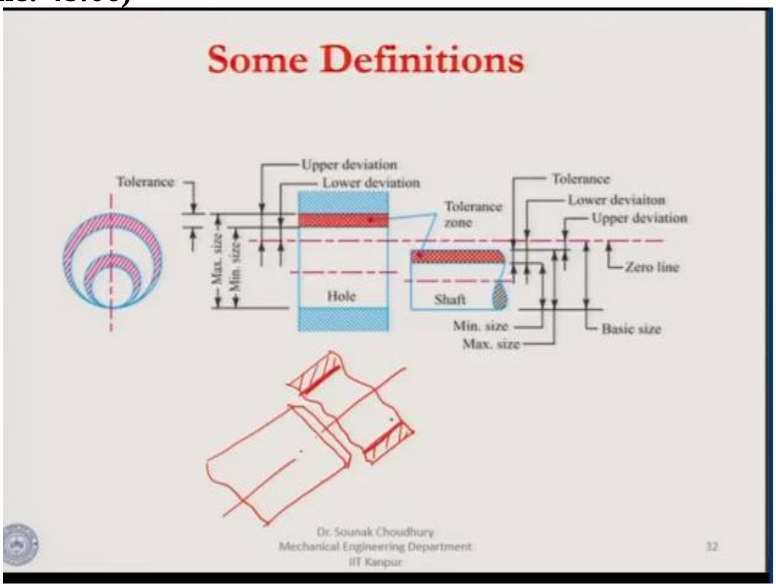
Accordingly you will see that the allowance that we are giving is from, this is as per definition it is from here to this line that is the lower limit of the hole and the upper limit of the shaft.

(Refer Slide Time: 47:56)



Tolerance zone, upper deviation, lower deviation that I have already shown you in this diagram.

(Refer Slide Time: 48:06)

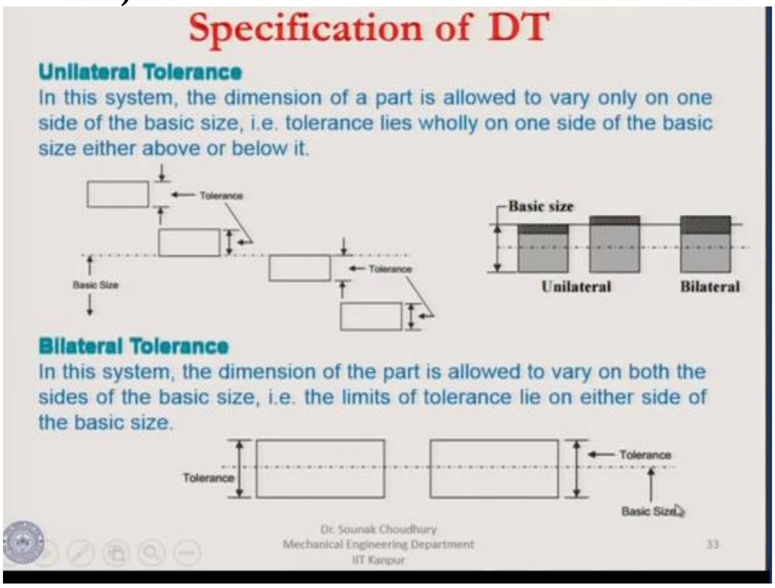


Now, here also it is shown the same way that I have already shown it to you those definitions that the tolerance zone for hole will be here it is given in a better way. You can see this now, here we are saying that this is the zero line. With respect to zero line, we will show the upper deviation, lower deviation and the tolerance here with the basic size of the shaft and the hole. So, with the minimum size of the hole here and the maximum size of the hole here similarly for the shaft also it is shown here.

Suppose we have a shaft and this shaft has a chamfer and it goes in a hole here. Our basic concept is the following that once it goes into the hole let us say this is the shaft and they are coaxially made. If suppose we want this shaft to be snugly fitting inside the hole, snugly fitting is tightly fitting. In that case, the internal diameter of the hole should not be more than the outer diameter of the shaft.

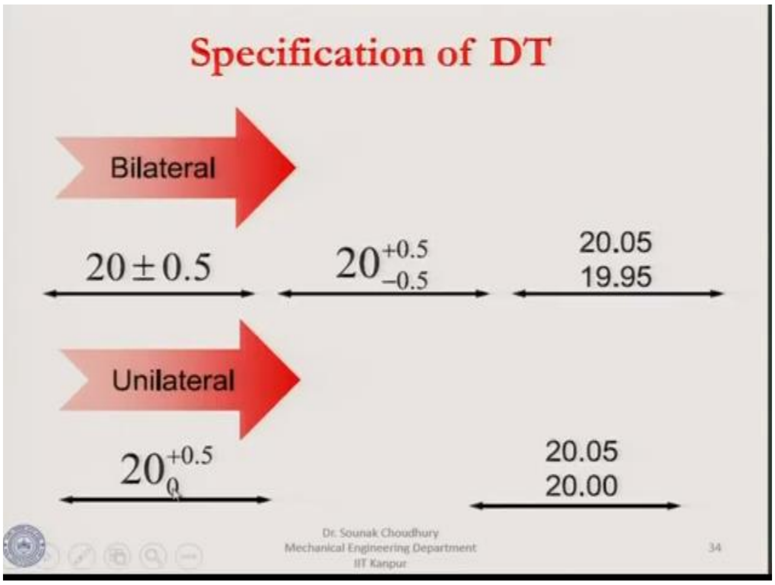
That means the shaft diameter should be little more than the internal diameter of the hole so that it can sit very tightly. The tolerance has to be given accordingly. I will show you what are those groups of tolerances, depending on which it is given the two mating surfaces are tightly fit with each other or it is not so tightly fitting with each other.

(Refer Slide Time: 50:12)



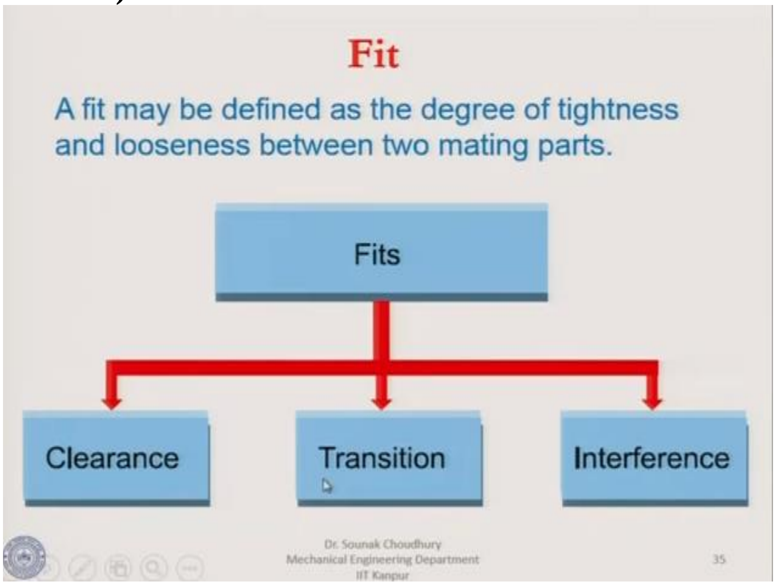
Tolerances are given as the unilateral tolerance in one side or it can be the bilateral tolerance where tolerance will be given from both sides.

(Refer Slide Time: 50:31)



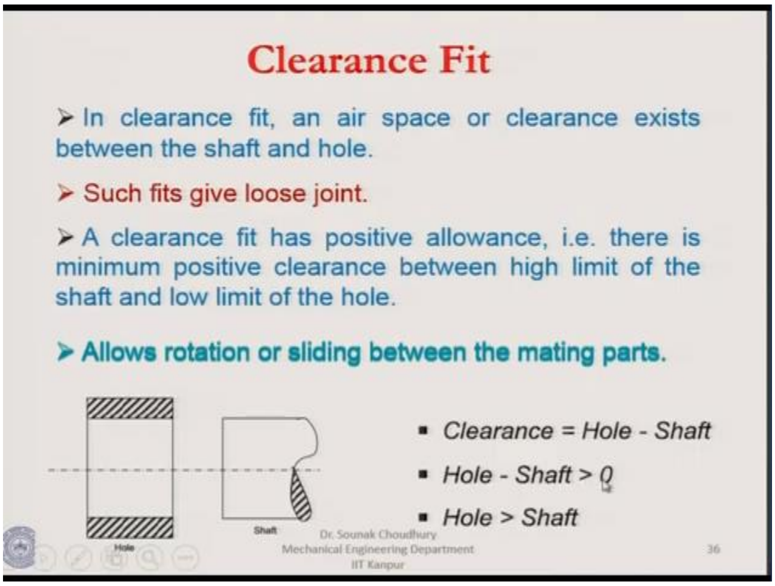
In case of unilateral tolerance, one side will be 0 and in case of bilateral tolerance, that both sides will be tolerance that means upper limit and lower limit both will have some limit.

(Refer slide Time: 51:03)



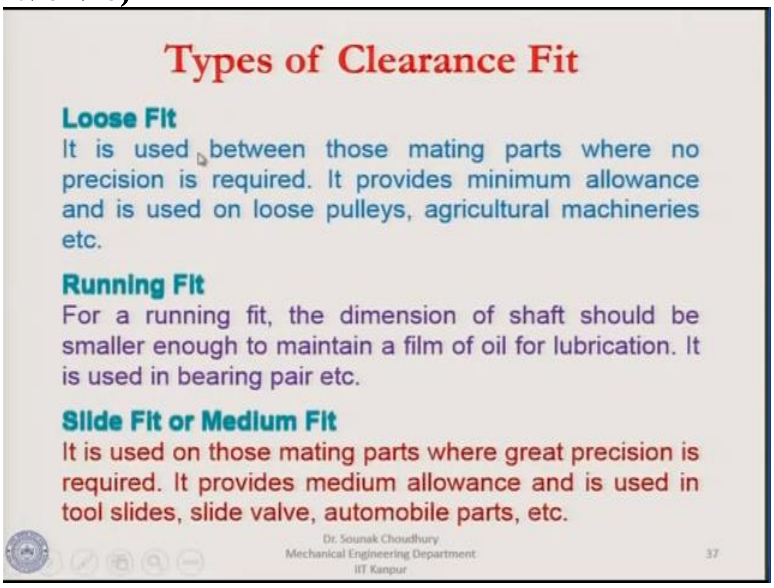
Next we will talk about the fit. A fit may be defined as the degree of tightness. I just started telling you about the shaft going into the hole. When we are having an assembly drawing where parts are shown with respect to each other then we have to give the fit and tolerance on each part, how the part dimension can be deviated from the actual diameter or the length that is called the tolerance and the fit is when the parts are in the assembly, how they are fitting with each other. Fits can be clearance fit, transition fit or the interference fit.

(Refer Slide Time: 51:47)



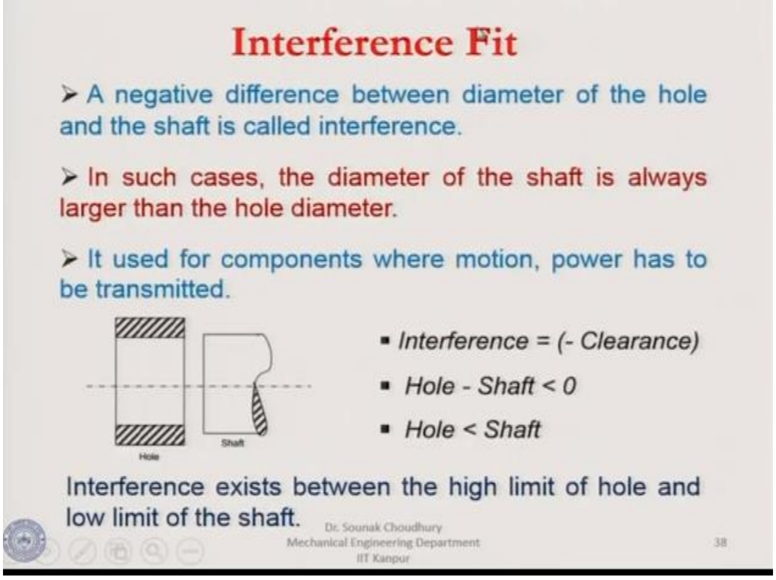
In one word, clearance is the the hole minus shaft diameters. Clearance fit is that the shaft diameter is little less than the hole diameter, so, this is the loose joint. The clearance is the hole minus shaft and the hole is bigger than the shaft. So, hole minus shaft is more than zero.

(Refer Slide Time: 52:13)



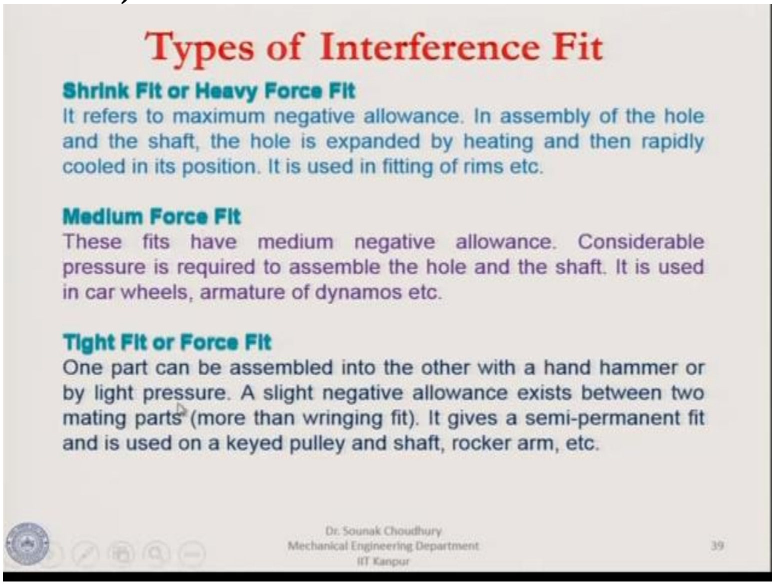
The types of clearance fit can be loose fit, running fit, slide fit or medium fit depending on the level of fitness.

(Refer Slide Time: 52:25)



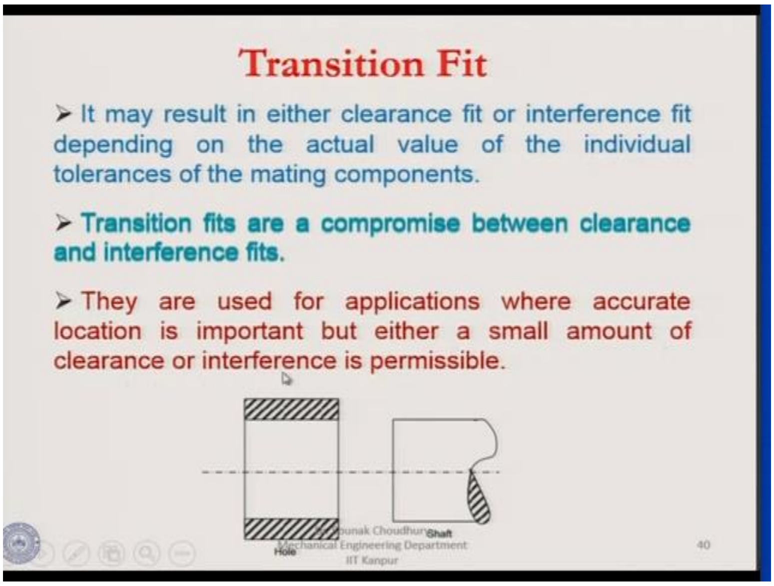
Similarly, we have the interference fit when we will have the hole minus shaft is less than zero that means the hole diameter is less than the shaft diameter. This is the interference fit, diameter of the shaft is always larger than the hole diameter. So, it snugly fits, tightly fits.

(Refer Slide Time: 52:50)



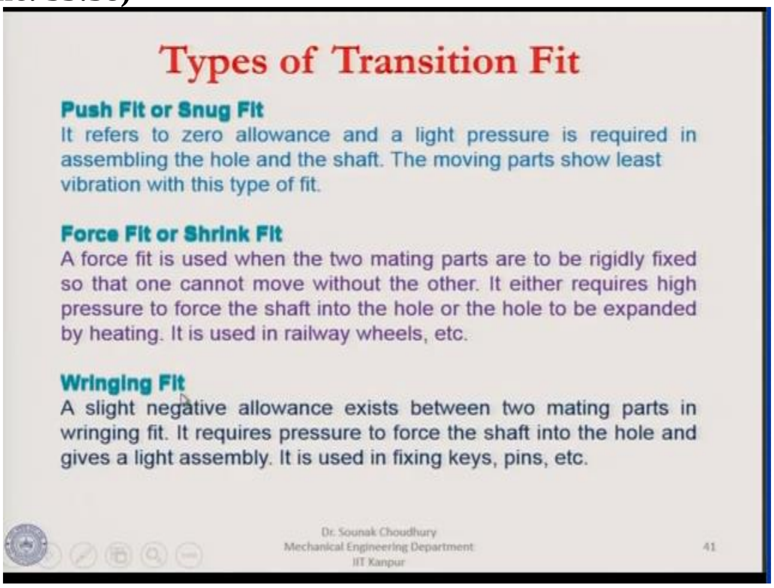
The types of interference fit also are different, shrink fit or heavy force fit where heavy forces are required. The diameter of the shaft is much more. Medium force fit and the tight fit or force fit are the different kinds of fits that are all interference fit.

(Refer Slide Time: 53:11



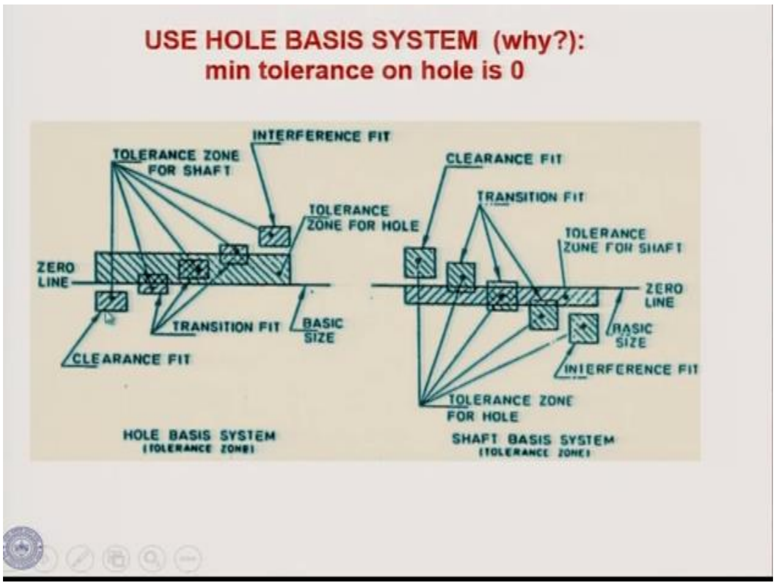
Transition fit it is neither very tight nor very loose. It is a compromise between the clearance and interference fit, they are used for applications where accurate location is important, location has to be very accurate, but either a small amount of clearance or interference is permissible.

(Refer Slide Time: 53:38)



The types of transition fit can be push fit or snug fit, force fit or shrinks fit or wringing fit. These are the various types but all of them are transition fit depending on how much is the level of the transition in each of them. Here it is defined and that is defined based on the level of the transition in the fit.

(Refer Slide Time: 54:10)



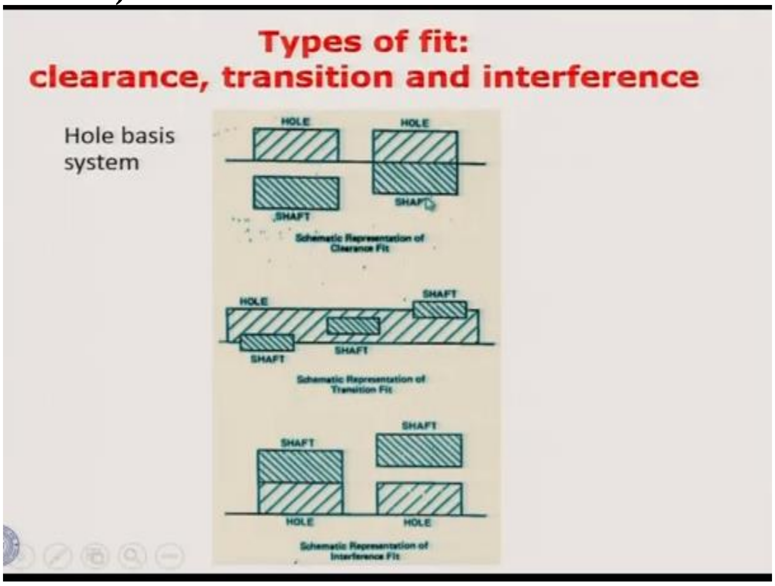
This is very important that we can use the dimension, which is a hole basis or the shaft basis. Normally we use the hole basis and why it is that it is shown here clearly. Use hole basis system so that the minimum tolerance on the hole is zero. Here you can see that this is the hole basis system. Minimum tolerance of this hole is zero and depending on that, we have the tolerance of the shaft. You can see that these are the different kinds of tolerances.

Depending on that if you have the tolerance zone somewhere here, it will be clearance fit because then the shaft will be loosely fitting in the hole and if the tolerance level is here, so this will be the interference fit because it will be very tight and these are the transition fits in between. Now, similarly, if it is shaft-based system then we will have the minimum tolerance zone here of the hole and the tolerance zone for shaft will be here with the zero line and since here this was the hole base and for the hole base zero line is concerning here.

Then for the shaft base, zero line will be considering and shaft will be on this is the tolerance level. Hole then is changed, but why we said that you use the hole basis because the tolerance on hole is minimum, tolerance is zero; here you can see that minimum tolerance is nonzero for the hole which is very difficult to make in the sense that when you have made a hole you have a particular minimum tolerance, you can increase the diameter of the hole, but you cannot decrease it.

But in case of shaft, you have the diameter that you can decrease but you cannot increase. Therefore, the hole system should be used normally because the minimum tolerance on the hole is zero.

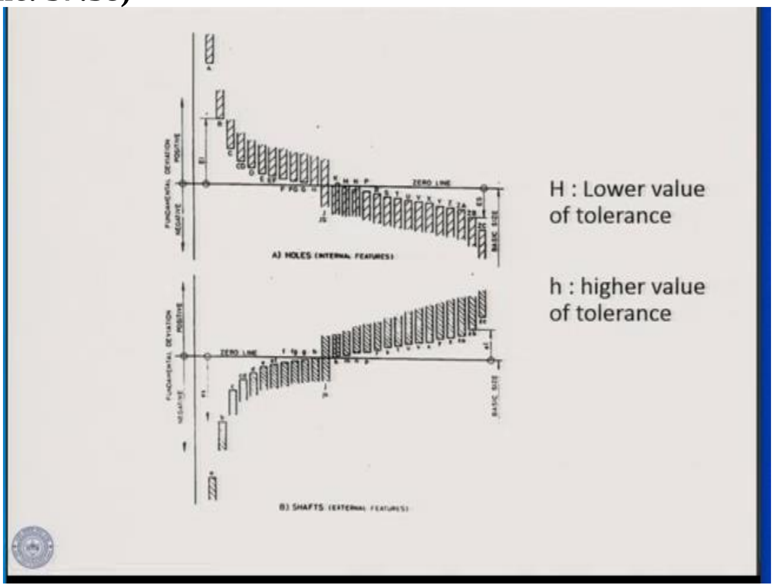
(Refer slide Time: 56:43)



Types of fit are given here. This is the hole basis system and the hole is constant and then the shaft diameter is changing. Depending on that we have the clearance fit or the transition fit or the interference fit. We said that the clearance fit is when we have the clearance between them but that means the hole and shaft are relatively loosely fitted.

Shown in the slide is a schematic represent of a transition fit that is in between the very tight and the loose we will have the transition fit. This is all in the hole basis because once again I would like to repeat that in the hole basis the hole diameter is fixed and then depending on that you adjust the shaft diameter.

(Refer Slide Time: 57:38)



Here is the system given in more details. So, these are the internal holes and these the shafts, external features. Here you can see that H is the lower value of tolerance and h here is this shaft system. Normally we say that we use the hole system internal features and lower value of the tolerance given so A B C D all the alphabets are used and this is that difference between them as shown in this diagram.

Depending on that you can have the fit and the tolerance given on the part a to give tolerance and fit in the assembly drawing or the sub assembly drawing. Here depending on the clearance between them, you can define whether it is a clearance fit, whether it is a transition fit or whether it is interference. That is all I wanted to tell you about the fits and tolerances and overall about the course that we have discussed within the 5 modules.

All the 5 modules together constitute the course. Lab parts we will show you in the lab, different kinds of processes, including the NC and CNC machines. So, this is the production technology theory and practice. Thank you for your attention.