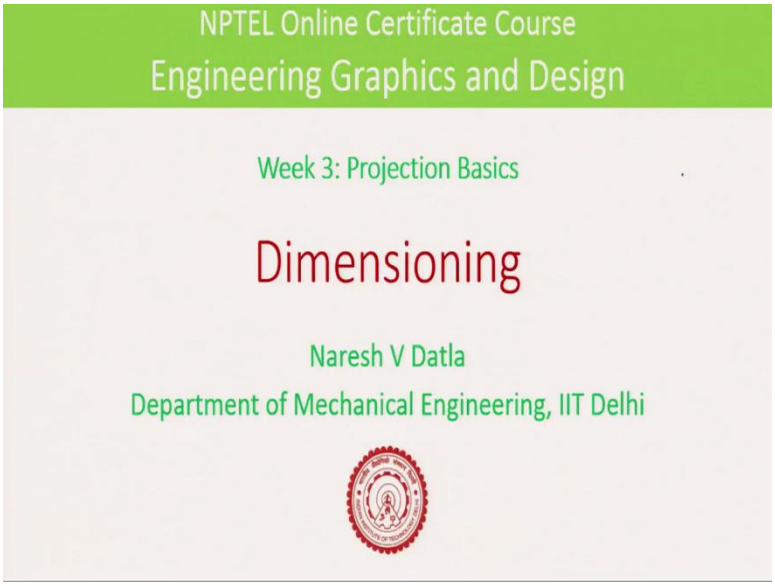
Engineering Graphics and Design
Professor Naresh V Datla
Department of Mechanical Engineering
Indian Institute of Technology, Delhi
Lecture 3
Dimensioning

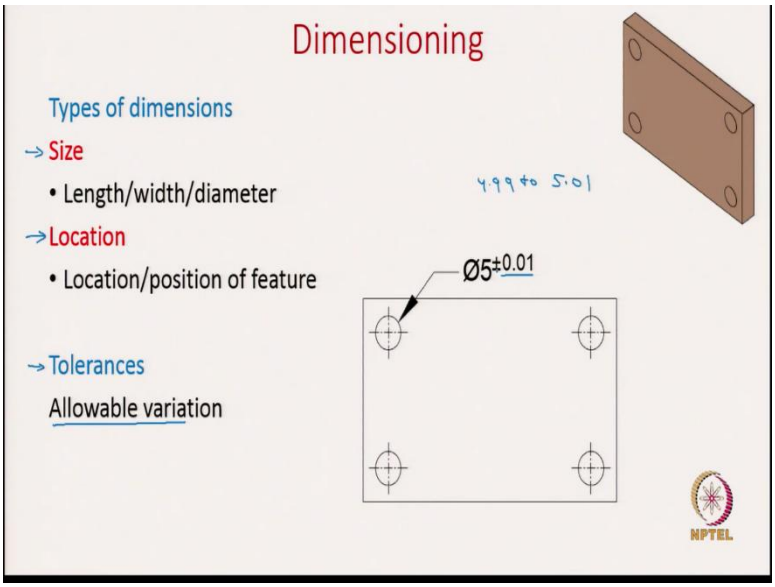
(Refer Slide Time: 00:09)



Welcome to week 3 of Engineering Graphics and Design. We are on projection basics. In the last two lectures, we have looked at formal aspects of this engineering drawing like we started by looking at the drawing sheets, the different sizes, scales and the layouts. Then we also looked at the different kinds of lines we use in an engineering drawing.

Now, in this lecture we will be looking at dimensioning because which is a very important aspect if you want to convey the complete information about the object on an engineering drawing. So, there are a different aspect to the dimensioning that will be covering in this lecture.

(Refer Slide Time: 00:59)



So, let us start with looking at what are the types of dimensions we need to represent. So, here we have a plate with four holes at the corners. So, if I represent this as a drawing, is this information enough for somebody to take it and reproduce this object as it was intended to? The answer is no because we need some additional information. What is that additional information?

To start with I can say I want to know what is the length and height of this object so which we call it as we are looking at size of this object. So, it could be length, width, depth of any feature. It can also be about let us say since we have diameters here, we want to know what is the diameter of this hole.

You may also want to know what is the depth of this hole. So, is it a through hole which means goes from one face to the other face or does that hole stop in between which means you need to specify at what depth the hole stops. So, this is one aspect of the dimensioning but is this sufficient to build this object. The answer is we need more information what is the other information we need?

We need to know where these holes are spaced. So, in other words we need to know the location of these features. So, location or the position of this feature, for example, if we are talking about this particular hole, we want to mention how far it is from the left edge or how far it is from the top edge. So, this is also dimensioning but here we are not talking about how big or small this particular hole is.

We are talking about how far this is from a particular reference plane. So, these are the two different aspects we need to dimension, the size and location but there is even more information one can give. For example, one may also specify the tolerances. So, first let us define what is tolerance.

So, as like humans are imperfect, the same thing with machines so for example even if I use the best of the machines and I say I want the length of this object to be let us say 10 centimetres, even with the best of the machines you cannot, you can never expect to get it exactly 10 centimetres, there is always some variation.

The question is whether this variation is small or large and again is this variation acceptable or not. So, that one needs to specify. So, for example, here we are saying that tolerance is nothing but the allowable variation. In this example we are showing this particular hole diameter. We are saying the allowable is plus or minus 0.01 units.

So, plus or minus 0.01 from 5, so which means 4.99 to 5.01. These are the allowable limits with which this will you are fine with it and the functionality would not hinder. So, and these tolerances changes from place to place or application to application.

For example, in construction when you are talking about big buildings maybe the height of this room or the length and width of this room, the allowable variations probably are slightly more compared to your mechanical components or some precise machines.

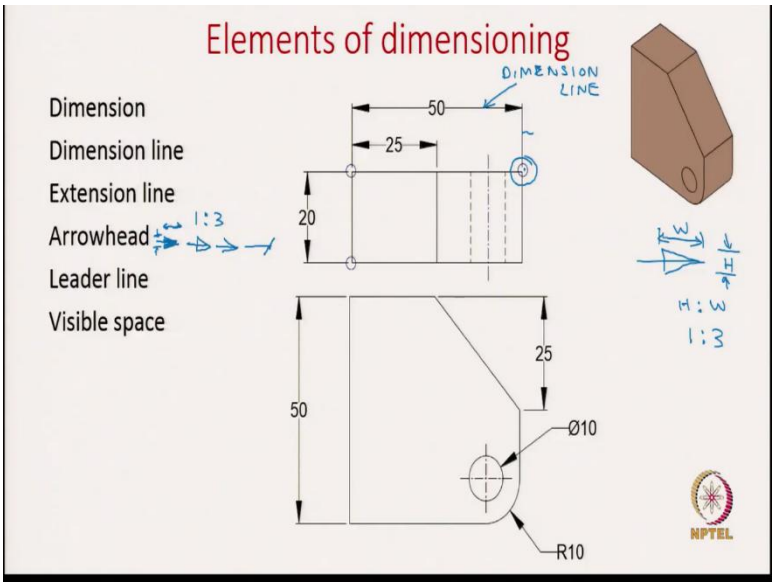
So, because the height of this room probably I am fine with it if it is off by a few millimetres probably that would not affect the design hugely but that is not the case with some of these precision machines or the precision devices where you want the tolerances to be as close as in microns, maybe 10 microns. At least in this case if you are talking about if the units are millimetres, the 0.01 is 10 microns but this is an information we need for a person who is fabricating because then depending on these tolerances, he need to make few choices.

So, if you ask for very stringent tolerances then he may, he has to use a very high precision machine to fabricate but then the costs are also high. But if the tolerances are a bit lenient then probably, he can use a normal workshop tool to drill that hole and then the cost might be more reasonable. But this all depends on your application and what is the tolerances one can live with for a particular device or a machine.

But this being again an introductory course, we are not going more into the tolerances just for the sake of completion I just mentioned that this information can also be given but yeah of

course, if you take any advanced course after this depending on your specialization then you may need to discuss more or learn more about these tolerances. So, for the rest of the lecture we will be looking at dimensionings, essentially the size and location dimensionings and see what are the different ways we can depict this.

(Refer Slide Time: 06:58)



So, first let us look into what are the different elements we have in dimensioning. So, each of these show the dimensions. Some are linear, some show the radius, some show the diameter of a hole. So, let us look at one at a time. When I say dimension, it is a number which represents the size or location. So, it is you are giving a number, it is a value to the dimension. So, for example here you say the width of the object is 50, the overall dimension is 50 and maybe the height of the object is also 50 here and the depth is 20.

All these numbers represent the dimension value. In short, we simply call it as dimension. And then the dimension line, so this is what we are mentioning as the dimension line. So, as you can see the ends of the dimension line hit another line called as the extension line. So, these are the lines which are used you extend it from the feature, you start from the feature where you want to dimension and then extend it to the dimension lines and again, I am repeating you need to extend it beyond the dimension line.

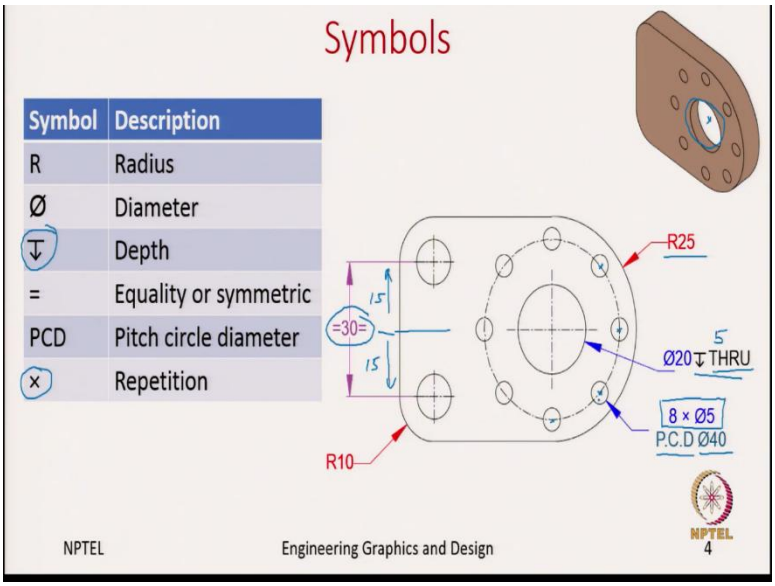
What else do we see? We also see that this dimension line ends with a arrow and what are the different kinds of arrows one can use? What we see here is a solid arrow, where you can also use an open arrow or may be just like this. So, usually in construction they also use this oblique arrowhead. So, these are all the different kinds of arrow heads one can use but the important point is what is the aspect ratio.

So, for example, if we look at this as the length of the arrow and the height of the arrow, these are placed in the ratio of 1 : 3. So, one being the, so if we say this is the height of the arrow and this is the width of the arrow, so the ratio of height to width is usually is 1: 3. And what are the other things we have here? We also have the leader lines.

So, for example, when we are representing this hole, we represent it using a leader line and then here this phi symbol represents the diameter. So, it says the diameter is 10 units. Similarly, on this curved edge we use the leader line to point that this is a circular arc of radius 10 and lastly the visible space.

So, if you look closely here, you see that there is a space you need to leave between the object and the extension lines. You cannot join them together because once you join them then you create unnecessary confusion that what is the object and what is the extension line. So, once you leave the space it helps you to focus on the object and you know clearly without ambiguity what is an extension line. So, these are the different elements of the dimensioning.

(Refer Slide Time: 10:43)



We will start with symbols and see that what are the various symbols one can use. In the previous slide I have already showed you 2 symbols. One is R for radius, here I am using it to say that this is a circular arc of radius 25 and phi for diameter. So, for example, here the inner circle we are saying the diameter is 20 units.

We will come back to this in a moment. So, this symbol what we have is the symbol for the depth saying that if you are looking at this hole, we notice that it is a through hole because

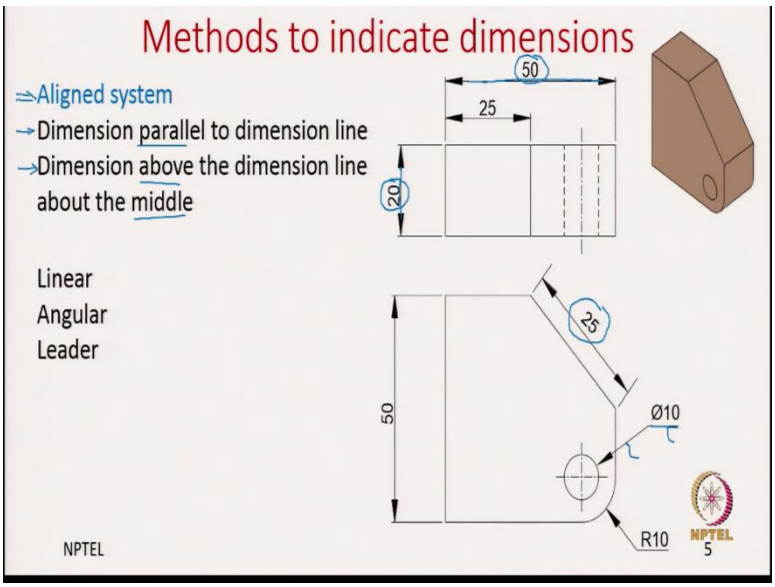
you are able to see the back of the object. This‘THRU’ represents it is a through hole. So, you are saying the depth is through, not every time it is through.

So, let us say for example this depth is half the thickness of the sheet then you can mention the units. So, let us say 5 units. So, you can strike the THRU saying the through hole and mention a number. Then it says you have a hole of depth so and so units. We can also use this equality of symmetric for example here we have used it, what it means is these two holes are symmetric about the central line.

So, then you know for example when it is the dimension is 30 these are 15 on each side, 15 plus 15. Similarly, the pitch circle diameter, so when you connect all these centre’s we notice that they all fall on a circle and this circle which is what we are calling as the pitch circle diameter. So, we use P.C.D to represent a pitch circle diameter and here we are mentioning that this pitch circle diameter is of 40 units.

So, the diameter is 40 units and lastly, we can also do, use the cross mark for repetitions. So, for example, here we have seen there are eight holes. So, instead of going to each and every hole and saying that it is of so and so diameter, we can simply say 8 cross diameter of 5. So, which means each hole is of diameter 5 units and then we have eight holes with the same dimension.

(Refer Slide Time: 13:26)



Now, let us look at different methods to indicate dimensions. We essentially have two different methods. First, we will look at an aligned system, so it has two specific rules saying that the dimensions parallel should be placed parallel to the dimension line and the dimension

should be placed above the dimension line about the middle. So let us place the dimensions and see what these two principles mean.

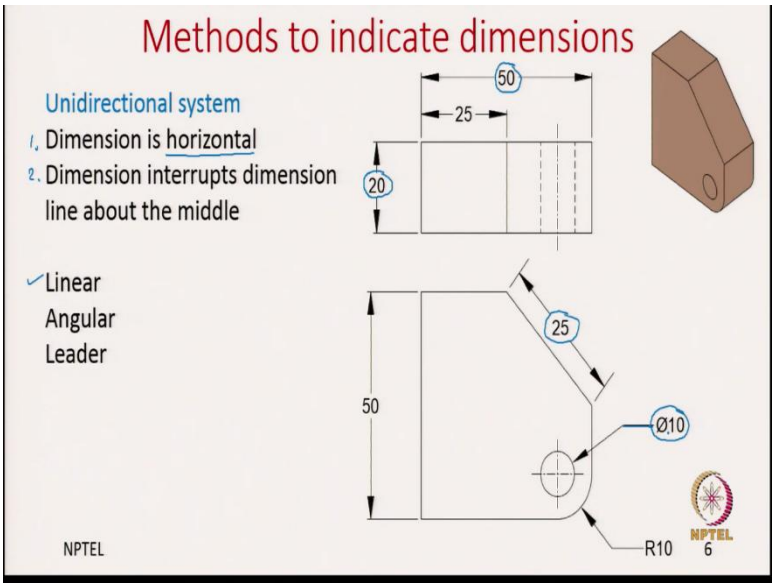
So, the first one said the dimensions when I say dimension it is the dimension value, the number are placed parallel to the dimension line. So, for example, this is the dimension line and this number is horizontally orient. So, this is parallel to the dimension line but now let us look at a vertical dimension line and now if you look at this number 20, this is tilted and how is it tilted? It is parallel to the dimension line since the dimension line here is vertical, even the text or the number is written in the same orientation.

And the second principle is we are saying we need to place this dimension value above the dimension line. So, this 50 is sitting on top of the dimension line. Similarly, this 25 is sitting on top of the dimensional line. For a vertical line, you need to place it on top. The top is not so obvious here, but we place it to the left in this case.

But what if we have angular dimensions? So, for example, we have a angular dimension here. So, again you can see this 25 is written parallel to the dimension line and it is placed on top of the dimension line, and it says that it is always preferred to place it at the middle. So, it is not a hard and fast rule, but we are saying it is good if you can place it this number almost at the middle of this dimension line. But there are some instances if you think that it is more convenient to move it a little left or right you are free to do so. But yeah, you should have a strong justification why you want to place it at one side of the dimension line.

And for leaders, as I said, this is a leader line, but this is the reference line. So, these numbers again follow the reference line. Since the reference line here is horizontal, the text is also placed horizontally. So, this is the first method we have seen what we call it as the align system.

(Refer Slide Time: 16:28)



So, now we will present the other system called the unidirectional system. Here the rules are slightly different. First, we say that the dimension value or the number is always horizontal. And the second saying that the dimension interrupts the dimension line about the middle. So, let us look at it how we do these dimensions. So, first starting with the linear dimensions.

The first principle says that the dimension is always horizontal. For if you have a horizontal dimension line the number is placed horizontal, even for a vertical dimension line the number is placed horizontal.

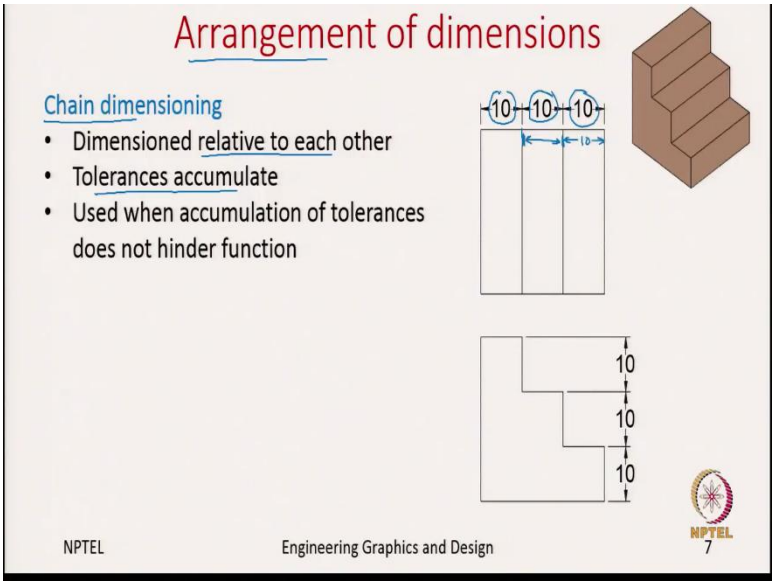
So, as you can see it is very easy to read because you need not tilt your head or you rotate to look at what is the dimension of this vertical dimension or an angular dimension. And the other important aspect is that number is interrupting your dimension line. So, previously you remember we are placing it above the dimension line but here we are placing it by interrupting the dimension line.

For angular, again you see that this number is placed horizontal and it is interrupting the dimension line. And lastly the leader lines again since it follows the reference line, here the reference line is horizontal, so the text also remains horizontal. Irrespectively this unidirectional system we always retain horizontal, so the text is horizontal.

Here we have seen the unidirectional system where the text is always horizontal, and it is interrupting the dimension line. And in the previous align system depending on the dimension line your text is always parallel to the dimension line. So, most often people follow this

unidirectional system depending on the industry you work it changes whether it is unidirectional system or a aligned system.

(Refer Slide Time: 18:40)



So, now we are looking at a different aspect which is the arrangement of dimensions. So, let us say you have multiple dimensions how do we arrange? Again, here we notice that there are multiple ways of doing it. The first way we will be discussing here is the chain dimensioning. What do we mean by chain dimensioning?

We say that all dimensions are dimension relative to each other. So, let us focus on this dimension of 10. So, after we are done with this dimension, we have shown this particular feature saying that this is at a distance of 10 to the left edge. Then we are using this feature as a reference for the second. So, for the second dimension of 10, we are using the reference as the previous feature and then showing the new feature which is this vertical line. So, now we are saying that these two are placed at 10.

Again, we repeat so once you go to the third dimension, the reference we take from the previous feature and then say that this is at a dimension of 10 from the next feature. So, this is like a chain dimensioning. So, where you show one dimension and then you place the next dimension next to it and keep continuing. There is a problem with this kind of dimensioning. It accumulates the tolerances. Previously you remember I mentioned what a tolerance is.

This is the allowable fluctuation. So, let us say we are fabricating this block, we go to a machine shop to fabricate this block the first we make sure this 10, the first step is maintained

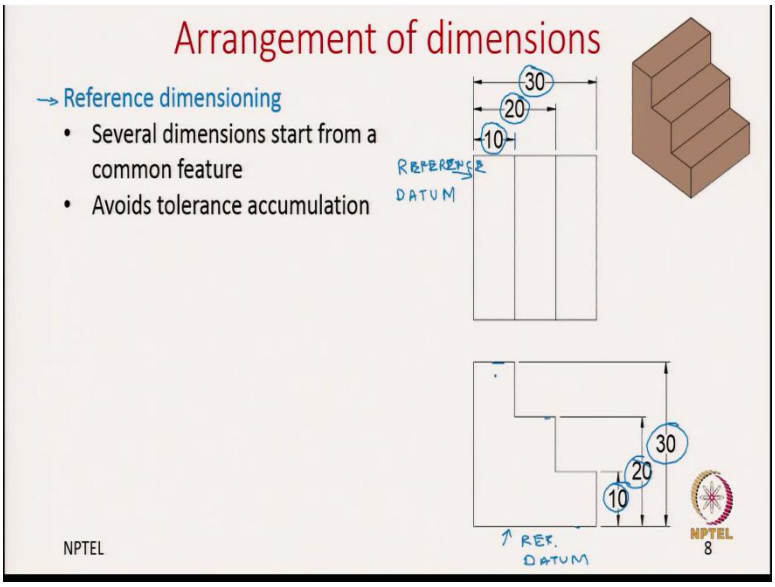
at 10 and then since the machining is not accurate, it can have a fluctuation. Probably instead of 10 it has plus or minus 0.1.

Let us assume that instead of 10 we got it as 10.1. So, next time we prepare this step this error accumulates because the first step is 10.1 and the second step will also let us say is 10.1 and the third step is 10.1. So, overall length will be instead of 30 you have it as 30.3 so which means the error of zero point, 0.1 units is accumulating.

So, maybe here it is only three steps but let us say we have a object where there are this chain dimensioning is done for 10, 20 or even 100 features then what is happening. That small tolerance keeps accumulating and then becomes a very big error.

So, though we say there is a trouble with tolerancing, we still use it because it depends on some places these tolerances may not hinder your function of the object. At that time, you can still follow this chain dimensioning. So, even though there is an accumulation of the tolerance, probably that would not affect your function of the object. So, you are fine with it.

(Refer Slide Time: 22:05)



So, then you can go for chain dimension but let us say this is a problem, then the alternative is to follow what is called as reference dimension. What do we do with reference dimensioning? We say that we pick a reference. So, for example, for all the horizontal dimensions we pick this as the reference.

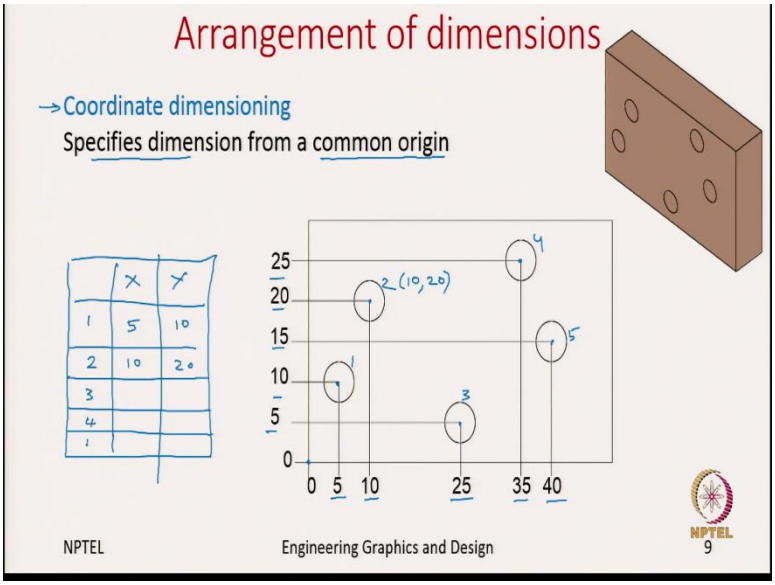
It has many other names maybe you can name it as datum or reference line. Essentially you can see all the horizontal dimensions are made using this reference line. So, for 10 it start from this reference line and goes to the feature and even for this dimension of 20, it starts

with this reference line and goes to the feature and similarly for this 30, it starts with this reference datum.

Similarly, again for these vertical dimensions we use this bottom edge as the reference or the datum. So, for all these dimensions of 10, 20 and 30 we have this as a common reference using which the dimensions are specified. So, the advantage here is your allowable tolerances does not accumulate. In the chain dimension we said the tolerances accumulate but here it would not accumulate because you specified it to be 10 based on one reference.

So, once you are mentioning about the second step or the third step, they have their own dimension. We are again basing it from the reference plane. You are not adding these tolerances to accumulate it. So, this is one good advantage and most often people follow this reference dimensioning but again as you can see it takes more space so depending on that sometimes people prefer the chain dimensioning because it is more compact.

(Refer Slide Time: 24:15)



And there is a third alternative. Third alternative is the coordinate dimensioning, let us say, in this example you have a block and there are plenty of holes on this block and each centre of this hole if you need to specify the location of these holes, it is easier if you specify the centre of this hole using a reference point.

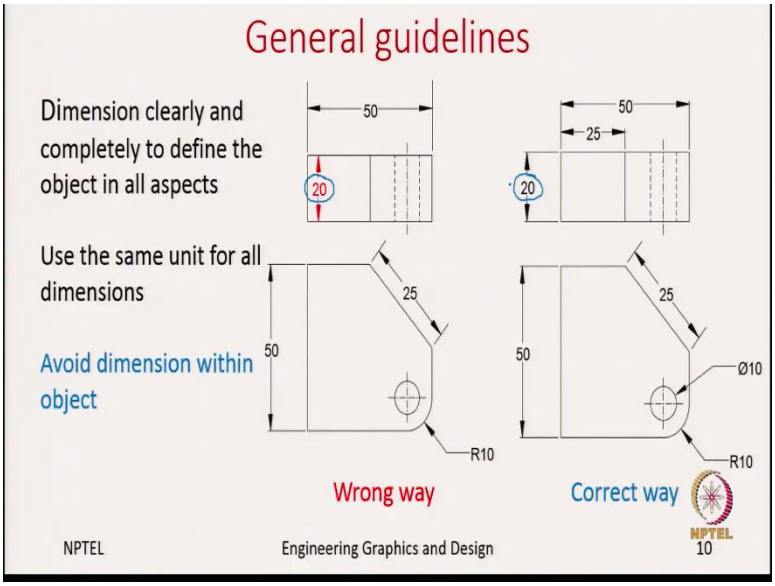
So, for example, if I take the left bottom corner as my common origin, then I can use this to specify the other dimensions. So, let us specify on this let us name this circle as 1, 2, 3, 4 and 5. So, the first circle I know in the x direction it is at 5 units from the common origin in the

horizontal direction and in the vertical direction it is 10 units from the common origin in the vertical direction.

Similarly, for 0.2 in the horizontal direction 10 units and vertical direction 20 units. So, you need not use those lines there are multiple ways of showing these dimensions saying that since we are simply focusing on this x direction, we specify only the dimensions in x direction. So, 5, 10, 25, 35, 40. Similarly, in the vertical direction we have this 5, 10, 15, 20, 25 or what other people do is maybe you can also have a table, saying that what are the dimensions of the first circle and the x coordinates and the y coordinates.

So, for the first circle the x coordinate is 5 and the y coordinate is 10. Second circle, the x and y are 10 and 20 and so on and so forth you can give it for the other centre circles, circle centres. So, we have followed three different dimensioning, the chain dimensioning, reference dimensioning and the coordinate dimensioning. Sometimes you can mix and match which means like you can use a combination of any two or any three of these dimensioning. That is also possible.

(Refer Slide Time: 26:44)



So, until now we are looking at what are the ways to arrange these dimensions but now let us look at what are the common guidelines. So, of course before I give you those guidelines, I should mention that there are different standards based on the industry or the kind of nature of the work you are working.

So, for example automotive industry might have its own standard where it will specify how to dimension things and an electronic industry, or a construction industry might have its own

standard international or national standards which they follow. But what I am trying to say in this introductory course is some basic principles which apply to most of these standards. You may have to check it again, but these apply to most of the standards. So, let us get started to looking at the common guidelines.

Let us start with this saying that you might be thinking, should I dimension everything, or should I avoid a few things? The answer is you should dimension it keeping in mind that the third person who has, who is reading this engineering drawing, let us say he cannot call you which means you need to convey all that information either through your engineering drawing or probably some other means like maybe a report or there are other engineering drawings where that information is already specified.

Even in that case you need to give some pointer to them saying that okay some missing features can be found at some other place which means since the designer or the person who is preparing the drawing is not being present, you need to specify all that information either on the engineering drawing or some alternative means.

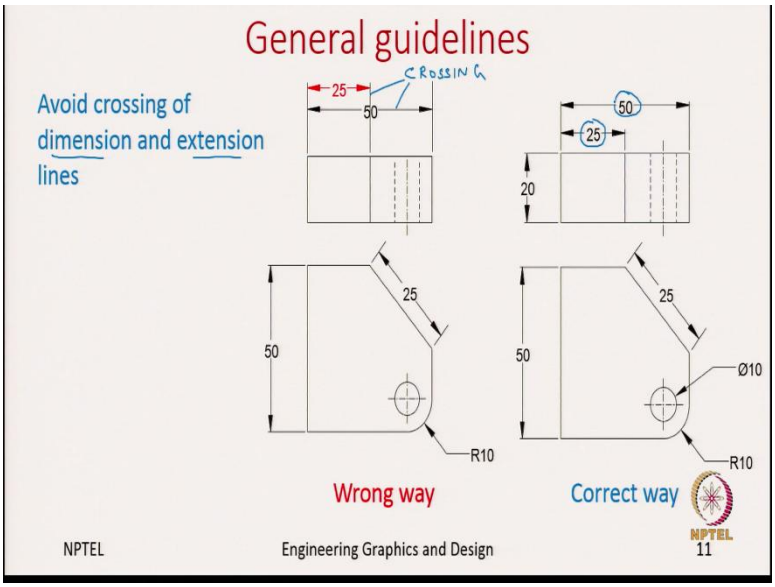
So, usually what we do is once we choose a unit, let us say it is a millimetre or an inch, we tend to stick to that units all throughout the drawings and the other important point is since we specify these units already in the title block, you need not write that millimetre at each and every dimension you specify.

You can avoid that and simply use the, give the number value, you need not specify the units because that is anyway specified at the title block. Now, let us look at some wrong ways of doing things as well as the right way of doing things. The first one is to avoid dimension within the object.

So, let us see what is wrong here. So, this dimension 20, as you can see is being specified within the object and the right way is to move the 20 out of the object. Probably you should be thinking why is this a right way. See if you keep the dimension within the object and you keep doing that, then basically what happens is your object gets cluttered.

It becomes messy, it becomes complex, so it becomes hard to read. For these reasons, we have rules like this, I mean I would not say these are rules but let us say these are guidelines. The difference is though we say dimension it out of the object there are some special instances where it is unavoidable. That time you can break the rule and dimension is within the object, but the guideline says try to avoid it as much as you can.

(Refer Slide Time: 30:37)

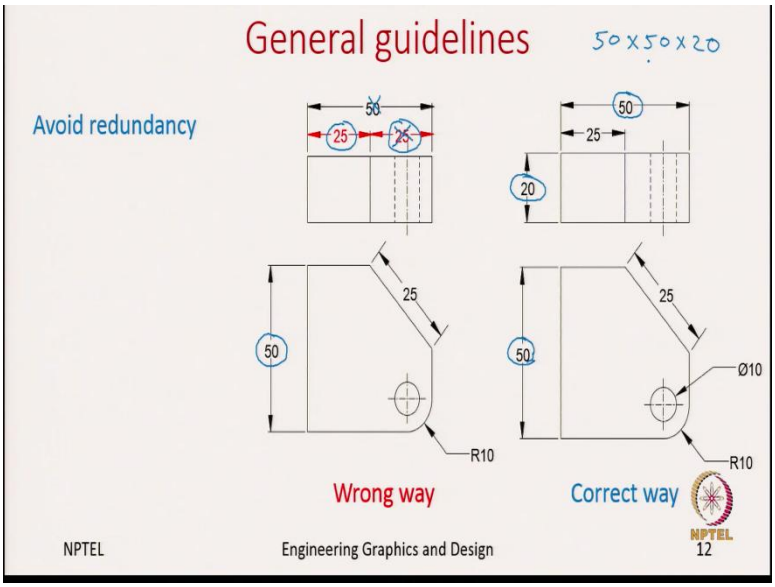


The next thing is try to avoid crossing of dimension and extension lines. So, for example here what is happening is this extension line is crossing this dimension line. We need to avoid this crossing. How can we do this? We can do this if you can first dimension the smaller dimensions closer to the object and then dimension the larger numbers away from the object.

So, you can use this technique so that they do not overlap with each other essentially, we are talking about crossing of dimension line and the extension lines. So, the technique what we are trying to say here is always try to dimension first the smaller dimensions which will be closer to the object and if you are looking at a bigger dimension, then you can put it slightly away from the object.

And always make sure that there is enough space you leave between the object and the dimension so that the whole idea is if you look at an engineering drawing, your eye should first look at the object and then you can pay more attention and look into the other details like dimensioning. The dimensioning should not come in the way of the object is what we are trying to say.

(Refer Slide Time: 32:07)



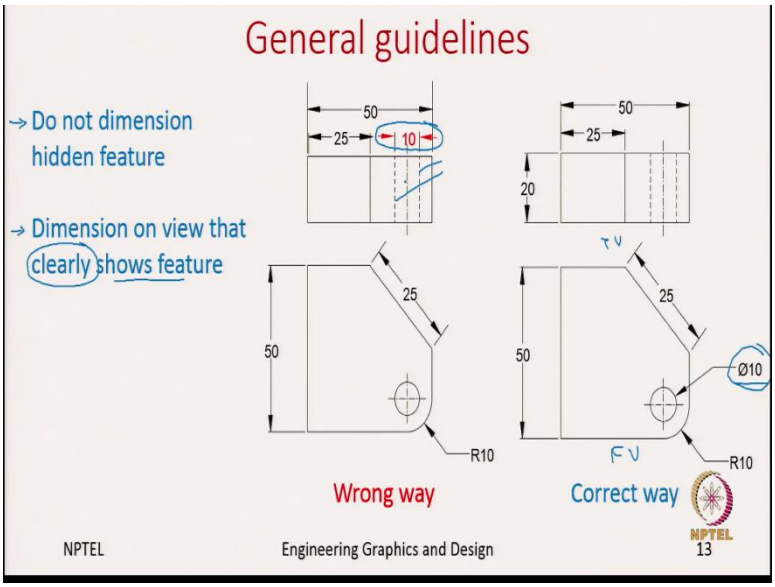
Let us look at another guideline saying that avoid redundancy. Of course, we do not want to repeat the same information again and again because then you are unnecessarily taking more space, making the object more complex. Let us see here why you think there is a redundant information.

For example, here if I say this is 25 and this is 25. If I add them together it is 50. So, I can probably avoid mentioning that it is 50 or that is one option and the other option is maybe I can avoid the second repetition of 25 and simply mention the 50. I would prefer the second option; the reason is because this 50 also gives you what is the overall dimension of the object.

Always remember that to give the overall dimension of the object because sometimes once someone is reading an engineering drawing, probably their aim is to see how much space it takes. So, they are looking into what is the overall dimensions of this object.

So, for example, here it helps because the height of the object I can look here and say it is 50, the width I know here is 50 and here we have it the depth is 20. So, we have all the overall dimensions. So, I know that it is 50 by 50 by 20. So, that is a good practice to follow because knowing the overall dimensions is more often required.

(Refer Slide Time: 33:48)



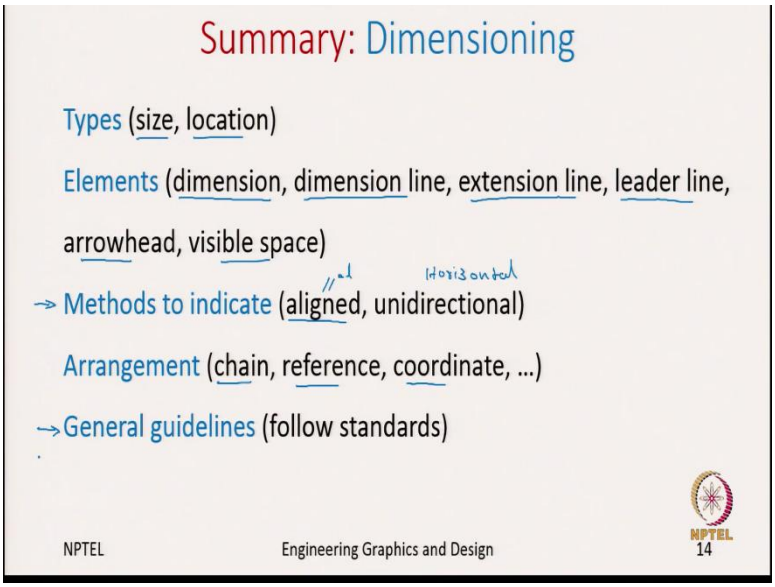
So, we also say that do not dimension the hidden features. So, hidden features as you know we are showing just for an additional information we do not want to show too much of information and so we say that for example, here this hole which is of diameter 10, we need to avoid using that by using this hidden line.

So, for example this 10 right the extension lines are going from this hidden line 1 and 2. So, we need to avoid that but how do we mention that it is 10? So, we can mention that by saying that we look into the front view instead of the top view and in the front view, we say that the diameter of this circle is 10.

There is another thing, other important guidelines saying that the dimensions of the view that clearly show the future. So, let us say the same information, I need to say this circle is diameter 10. I have two options to do, either I can do it in the front view or in the top view. So, now the guideline is to say where can you see the feature clearly?

Where can you see the circle more clearly in the front view or the top view? At least in this example, you can see this circle much better in the front view. So, you should always pick the front view to show the dimension instead of showing this dimension in the top view. So, that is why we say in the correct view this is given in the front view circle diameter is 10 and in the wrong way we do it in the top view.

(Refer Slide Time: 35:40)



So, let us summarize now what we have learned in this lecture. So, first we said there are two kinds of dimensions, one is the size and the location. So, the location says where a particular feature is positioned with respect to some reference line or surface or a point but the size talks about the actual dimensions or how large or how big or how small that particular feature is.

Then we also looked at different elements which are there in a dimension. So, when we say dimension, it means the number the dimension value to the dimension we are talking about. So, we show that using a dimension line with the help of an extension lines. We also use this leader line to give additional information about a particular feature.

And finally, when we talked about arrow head, we said there are multiple ways you can choose an arrow head and then when we look into the width and height there is a particular ratio of 1 : 2 and maybe you should be thinking how big or small you should choose a arrowhead that depends on whether you are using a big sheet or a small sheet you should find a reasonable value. Of course, if you look into proper standards that will give you the actual dimensions to use and you should always leave a space between the feature and the extension lines.

So, then we looked at two different methods by which you can indicate a dimension. First, we looked at the align method where we said the number, or the dimension value will be parallel to the dimension line. So, we said it is parallel but in unidirectional we said the number value, or the text is always horizontal.

We also looked at the different arrangements, if you have multiple dimensions, you can either follow the chain rule, but we said in the chain rule the problem is the, it accumulates the tolerance. To get rid of it you can use the reference which is more commonly used where we fix a reference line or a feature or a datum and from there you dimension different features. And finally, we said we all can also use coordinate systems where we have a common origin and then we can specify the centre of circles for example.

Of course, you can use a combination of these depending on the need of the drawing and lastly what we discussed are few important guidelines. I will repeat again saying that these guidelines are very common but depending on again the kind of application or the nature of job you are at, you need to follow a specific national or international standard which will give you exact details about how to do the dimensioning's.

So, with this we conclude this lecture on dimensioning's. In the last three lectures we looked at first starting what are the drawing sheets we can pick, later we looked at the different lines and we looked at dimensions in this lecture. With these three lectures talk about the principles or the formal way of starting with an engineering drawing.

What we look in the next lectures is, to get started we will actually go into the drawing process, we will start with some very simple objects and start by looking at how we go ahead with these drawings. Thank you for your attention.