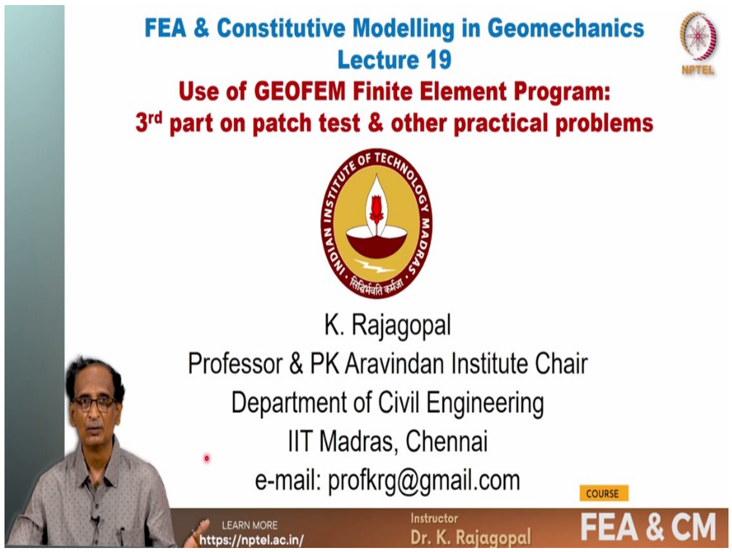
FEM and Constitutive Modeling in Geomechanics

Prof. K. Rajagopal
Department of Civil Engineering
Indian Institute of Technology – Madras

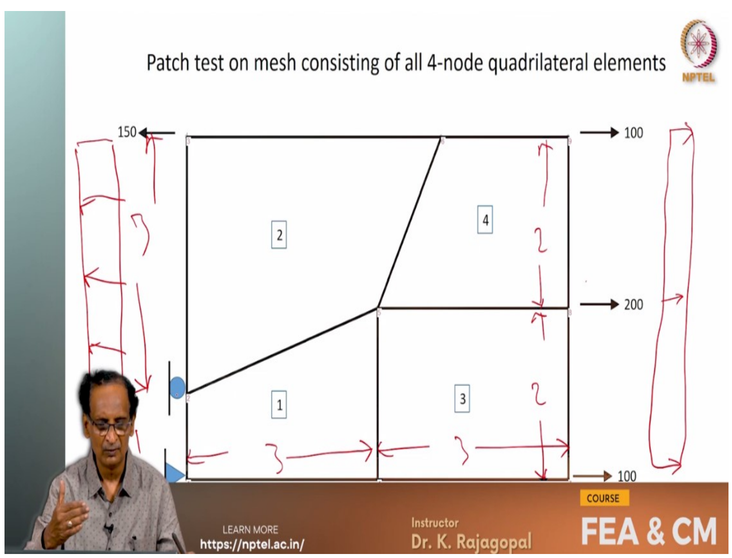
Lecture – 21
GEOFEM Part III

(Refer Slide Time: 00:20)

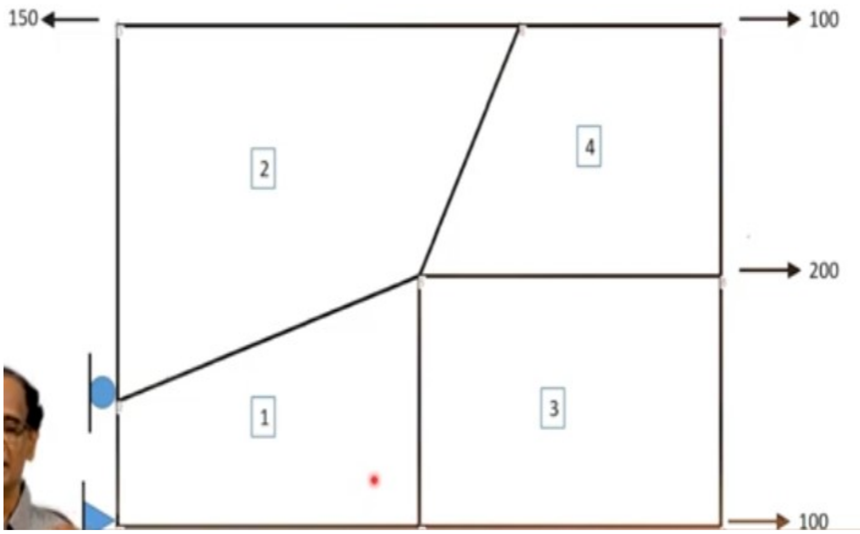


Let us look at the use of finite element program for doing the patch test and other practical problems that we had discussed in the previous class.

(Refer Slide Time: 00:33)



One of the patch test that we performed was on this type of mesh that is consisting of totally 9 nodes and 4 elements each of 4 node type quadrilaterals and what is done is we applied a pressure loading on both right hand side and the left hand side and then just given some boundary conditions that will prevent any rigid bodies modes of deformation and to prevent the rigid body motions we had fixed node 1 and hinge.

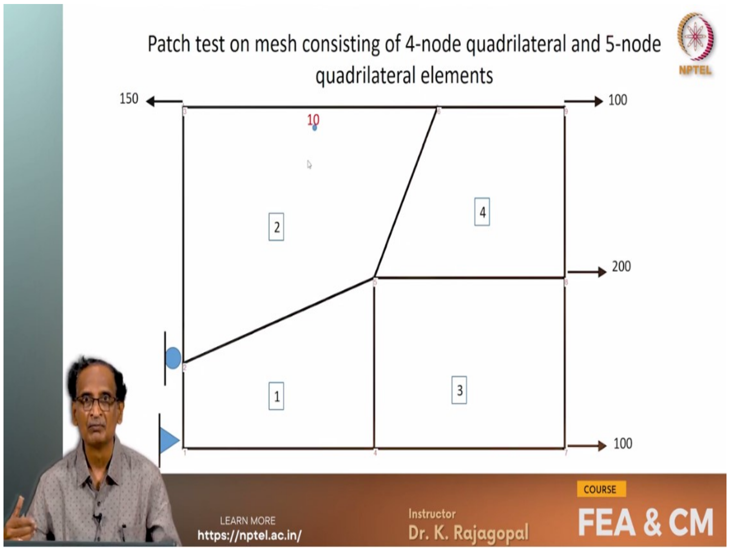


So that both x and y direction displacement is constraint then at node 2 it is on a smooth roller so that the material can move in the vertical direction, but not in the horizontal direction and then on the right hand side the pressure loading of 100 is given and each of these is of length 2 units and let me just indicate the units here. This is of length 2 and this is of length 3.

And this is of length 1 and 3 and so if you apply a pressure loading of 100 KPI here and because this being 2 node surface the load is equally distributed. So, 100 times 2 is to 200 so you will have 100 here and then another 100 here and then on the same pressure on the other element 4 will have 100 here and 100. So, at node 8 we will have totally 100 + 100 200 and then on this surface also we have given the loading in the other direction like this.

And then on this element 1 both of these nodes are fixed in this lateral direction so there is no force and there is only force on element 2 and this pressure of 100 is acting on a length of 3. So, it is totally 300 and we distribute half of that; that is 150 here at node 3 and another 150 at node 2, but node 2 being a fixed node we will not have any load transferred there.

(Refer Slide Time: 03:59)



And then the same test is repeated by so the same test is repeated for by changing this element into a 5 node element I have introduced another node 10 and then this will be 5 node element the nodes are defined as 6, 3, 2, 5 and 10 and that we will see the GEOFEM program and here also we have this loading of 100, 200, 100 then 150 **(Video Starts Here: 04:46)** And now let us see how do we do this in the program.

And let me just so here is our data file for the patch test, let me just zoom it a little sorry I am not able to zoom it, but let me just illustrate this little bit okay I cannot use the pen it is okay I think we will do it. So, here this mesh has 9 nodes and then have made all the 9 nodes free, but then later I have given some boundary conditions and the loading I have given instead of points load I have given it as a uniform pressure.

So that the program we can calculate on both the right hand side surfaces applied a pressure loading of 100 and the left hand side at node 3 there is a load of 150 applied and then we have this all this quadrilateral elements they are defined with 4 nodes 5, 2, 1, 4 then 6, 3, 2, 5, 8, 5, 4, 7, 9, 6, 5, 8 and then node 1 is fixed in both x direction and y direction. So, you see here 1 and 2 it is fixed whereas node 2 is fixed only in direction 1 that is in the x direction.

So, if you look at this result let me see whether I can okay I think I can increase the font a little bit let me increase it a little bit more. So, here we have a mesh with 9 nodes and then number of element types is 1 because the entire mesh is made up of only continuum elements then there are no body forces considered, analysis type is only linear elastic 0 and then these are the 9 nodes and then the coordinates and all the displacements are free at the element level.

But then later we put some constraints and this is a plane stress state the Young's modulus Poisson's ratio and then there are 4 elements. The first element is connected between node 5, 2, 1, 4 second third and fourth and then there is one parameter IRCT that refers to the shape of the element. If this value is 0 then it tells the program that the element is a triangular element and if it is given as 1 it is a quadrilateral element .

Then there is another type 2 that refers to the infinite element then integration order 1 is for 1 point, 2 is for 2 points; 2 points in each direction both Xi and eta directions and then the stress grouping is actually if you want to print these stresses at some points you can use this option and then the loading at node 1 there is a load of 150 given in the negative x direction because the NCODE is 1.

And then on the right hand side we have the nodes there are two surfaces one is 7, 8, 8, 9 and then I have given a negative pressure. So, as you recall if it is a positive pressure it will be acting into the element and now on the right hand side I want the pressure to act outside outwards so I have given that as a negative and then if you look at this we have 4 elements and all the elements they have a tensile stress of 100 everywhere.

And there is no sigma y and no sigma x y and these are all 0. So, 10 to the power – 13 means practically 0 then the sigma x is 100 exactly 100 that is what we applied. So, we can say that this particular patch of elements is able to pass the patch test because we applied a constant stress of 100 and the same thing is predicted inside the elements. So, we can say that this particular patch has passed the patch test.

And one of the requirements is that we should apply boundary conditions corresponding to help the patch to develop the constant stress state and now let us do slightly different thing, let us prevent the mesh from moving in the other direction by putting one more boundary condition. So, here I am just doing one more thing at node 2 I am fixing the displacement in the y direction and now let us see what happens.

So, here we are not allowing the mesh to deform in the other direction. So, now we see our sigma x is not constant it is nearly constant. So, it is within element 1 at one integration point 103.56 another point 98.7 99 and actually if you are away from the boundary it is actually element 1 is very near to the boundary and so that got affected very badly in terms of the varying stresses.

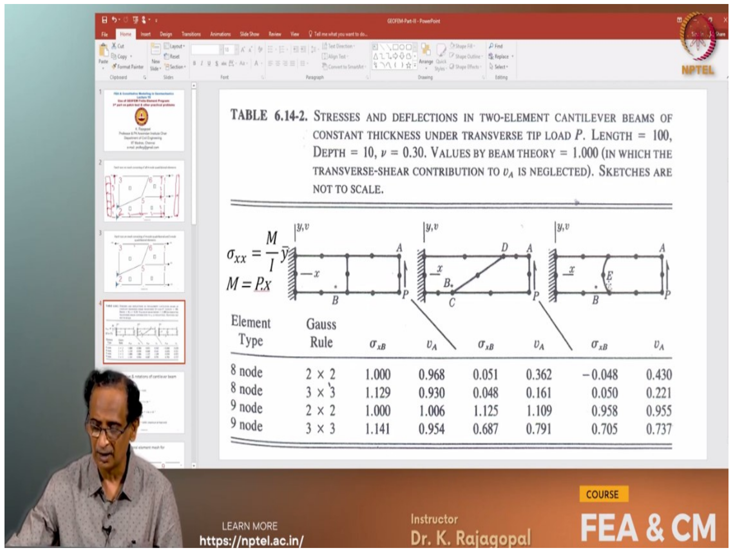
But if you take element 3 that is away from the boundary the stresses at all the 4 points are nearly the same 99.19 99, 100 and then you see here we did not apply any stress in the y direction, but then because of the Poisson effect there is some stress developed because we are not allowing the material to freely deform in the other direction and the same thing although we did not apply any shear stress the material has developed to some shear stress.

Whereas in the previous case let me open this and show you so this is our previous result where we allowed the lateral deformations to take place. So, that the Poisson effect will not be seen on the strains and the stresses. So, we have a perfectly sigma x is 100 there is no sigma y stress and there is no shear stress in all the elements whereas in this new case where we prevented the material from straining in the other direction we have a problem.

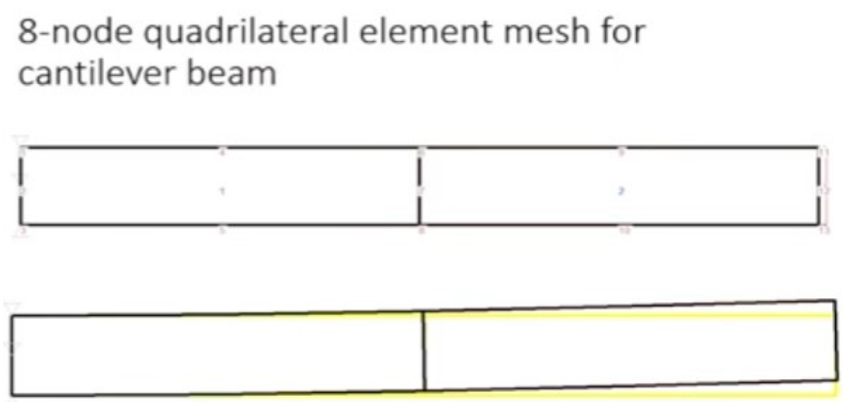
So, when we perform the patch test we must make sure that your boundary conditions also they enable the patch to pass the patch test that is very important and then the same thing if you do for the element with 5 nodes we see that see one of the elements 2 it has 5 nodes 6, 3, 2, 5, 10 and then everything else is the same and all the stress is inside the elements are all constant 100 in element 1, 100 in element 2, 3 and 4/

And there is no stress in the sigma y direction at tau x y. So, we can say that even combination of 4 node and 5 node quadrilateral elements is able to pass the patch test. So, we should expect the monotonic convergence by using this type of elements **(Video Ends Here: 16:00).**

(Refer Slide Time: 16:01)



And in the previous class we had also seen the influence of the type of element and then the order of integration. So, this particular one is from the textbook by Cook, Malkus,.Plesha where the influence of the type of element 8 node quadrilateral or 9 node quadrilateral and then whether if you have a rectangular element or a distorted element what difference it makes.



This particular one is a perfect rectangular element this is a distorted element, but with a straight side and the third one is a distorted one with a curved side and for the first one both 8 node and 9 node element measures they are able to give very good result with reduced integration 2 by 2 integration and when we use higher order of integration 3 by 3 this solution accuracy has reduced a little bit especially in the sigma x.

Sigma x at this point at one of the integration points and then when we distort the element the 8 node element has performed very badly when you have 2 by 2 integration the displacement is only 36% of the actual result and if we use the higher order of integration 3 by 3 is even worse 0.161 and the same thing when you have this curved edge the solution is very bad, but then if you have this 9 node element with a 2 point integration it is not bad **(Video Starts Here: 18:05).**

And let us see this. So, this is the analysis with sorry I think this is 8 node or 9 node let me open the 8 node so this is the one with 8 node quadrilateral element. There are totally 13 nodes. So, here we have two elements and 13 nodes 1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13 and I think this is the 13 nodes and then this is the deformed shape the yellow one is the original mesh and then this black colour one is the deformed shape.

And let us see how the displacement and everything are predicted. So, the first one is 8 node element or there are totally 13 nodes and nodes 1, 2, 3 are fixed completely because we have a fixed and cantilever and then 8 node elements the element one is connected between these nodes 6, 1, 3, 8, 4, 2, 5, 7 then second element is 11, 6, 8, 13, 9, 7, 10, 12 and so on and then the shear load of 100 is applied at the tip.

And the tip displacement should be 0.02, but we are getting reasonably good value like 0.119 and then if you look at this horizontal deformations the center point there is no horizontal deformation because that is a neutral point see within depth of 10 I think this depth is 10 yes so this the depth of the beam is 10 and the length is 100 units. So, at mid depth there is no lateral deformation so that means that it has undergone a pure flexural type deformation.

And the upper surface has deformed to the left by 0.0014 and then the lower surface has deformed to the right by the same amount 0.0014. So, the total is 0.0028 that is the relative deformation divided by 10 is 0.00028 and if you look at the theoretical solution. So, this theta is 3 times 10 to the power of – 4 and our finite element is predicting 2.8 times 10 to the power of – 4 which is not bad.

Considering that we have considered only two elements and then let us look at the stresses. See this stress sigma x is our flexural stress and that is the maximum value is 6,000. I think that is given in the power point the flexural stress is M by I times y bar they are the fixed end your moment is 1,000 times 100; 100 is the length, 1,000 is the total shear load divided by 1 12th 10 cube and the maximum distance of the fibre from the neutral axis is 5 that is 6,000.

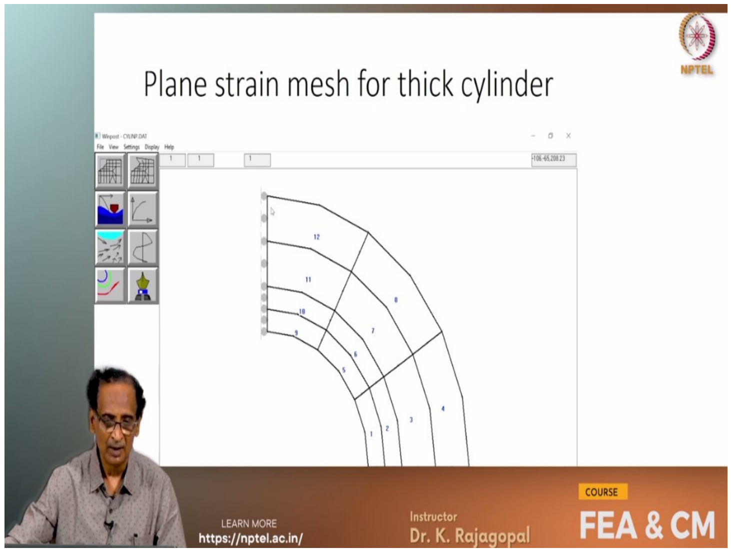
And that is what we see here. So, at the fixed end when x is 0 our sigma x is 6,000 and the upper fibre is actually it is bending in the anticlockwise direction. So, the upper surface is on the compression so we have – 6,000 and then the lower surface is + 6,000 it is tensile stress and then if you see this at the tip when x is equal to 100 our bending stress is 0 because that is free end.

And then our mid length at x of 5 both element 1 and element 2 they are giving the same stress of 3,000 depending on whether you are at the top it is a – 3,000 and if you are at the bottom it is 3,000 and then although we did not apply any stress in the y direction there will be some sigma y there is a formula for that and if you check these values are not bad sigma y then sigma xy we applied a shear stress of 100.

But then we are not able to predict that shear stress maybe because of this we have used only 2 elements so it is not able to predict the shear stress correctly, but if it is done correctly then it should have predicted exactly 100 and then let me show you another result with 9 node element. See with 9 node element mesh this is with 3 point integration. Our displacement is at the tip is 0.02 that is not bad.

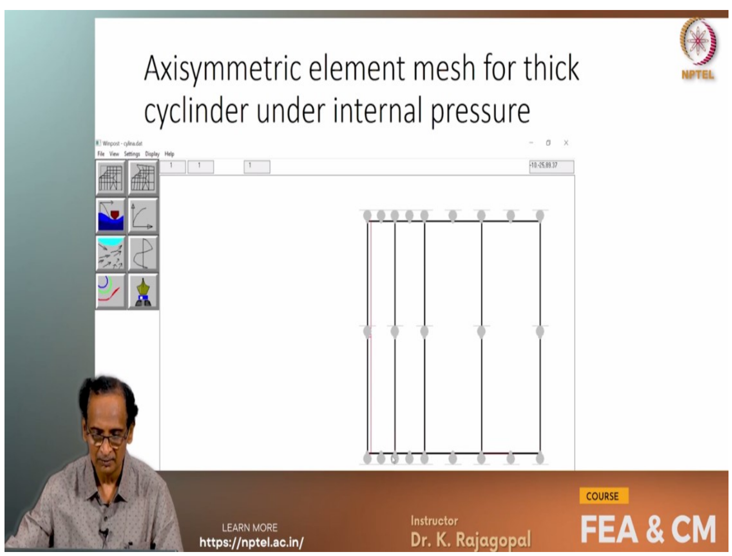
But then our flexural stress is not predicted correctly it is at x of 0 we are getting only 5,389 against a stress of 6,000 that is because of our integration order whereas with 8 node element we were able to get the exact flexural stress of 6,000. So, you see that our results not only depend on the type of element, but also on the order of integration. We have to repeat and see whether we are getting reasonable results or not. **(Video Ends Here: 26:03)**. So, now let me just show you quickly two other things.

(Refer Slide Time: 26:10)



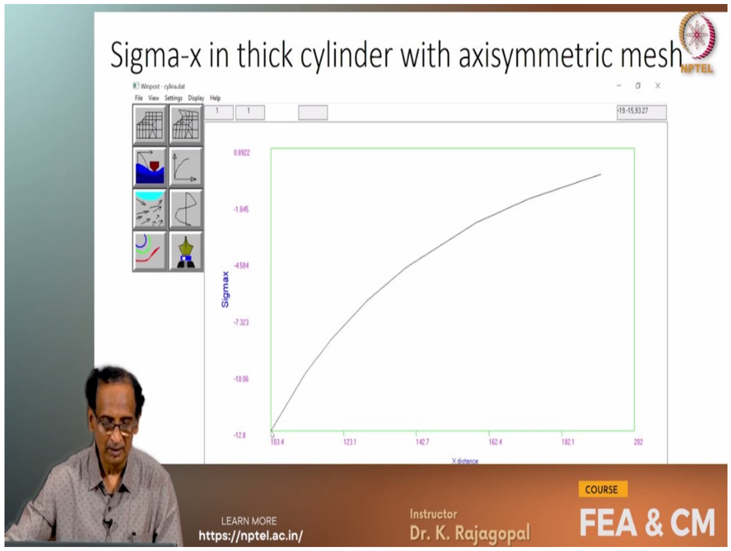
See imagine that you have a thick cylinder that subjected to internal pressure and in one of the classes before we have seen that there can be two different models. One is a plane strain model by taking section across the length we can take, we can model it as a plane strain case and this full circle this full cylinder is modeled with quarter symmetry and see here along the x axis all the nodes are moving only in the x direction whereas on the vertical boundary at mid section you have only vertical displacements.

(Refer Slide Time: 27:01)



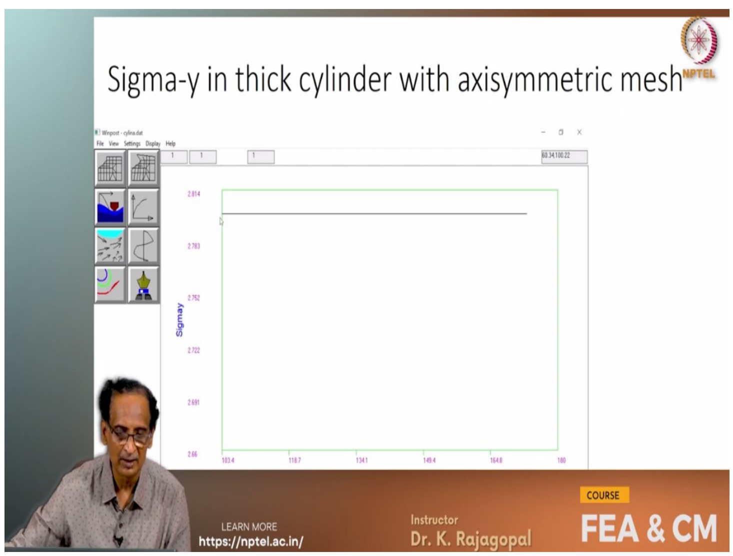
And then the same thing we can model by taking a longitudinal section as an axisymmetric one and so the axisymmetric one we can taken one thickness and for unit length and constraint all the lateral displacement so that the cylinder can move along the radial direction like this. So, here you see all these nodes with a circle so that means that they cannot move in the vertical direction, but they can only move in the horizontal direction.

(Refer Slide Time: 27:43)



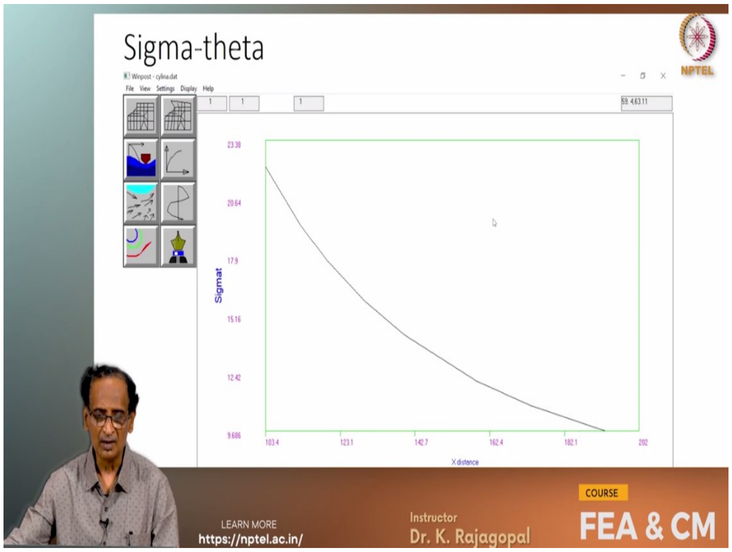
And this is the radial stress with axisymmetric mesh the maximum at the interior point is 12.8 then as you go away the pressure reduces.

(Refer Slide Time: 28:00)



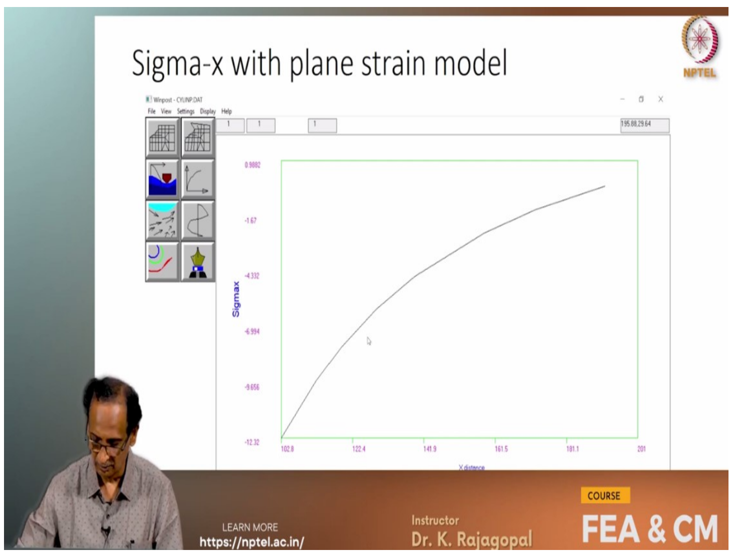
And sigma y that is actually in the other direction longitudinal direction it is actually it is very small because there is not much attacks along the length and it is constant across the thickness it is constant.

(Refer Slide Time: 28:20)



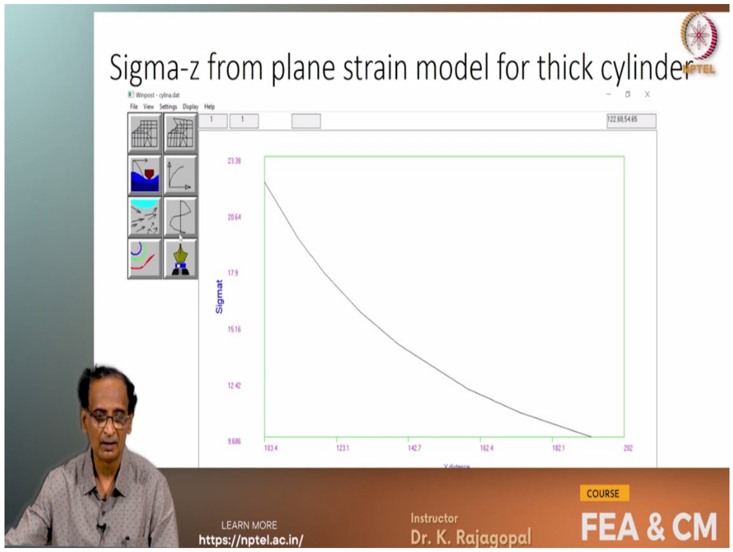
And sigma transverse that is the hoop stress are circumferential stress. This is also the maximum is somewhere around 22 and it is reducing as you go away.

(Refer Slide Time: 28:36)



And this is the sigma x with a plane strain model it is also predicting almost the same the maximum is 12.32 then it is reducing.

(Refer Slide Time: 28:48)



And this is sigma z in the plane strain model is equal to your sigma theta in the axisymmetric model. It is about 22 at the interior it is actually this is on the integration points this is not exactly at the tip. I will show you where the stresses are computed. **(Video Starts Here: 29:14)** And let me just go back. So this is our mesh that we have used it for this analysis and then let me show the boundary conditions.

So, this is our mesh actually ignore this straight just actually this post processor does not have the capacity to draw a curved line it is just simply drawing straight lines between all the points, but inside the finite element program it is a pure isoparametric element. So, it will consider as a curved round surface. So, we have totally 12 elements and then these are all the nodal points.

And let us see how the stresses have varied let me put sigma x we can select the integration points. So, these are all the points where this stresses are computed, these are the integration points and we can draw a line and then select all these points and then plot any stress and here plotting these sigma x I think it has not done it correctly because it is starting from 129 it should have started from 103.

Let me just do it again because I think there is a problem because it is displaying the so now let me select some other along some other line. So, here it is showing the sigma x variation along the length and somehow I am not able to select the points because our inner radius is 100 and outer radius is 200 just able to select the outer radius points, but not the inner radius points I think because of the accuracy of this system I think but it is able to do it on my laptop.

So, if you have a problem whether you have a plane strain model or axisymmetric model we see that we get the same result. In fact, you can compare the displacements and then the stresses and strains from both the plane strain and axisymmetric models and you will see that there is no difference. So, that means that whatever we have done is accurate and then we can compare this with theoretical results in most of the elasticity text books we have the solutions for thick cylinder subjected to internal pressure and that I am not doing here.

But I will give it as a hand out. You can refer to the handouts that are uploaded on the website. So, this is some introduction how you can use the finite element program GEOGEM for doing our class problems of the patch test and then the other problems like the cantilever beam and then the thick cylinder. I will give you more instructions and how you can use this programs and then do these analysis so that you can do your weekly home works. **(Video Ends Here: 34:47)**