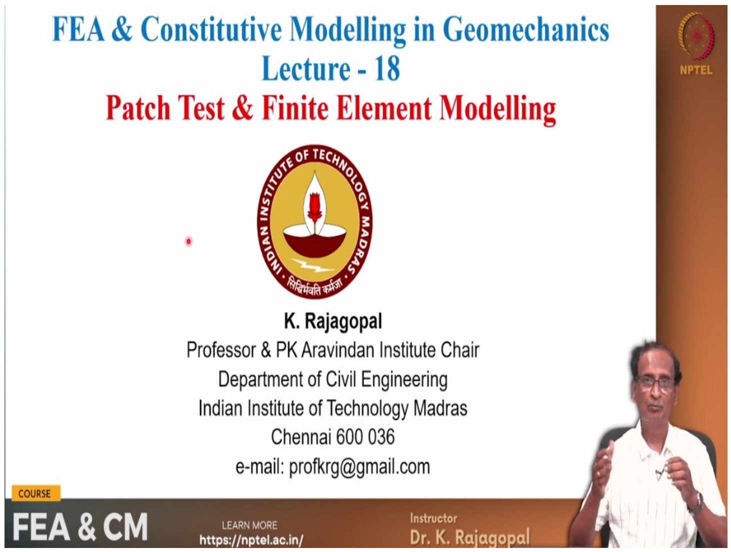
FEM and Constitutive Modeling in Geomechanics
Prof. K. Rajagopal
Department of Civil Engineering
Indian Institute of Technology – Madras

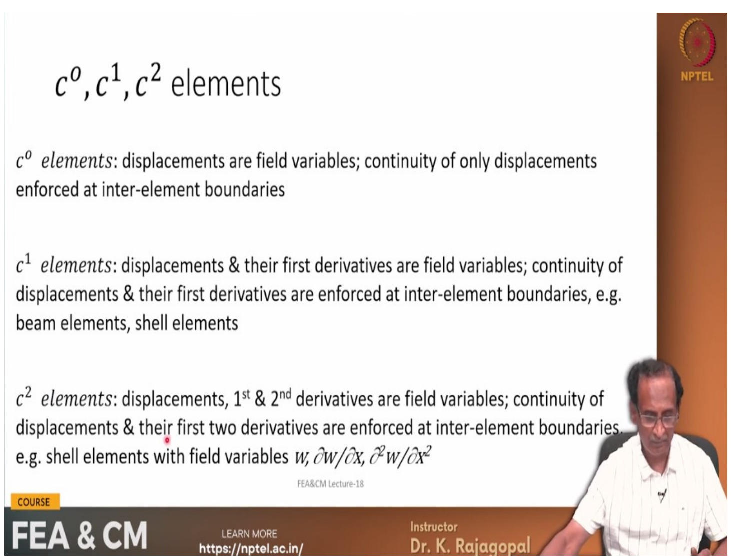
Lecture – 20
Patch test & Finite Element Modelling

(Refer Slide Time: 00:20)



So hello students let us continue our finite element calculations. In the previous lectures we have seen how to evaluate the stiffness matrix and then the load vector due to the body weight the stresses, the surface pressures using the different types of parametric concepts and now let us continue and see one very important aspect that is the patch test and then how we can use the finite element programs for doing the modeling

(Refer Slide Time: 01:01)

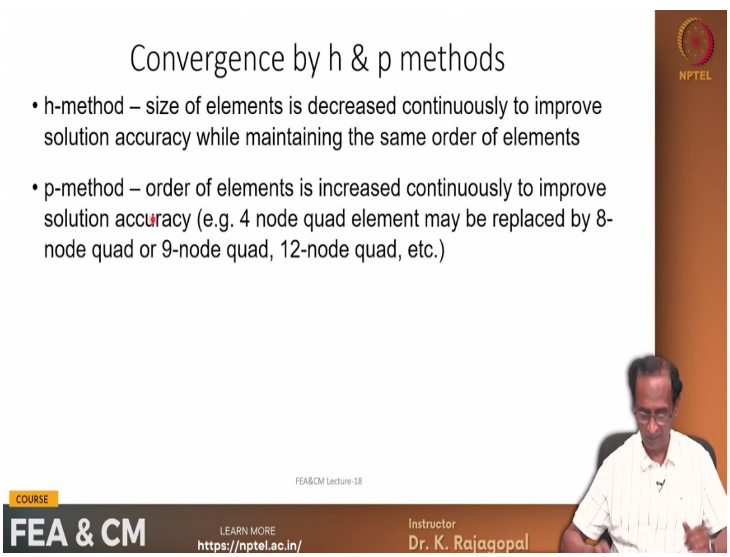


So, till now we have seen two types of elements c 0 and c 1. The c 0 elements these are the elements where only the displacements of the field variables and only the continuity of displacements is ensured at the interface between the elements so these are called as a c 0 because the first order derivatives may or may not be continuous whereas the c 1 elements we have seen the c 1 elements and the context of beam elements where our degrees of freedom are displacements and then the rotation like dou w by dou x.

$$c^o, c^1, c^2 \text{ elements}$$

And both the displacement and their first derivatives of the field variables and the continuity of beam elements and their first derivatives are enforced at the inter elements boundaries these are called c 1 and then c 2 elements are where we have displacements first derivatives and then the second derivatives the rotation and then the curvature as degrees of freedom and these are with thin shell elements, but we do not come across them in the geotechnical engineering. We will be only seeing the c 0 and the c 1 type element.

(Refer Slide Time: 02:28)

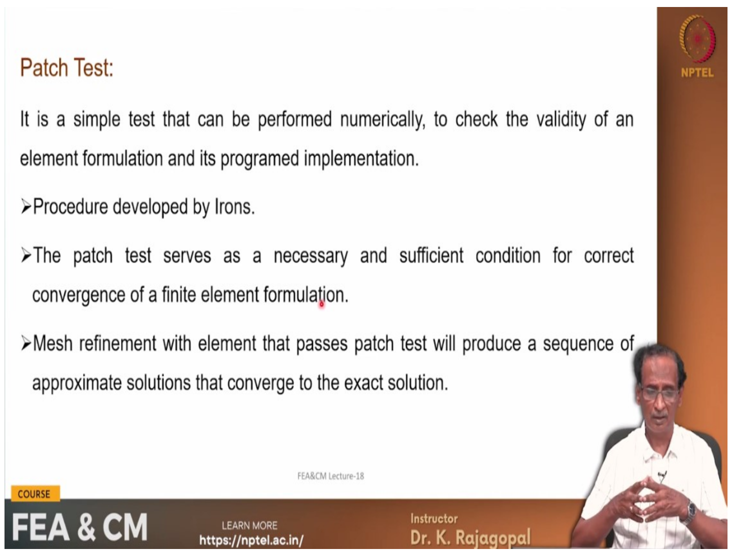


Then how do we get convergence. See there are two methods one is by h method and the other is by p method. H method is the size of the elements is decreased continuously to improve the accuracy. So you start with let us say 100 elements in the mesh next we increase it to 200 and then increase to 300, 400 and so on so that the size of the element goes on decreasing or in some cases what we do is we identify some critical point and then around that we can refine the mesh make it more finer.

And in this h method we are not going to change the type of element like if you have a 4 node quadrilateral we continue to use the 4 node quadrilateral everywhere and there is another method called as the p method where we try to get higher more accurate results by increasing the order of elements. So, you might start with a 3 node triangle, you might increase it to 6 node triangle or 10 node triangle, 15 node, 21 node and so on or with respect to quadrilaterals like you may have 4 node quadrilateral 8 node, 9 node, 12 node, 16 node and so on.

And this is the p method where the size of the element may remain the same like let us say initially you have 100 elements it will continue to have 100 elements, but we may have more number of nodes because we are increasing the order of the elements.

(Refer Slide Time: 04:13)



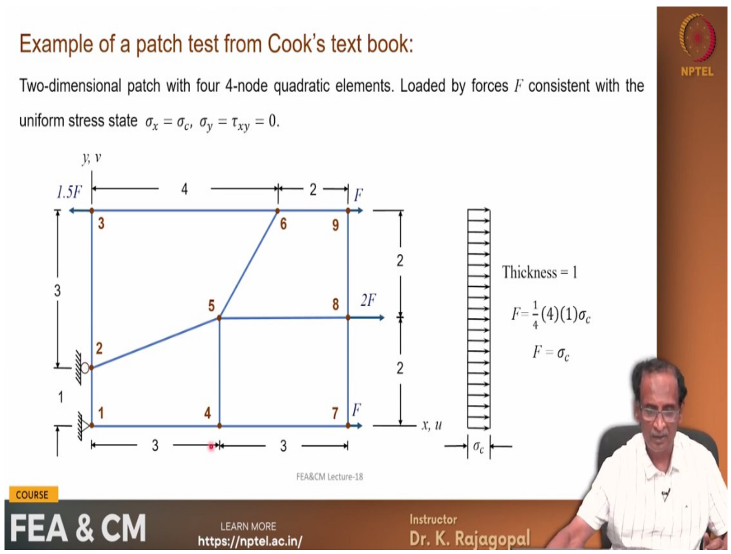
And there is one test that is called as a patch test that is performed numerically. So, we have seen the same test earlier in the context of our generalized coordinate method. So, we have seen that if you include the constant term we can simulate the rigid body deformations without developing any strain then if you subject an element to constant strain state or constant stress state we can predict the same thing within the element.

And the same thing we are going to demonstrate numerically and this particular one is done through the finite element program and this is to check the validity of an element formulation. See the element may be correct, but we might have made some mistakes in the program and by doing this we can verify whether the program is also correct and this procedure was first developed by Irons.

In fact Irons was the one who gave rise to lot of finite elements like the isoparametric elements and so on and the patch test also serves as a necessary and sufficient condition for correct convergence of finite element formulation. So, that if you are able to pass the patch test that means that our elements can satisfy the monotonic convergence requirements and sometimes a patch test mesh may not pass the patch test.

But then as you make the mesh more and more finer you might pass the patch test that is also acceptable.

(Refer Slide Time: 06:15)



And this particular example I have taken from the Cook's text book. So, when we take a patch of element it should be arbitrary patch of element and then we subject it to some loading and some boundary condition so that they represent the constant pressure state and here I have taken a patch that is 6 units long and 4 units height and then on the right hand side there is a pressure sigma c applied on the left hand side also there is a same pressure applied.

And then this node is fixed and hinge that both x and y direction displacement are not allowed and node 2 is on a roller that is the mesh can move vertically, but not horizontally then all these elements they are 4 node quadrilaterals and distorted only one element is a rectangular element and all the others are distorted and then we have the same pressure acting on this surface also.

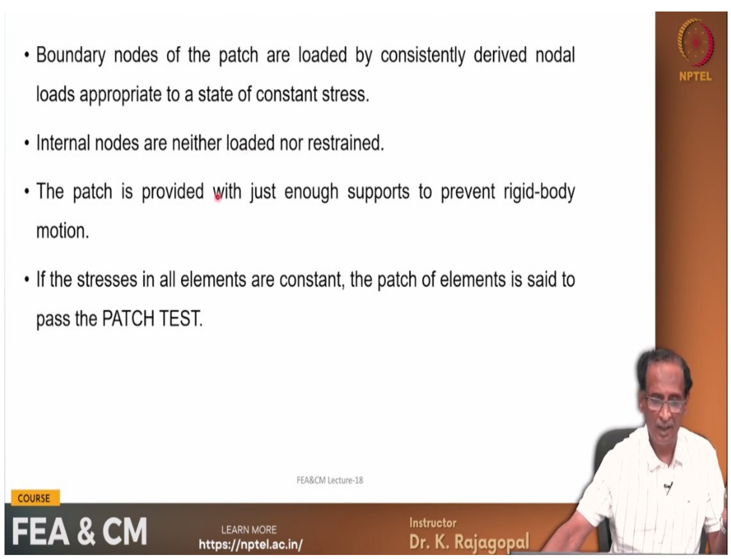
And we know that on any 2 node surface the load is distributed equally between the 2 nodes. So, here if our sigma c is the pressure applied the force at each node is sigma c multiplied by this height multiplied by unit thickness divided by 2 is your force at any node. So, the sigma c times 2 divided by 2 is your F and then same thing if you do for this element that is you get an F then at this node you have a F from this element.

And then you get another F from this element so you have 2 F. Similarly, if you apply on this surface you get 1.5 F and 1.5 F here. So, 1.5 F is applied in the negative direction and here this is put on a roller and then there is some Poisson's ratio and then some Young Modulus given then we do this analysis and if we calculate the same constant stress all through the mesh that means the elements are passing the patch test.

And here see the boundary conditions also should enable the mesh to pass the patch test. So, here by placing this node on a roller we are allowing for Poisson's ratio induced deformations. So, as you are elongating the mesh in one direction the other direction the mesh should contract because of the Poisson's ratio. So, if you had place this node and a hinge.

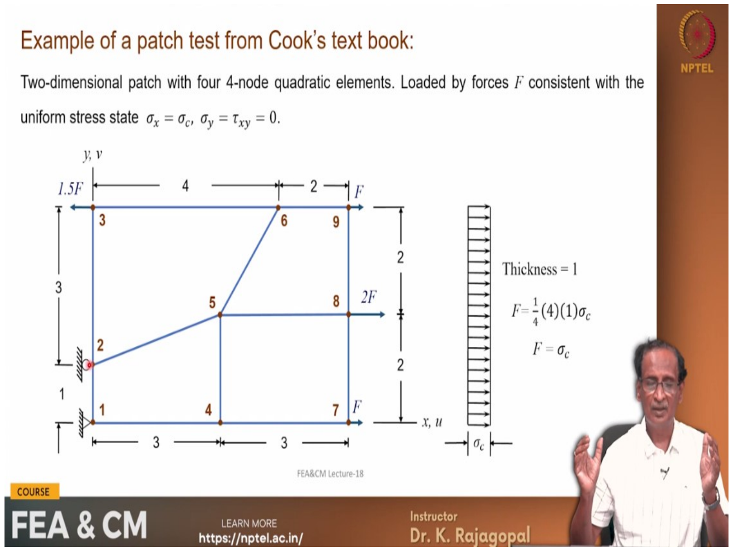
Then obviously this patch will not pass the patch test because it tries to contact in the other direction, but then you are now allowing. So, it will develop some extraneous stresses. So, if you apply some stress the stress that is predicted here inside the element should be the same that is what we are going to ensure from finite element analysis.

(Refer Slide Time: 10:32)



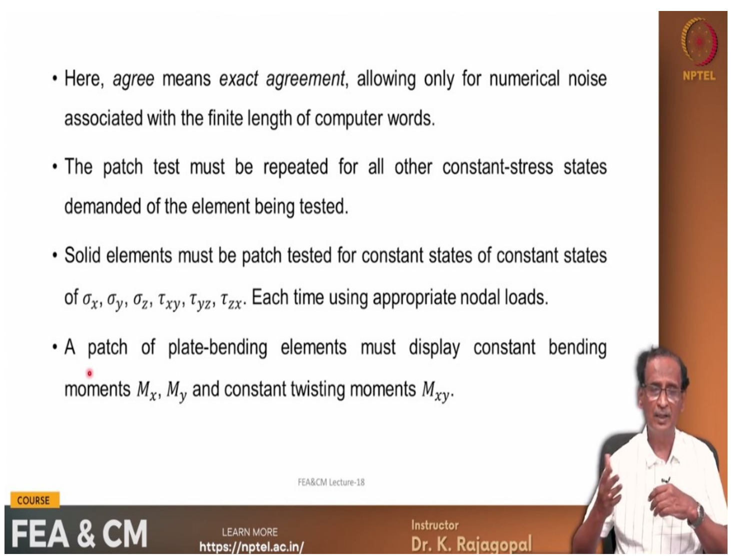
And in this test we are going to load the boundary nodes consistently derived from our N transpose t d s calculations and then the internal nodes are neither loaded or restrained and patch needs to be provided with a just enough boundary support to prevent any rigid body deformation and so that is what we have done here.

(Refer Slide Time: 11:02)



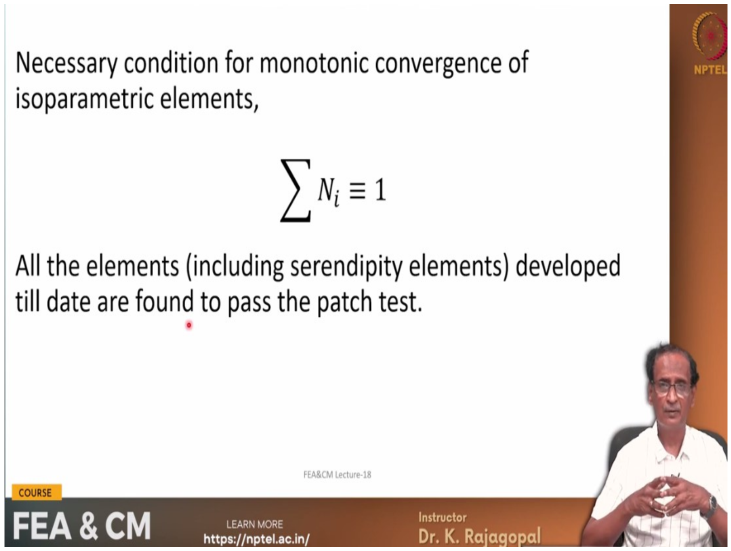
So, if you fix only one node and leave all the other nodes then it can undergo the rotation without any limit. So, we need to constraint one more node and that we have done by constraining this and you prevent the rigid body deformation and then we should be able to predict this constant stress that we will see through a program.

(Refer Slide Time: 11:35)



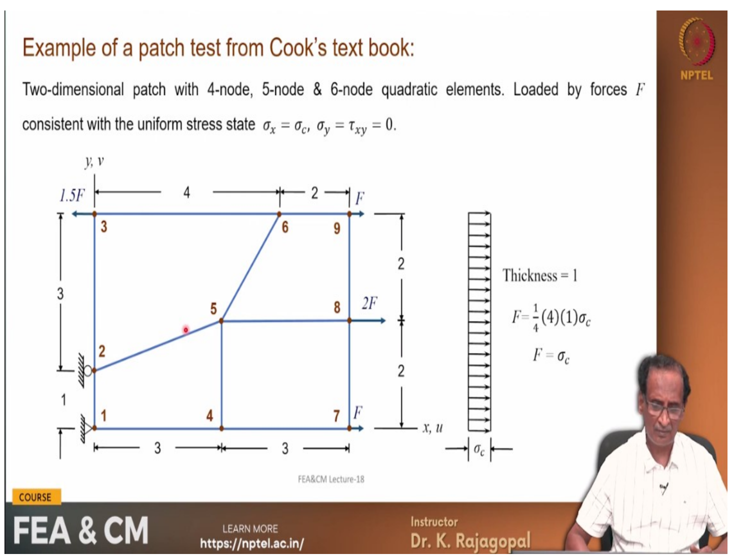
And so sometimes the elements may not pass the patch test, but as you make the mesh more and more finer you might be able to pass the patch test and that we call as the weak patch test and the constant stress should be there in all the quantities sigma xx, sigma yy, sigma zz, tau xy and so on then if you have bending element like a beam element even the moment should be constant all along the length.

(Refer Slide Time: 12:12)



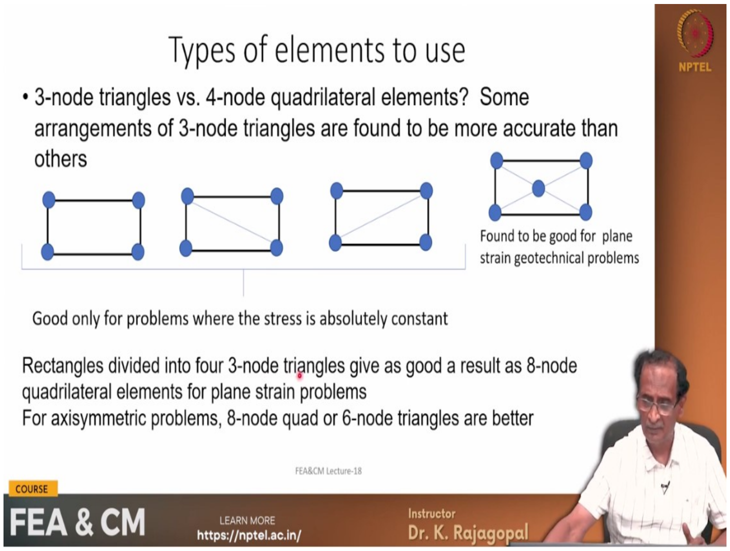
And we have already seen that sigma of N i should be equal to 1 for monotonic convergence and usually all the elements that we have they do pass this patch test because all the elements isoparametric elements that we have derived they have this condition of sigma of N i is 1.

(Refer Slide Time: 12:41)



And we can actually do a combination of different type of elements like 4 node, 5 node, 6 node and so on and that I will demonstrate when we go into the finite element program.

(Refer Slide Time: 12:58)

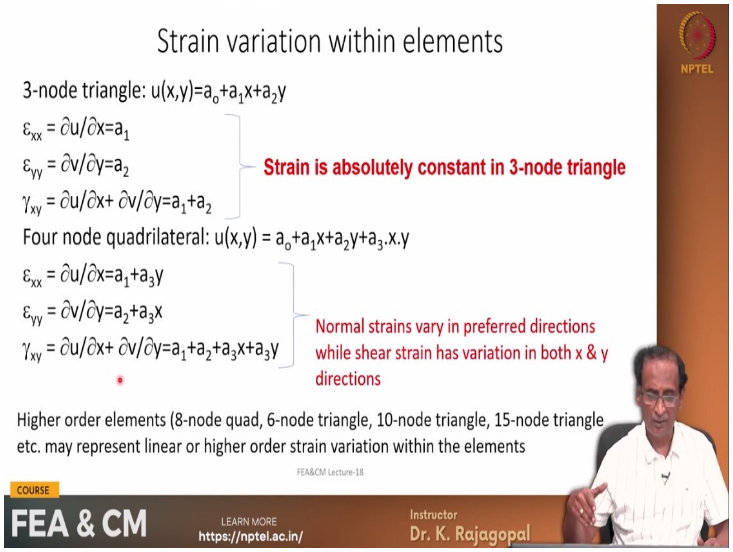


And there are different types of finite elements. So, the 4 node quadrilateral is what we have seen and the 3 node triangle also we have seen in our calculations and we can divide this 4 node quadrilateral into two triangles like this or like this and these elements are too simplistic and they are good only for problems where the stress is absolutely constant all through the mesh. For those cases we may be able to get reasonable results.

But otherwise we should not use them or we should refine the mesh so much that we can approximate the stress field that we get and within geotechnical engineering the experience is that if we divide any rectangle into 4 triangles like this we can get good results for plane strain problem for a strip footing or an embankment or something not for axisymmetric cases and the rectangles divided into 3 node triangles give a very good result comparable to an 8 node quadrilateral element for plane strain.

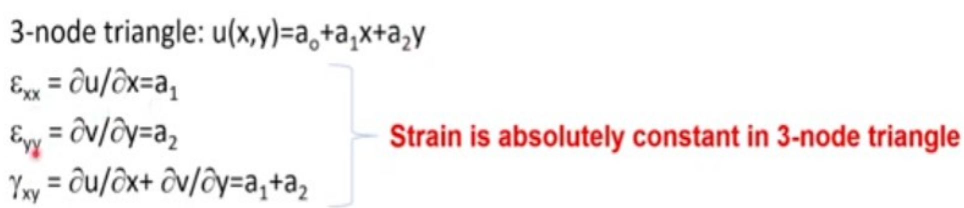
But for axisymmetric problems we need to go in for 8 node quadrilaterals or 6 node quadrilateral or 6 node triangles or even higher like 10 node triangles or 15 node triangles and so on.

(Refer Slide Time: 14:44)



And how we decide whether given element is sufficient for a problem or not we can guess it by looking at the variation of strain within the element and see that can represent reality. So, let us do that for 3 node triangle and 4 node quadrilateral. So, in a 3 node triangle our displacement field can be written as a 0 + a 1 x + a 2 y and our strains epsilon xx epsilon yy and gamma x y we see that they are all absolute constants.

So, the 3 node triangle per se is a constant strain triangle and so when you apply it we should be careful because if you use a very large element then within that large area you are assuming that the strain and stress are going to remain constant. So, that means you are unnecessarily constraining a lot on the system and your result may not be accurate and when we go into the 4 node quadrilateral our displacement field is a 0 + a 1 x + a 2 y + a 3 x y.



And our epsilon x is a 0 + a 3 y and epsilon y is a 2 + a 3 x and gamma x y is a 1 + a 2 + a 3 x + a 3 y. So, here we see epsilon x is a constant along x direction, but varying along y direction and epsilon y is constant along y direction, but varying along the x direction and gamma x y can vary both in x and y directions. So, when you apply this we should be careful because the 4 node quadrilateral is also directionally constraining the variation of the strain.

Four node quadrilateral: $u(x,y) = a_o+a_1x+a_2y+a_3.x.y$

$\varepsilon_{xx} = \partial u/\partial x = a_1+a_3y$

$\varepsilon_{yy} = \partial v/\partial y = a_2+a_3x$

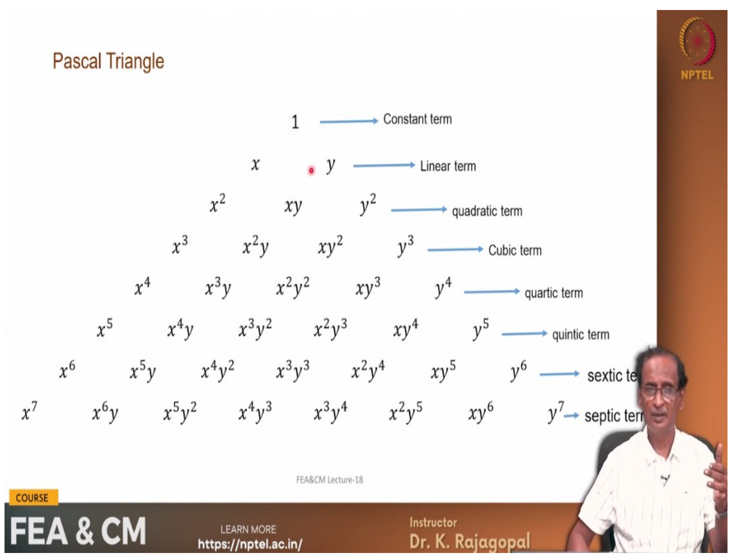
$\gamma_{xy} = \partial u/\partial x + \partial v/\partial y = a_1+a_2+a_3x+a_3y$

Normal strains vary in preferred directions while shear strain has variation in both x & y directions

And let us say if you use a 3 node triangle for analysis for a cantilever beam we will have a problem because a 3 node triangle assumes that the stress is constant and the strain is constant. So, that means that within cantilever we know that on the upper fibre we will have tensile strain whereas bottom fibre we will have compression strain, but then if you represent with a 3 node triangle it will either give you compression and tension so that is not correct.

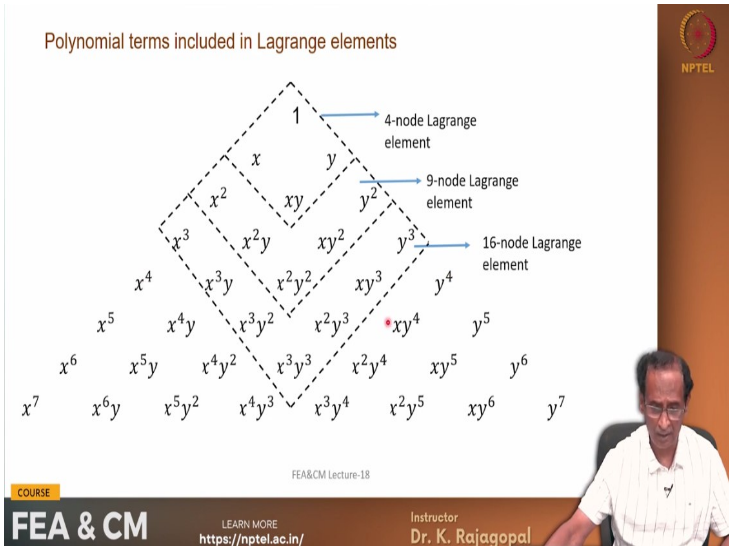
Then if you apply a 4 node quadrilateral the epsilon x that is the flexural strain is a function of y. So, it has a capacity to represent tensile strain at the top and compression strain at the bottom. So, that is why the 4 node quadrilateral maybe better compared to a 3 node triangle and then even more versatile than these are higher order elements like 8 node quadrilaterals or 6 node triangles and so on and by using them we can get better result.

(Refer Slide Time: 18:17)



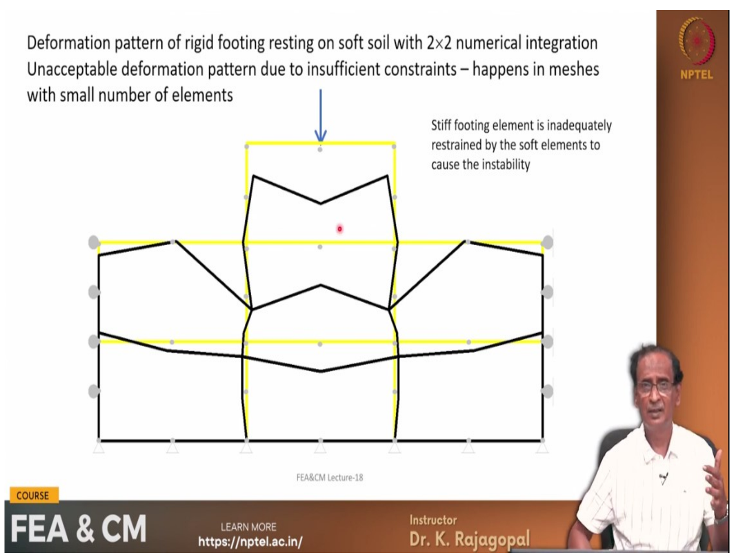
And when we choose the number of terms we have to carefully examine the pascal triangle in choosing the terms and this particular one is given in terms of Cartesian coordinates x and y, but you can write the same thing in terms of natural coordinates Xi and eta.

(Refer Slide Time: 18:40)



And all our Lagrange elements they include only the terms within this highlighted areas and the Lagrange element they include some high order term, but while neglecting some lower order terms like, for example, the 16 node Lagrange element it will have this x cube y cube term, but then it will not have this x to the power 4 y to the power 4 and so on all these lower order terms.

(Refer Slide Time: 19:17)

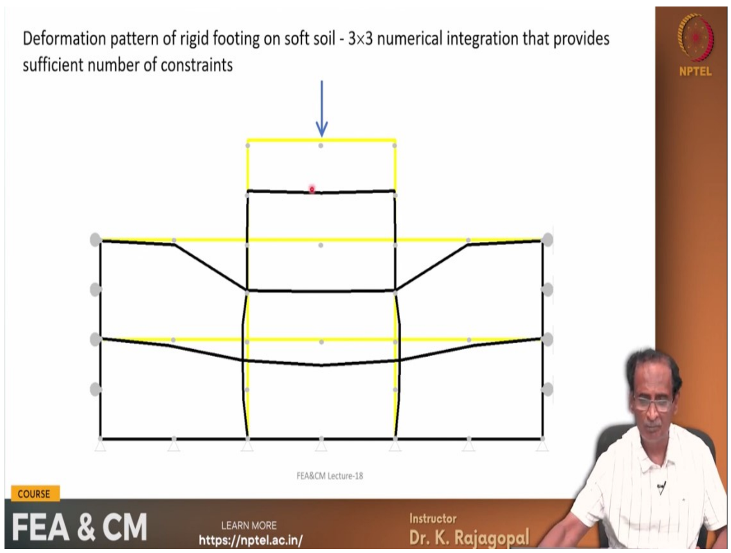


And so sometimes we may get some spurious results and that you might think that is because of the programming errors, but that is not true in some situations depending on the constraints that we have, we may get some result like this. So, here this yellow mesh is showing the rigid footing resting on a soft clay and it is subjected to some concentrated load and we know that it is a rigid footing resting in a soft clay and if you are subjected to some loading it should just simply sink into the clay.

But then the response that we get is very funny. So, this particular one it is a rigid element with very high modulus, but then it has buckled something like this, it has twisted out of shape and whereas even these elements they have distorted a little bit different and the main concern is see our rigid element with very high stiffness it is just buckled out of shape or it has got twisted out of shape that is because of insufficient restraint given by the soft clay.

And this particular result is obtained with by 2 by 2 integration numerical integration for all the elements this stiffness is evaluated using 2 by 2 numerical integration.

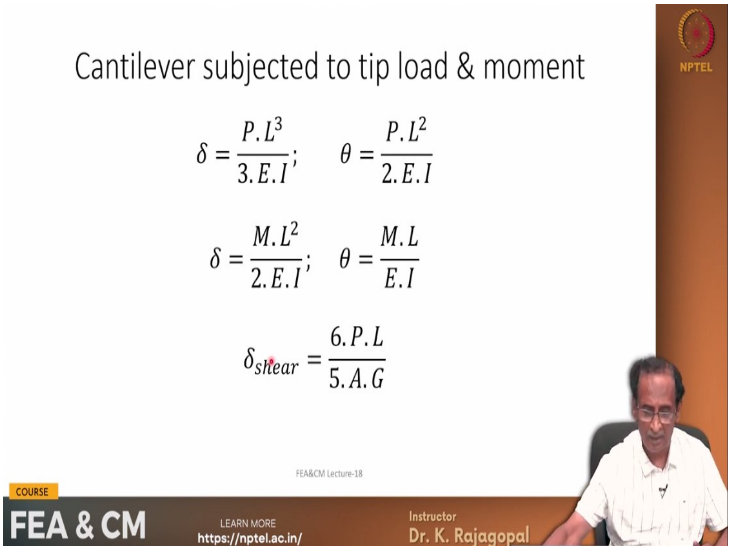
(Refer Slide Time: 21:01)



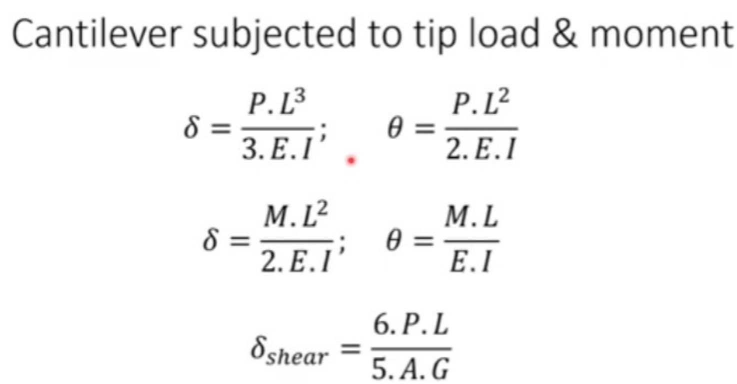
And the same thing performed with 3 by 3 integration is giving the result that we expect. So, if you have a rigid element it should just simply move like a rigid body, it should undergo uniform settlement all through whereas here it is not done. It has undergone some settlement outside, but then inside it has not undergone equal settlement and you might think maybe this mode of deformation is excited because of this concentrated load.

Actually in the plane strain there is nothing like a concentrated load, it is actually it is a line load that is because the concentrated load can happen in a full 3 dimensional problem, but not in the 2 D problems. So, even if you apply pressure loading like this we get the same spurious response because of insufficient restraint provided by the soft element.

(Refer Slide Time: 22:17)

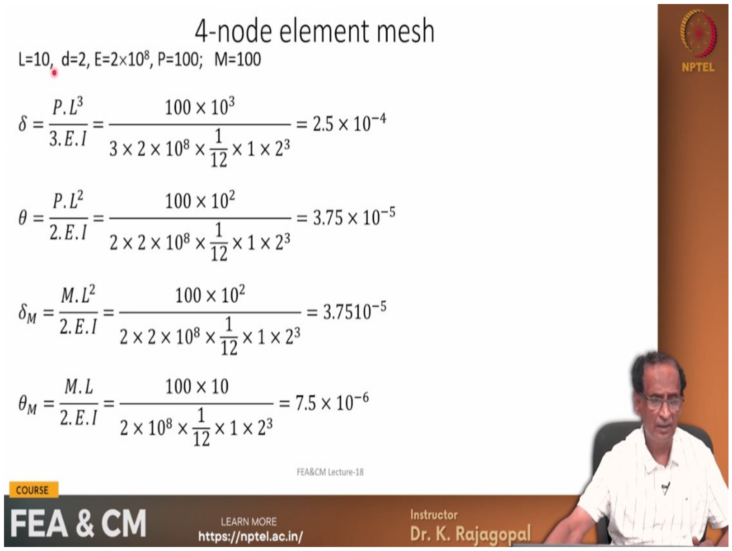


And I am going to show one more example at the cantilever beam subjected to tip load and moment and because this is problem that we understand very well and if you have a pure flexural member without any shear deformation these are the deformation and rotation delta is P L by 3 E I theta is P L square by 2 E I and delta is M L square by 2 E I and theta is M L by E I. These are the results that we should expect with pure flexural elements.



And but then if you use a plane stress element for representing a cantilever beam that being a continuum it might undergo some shear deformation and the shear deformation at the tip is 6 P L by 5 A G where G is the shear modulus A is the cross sectional area. P is applied load and L is the length of the beam.

(Refer Slide Time: 23:33)



And these are for different properties let us say the length is 10 and the depth is 2 and Young Modulus is so much the load at the tip is 100 at the moment is 100 and the deformation at the tip is 2.5 times 10 to the power of – 4 and the rotation of the tip is this and the deformation due to the moment is this and rotation due to applied moment is this and we can perform these the finite element calculations and see what happens.

4-node element mesh

L=10, d=2, E=2×10⁸, P=100; M=100

$$\delta = \frac{P.L^3}{3.E.I} = \frac{100 \times 10^3}{3 \times 2 \times 10^8 \times \frac{1}{12} \times 1 \times 2^3} = 2.5 \times 10^{-4}$$

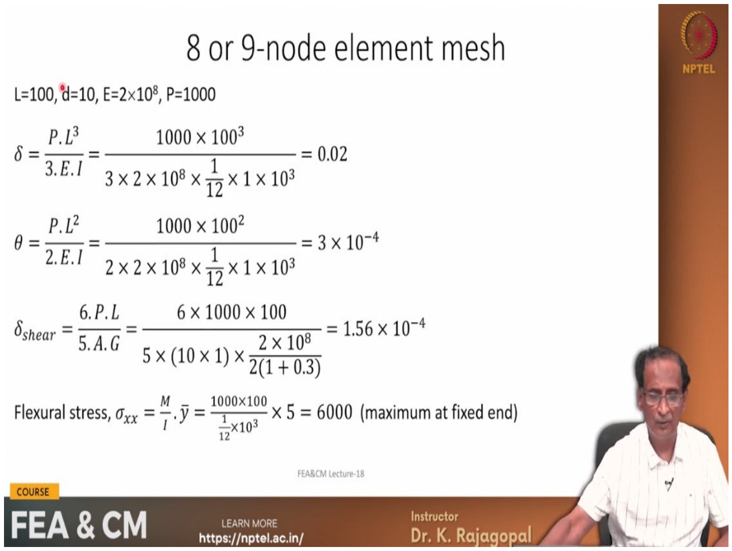
$$\theta = \frac{P.L^2}{2.E.I} = \frac{100 \times 10^2}{2 \times 2 \times 10^8 \times \frac{1}{12} \times 1 \times 2^3} = 3.75 \times 10^{-5}$$

$$\delta_M = \frac{M.L^2}{2.E.I} = \frac{100 \times 10^2}{2 \times 2 \times 10^8 \times \frac{1}{12} \times 1 \times 2^3} = 3.7510^{-5}$$

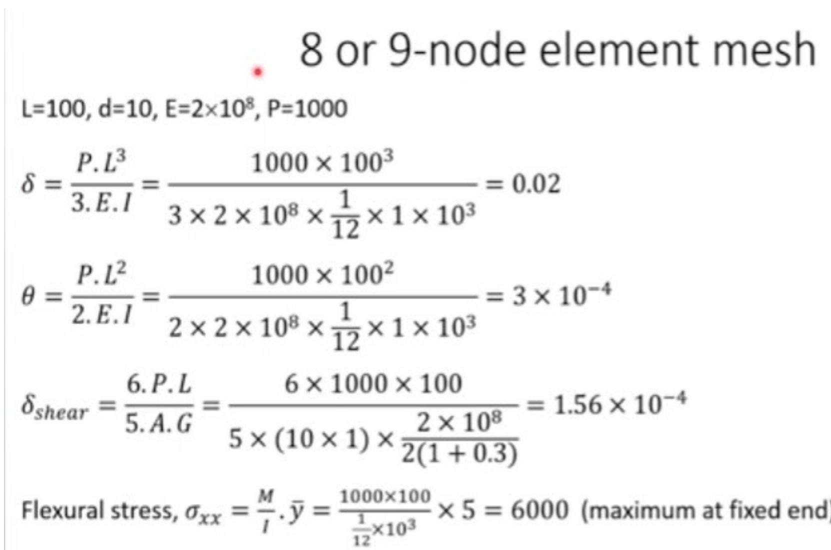
$$\theta_M = \frac{M.L}{2.E.I} = \frac{100 \times 10}{2 \times 10^8 \times \frac{1}{12} \times 1 \times 2^3} = 7.5 \times 10^{-6}$$

And actually there are two different geometries considered for a 4 node element mesh these are the geometric parameters, length is 10 and d is 2.

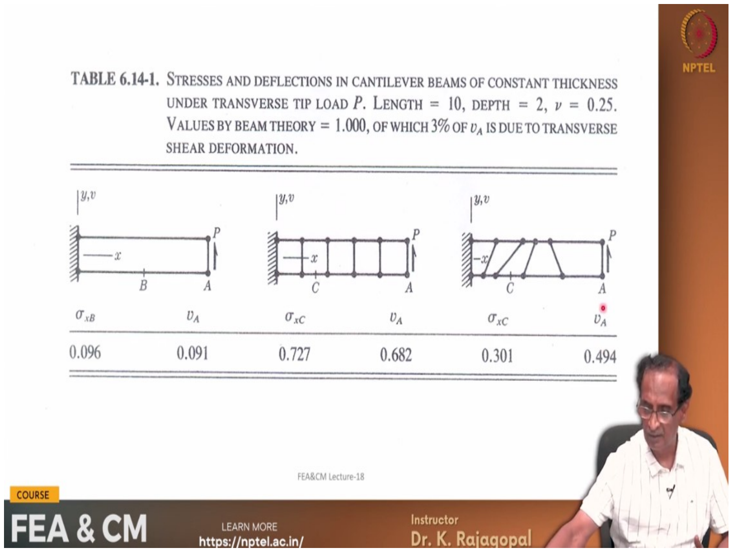
(Refer Slide Time: 24:33)



And for 8 and 9 node element the length is 100 and the depth is 10.



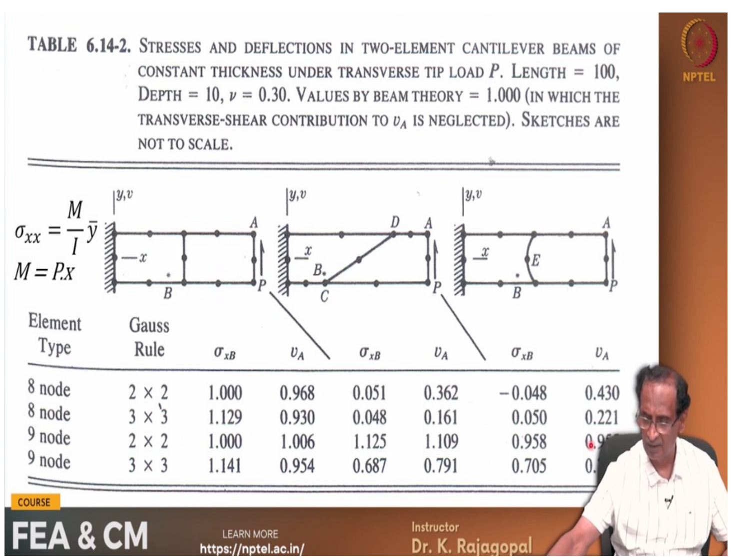
(Refer Slide Time: 24:40)



And Cook's textbook he has shown the result qualitatively what you get with the different type of elements and this is for a cantilever beam model by using the one single 4 node element and the deflection at the tip is only 9 percent of the actual deflection and then the stress at this point at the bottom fibre at the bottom is only about 10 percent of the actual flexural stress.

But then if you have more number of elements like this your deflection is slightly better it is 68 percent of the actual result and the stress at this point is about 73 percent of the actual stress, but then if you have the same 5 elements, but distort them like this your solution accuracy comes down, the tip deflection is only 0.49 times the actual deflection and then sigma c at this point is only 30 percent of the actual stress.

(Refer Slide Time: 26:12)



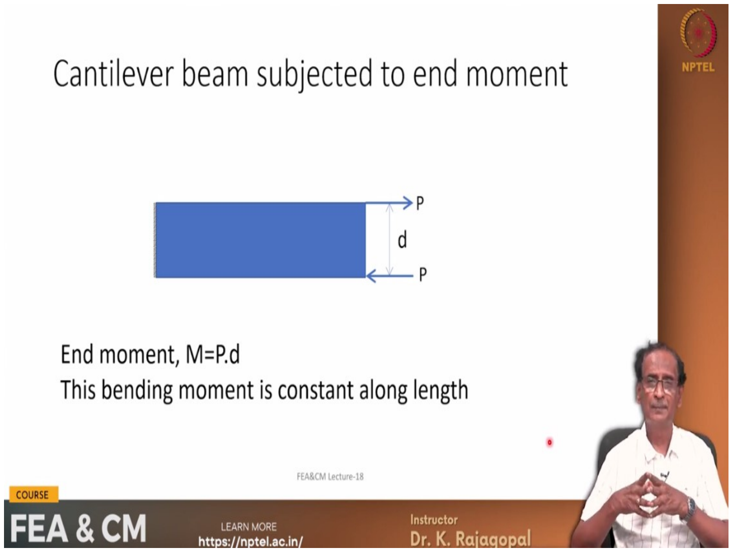
And if we use 8 and 9 node quadrilateral and three different cases are given. One with pure rectangular elements two of them and one with a distorted, but then this surface is a straight surface then the third one is distorted, but it is a curved surface and for this case with a 2 by 2 integration if we use 8 node element the sigma at this place is exact 1 times the actual stress so that is 100 percent exact.

And the tip displacement is not bad 0.968 times the actual displacement, but then if we use 3 by 3 integration the calculated stress is much more than what it is like, it is 1.129 and then the deflection is slightly lesser 0.93 times the actual displacement and if we use 9 node element with reduced integration you get very good result both in terms of the stresses and then the deflections.

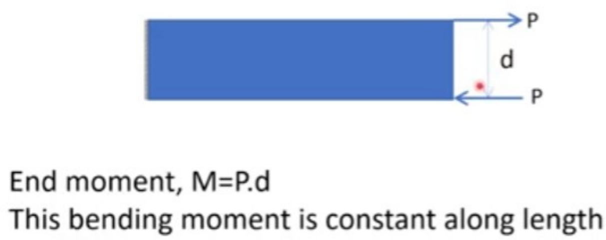
And 9 node element with a 3 by 3 integration it is not very good and when you distort it your results suffer a lot. See with 2 by 2 integration with 8 node element your vertical displacement is 0.36, but then if you use a 3 by 3 integration your solution is only 0.16 times. So, that is actually that is so you see as you are increasing the order of integration instead of improving the solution you are actually impairing the solution.

It is the accuracy is coming down consistently in both 9 node and 8 node elements then this is a curved element with a curve and once again this 8 node element it is not able to perform well whereas the 9 node element with a 2 by 2 integration it is able to reasonably predict and the deflection is 95 percent of the actual deflection and the stress is about the same 95 percent of the actual deflection.

(Refer Slide Time: 28:50)



So, this I will show through a computer program and so our continuum elements we have only facility to apply at the forces either in the x direction or y direction, but then how do we apply a couple or moment. So, what we do is we can actually apply the force like a couple. So, if you want to apply a moment of M here we can equate M to P d if d is the depth we can calculate the load that you need to apply P and apply in the positive x direction at the top.

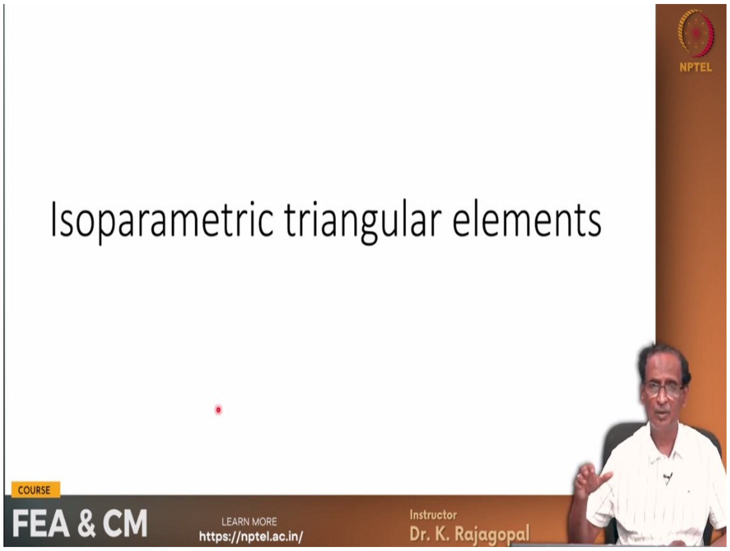


And the negative x direction at the bottom to get our moment. So, similarly if you want to get flexural stress so when you have a beam the flexural stress we get directly, but then sigma x is actually M by I times y bar. M is the moment that is the shear force multiplied by the distance and at the fixed end your moment is P times L that is maximum at the fixed end and 0 at this free end.

$$\sigma_{xx} = \frac{M}{I}\bar{y}$$
$$M = P.x$$

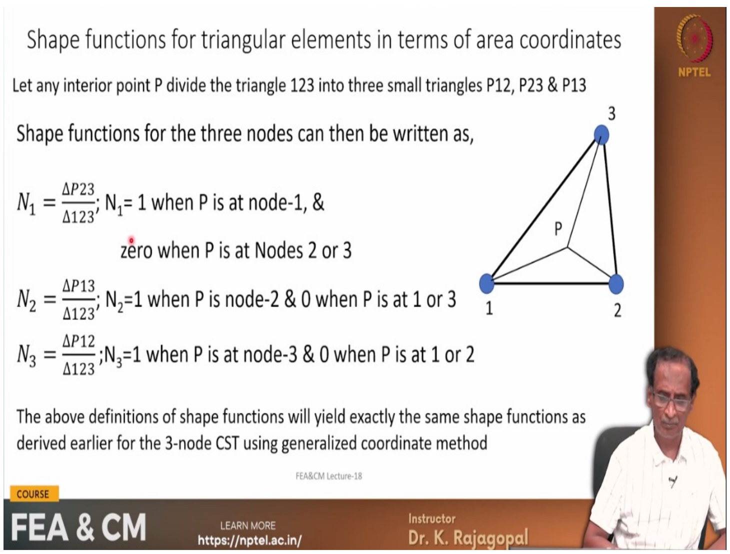
And at any place you can calculate the moment M by I and compare this flexural stress against this sigma xx and that is what is reported here the percentage error or how it compares with the actual result.

(Refer Slide Time: 30:51)

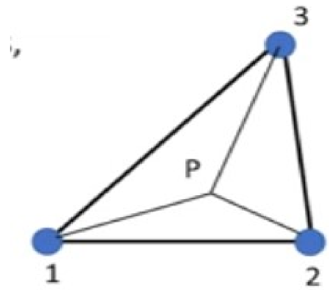


See till now were only discussing about the quadrilateral elements and I will want just briefly show you about the triangular finite elements.

(Refer Slide Time: 31:06)



And the triangles are also very popular. As you know the triangular elements they have the complete polynomial starting from 3 node triangles, 6 node and so on and so their stability is better. Some of the modes of deformation that you see with quadrilateral may not happen with the triangular elements and there is another method for getting the shape functions that is in terms of the area coordinates.



And here we can define any node P and then get the area of any of these triangles like P 23, P 13, P 12 and so on and the N 1 can be written as area of this P 23 divided by the total area and so that when P is located at node 1 N 1 is 1 and when P is located at 2 it is 0 or when P is located at 3 it is 0 and then in between it varies from 0 to 1 and this definition of finding the shape functions in terms of the area of this triangle it will give us the same shape functions that we got earlier.

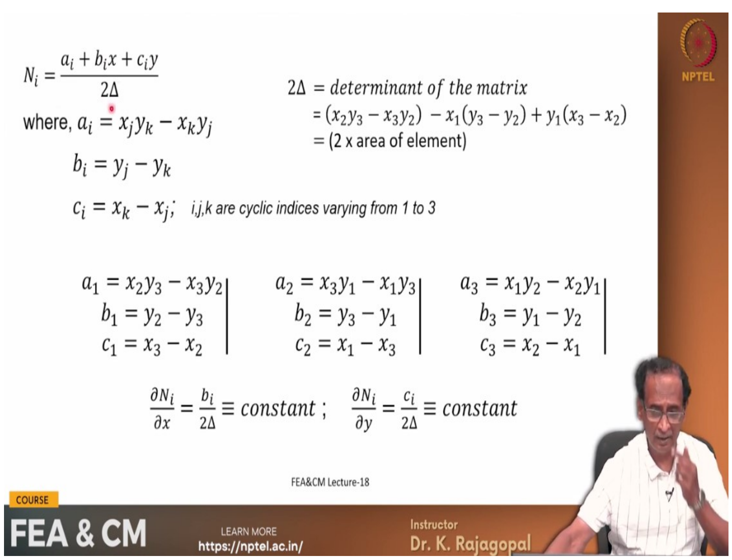
$N_1 = \frac{\Delta P23}{\Delta 123}$; N_1= 1 when P is at node-1, &

zero when P is at Nodes 2 or 3

$N_2 = \frac{\Delta P13}{\Delta 123}$; N_2=1 when P is node-2 & 0 when P is at 1 or 3

$N_3 = \frac{\Delta P12}{\Delta 123}$; N_3=1 when P is at node-3 & 0 when P is at 1 or 2

(Refer Slide Time: 32:40)



That N is a + b x + c y by 2 delta that is what we derived earlier by using the generalized coordinate methods we will get exactly the same things.

$$N_i = \frac{a_i + b_i x + c_i y}{2\Delta}$$

where, $a_i = x_j y_k - x_k y_j$

$b_i = y_j - y_k$

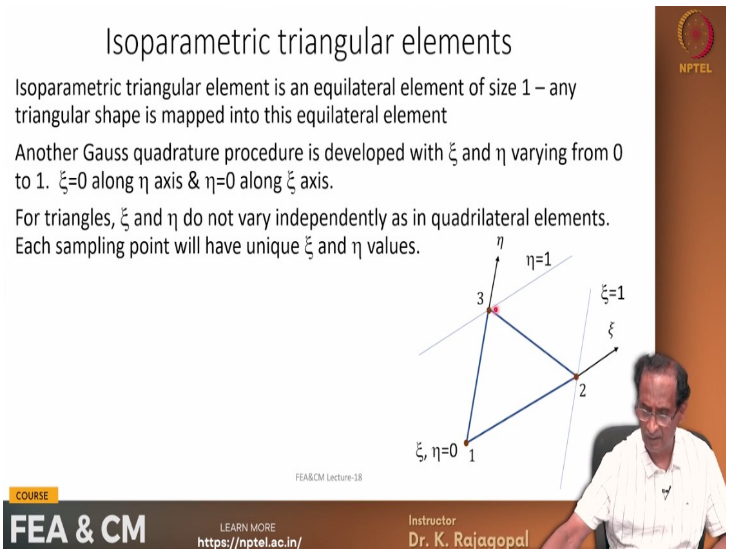
$c_i = x_k - x_j$; *i,j,k are cyclic indices varying from 1 to 3*

2Δ = *determinant of the matrix*
$= (x_2 y_3 - x_3 y_2) - x_1(y_3 - y_2) + y_1(x_3 - x_2)$
= (2 x area of element)

$$a_1 = x_2 y_3 - x_3 y_2 \quad a_2 = x_3 y_1 - x_1 y_3 \quad a_3 = x_1 y_2 - x_2 y_1$$
$$b_1 = y_2 - y_3 \quad b_2 = y_3 - y_1 \quad b_3 = y_1 - y_2$$
$$c_1 = x_3 - x_2 \quad c_2 = x_1 - x_3 \quad c_3 = x_2 - x_1$$

$$\frac{\partial N_i}{\partial x} = \frac{b_i}{2\Delta} \equiv constant \; ; \quad \frac{\partial N_i}{\partial y} = \frac{c_i}{2\Delta} \equiv constant$$

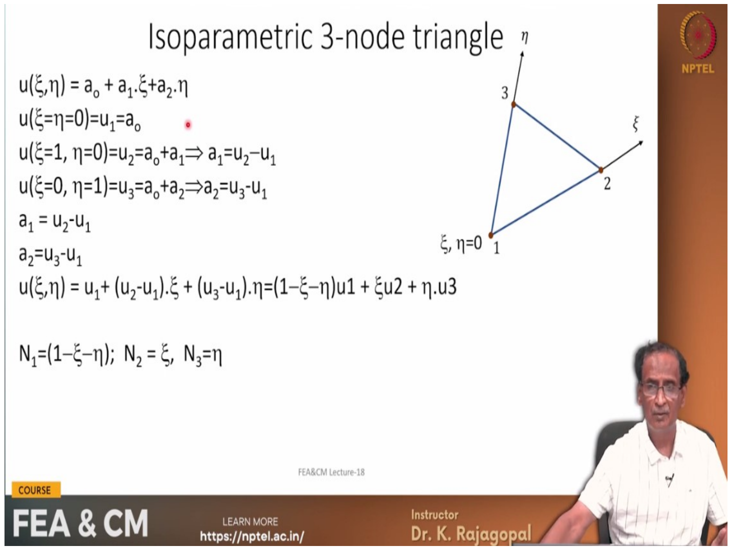
(Refer Slide Time: 32:52)



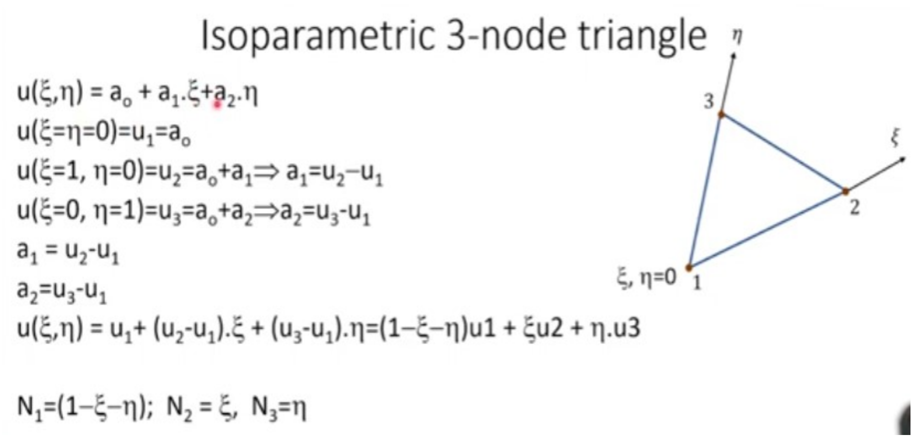
And it is possible to develop shape functions for triangles using our parametric procedures and the Xi and eta for the triangles they vary from 0 to 1 not from – 1 to + 1 and Xi and eta are not orthogonal in this case whereas in the case of quadrilaterals we have taken them as orthogonal, but here it is not so and so they do not vary independent of each other. So, each sampling point will have unique Xi and eta values.

And we can define say Xi along the surface 1, 2 and eta along this lone 1, 3 and let us say that Xi and eta are 0 at this node Xi is 1 here and eta is 1 there and Xi along this line is 0 and along this line is 4 it is parallel to this surface and then eta is 0 along this line and then 1 along this line and in between it is varying from 0 to 1.

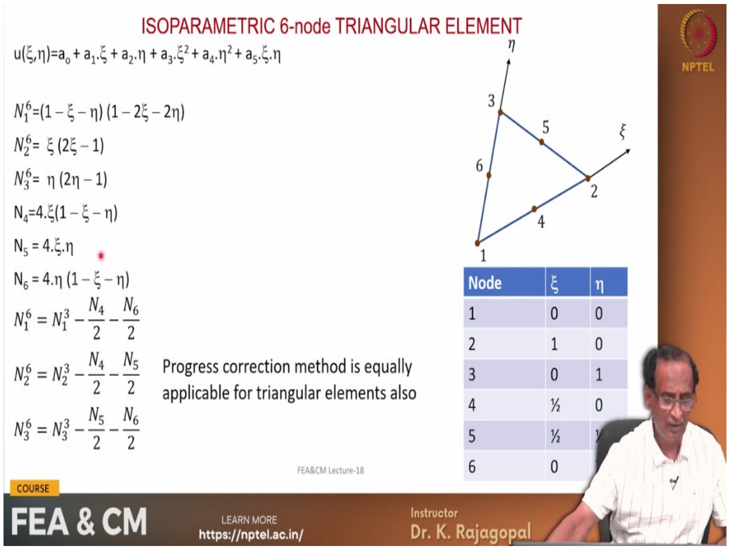
(Refer Slide Time: 34:12)



And we can get our shape functions in terms of the natural coordinates Xi and eta using our conventional generalized coordinate method. So, our u can be assumed as a 0 + a 1 Xi + a 2 eta and then we derive go through the procedure and derive we will see that N 1 is 1 – Xi – eta N 2 is Xi, N 3 is eta and our N 1 + N 2 + N 3 is 1 because it is Xi – 1 + Xi + eta.



(Refer Slide Time: 34:54)



And we can get our shape functions for higher order elements by going through our generalized coordinate method for the 6 node triangles the u is a 0 + a 1 Xi + a 2 eta + a 3 Xi square + a 4 eta square + a 5 Xi eta this we can easily get by looking at the pascal triangle and if we go through them and substitute different Xi eta values here and then determine the a 0, a 1, a 2 and so on we get the shape functions like.

$$u(\xi,\eta)=a_o + a_1.\xi + a_2.\eta + a_3.\xi^2 + a_4.\eta^2 + a_5.\xi.\eta$$

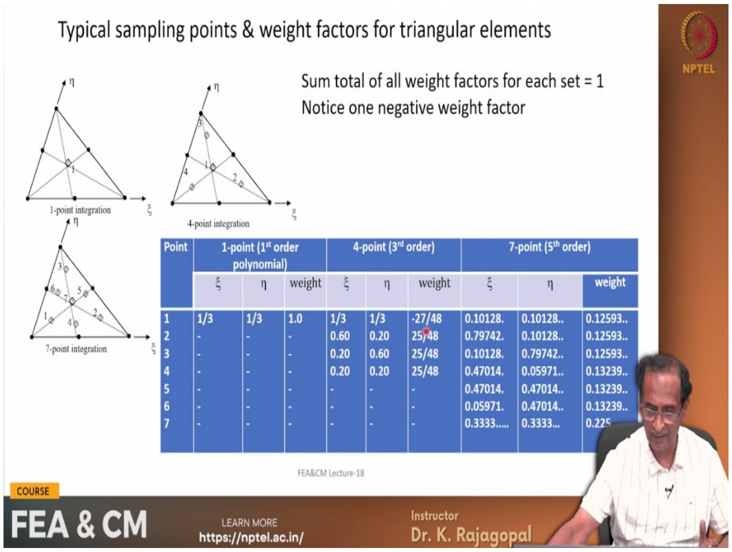
N 1, N 2, N 3, N 4, N 4, N 5, N 6 so whatever progressive corrections that we had done with quadrilateral elements can also be done with triangular elements. So, the N 1 for the 6 node element we can get from by correcting the N 1 of the 3 node element N 1 3 we can get it we can subtract N 4 by 2 and N 6 by 2 and get the shape function for 6 node element. So, our N 1, N 2, N 3 they are given here for the 3 node element.

$$N_1^6=(1-\xi-\eta)(1-2\xi-2\eta)$$
$$N_2^6=\xi(2\xi-1)$$
$$N_3^6=\eta(2\eta-1)$$
$$N_4=4.\xi(1-\xi-\eta)$$
$$N_5=4.\xi.\eta$$
$$N_6=4.\eta(1-\xi-\eta)$$
$$N_1^6=N_1^3-\frac{N_4}{2}-\frac{N_6}{2}$$
$$N_2^6=N_2^3-\frac{N_4}{2}-\frac{N_5}{2}$$
$$N_3^6=N_3^3-\frac{N_5}{2}-\frac{N_6}{2}$$

Progress correction method is equally applicable for triangular elements also

And for the 6 node element we can get this and in between we can have a 4 node triangle or 5 node triangle or a 6 node triangle it does not matter because whatever procedures that we discussed earlier are also valid for this particular element.

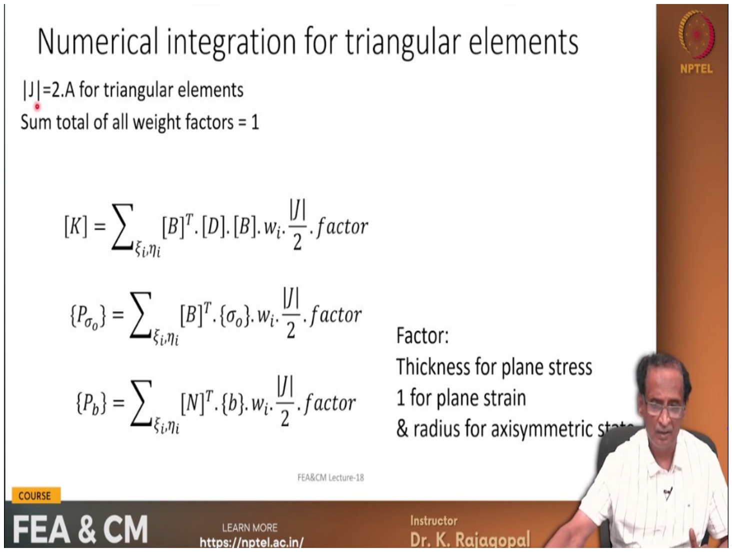
(Refer Slide Time: 36:40)



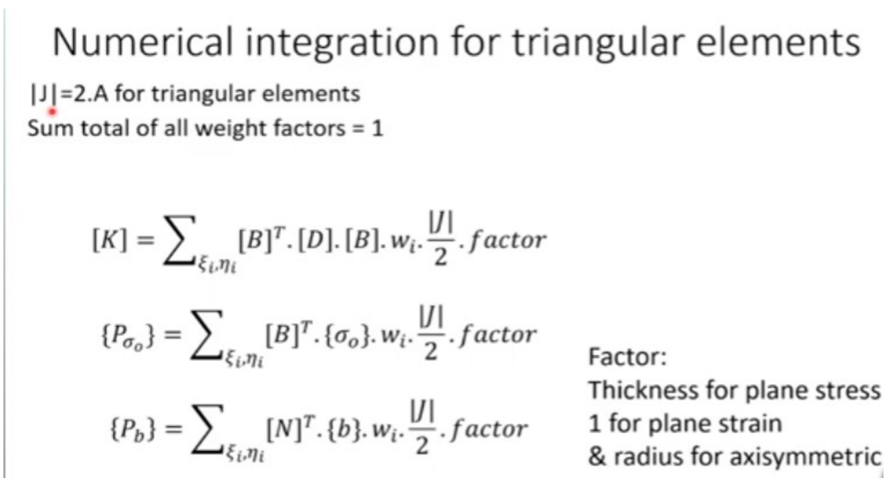
And these are the sampling points for the triangles and 1 point integration this is good for first order polynomial, the Xi and eta are one third, one third and the weight factor is 1 and we can have a 4 point integration that is good up to third order polynomial Xi and eta are here and the weight factors and you see one of the weight factors is negative for the triangles for the 4 point integration it is negative – 27 by 48 then 25 by 48 at all the other nodes then 7 point integration it is good up to 5th order polynomials.

These are the different locations and then the weight factors and the sum total of all the weight factors for the triangles is equal to 1.

(Refer Slide Time: 37:42)

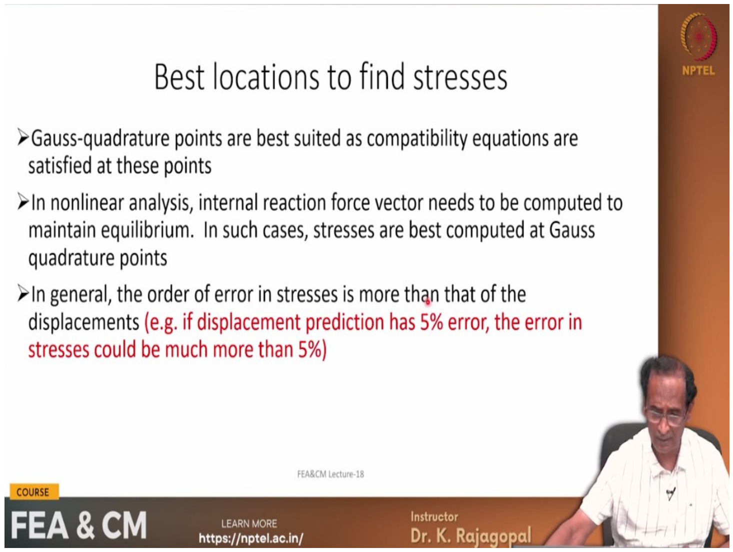


And our equations for triangular elements are written like this and the determinant of the triangular elements of the determinant of the Jacobian matrix for triangular elements is two times the area 2 a whereas for quadrilaterals the determinant is a by 4. So, our stiffness matrix is written like this B transpose D B w i because now we are not independently doing in Xi and eta directions.



We are at each and sampling points we can have a different Xi and eta values. So, for say 1 point integration Xi and eta are just one third, one third w i is 1 and so on and if you have more points you have this Xi and eta like this varying like this one third, one third, 0.6, 0.2, 0.2, 0.6, 0.2, 0.2 something like this I think it is illustrated here and the load vector due to initial stresses and the load vector because of the body weight they are all given here.

(Refer Slide Time: 39:09)

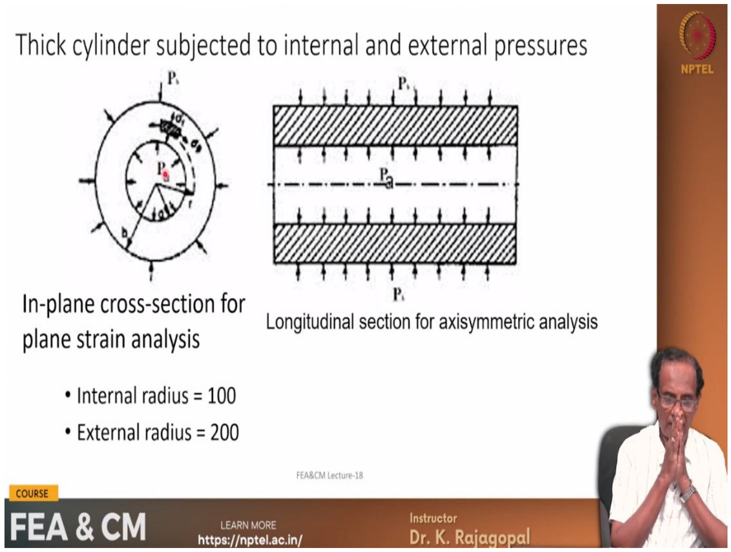


And what are the best locations to find the stresses. See till now we were looking at only the displacements and the displacements are obviously determined at the node points and so when it comes to the stresses we have to see exactly where we are enforcing the compatibility conditions. So, in the compatibility conditions are enforced only the sampling the sampling points the Gauss quadrature points.

And so this strains are evaluated at this Gauss quadrature points and then the stresses are also best evaluated at Gauss quadrature points or the Gauss sampling points when we determine the stresses at the nodal points they may or may not be correct and in the non linear analysis we always calculate the stresses only at the integration point so that we can apply B transpose sigma and equate the internal force to the external force.

And in general the order of error and the stresses is more than that of the displacements. So, if your displacement prediction has 5 percent error the error in the stress is need not be exactly equal to 5 percent it could be more or much more sometimes.

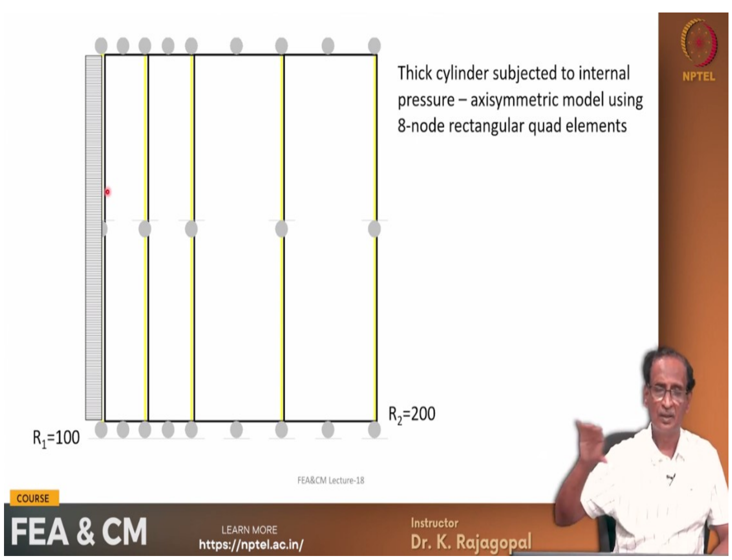
(Refer Slide Time: 40:38)



Now, let us see how we can simulate thick cylinder subjected to internal pressure like this and the dimensions internal radius is 100 and external radius is 200 and we can do the analysis in two ways one is in like a plane strain and the other could be like an axisymmetric and depending on how you take the cross section and if you take a cross section like this along the length you take unit slice we can model it as a plane strain case assuming that in the length direction the strain is 0 or you can take a longitudinal section like this.

And then do this problem as a axisymmetric analysis. So, here actually we can take along the length direction and then when you see the section you will see only the thickness and thickness that is rotated around the central point will give you the cylinder. So, that is what we do here.

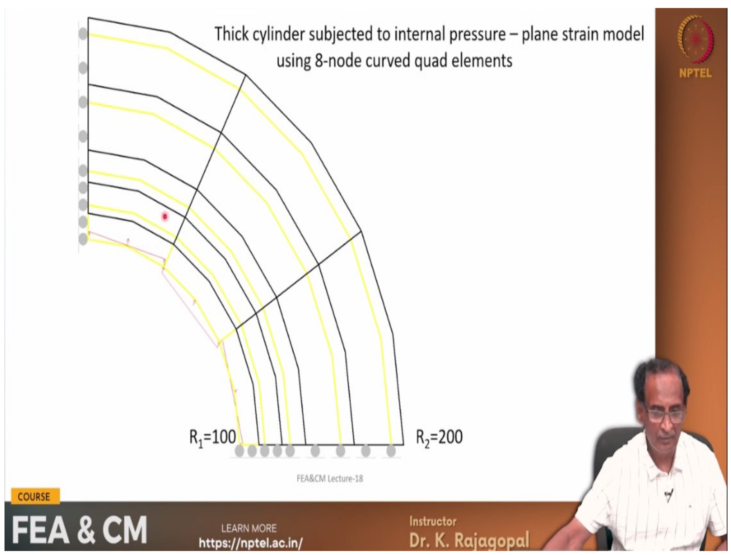
(Refer Slide Time: 42:06)



And so this is the mesh for doing the analysis of thick cylinder subjected to internal pressure. What we do is we take a unit length and support all these the nodes and rollers so that the mesh can deform only radially and that is the ideal case. If you have a very, very long cylinder and take some length in the mid section we have predominantly radial displacements and that is what we get with this mesh or the other possibility is using plane strain model.

So, if you consider unit slice this plane will get a thick cylinder like this and it is not necessary to solve the entire things you can draw two symmetry lines, two symmetry planes here and here and then whatever is happening in this positive quadrant the same thing is going to happen in the negative quadrant and so on.

(Refer Slide Time: 43:18)



So, this is your deformed mesh and the black one is the original mesh and then the yellow one is the deform mesh and just ignore because in this program it cannot draw curved surfaces. So, it has just drawn with straight lines like this, but it is a curved surface then it can also not plot these the pressure loading correctly. So, that is the brief introduction to patch test.

And some aspects of some finite element analysis and if you have any questions please write an email to profkrg@gmail.com. **(Video Starts Here: 44:11)** And before I wind up the class just want to show you one program that you can utilize for doing all these parametric calculations. So, you are going to get one program called Isocalc and this is a very simple program and it can do the calculations for quadrilateral elements, triangular elements and then the traction calculations.

And then you can do two types of calculations or the body force vector or these the force vector due to the initial stresses and then you can do calculations for plane stress, plane strain or axisymmetric and then for different orders of integration, order of integration 1, 2, 3 and 4 up to 4th order of numerical integration you can do and for the plane stress case you need to define some thickness and then we need to define the coordinates.

And so here for each node we need to give x and y coordinates and then this input box is common for both the traction and also for quadrilateral and then for triangles. So, let us see so if you choose the quadrilateral automatically you get option for giving the coordinates for 4 nodes because that is the minimum 4 nodes you need to define or if you choose a triangle you get only 6 node like quadrilateral let me go back to quadrilateral.

See these quadrilateral elements are defined with variable numbers like minimum 4 nodes up to maximum 9 nodes and you may or may not involve the other higher order nodes. So, you can uncheck these boxes so that you will use only the 4 nodes and let us do some calculations. Let us give some numbers let us say our node 1 is at 10 and 10 and at node 2 x is 0 and y is 10 and node 3 is at the origin 0 and 0 let us say and node 4 is at x of 10 and y of 0.

And let us say that we have a plane strain case and the thickness is automatically 1 and our order of integration we can select let us say we select 2 and then let us say we calculate the body force vectors. So, the unit vector I am taking as 20 and the gravity factors 0 and – 1 and then you press this calculate then it will do the calculation and then it is showing – 500 and all these 4 nodes and your positive coordinates are like this x and y.

And so now let us see let me just choose axisymmetric and let us calculate the results with 1 point integration the body force vector you calculate and so with single point integration the axisymmetric element has given – 2,500 at all these 4 nodes which is not correct because our shape functions their first order variation and then our radius also has a first order variation so you have totally second order variation and then you need minimum 2 integration points.

So, let us try with a 2 integration points for this axisymmetric element. So, I have 2 point integration and if you do the calculation we will see that it is different the distribution has changed completely – 3333 at the outer nodes then – 1666 at the inner nodes. So, it is now let me demonstrate for an 8 node quadrilateral I am choosing all the 8 nodes and then our node 5 is at the top surface at x of 5 and y of 10.

And then node 6 is at x of 0 and y of 5 and node 7 x of 5 and y of 0 and node 8 is 10 and 5 and let us choose our plane strain case and let us try with one point integration and we have an 8 node quadrilateral N transpose B with single point integration. So, if you do this you see at the 4 corner points you have a force of 500 whereas at the mid side nodes you have 1,000, 1,000, 1,000.

So, obviously this we do not know whether it is right or wrong, but you know from our previous calculations that the mid side node should attract 4 times the load at the corner nodes, but here we have only 2 times. So, our order of integration may not be sufficient. So, we can go back and change the order of integration to 2 and let us do the calculation and here we get see + 166 at the four corner nodes whereas – 666 at the mid side nodes it is going down at the 4 mid side nodes whereas at the corner nodes it is going up.

That is what we had seen earlier and now let me define the 9th node at 5 and 5 and let us do the calculation and now the nature of distribution has changed completely it is at the middle it is attracting the maximum force of 888 whereas at the 4 corner nodes there are only 55.55. So, actually these and let me do the calculation for this initial stresses let me just take a simple case at one integration point.

And let us say our stresses are sigma x is – 50, sigma y is – 100 and then shear stress is 0 then if you do the calculation you will get the forces interestingly there is no force distributed at the 4 corner nodes because I have used only one point integration and it is not able to give us the correct distribution and – 1000, 500, 1,000, - 500 and we can actually do a 2 by 2 integration and then see what happens.

Let us give the same – 50 – 100 0, 0, 0, 0. So, we are assuming that the element is subjected to some constant stress state so we are using the same constant stresses. So, if we do the calculation now we get different result. See we are getting forces at all the 4 corner nodes and also at the mid side nodes then after you do the analysis you can save the result by doing this the report generator.

So, I am giving result data out and if you press this report generator the results will be saved in this file and then we can open that file and then look at the results not sure exactly where that is saved. These are the results that we get let me blow it up zoom. So, we have 8 nodes corresponding x and y coordinates and then we have used 2 point integration and the stresses are the same at all the points and then the force in the x and y direction at each of these nodes.

So, this is what we say as a consistent term distribution of the load and that is what we need so that our stiffness and the forces are consistent with each other so that is a brief introduction to this program **(Video Ends Here: 55:51)** And in the next class I will demonstrate for more number of cases then we will also see the patch test because that will take some time and in the meantime if you have any question please feel free to send an email to this to my gmail address then I will respond back. So, thank you very much we will meet next time.