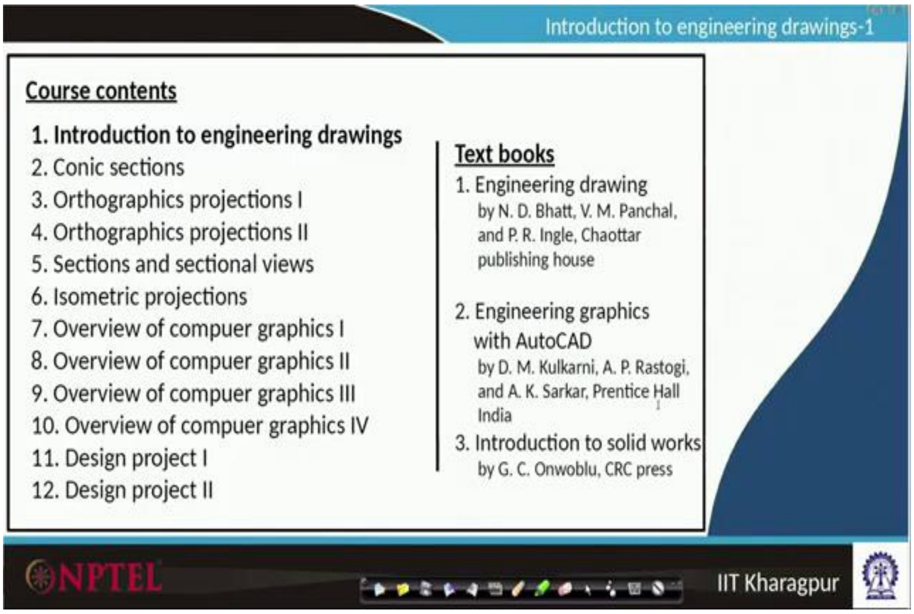
Engineering Drawing and Computer Graphics

Prof. Rajaram Lakkaraju
Department of Mechanical Engineering
Indian Institute of Technology, Kharagpur

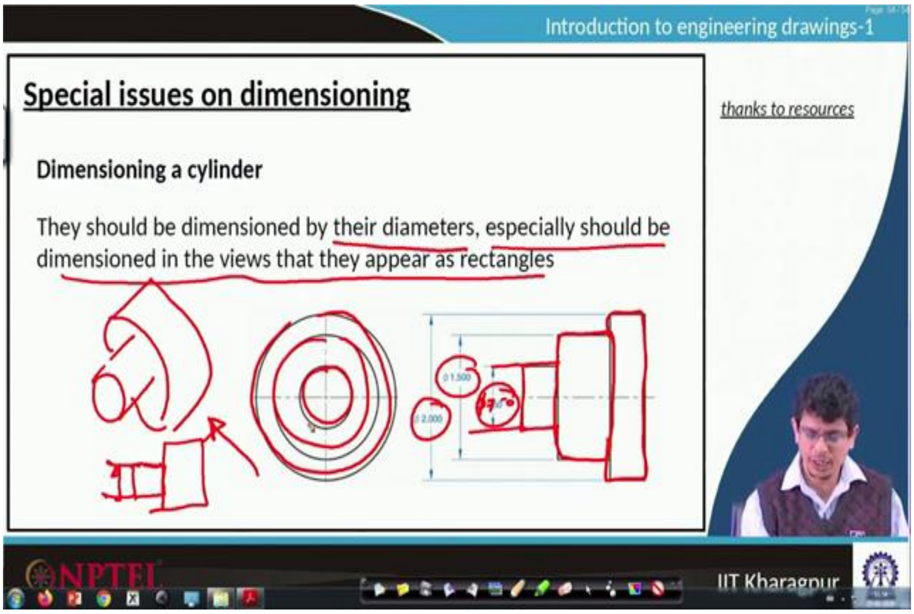
Module – 01
Lecture - 06
Introduction – VI

Hello everyone. Welcome to our NPTEL online certification courses, on Engineering Drawing and Computer Graphics. I am Rajaram from Mechanical Engineering IIT Kharagpur. We are in module 1, lecture number 6 on introduction to engineering drawing. In today's class, we will briefly look at a special dimensioning thing and also tolerances.

(Refer Slide Time: 00:49)



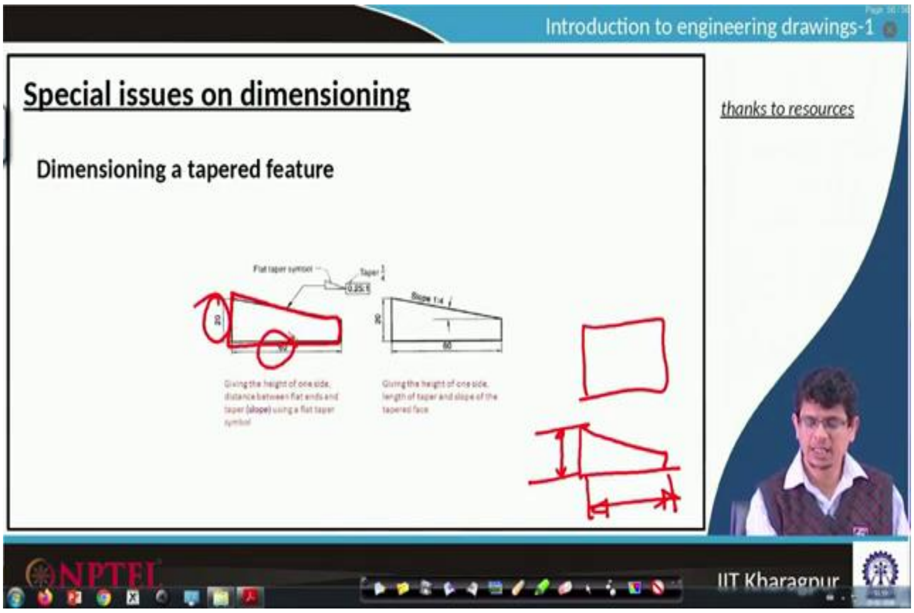
(Refer Slide Time: 00:51)



On special issues on dimensioning in the last class, we looked at different ways of dimensioning it. For example, if it is a cylinder having multiple steps. So, that on the other side of the view we look at like a step one, try to look at diameters if it is diameter we use symbol phi, and smallest one inside and the largest one outside that is the way we try to look at.

So, they should be dimension by their diameters, especially should be dimensioned in the views that they appear as rectangles. So, if we have multiple cylinders such kind of thing, if we are looking from this view, they appear like rectangles. In that case diameter, we are going to show it in this way. We do not usually show diameter in this view everything supposed to be from this one.

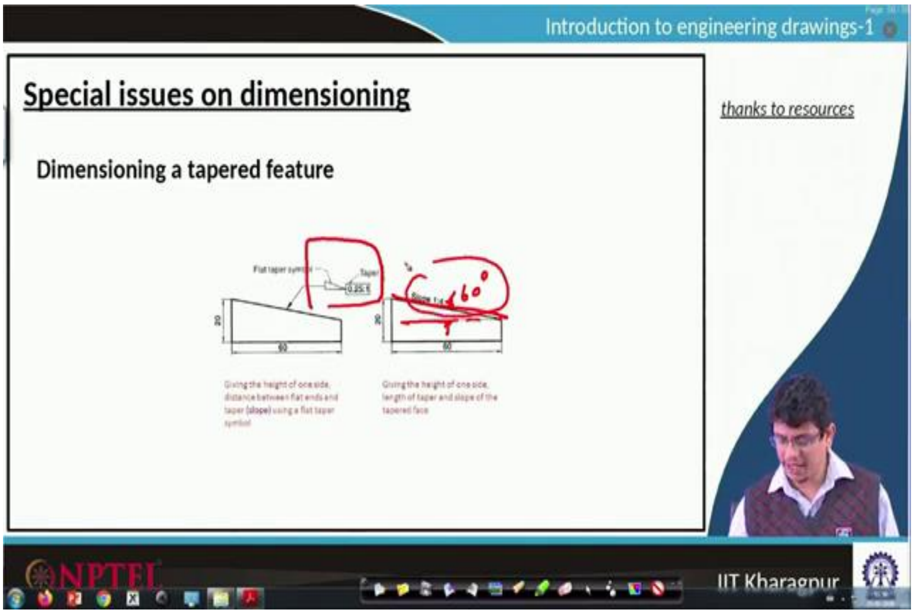
(Refer Slide Time: 02:38)



If there is anything like tapered features, for example, here, we have a tapered feature instead of having a rectangle, perhaps the object might be having such kind of feature. In that case, we show its height, and also we show its length, 20 units and 60 units.

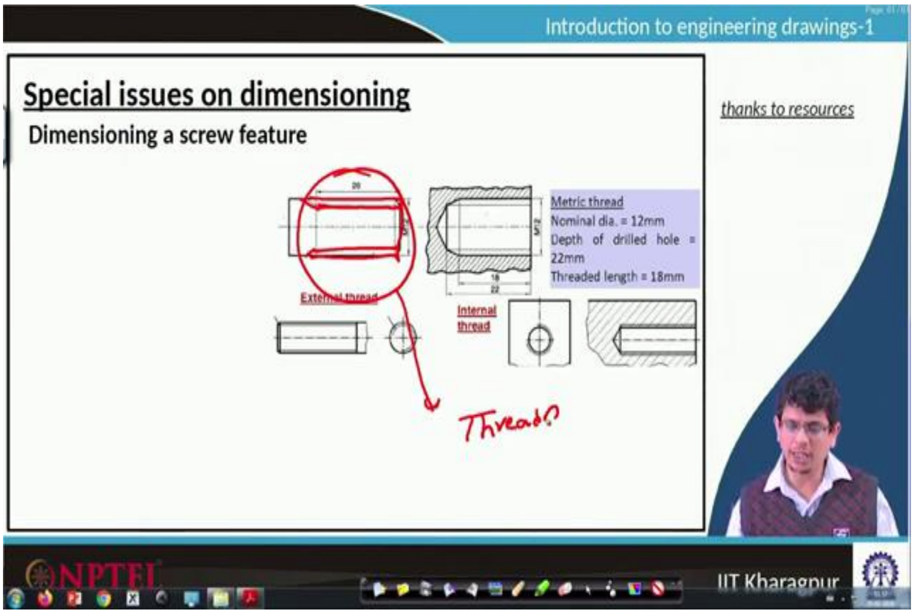
Giving height of one side distance between flat ends and their taper slope using a flat taper symbol, we usually go for tapered features. Here we are showing these dimensions and also something like a tapered one with a symbol of a tapered shape. This is the way we show and this taper perhaps increasing in that ratio; this is the way any taper things we go ahead and show it. The other way is, giving the height of one side perhaps this one length of taper and slope of the tapered face.

(Refer Slide Time: 04:00)



So, the length of the taper is 60 units, and it makes an angle of 60 degrees with the horizontal. So, that way also we can show it one by giving how this taper symbol ratio is going to change, the other way is through slope and angle these are two ways of special dimensioning.

(Refer Slide Time: 04:33)



Similarly, if we have thread screws, etc., we do not put threads directly on that for dimensional purpose and other things. We show particular dimensions that indicate that these are threads.

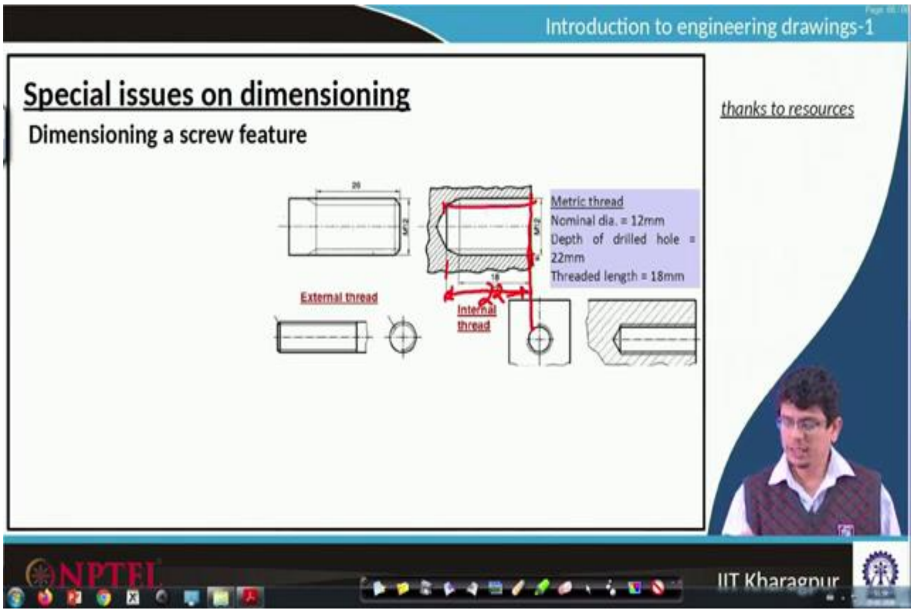
For example, if we take a pen which can be opened into two parts with screw and thread kind of combinations. So, one of them may be having external threads, and the other shall have internal threads. So, when we show these dimensioning for bolts, nuts, pen, screws, caps, etc. there always be external threads and internal threads.

So, there are different ways of showing these internal threads and external threads, more details we will learn in the future classes. However, whenever we see this kind of rectangular portions, tapered ones and curves and lines, it indicates that it is supposed to deal with threads. Moreover, we usually show these threads by metric M portions we will show. The metric threads for that purpose there is a symbol M perhaps it might be having a diameter of 12 mm.

So, if it is external threads we show it from the outer side this is the thread on which we have it. So, M 12 we will show if it is something like the white colour portion of that Reynolds pen, which is having internal thread then, in that case, internal portions we are going to show it.

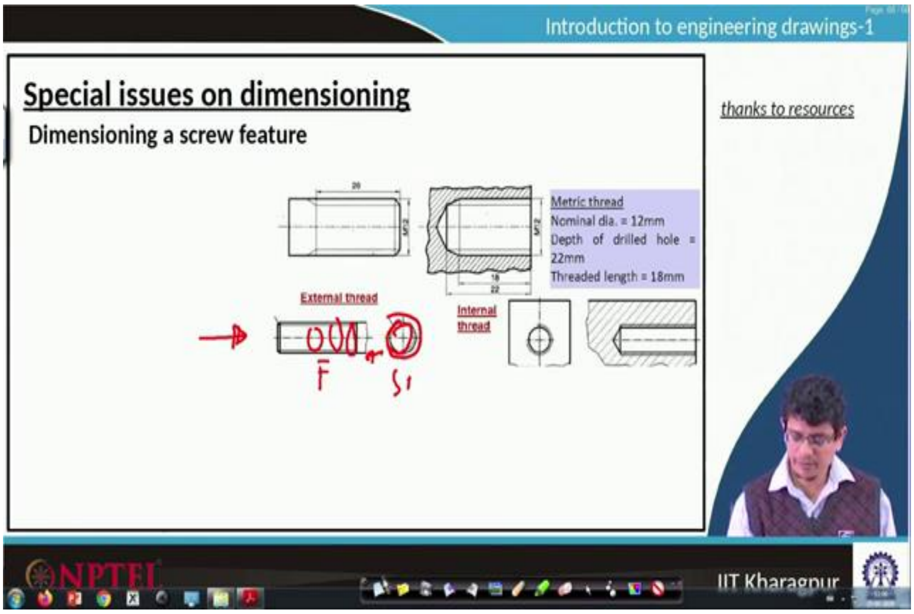
Similarly, whenever these threads bolts nuts and other kinds of things are involved, we usually go with the depth of the drilled hole for example, for this object inside there is a hole in which we have a thread. So, depth is always be required, for example, here in this case 12 mm. So, from one end it is shown 22 mm this one.

(Refer Slide Time: 07:18)



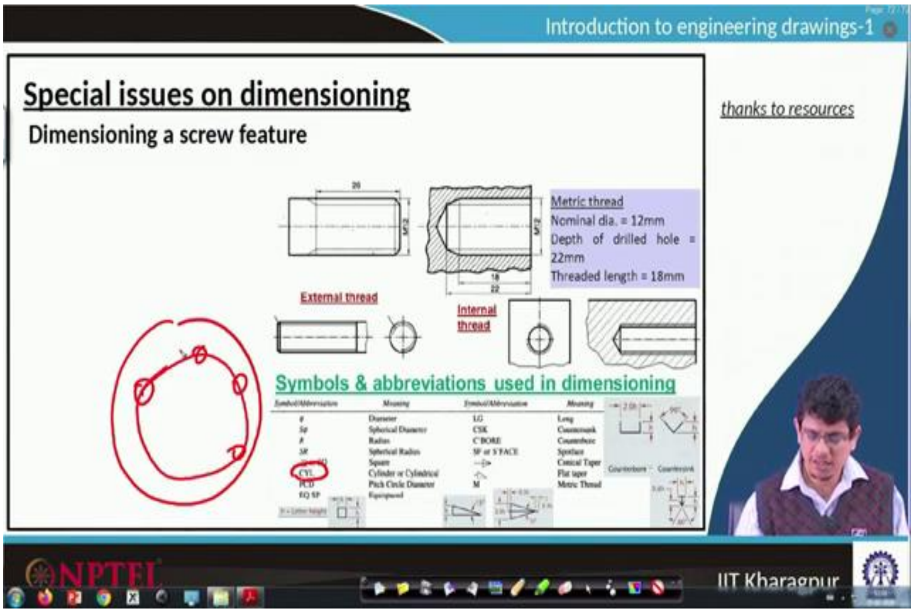
So, from here it is 22 mm up to this portion and thread perhaps the object size is up to that hole might be created up to this level, but threading is done up to this level only. So, in that case, the thread length 18 mm also has to be shown this is the 18 mm. This is the way threads and special screw features are shown.

(Refer Slide Time: 07:54)



We always show its different views like frontal view and side views. Because its a thread kind of thing external thread, if you are looking from this side or perhaps from this side it looks like a circle. On which we start saying the threads. So, in that case, we have inside dimension also, more details we will learn about that in the future classes.

(Refer Slide Time: 08:17)



Usually we go with symbols for any kind of dimensioning by just looking at that symbol, we will be in a position to understand whether the drawing consist of diameter or spherical diameter radius and so on. If we are using phi it represents diameter, if it is something like S phi its indicates that spherical diameter.

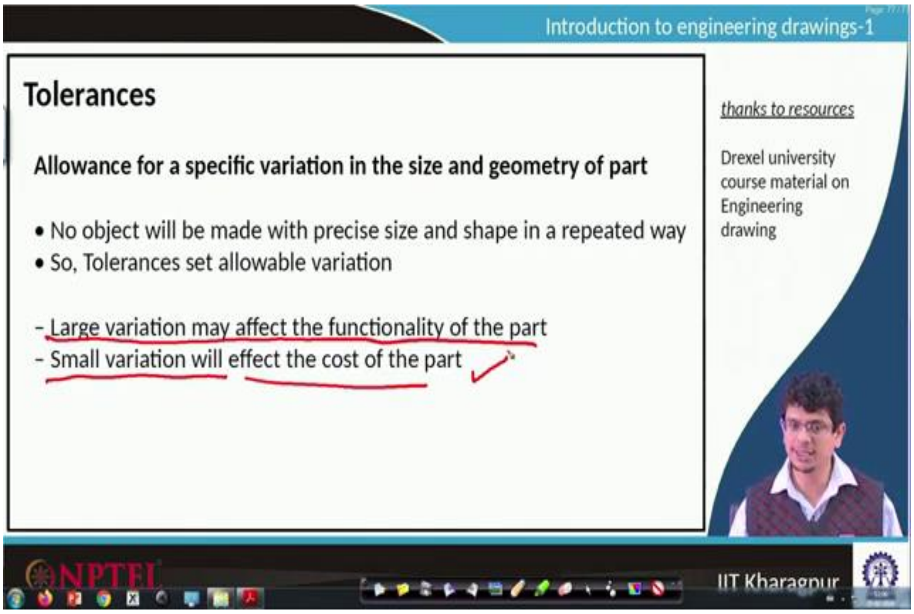
For radius we go with R and for spherical radius we go with SR symbols. If these are square components, we use symbol squares. Sometimes, we might be going to represent cylindrical objects in that case we use CYL as cylinders.

Similarly, whenever these multiple holes are made on a plate, there will be a pitch diameter represented in that case we usually go with this PCD pitch circle diameters. If things are equi spaced we go with EQSP equi space kind of things.

Similarly, other things like if it is conical taper, we use it in that way if it is just a flat taper we use it in this way, something like flat taper symbol is this. If it is conical kind of taper we use this one. Many other things like typically for threads and other things, we usually see these symbols M; that means, metric thread one way of creating these threads.

There are variety of holes and so on so, things like counter bores countersinks and so on there are special symbols people usually use it. So, on drawing sheet symbols dimensioning these units are the most important thing, if you want to convey a picture to machinist.

(Refer Slide Time: 10:35)



Now, all these dimensions for drawing are perfectly fine like saying 12 mm, but if we ask a machinist to prepare 12 mm kind of object, perhaps it can be the length it can be hole its not possible to precisely make 12 mm. There are always be plus or minus deviations possible because we are dealing with machines, there are many factors which influences this machine.

For example, we would like to make a turning operation where a cylinder of 20 mm we would like to make. However, a small deviation, because of mechanical vibrations temperature variations is the way how we feed this lathe machine and so on. So, things there is always chance that it might be excess it might be low of that object, many factors influence these things.

So, 12 mm or 20 mm usually is not a good idea to indicate, it is always good to mention 20 ranges to 20.5 mm. And also 19.5 mm or perhaps something like 20 plus or minus 0.05 mm. Such kind of dimensions what we represent are called tolerances. Let us learn more about these tolerances. The basic definition of tolerances is an allowance for a specific variation in the size and geometry of a part.

No object will be made with precise size and also shape in a repeated way. If we want to repeat it for multiple instances, there always be variations within that. So, it is not precisely possible to make the same object of the same size and geometry.

So, tolerances set allowable variation so, any quality inspections and so on, further purpose these tolerance limits are strictly imposed, if there is a large variation, it may affect the functionality of the part. For example, ah they ah think what we have taken a simple Reynolds pen if we are taking it has both the screw and the other part.

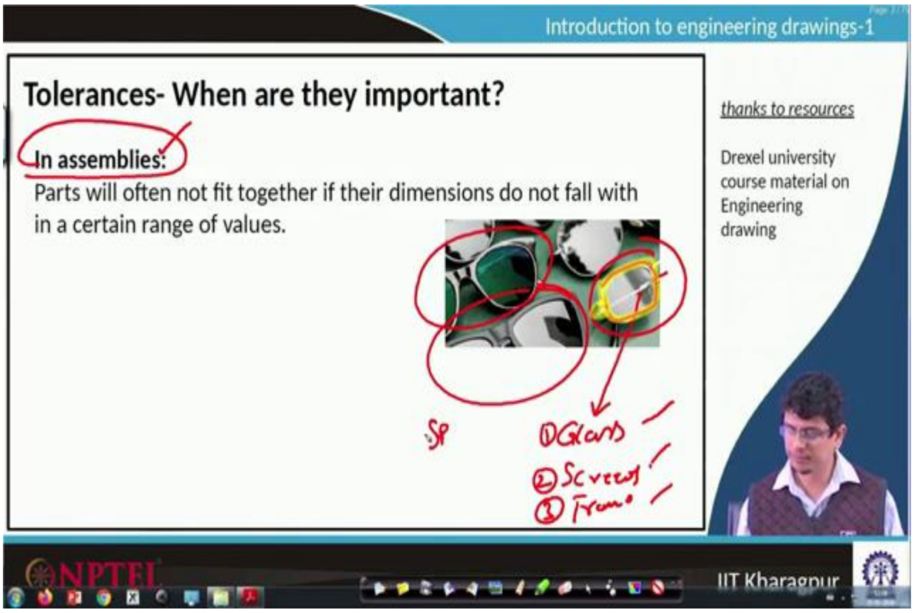
So, when we are trying to screw this Reynolds pen the blue colour part, if the thread is too small, it immediately loses that pen and the functionality of the part severely affects. If that thread is very large, perhaps it might not fit into that object also.

So, a large variation always affects the functionality of these parts. Even, for example, the finger rings what we use if it is too large we will not be in a position to effectively use that ring, if it is too small it does not fit into our finger also.

So, any object whatever we take with these tolerance things, if there is a large variation, it severely affects the functionality of the part. However, if we want to make it precise; that means, we are going to make a various small variation, then one has to be at most care and to make that component it will be very costly. So, one has to play with this small variation to significant variation carefully.

So, that on one side it would not increase the cost this cost can be time-consuming it can be in terms of monetary things, in terms of production how much how many quantities you would like to ah prepare it that also cost and many things. So, usually, industries try to optimize this small variation and large variation.

(Refer Slide Time: 14:48)

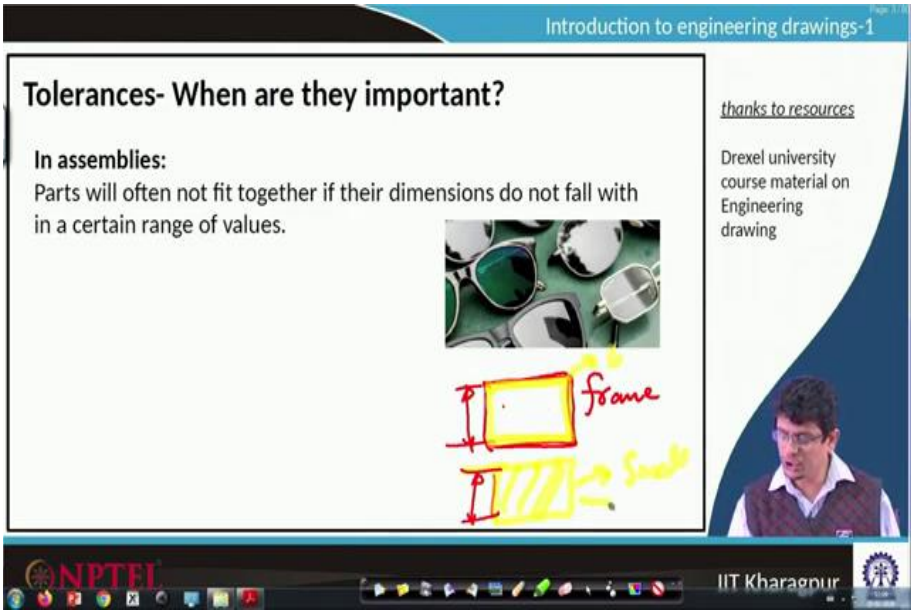


Tolerances:

Let us look at a standard example, in machining processes; we call somethings things as assemblies. These are individual components we fix them and make it like a final product.

Let us pick an example like spectacles. This is an assembly this consist of the typical glass, perhaps the frame. This is the frame. Perhaps there will be joints those joints might be having screws glass screws and frame. These are called parts of this spectacle when you club them in a proper way you will be in a position to get an assembly that assembly makes the functionality like spectacles.

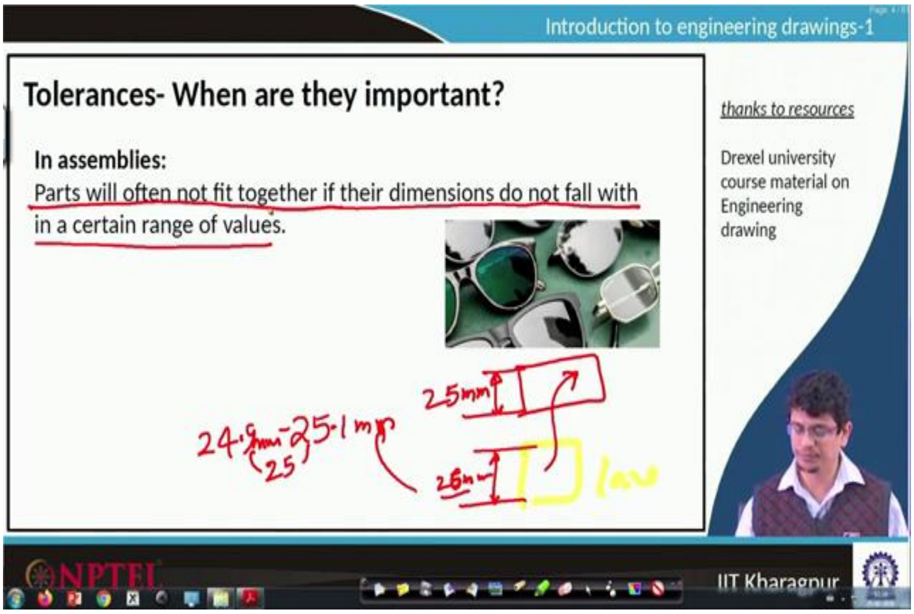
(Refer Slide Time: 16:09)



Now, let us look at the individual thing in that frame; this is the frame. We would like to put a glass, let us make this glass this is the glass what we would like to fit if this dimension let us look at this dimension if this dimension first of all its not easy to prepare the same dimensions.

Even if we make it a small variation in terms of fixing it like mechanical stresses perhaps might be twisted. So, the main functionality like fitting this glass inside that we may not be in a position to achieve, it a small increase in mechanical stresses perhaps temperature variations, or perhaps surface abrasions and other things. It may severely affect the functionality of that, but in case if we are going to make it a smaller one this is a small size, but this one large it just comes out of that object.

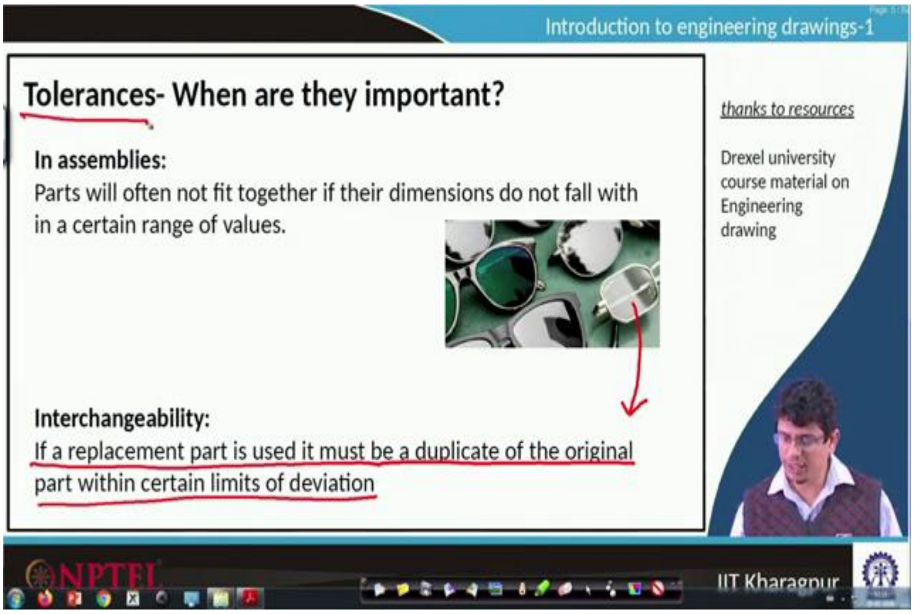
(Refer Slide Time: 17:20)



If it is too large if this one is large and if this one is small it cannot really fit into that. So, parts will often not fit together if their dimensions do not fall within a certain range of values. For example, let us consider this dimension is 25 mm, this one, 25 mm is the correct fit.

Perhaps when you are making this is ranging from 25.1 mm to 24.9 mm within that range. If this one is 25, it fits in that object, if it is 26, definitely it will not fit into that object one has to do other ways of trying to fit that. So, parts will often not fit together if their dimensions do not fall within a certain range of values.

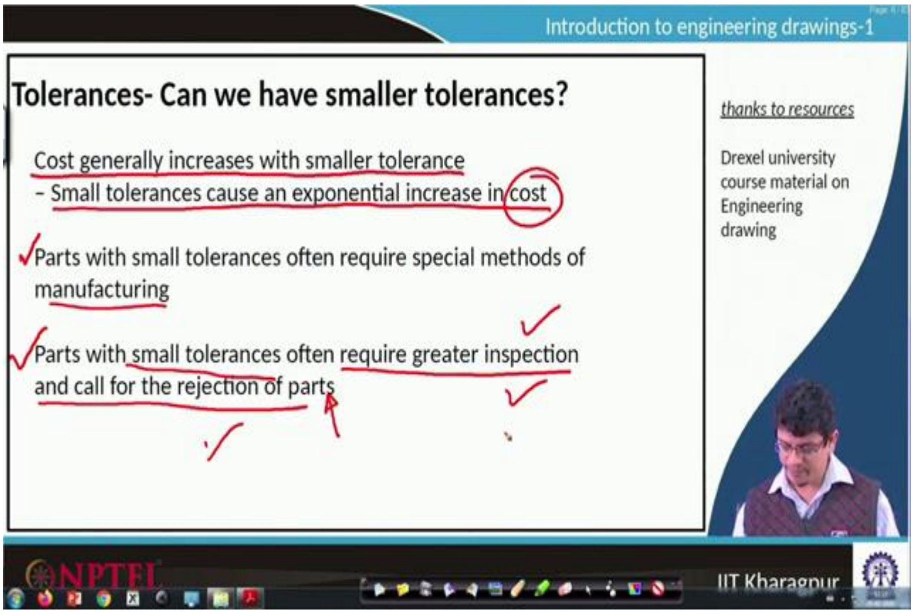
(Refer Slide Time: 18:32)



For example, you broke your one of the glass; you would like to replace it this one you would like to replace it. If a replacement part is used, it must be a duplicate of the original part within certain limits of deviation.

So, when industries make this kind of products in multiplication many parts, one has to be very careful with this tolerances. Repetition is very much required because you broke it and you would like to replace it. So, to replace that you require a similar kind of component of precise dimensions, if not possible, then something one has to play with this tolerances.

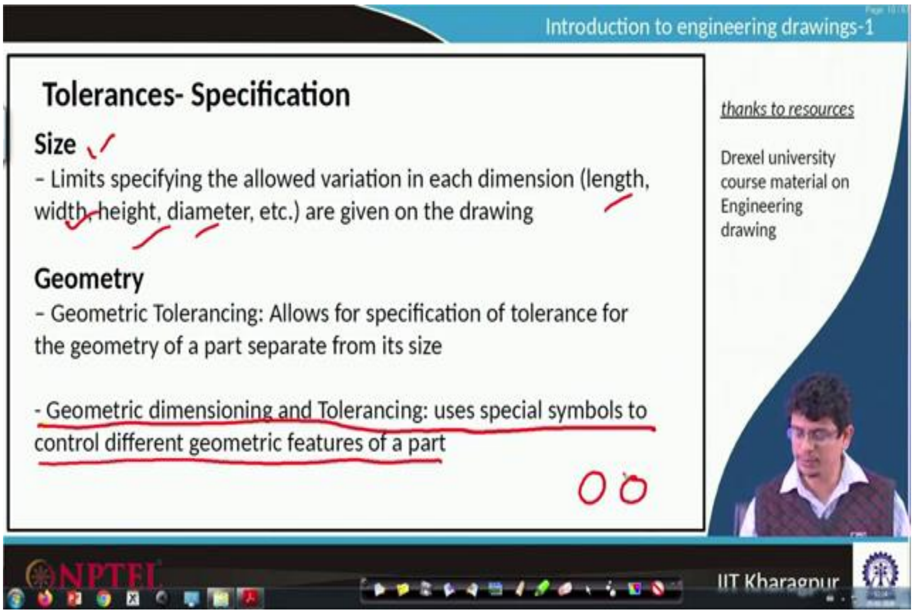
(Refer Slide Time: 19:25)



It is a common practice cost generally increases with smaller tolerances small tolerances cause an exponential increase in cost to make it. Parts with smaller tolerances often require special methods of manufacturing, its not only about cost if you are going for very small tolerance values to prepare that itself we require special equipment.

Parts with small tolerances often requires greater inspection and call for rejection of parts, how much care we might be going to take to care such kind of small tolerances, end of the day it has to go through inspection. A small deviation from the required tolerances, there is a high chance that part will be get rejected. So, small tolerances always have a greater inspection thing, and high is highly likely that many parts will be rejected, though it is a good practice to have smaller tolerances, making that itself is challenging.

(Refer Slide Time: 20:58)



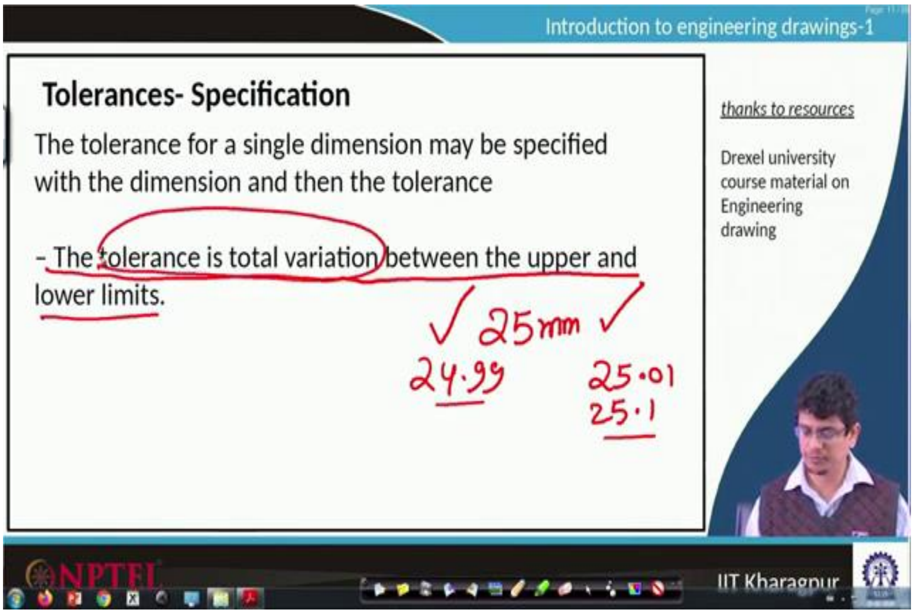
We specify these tolerances in different ways; for example, it can be based on the size it can be based on geometry also. For size, let us look at it. It limits specifying the allowed variation in dimension; that means, it can be length width, height or diameter etcetera are given the drawing. These kind of things if we are talking about those are about size tolerances regarding geometry also we talk about this tolerances.

For geometric tolerancing, it allows for the specification of tolerance, for the geometry of the part separate from its size. I want to draw a rectangle, but perhaps there is a small variation it might look like trapezium kind of thing, these variations are very small. Such kind of thing instead of having a square perhaps this trapezoidal kind of thing is also tolerated, and that is what we call geometric tolerances.

In terms of lines lengths diameter that kind of thing what we call size tolerances. As I said we would like to draw something like a cylinder, but it might look slightly oval, that kind of things are also geometric tolerances one has to maintain.

Geometric dimensioning and tolerancing uses special symbols to control different geometric features of a part. However, we are clearly mentioning it in terms of size like width height diameter and so on. Perhaps a circle to a slight ellipse or oval kind of things we use special kind of symbols; that means, it is allowed up to that level.

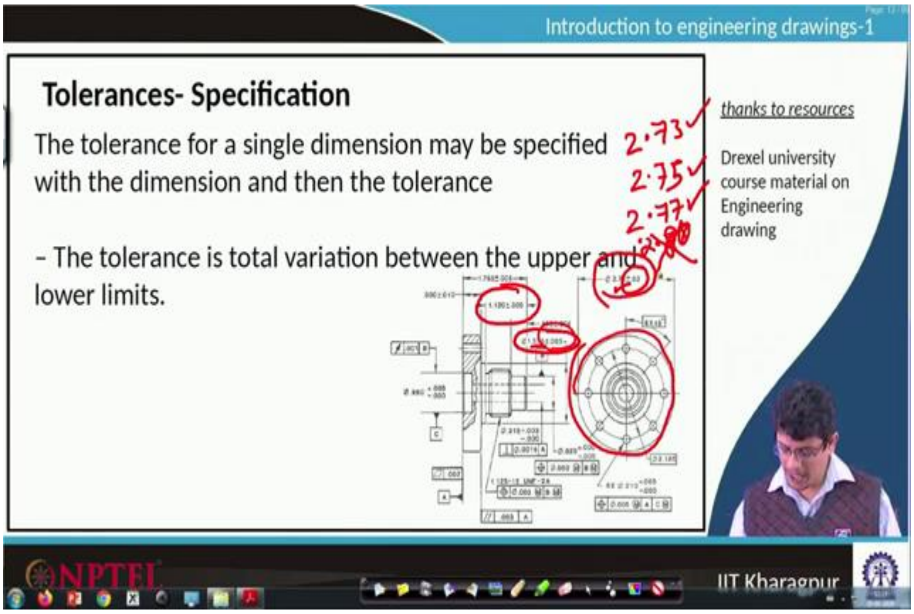
(Refer Slide Time: 23:01)



The tolerance for a single dimension may be specified with dimension and tolerance. We will shortly look at it how to represent that. One thing what we have to notice? The tolerance is a total variation between upper and lower limits if I am saying something like 25 mm.

On upper side up to which level I can really go on the lower side up to which level I can really go with that dimension, is it 24.9 or is it 24.99 or is it something like 25.01 or is it 25.1 such kind of upper-lower limits something to deal with this total variation.

(Refer Slide Time: 23:51)

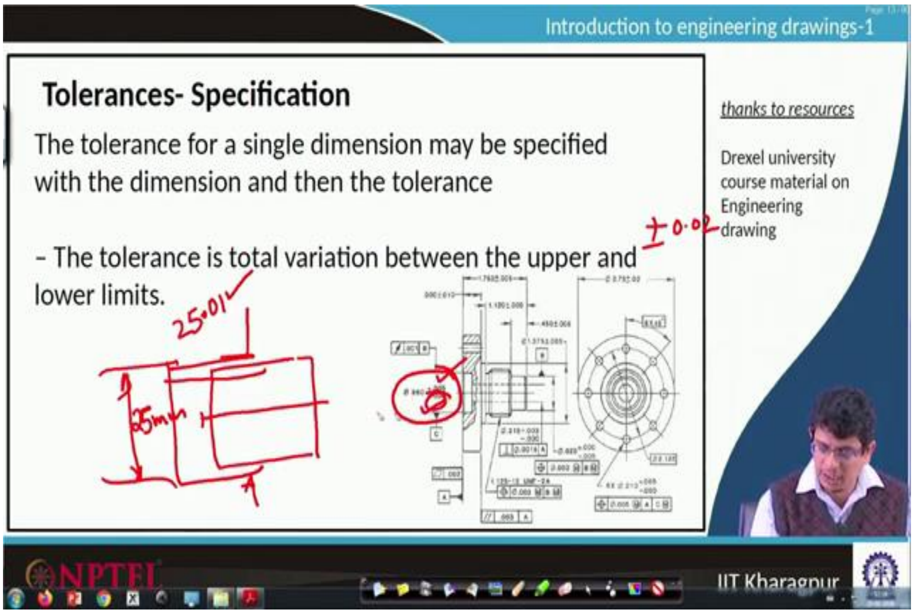


Let us look at an example here we have a drawing sheet on which there are many components mentioned, for example, its a for a turning operation the chalk screws and other things here is represented, for example, let us pick this one. We can see that something like phi representing diameter 1.375 units with plus or minus 0.005 units.

Similarly, let us look at this one 1.120 plus or minus 0.005. Similarly, let us look at this diameter phi 2.75 plus or minus 0.02. So, when machining is done, and perhaps product comes out through this engineering drawing sheet when it goes to quality check mainly quality control gas check whether these tolerances are met or not for that object.

If someone measures the diameter of that outer circle, if it is 2.75 perfectly fine, if it is something like we 2.77 that is also perfectly fine, if it is something like 2.73 units that is also perfectly fine, because this plus or minus 0.02 represents. If anything is perhaps 2.78, they will reject that part. So, further purpose these tolerances are always helpful.

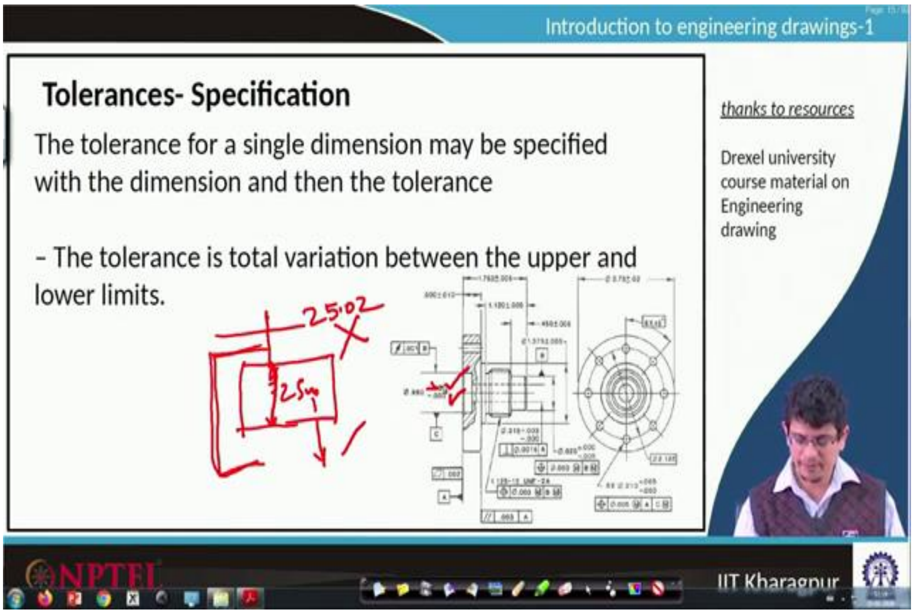
(Refer Slide Time: 25:42)



Similarly, let us look at this one it is not necessary that you always have plus or minus 0.02 on the positive side and negative side. Sometimes on the positive side, it is preferred, but not on the negative side when you want to fit the things and so on. If it is lower than that it may not hold the object, it will be very tight kind of thing. Anything above, for example, if this is the object in that, I would like to put a cylinder.

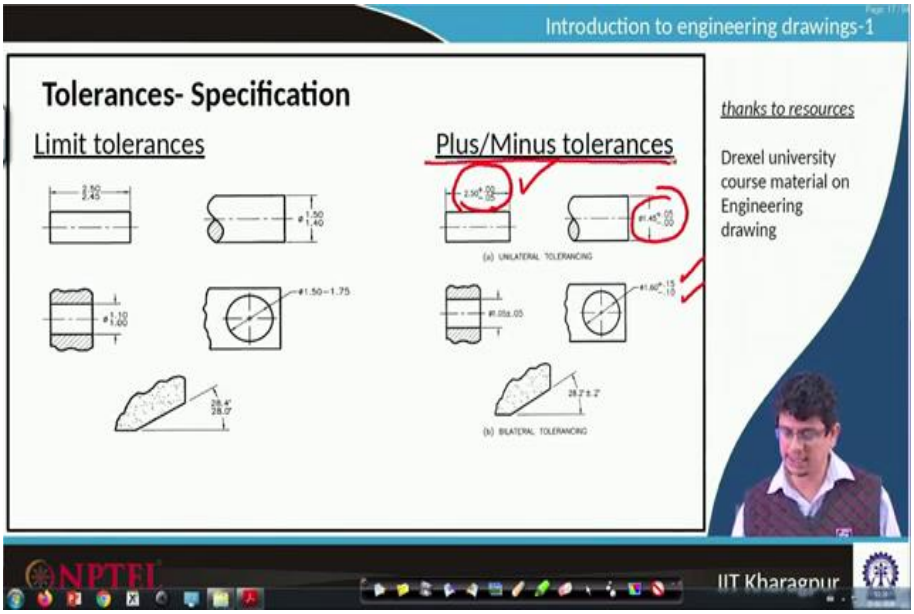
Now, when I am going to measure these dimension or perhaps this one in a more appropriate way. If I want 25 mm for that anything above 25.01 is preferred, but if I go below 25.1, it will be somewhere here around. So, this cylinder cannot be entered into that object. So, sometimes this lower dimensions supposed to be strictly imposed like 0.

(Refer Slide Time: 26:52)



Anything above is perfectly fine if it is that kind of objects, for the same thing if we are talking about cylinder which has to be fixed in these object. If I am going with that 25 mm, here this is fine perfectly fine. But anything above that maybe 25.02 it will be out of that box so, we cannot really fit it. In that case, what we will do is lower side is preferable upper side is not preferable. So, based on that your plus and minus variations may always happen..

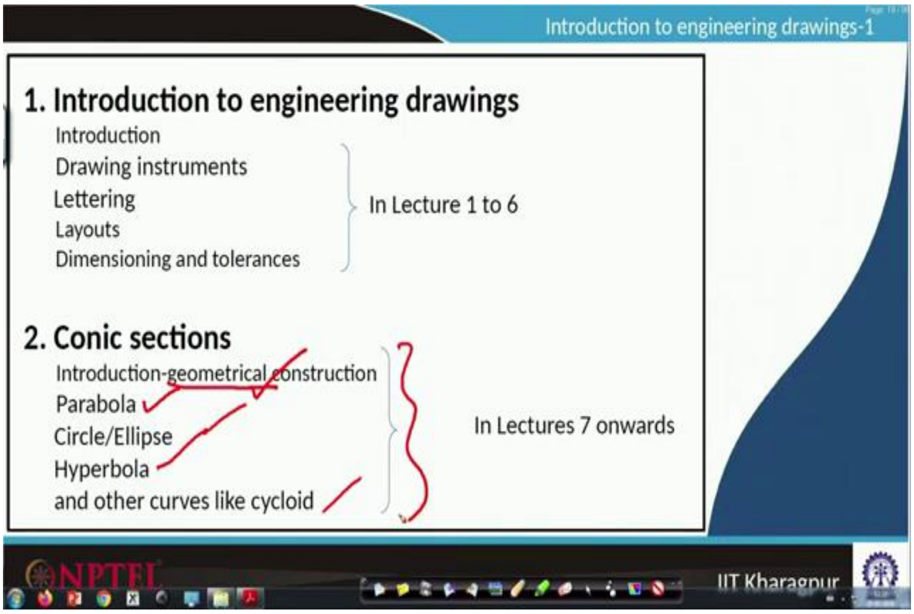
(Refer Slide Time: 27:38)



And these tolerances are broadly divided into two parts one is limit tolerances, where we show the dimension 2.5 2.45 both the things. Similarly, limit tolerances for which limits are mentioned it can vary from 1.4 to 1.5, this diameter can vary from 1.5 to 1.75 and so on so things.

Similarly, the angles 28.0 to 28.4 such kind of limits if we are showing that is called limit tolerances. If it is plus-minus kind of tolerances, we usually show it in terms of 2.5 plus or minus 00 minus 005. Such kind of thing a common practice we see it on drawing sheets something like 1.45 plus or minus 05 00. Something like plus 0.15 minus 0.10, such kind of things are called plus-minus tolerances.

(Refer Slide Time: 28:59)



So, to summarize our introduction to engineering drawings, we first looked at drawing sheets, drawing instruments, lettering layouts complete picture on dimensioning tolerances in the first six lectures. In the next class onwards we will learn about how to draw lines curves, bisector lines and so on how to utilize these instruments to construct geometrical things, how to construct common conic sections like a parabola, circle, hyperbola and other curves like cycloids and so on.

So, in lecture 7 onwards, we cover these conic sections and geometrical constructions.

Thank you very much.