

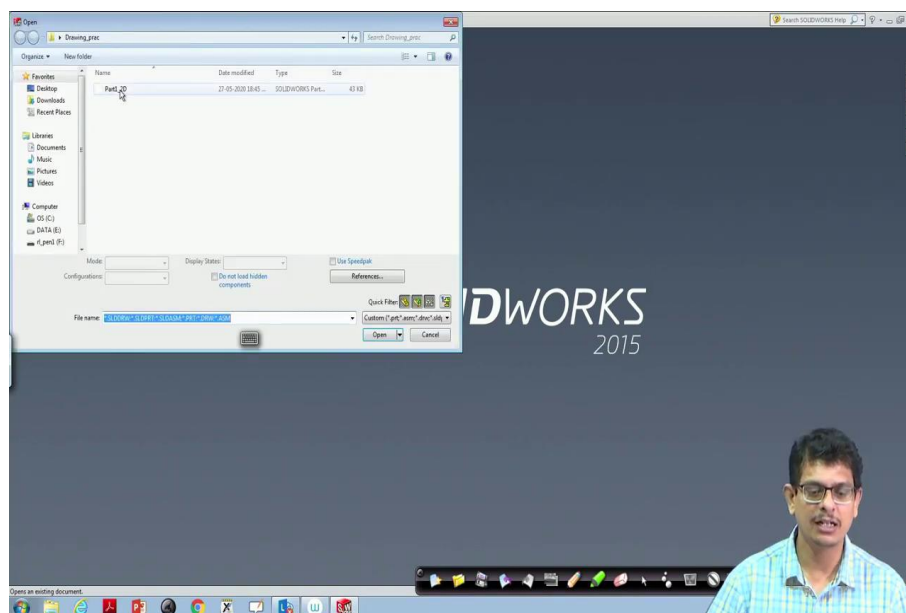
Engineering Drawing and Computer Graphics
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Lecture – 56
Solidworks (Contd.)

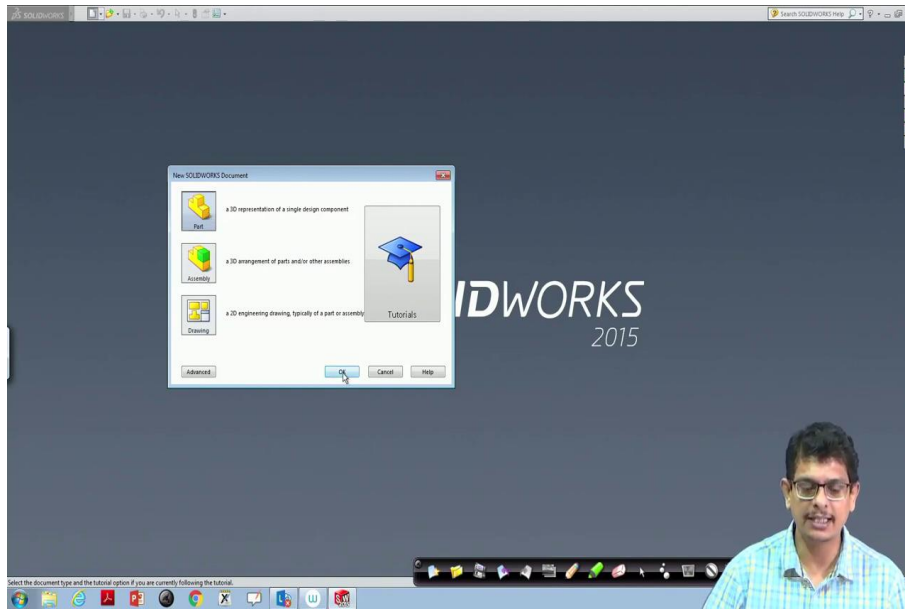
Hello everyone, welcome to our online certification courses on Engineering Drawing and Computer Graphics. In the earlier classes, we have learned a little bit about Solidworks. Now, we are going to use a variety of options involved with Solidworks.

So, in the last class, we have learnt that after double-clicking the Solidworks icon, we will end up with this kind of screen. In that, we have constructed something like the already previous one, if I want to open that we can use this open option click that part and open it.

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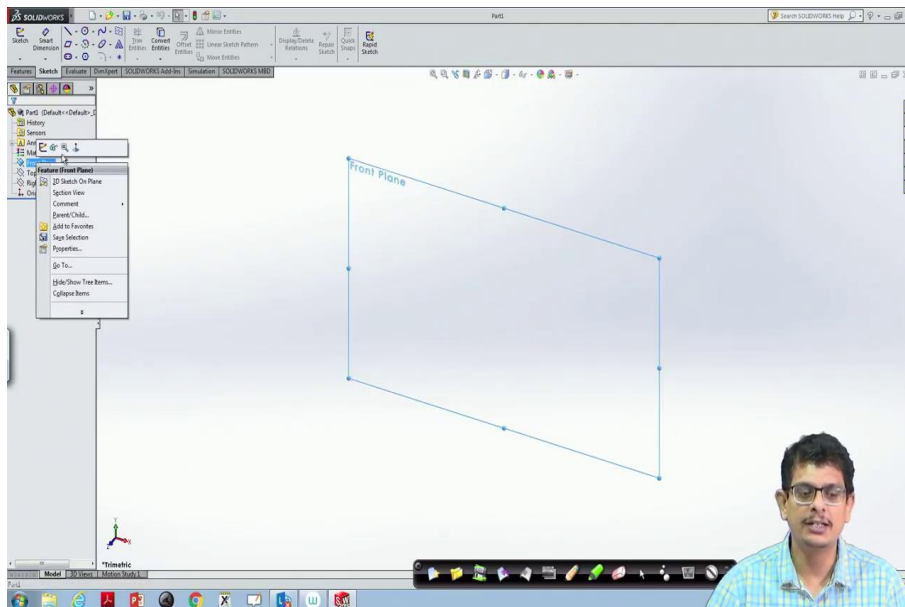


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If we are interested in constructing a new drawing, the first part what we have to do is go to this new option, click means left button click, part by clicking that, then OK.

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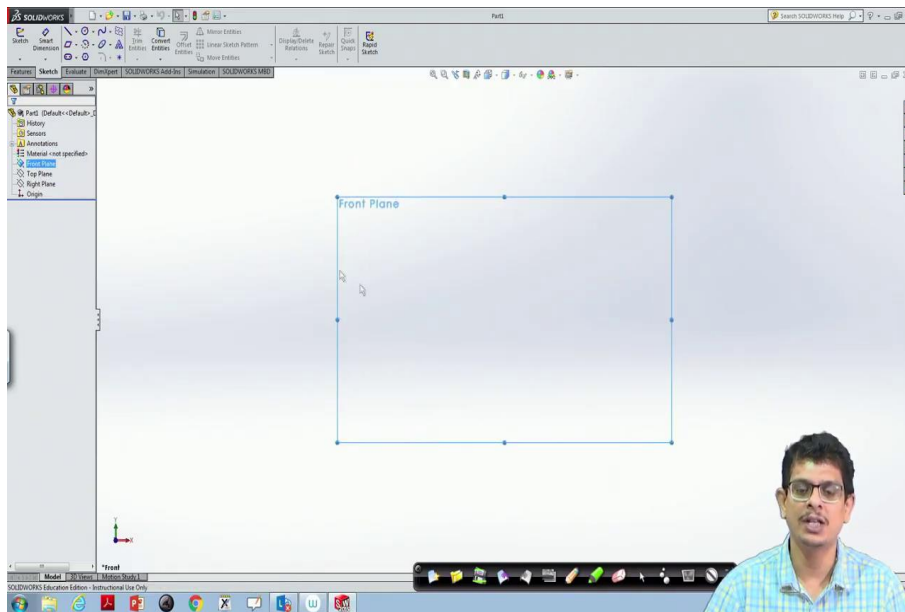
Then it opens a blank screen. In that blank screen, we see a variety of options like the taskbar, and something like feature manager and the other things.

Any drawing we would like to begin as a new one the recommended plane to begin is the front plane. So, this front plane, top plane, and right plane are not our original two-dimensional sketches when we have constructed the front view, top view, side view. Those views are different; here planes are

different. These planes are just to help the person to construct the drawings, these kinds of keywords have been used.

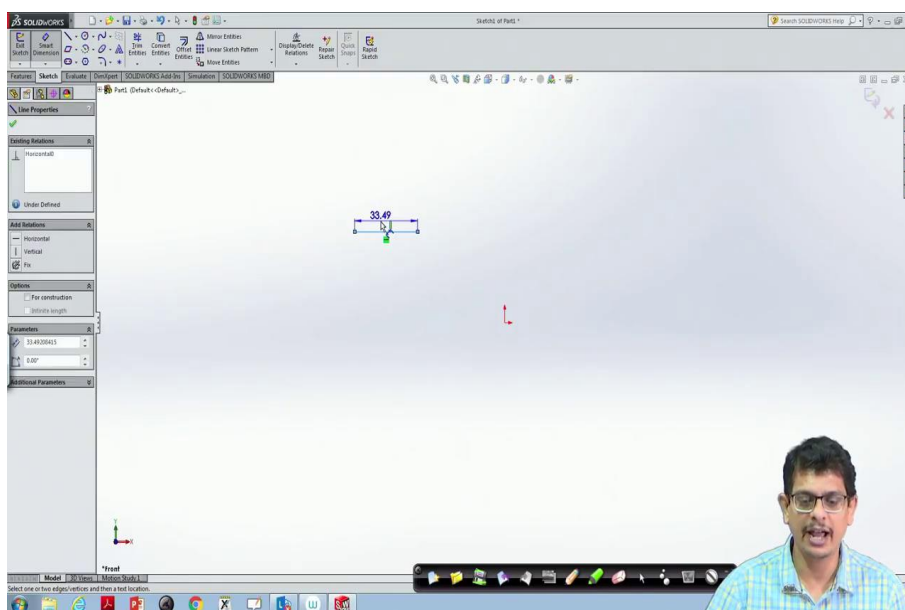
It is nothing to deal with front view, top view and side view. It is just a plane representation whether we would like to begin it to give a reference coordinate system, for that purpose only we use.

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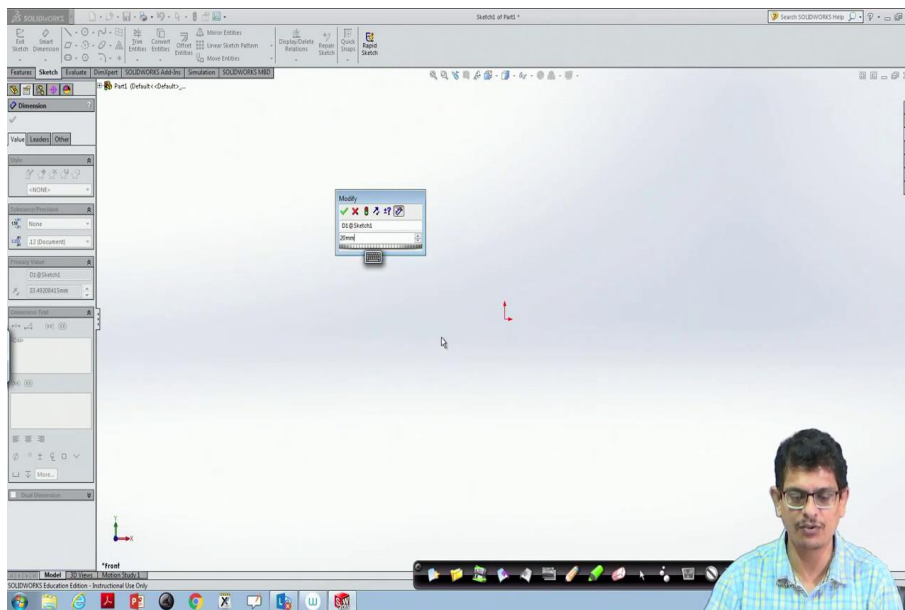


So, what we have to do? Mainly or all the time we pick only front plane, then give a right-click, it open something like a normal button, normal to that plane we have to click. Then the working space always be coming.

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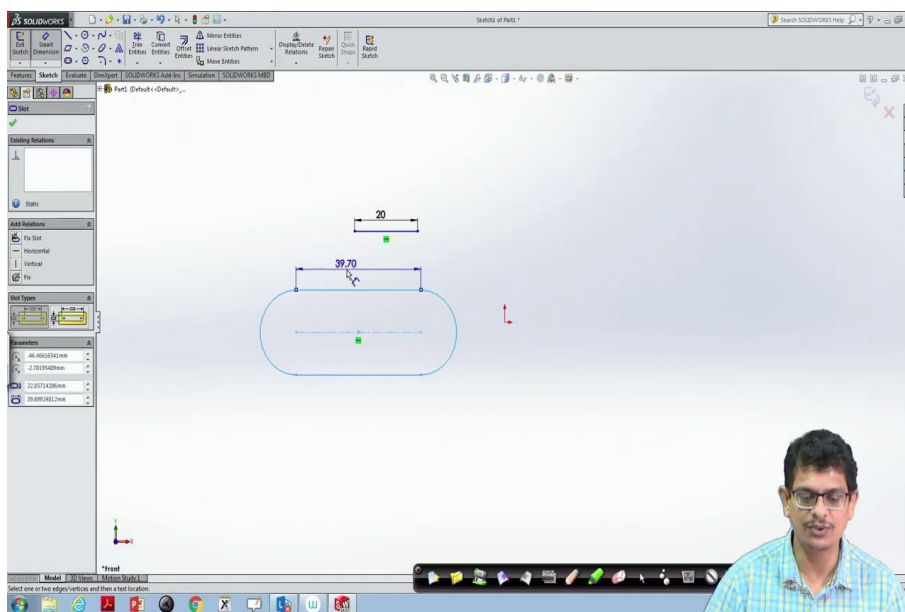


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In that, in the last classes, we have seen if I want to draw a line, I can draw it in that way. Then go to the smart dimension, adjust it to some 20 units. And units we always be careful like 20 mm then click OK. This is the way we have constructed line diagrams.

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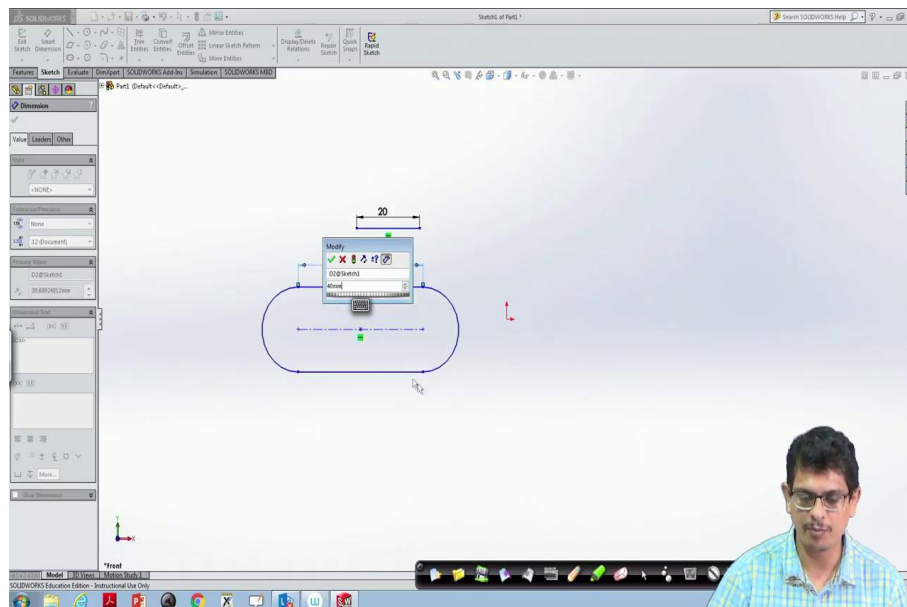
And also we went with rectangles. Now, there are specialized objects like slots, something like you have a plumber block, ah maybe any mechanical device where you want to keep something like a plate with holes and other things that kind of things what we call slots.

And those slots if I am interested to construct it, what I have to do pick straight slot. The object shape is mentioned there as a straight slot. You see there is something like a corner kind of thing with connected lines, pick that one.

So, pick a first line, second line, second point, so it constructs on the centre, now move your mouse. It constructs such kind of object. Again, first straight slot, click, do not hold any button, just freely move your mouse. Click the first point, then click the second point, ah freely move your hands.

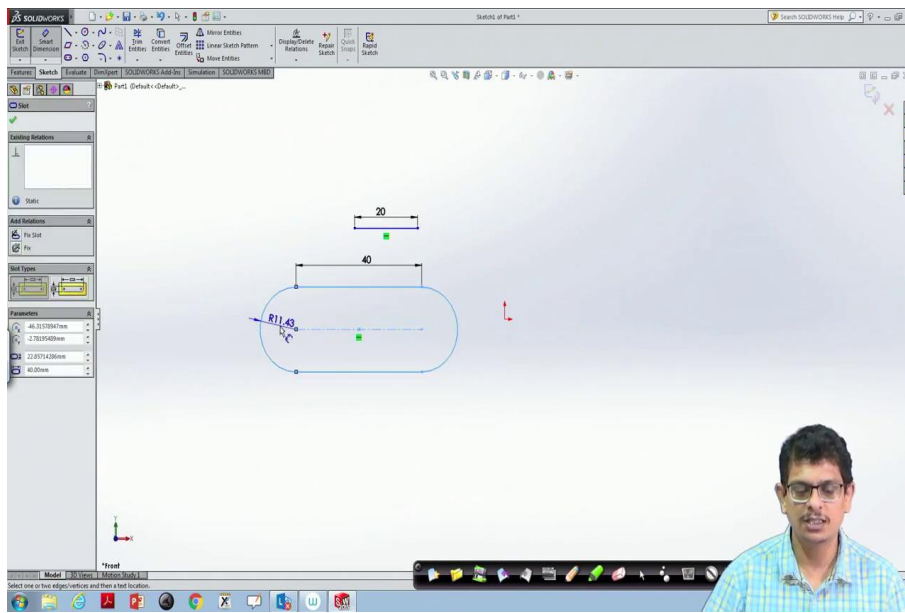
Now, just move your mouse to some other location it shows some kind of thickness width, then click, then this is done. So, if you are pressing escape, the object has been constructed.

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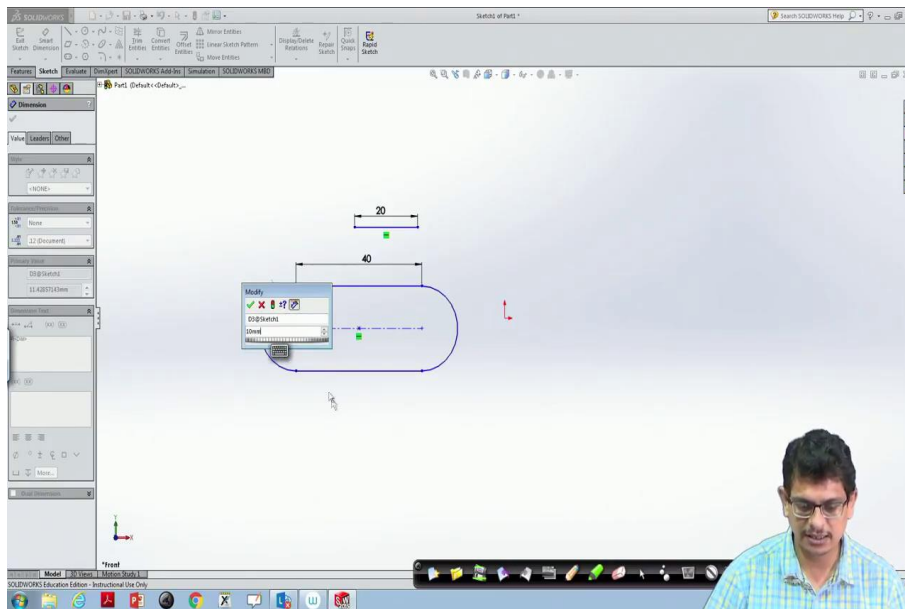
Now, you want to use the smart dimension. This one supposed to be something like 40 units you would like to have mm click ok. Then press escape.

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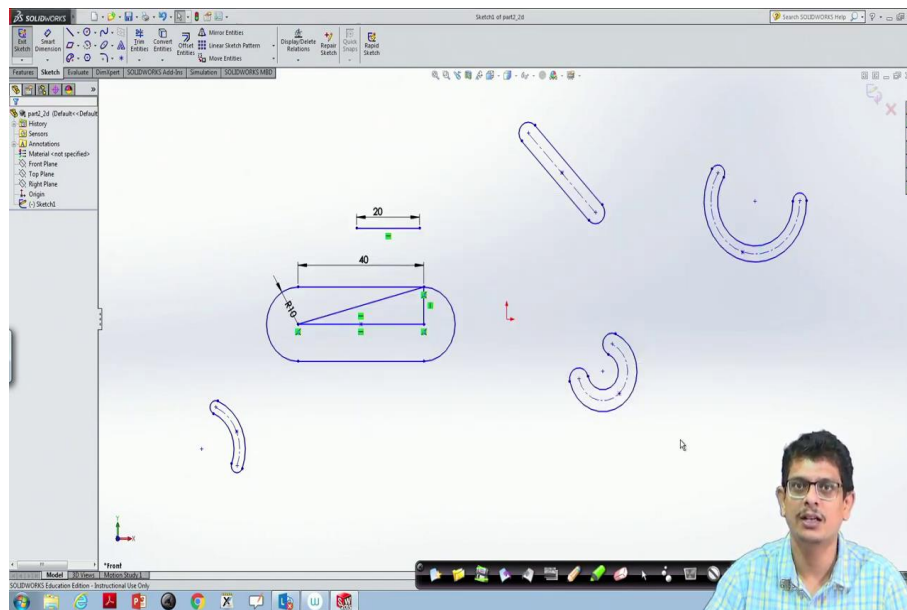
Now, I would like to adjust the radius of that click smart dimension then pick that, then it shows the centre from where this radius is present.

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So, radius I would like to have something like 10 units, then click ok.

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Then on both sides, because this is a symmetric object on both sides it naturally adjusts to that. This is the way you construct a straight slot. We will see what is the advantage of constructing this.

We can use the same kind of thing by using drawing lines, then drawing arcs, again drawing a line and closing it by arc also, semi-circle, we can do that. But there is an object straightforwardly available as straight slot; so, we are using that, then press escape. So, the dimensions always we mentioned like 40 units and arc 10 units.

In the same straight slot, now we are going with something like centre point straight slot. This example we are doing it just to have a grip on using this mouse, and various options whatever available at solid works. Now, pick the centre points straight slot. Click that. It shows a different kind of options like the first option, second option, third option. Pick the first one, centre point, now maybe I would like to construct it in that way too, then move your mouse, done.

So, these kinds of objects we use it for rigid body links maybe four-bar mechanism, this kind of things if I would like to construct at mechanical engineering. Then these links kind of objects always is helpful which under the keyword name straight slot available. Dimensions we have already learned, using smart dimensions I can always adjust this.

Now, I would like to have something like along an arc, I would like to construct that kind of link, mechanical link. So, I will pick three-point, arc slot, first point, second point, third point, and I just move press escape. So, along arc also we can construct.

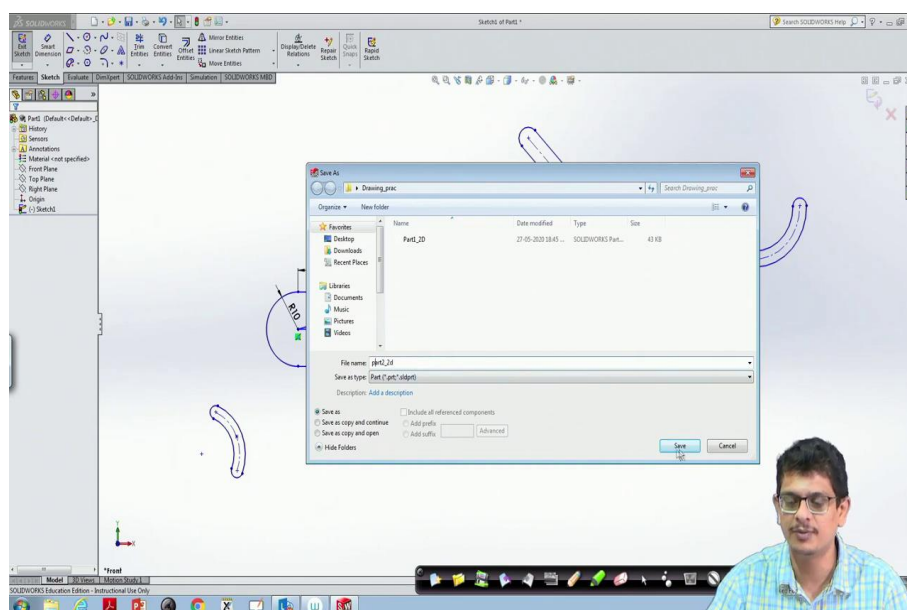
Let us do it once again. Pick three-point arc slot, the object whatever the thing we are going to draw is shown here as an icon, click that one. Pick first point somewhere click, left-click, right-click, ah sorry left click only, the second one, the third one click then move it. So, all the time left click, left click, left click, properly adjusting that has created this kind of links.

Now, we will look at other one centre point arc slot. Again on the arc, we would like to construct. So, for that we have to specify somewhere like centre, first one, the second one, the third one, then move, press escape several times. So, with respect to this centre, we have constructed an arc, then move to the third dimension to decide the width moved and clicked it. This is the way we use a different kind of things.

Now, if I would like to use a line, I just click a line, connect from there to there. Now, I would like to join that one with another one, again I click that one, click that point. Now, I would like to use one more line to join like a triangle, then I will pick that one and click that one. Whenever you would like to come out of that sketch ah the local sketch mode, you press escape several times; twice is good enough to come out of that sketch.

You are still at the sketch mode, but that local level where you are drawing you will be coming out so that you can use some other two like smart dimension, or rectangle, or any other kind of things. Now, I do not want to save this drawing, then I can straightaway click mark it. I would like to save the drawing, then naturally I have to go file save as.

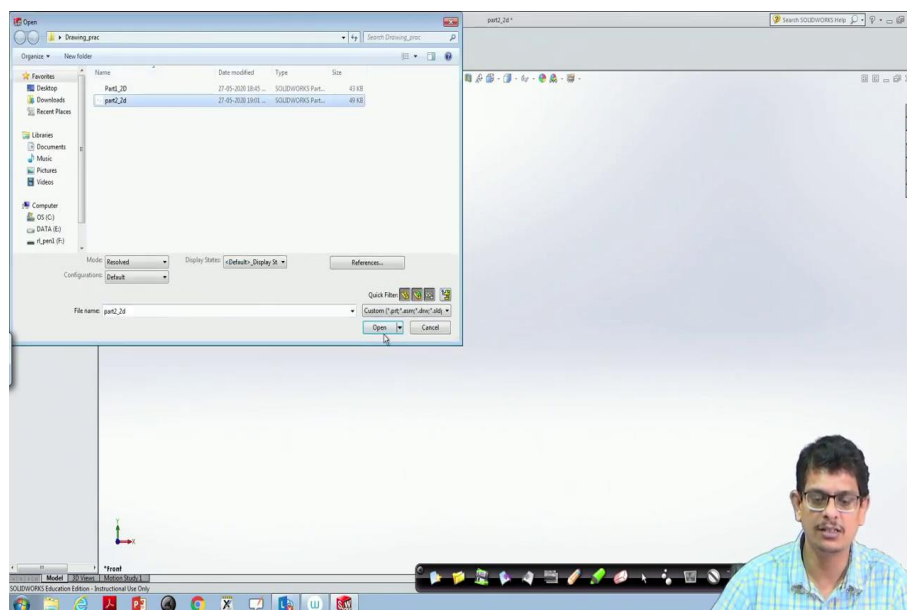
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Now, I have already constructed part12d. Now, what I will do is part 2 underscore 2 d drawing. ah Just be consistent with the names also. Use recommended is all the time lower case letters, better not to use any space in these words what we are using. For example, part 2 space 2d, I would not recommend you, because this remembering spaces will be a bit difficult.

So, if you want to specify some space has to be given use underscore button, so that that joins both the part2 and 2d thing. And it would be recommended if you go with everything in lower case letters, part2 underscore 2d everything in lower case letter is a good choice, to begin with. So, save. So, drawing is done. Now, I would like to close it; I can always close it.

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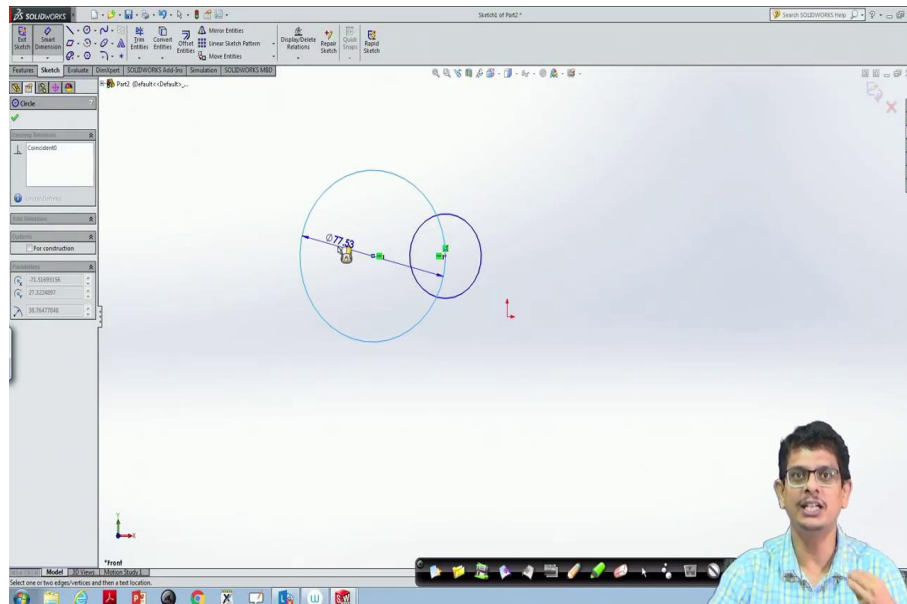
Now, if I would like to reopen that, open that. There is part2d. Open that because we have saved it the time when we click that sketch mode coming out of that again we have to save it we did not save it. So, now, the entire drawing you has been lost. So, it is always recommended to come out of sketch mode, then save the drawing, and later you can reopen it.

In the sketch mode you saved it, after that you straight away without coming out of sketch mode and saving it if you click that into the mark, it takes the recent version as you have already saved that is [FL]. Now, you are trying to close the drawing, that means, do not save anything. So, it erases everything. So, one has to be careful even at saving the drawings at regular things. Always come out of sketch mode, save the drawing.

Now, I would like to begin with a new drawing. So, begin with a new part OK. Now, for any new drawing, we have to go to the front plane, click normally to that. Now, we have learned about line,

rectangles, straight slots. Now, I would like to construct a circle on this 2d sheet. So, for that purpose, what I have to do? Click this button circle.

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Now, to draw any circle, the icon says that there is something like the centre point you have to pick, and something like a radius. So, for that purpose, what we are going to do? Right now my hands are not touching the mouse, but I have already clicked that circle icon, so still, it is maintaining on that ah monitor.

Now, what I have to do, give a left-click on one of the points. So, the centre has been picked. Now, move your circle to some other location, that means, we are specifying radius, click that. Now, we are done with that circle.

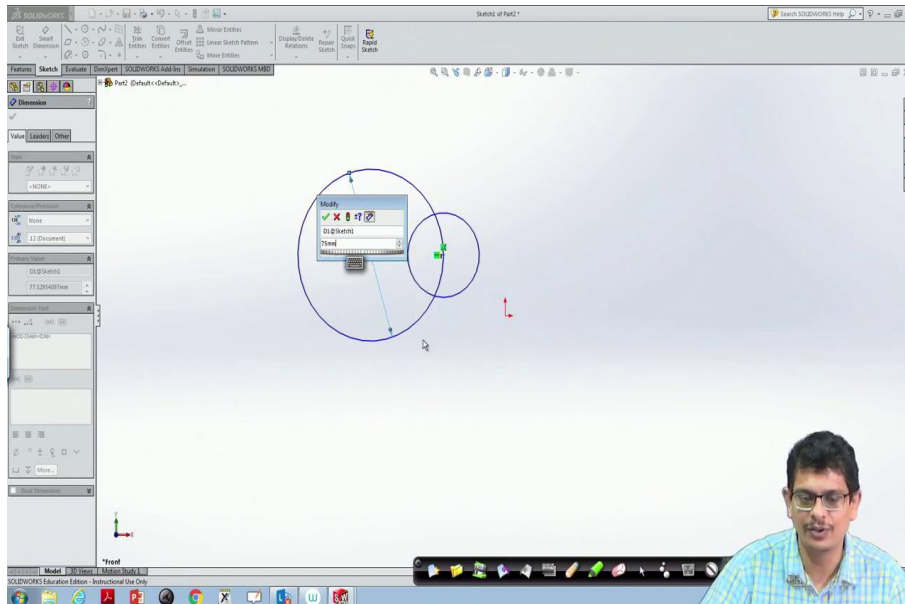
But your mouse, if you are moving is still in the circle mode, that means, if I would like to construct one more circle, for example, using this point I would like to construct one more, I can still use it. I would like to come out of that circle mode, then press escape twice.

Once you press, you are out of that circle mode. Now, I would like to use smart dimensions to realign these numbers. So, pick a smart dimension, click that circle, then it shows something like the diameter of that circle.

If you recall from our earlier lectures, whenever we use something like diameter, the symbol phi comes into the picture. And the same symbol same terminology here on software also we are seeing. Now, I would like to give internally the diameter.

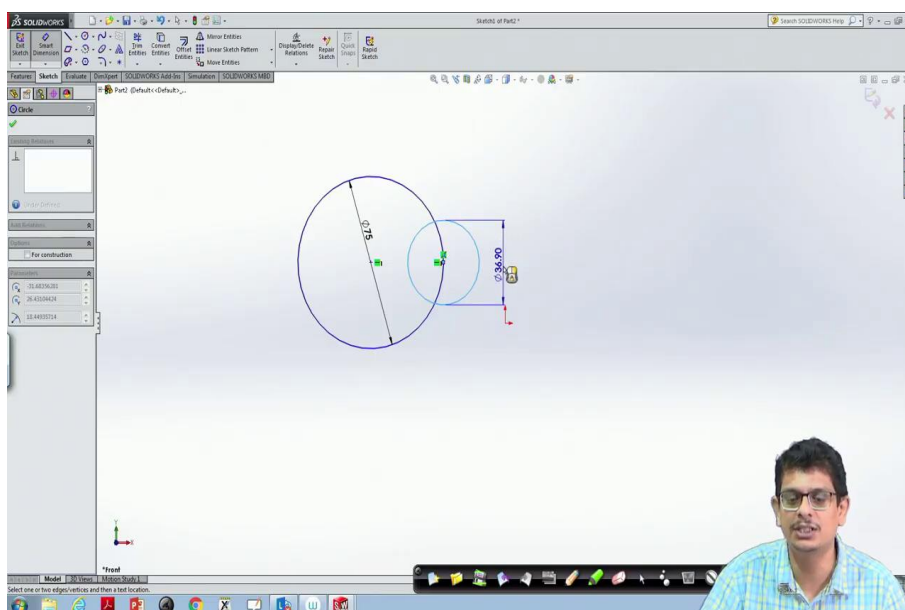
So, for that purpose, what I am going to do, one of the thing what we have learnt in our earlier lectures we should not give this kind of dimension, where your arrow is going to touch other circle and it unnecessarily creates ambiguity with the picture.

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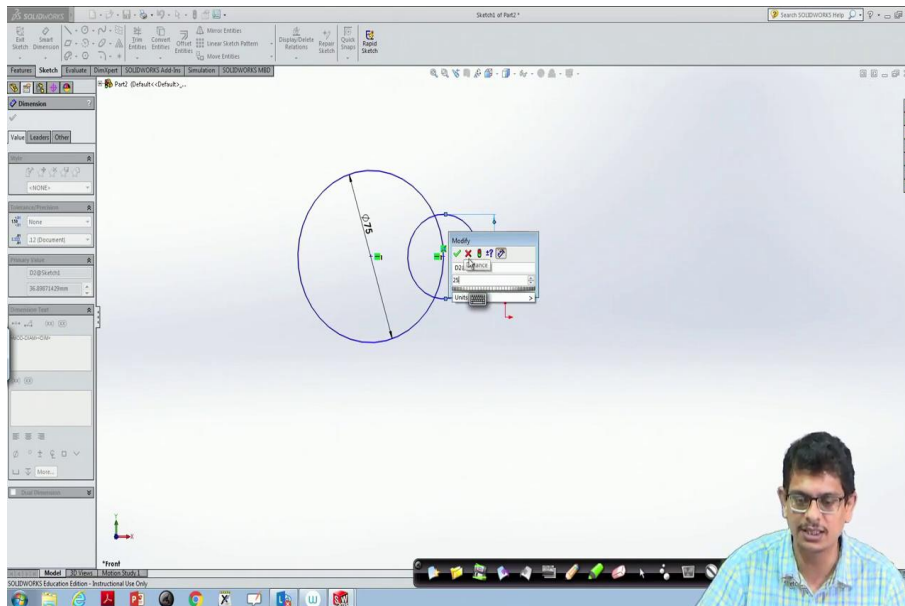
So, for that purpose, you have to carefully move your mouse, so that suitable diameter you can pick and click that. Now, you would like to have to give something like 75 millimetres. So, for that we are using 75 millimetres OK, then press escape. So, your circle will be of 75 millimetres.

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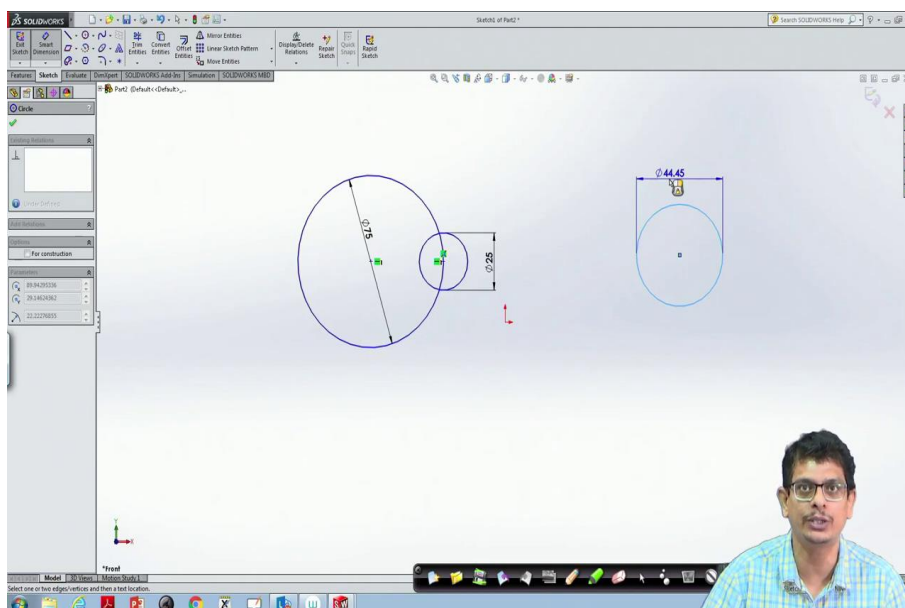
Now, the other circle I would like to adjust. So, smart dimension, click. I do not want to give internal, because now there is an ambiguity in terms of dimension. So, I would like to just go out of that. For that purpose, what I have to do, move your mouse out of that drawing. Do not click any button, just move your mouse.

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Now, if I draw it very long length, then it shows only one kind of arrow. If I come in this direction, it shows diameter in this way where I can always give something like 25 units, and click OK, then press escape. This is the way we can give dimensioning using centre base circles.

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Now, let us look at another example. In the same circle, there is something like the perimeter circle we have. Let us click that perimeter circle and see what it is doing. If you are seeing on the left-hand side, there are two things like the first one is the centre and radius, the second one is some three points around which this circle is passing. So, you pick three different points around which we are going to construct a circle.

In the manual drawing, we have a specialized method to construct that three-point circle, using tangents and other things. But here software uses that different kind of algorithm tries to fit a circle around that by minimizing these data points we have learned something about surface modelling similar to that we have learned something about curve modelling also.

So, because now we are picking up three different data points, it tries to assume $x^2 + y^2 + ax + by + c = 0$. Based on that a, b, c for three different points, it optimizes and provides what should be the centre of that circle where exactly circle has to pass.

At the manual drawing, we are going to construct a geometric procedure to draw that. But at software level, what we are trying to do is we straight away have a mathematical function, minimize these entire equations based on what should be the a, b, c values, that is the way most of the software works.

Now, we are using that internal structure. Though we do not know what is happening at that backend, because a professional engineer will be in a position to only click the options try to draw that, but at the back end of the software what it does is it uses this mathematical functions try to fit the curve and get the information, then that you will again get it in terms of GUI.

Now, we are picking that perimeter circle, three points I have to pick. So, click the first point, then the second point, I would like to have third point here somewhere here, then press escape.

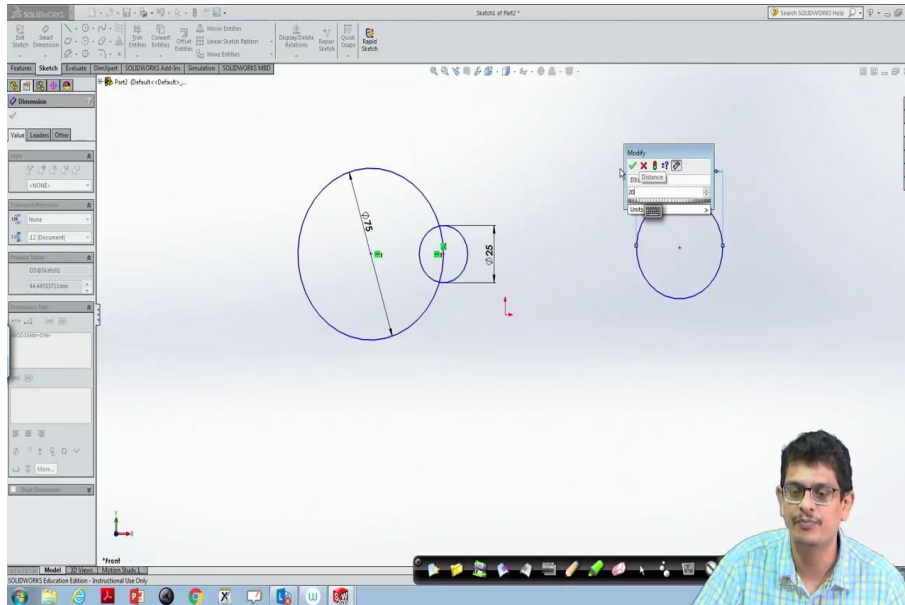
Because when I pick the first point, second point, third point, I do not have any control over what should be the radius I cannot control it. I cannot even control what should be the centre, because these are the outputs supposed to come out from that mathematical framework. So, your software does that. Straight away gives you where should be the centre of the circle and also what should be that radius.

Now, I would like to make this object may be a bit big or small using smart dimension. Let us see whether that takes care of it or not. So, what I will do? Smart dimension, pick that.

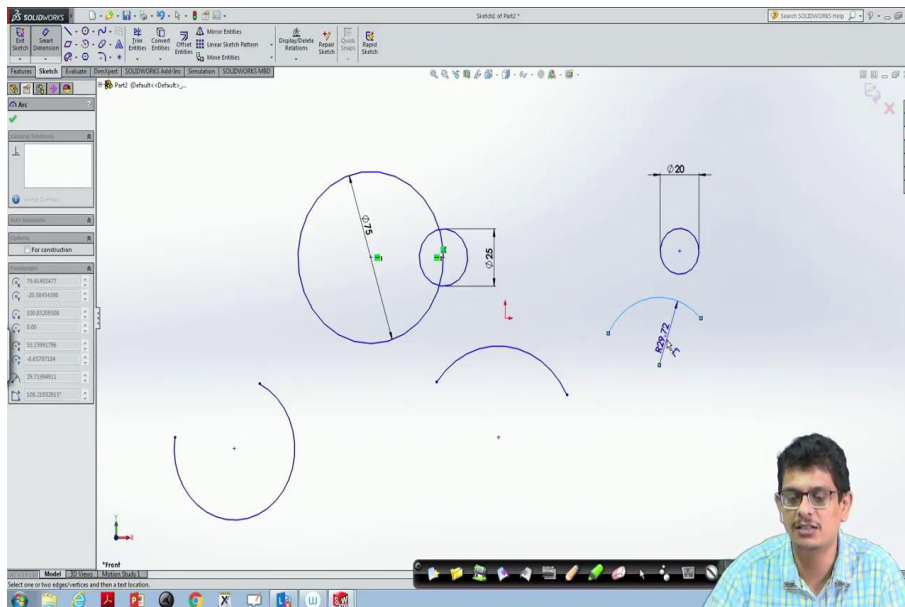
When I pick that, it immediately shows me diameter, that means, even though I am going to draw three-point circle saying that this is supposed to be the centre, this is supposed to be the curve which

supposed to pass through that. Now, if I am going to adjust at this first front it is going to take these dimensions. But let us try whether the dimension takes care or not.

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I do not want 44 units, but something like 20 units, I would like to specify. Then if I specify 20 units, it is adjusted. Now, those three points are not anymore three points. But what it has done is using those three points optimize the equations to fix the centre. Now, once the centre is constructed from any point of that, you are going to draw a circle.

Now, over that, you want to put additional constraint; no, no, I do not want this circle, but a circle supposed to have 20 units of dimension, then naturally the software does because now you are again changing your view. So, keep that centre, I am going to adjust this circle to 20 units 20 diameters. So, it goes unconstructed.

Do you remember like how we have constructed that three-point circle ah at manual level? From those points, we try to draw the arc and construct sectors and draw normals, where these normals are intersecting, then we locate the centre that is the geometric representation, but mathematical level it works differently.

So, whenever you are using software, you always go back check how a geometrical construction by hand drawing we have done, and how this software works. That gives you a flavour in terms of both geometrical ways of constructing the pictures and mathematical functions which might be used to construct that more like based on coordinate geometry rather than plane geometry, great.

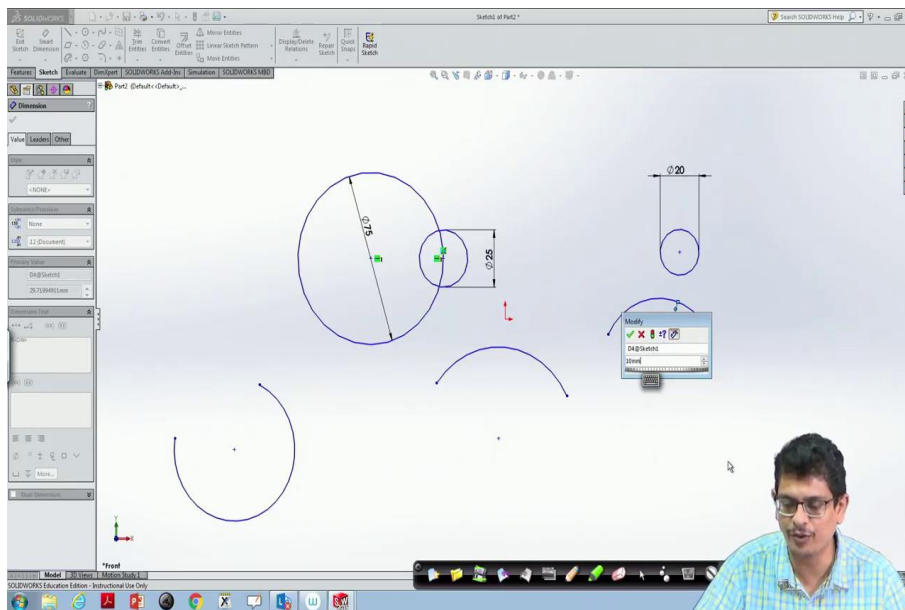
Now, we would like to construct only part of the arc, not a complete circle, so that option also available at Solidworks. So, pick centre point arc, the first one. Center point arc. So, first of all, I have to pick a centre point, go to some location, click arc I want. From there, I would like to construct an arc. So, I have to be careful in terms of using this mouse, where I am going to move and click that portion it constructs arc up to that level.

Let us do it once again. Center point arc, first one, now the second one I have picked. Now, I am moving my mouse in the counterclockwise direction, then click the option, then it constructs it. So, the direction along which you are moving it is trying to construct that. Now, we will let us go in the clockwise direction for the same arc. Center point arc, first point arc, now I want to move clockwise direction.

If I am going to do clockwise direction, your GUI always says that you pick the first point, but you want to end the point in the clockwise direction. So, what it does is it always goes in the counterclockwise direction, finish the circle. So, you have to be careful in terms of deciding arc up to which point level you would like to go. Though you would like to go in the clockwise, your arc always is taking in the counterclockwise direction. So, done, then press escape.

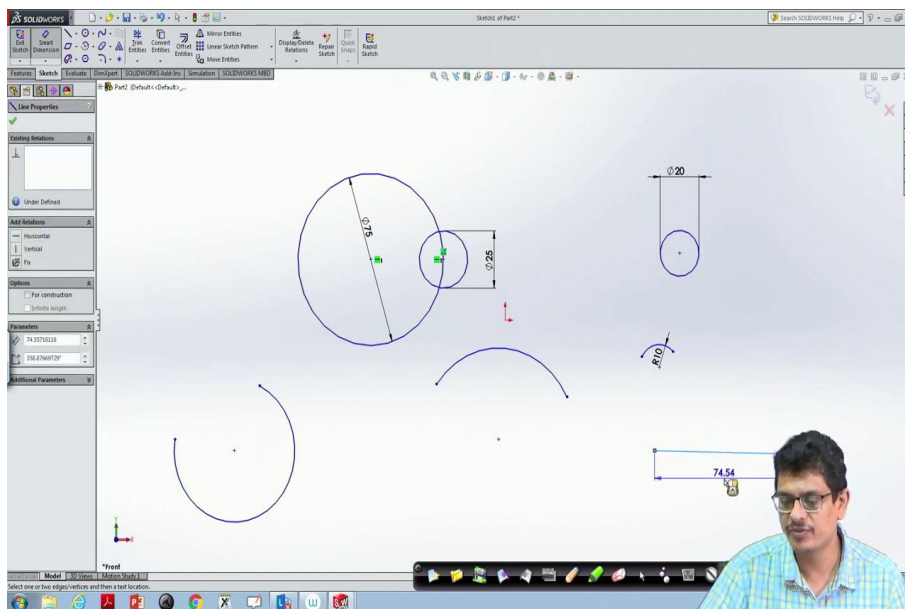
Now, I want to change the arc radius. Again our smart dimension is wonderful to go there, adjust that to something like these many units.

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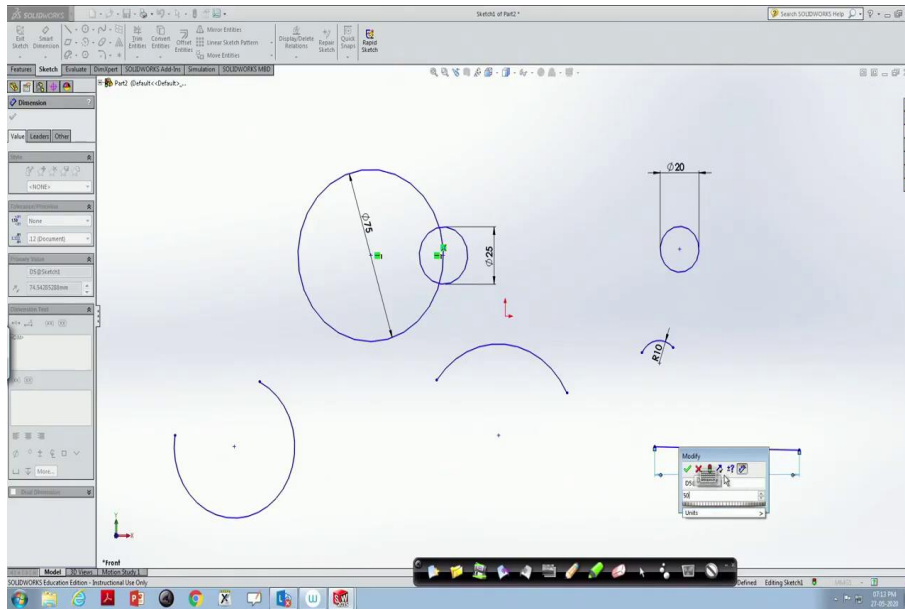
Now, I would like to have something like 10 radius units always be mm. We are going with mm OK, then the arc will be done.

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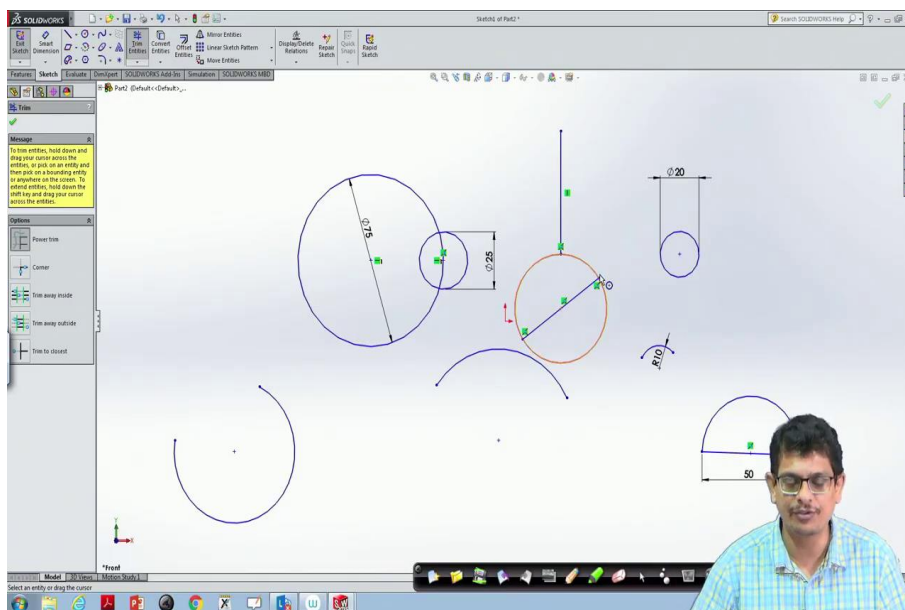


Now, let us try a combination like a line I would like to draw, on that I would like to construct an arc. Now, because I am online I would like to come out of that what I have to do press escape, escape, we are out of that line mode.

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Now, the line I would like to adjust it to some 50 units ok, done. Now, I would like to draw an arc centre point arc I would like to draw. This might be the centre point, I would like to pick the centre. The first point I have picked. I would like to join that, then escape, press enter. So, I can construct that arc in that complete 180 degrees.

Similarly, I would like to use multiple objects something like a line I would like to draw. We can see sometimes we can see this option also whether it is 90 degrees line or not, escape mode. Now, I would like to draw a circle around that. Circle I would like to draw, maybe the centre of that circle is this

point. I just go ahead, click the second one, done. So, any additional stuff if I would like to use, one over other I can construct.

Now, I am not interested in this point. Then I can always use trim entities object. We will shortly see how to use these trim entities. Now, because this circle I have it. I have drawn something like line; I want to either remove this bottom part of that line or top part of that line.

For that purpose, if I just straight away click that line, delete it, it deletes a complete line. Instead of that, we have an object like trim entities. There is something like scissors. Click that. Then it shows you many dialogue boxes something like there is power trim, corner trim, trim away inside, trim away outside, trim to close it, this kind of options always be there. You have to practice how to use this.

So, for example, let me pick power trim. Power trim is powerful. In that sense, you just swipe on particular line; it just immediately removes that line. Let us click that power line, then click that button hold it, drag along that line, it removes that line. Let us do it once again. I would like to draw a line from here to here.

What I want to do is I would like to remove this outside part of this line which is exceeding that circle. For that purpose, what I am going to do use trim entities, pick power trim, carefully click hold your left button click hold, and move your mouse along that line, then release your mouse button. Once you are done, that line will be removed.

Let us use the same thing for other lines. The other line what we are going to use is now we have this point, this point, maybe I would like to remove this part of the curve here. For that purpose again trim entities, power trim. Right now I did not move anything, now I am going to click the left button, hold that, and move over that line. Now, release your mouse button. Once you are done, whatever those intersecting points are there, that part will be removed.

Similarly, power trim, now I would like to remove this bottom portion, for that purpose, click left button move along that line. So, I have that object. So, even though I am constructing typical geometrical things, I would like to remove some of the lines, I can use this power button power trim button to click-drag over that to remove that part of it. In the next classes, we will learn about more options available at SolidWorks.

Thank you.