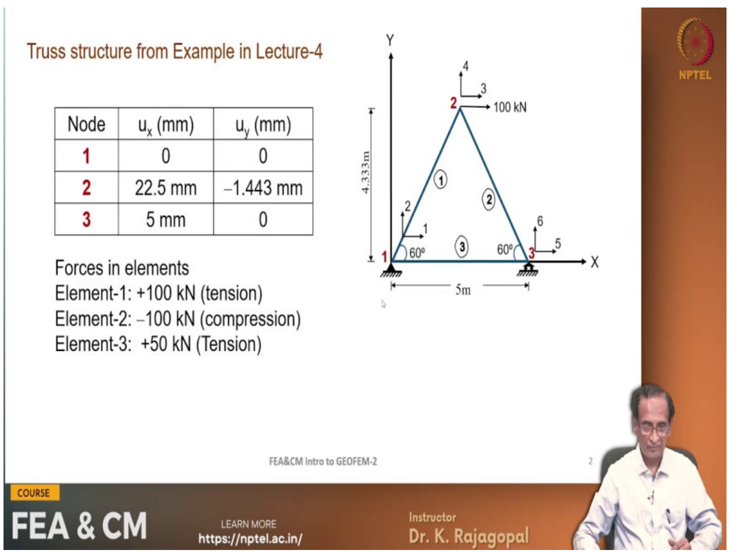
FEM and Constitutive Modelling in Geomechanics
Prof. K. Rajagopal
Department of Civil Engineering
Indian Institute of Technology - Madras

Lecture: 9
Use of GEOFEM finite element program_ Part - II

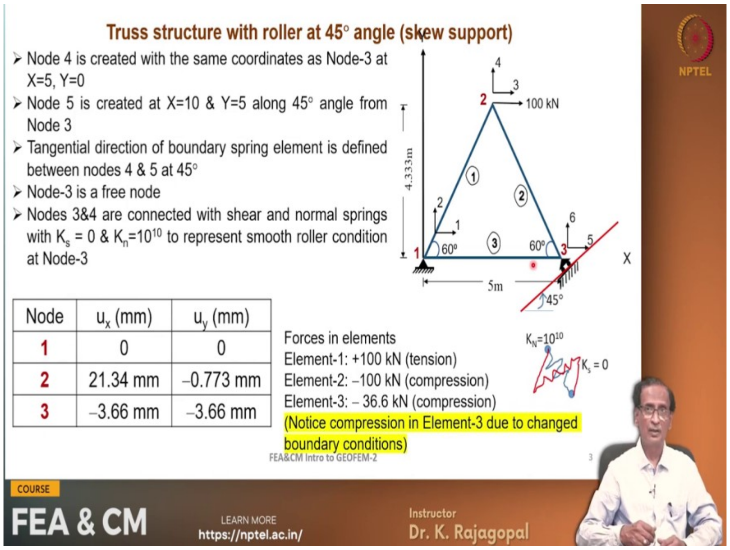
So, hello students very good to see all of you in the previous class we had looked at some applications of the geofem program for the analysis of bar element structures with bar elements and let us continue on the same thing and see some more examples to illustrate how we can use this program.

(Refer Slide Time: 00:50)



And the first example is the same plane frame or plane structure that we had seen earlier that has 3 nodes and 3 bar elements and the way we saw it the left hand side support is on a hinge where you do not permit both x and y direction displacements whereas the node 3 is on supported on a roller it can translate horizontal or in the index direction but cannot move in the y direction.

(Refer Slide Time: 01:39)



And let us see what happens in some other boundary conditions let us say that you have this structure built in a highly mountainous terrain and the right hand side support is on a slope. So, you want to provide the roller parallel to the slope on the other side. And so, we will have a problem because we did not discuss that because all the cases that we considered the boundary conditions were applied in the along the Cartesian coordinates along the x and y axis.

But now we want to apply a boundary condition here saying that you have a roller that is at an angle to the horizontal at an angle to the x axis this type of boundary conditions we call them as a skew supports or this skew boundary conditions and in this case this roller is a at an angle of 45 degrees to x axis. So, how do we do it? And I will illustrate that using our bar elements and then we have spring elements that are connected between 2 nodes and we can exploit their features.

So, what we can do is we create an additional node 4 at the same location of node 3 that is at x is equal to 5 and y is equal to 0. So, that we can introduce a nodal link element here and then we create another node an extra node 5 at x is equal to 10 and y is equal to 5 at 45 degrees from the node 3 along this line. So, that we can define the tangential direction for the spring element because the spring is connected between 2 points that have same coordinates.

So, we cannot really define any direction for that and just to be able to define the directions we give 2 additional nodes that that are placed the apart from each other and then now we make this node 3 as a free node. So, previously we had the node 3 with the free x direction degree of freedom but along the y it is fixed. But now this node 3 is a completely free and then we connect the node 3 and node 4 with some shear and normal Springs.

And the shear spring has a stiffness of 0 so, that the system is free to slide along this surface this inclined surface and the normal stiffness is given very high value. So, that there is no deformation in the normal direction to this inclined surface and we can run the program and these are the results that we get. Say the u x and u y at node one are 0 because this is a hinge and the displacements at node 2 now they are slightly different.

And at node 3 is actually this entire structure as has rotated around because this is a free to move along the downward direction here and because our surface is at 45 degrees angle both x and y direction displacements are the same at node 3, 3.66 in the negative direction and when we calculate the element forces we see that both element 1 and 2 they have the same forces element one has a tensile force of 100 and element 2 has a compression force of 100.

And then element 3 previously it had a tensile force of 50. See that is element 3 has a tensile force of 50 but now it has a force of 36.6 and the 2 in compression. See the actually the nature of force in element 3 has changed from being tensile to compression because of the change in the boundary conditions. Say here our boundary condition is different so, the entire structure is rotated and it has induced some compressive type deformation in element 3.

So, we have this negative force. So, it is actually it is this the spring elements can be used for even applying skew boundary conditions and later we will see that these spring elements can be used for modelling the interfaces between 2 surfaces and so, on. And now let us look at the data file corresponding to this analysis.

(Video Start: 07:29)

So, this is the data file for this analysis. See previously we had only 3 nodes in this in this problem now we have 5 nodes because we need some extra nodes to introduce the spring elements and then to define the direction of this the tangential direction. So, I have defined node 4 with the same coordinates of x is equal to 5 and y of 0 and then I have eliminated all the all the degrees of freedom in that particular node.

And then there is one more node defined but ten and 5 node 5 and these are our bar Element Bar elements 1, 2 and 3 and then this is the spring element this node the type 5 in the geotherm program refers to the nodal link elements and there is only one nodal link element and its connected between nodes 4 and 3 that is at the right hand side support. And then the 2 external nodes are 4 and 5 see the 4 is at x is equal to 5 y of 0 and node 5 is at an inclination of 45 degrees the x is ten and y is 5.

And then we applied the 100 kilo Newton force and now let us see what happens. So, these are this is the result file. So, here this is the definition for spring elements the tangential stiffness is 0 and the normal stiffness is a 10 to the power of 10. So, that there is no deformation normal to the surface and then these are the deformations. And then the forces and the bar elements and the element 3 has a compression of 36.6.

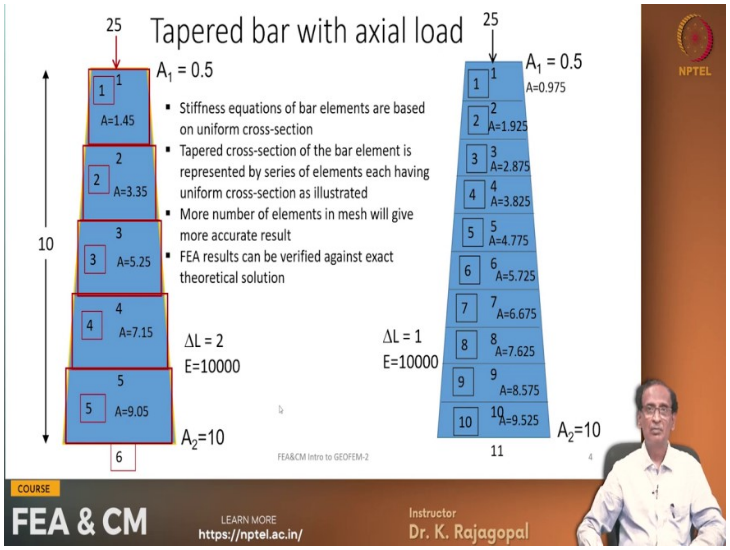
And then when we look at the force in the spring is coming as compression 122.47 see previously when our roller support was a horizontal flat ground it was 86.6 and now we are getting 122.47 but then if you multiply with sine 45 to resolve it in the vertical direction we will get back our 86.6. And let us see one more thing for a flat surface. Let us so, let us define the node 5 along the horizontal direction the long y of 0 corresponding to the 4th node and see what happens.

So, here I am defining both node 4 and 5 along the x axis x is 5 y y of 0 x of 10 and y of 0 and then you see here we get back our previous result of the 50 kilo Newton tension in the element 3 and node 3 has extended by 5 millimeters and then the normal force at node 3 is 86.6 because that is the reaction force that we should expect because of the applied force. So, this example actually explains how we can use the this the nodal spring elements for simulating skew boundary conditions.

So, otherwise we have to go through some transformation or the stiffness Matrix level but that is not implemented in this geofem program but the option is given to implement this skew boundary conditions through the boundary spring method. So, let us continue.

(Video Ends: 12:37)

(Refer Slide Time: 12:38)



So, now let us look at one more problem and this problem is the case of one dimensional bar element it is actually it is a column with a load. And the solution for this is a delta of P l by A E but only thing is our cross section is not uniform it is a tapered bar having an area of 0.5 at the top and an area of 10 at the bottom and you please mind that this is not a scaled diagram just its shown with some taper.

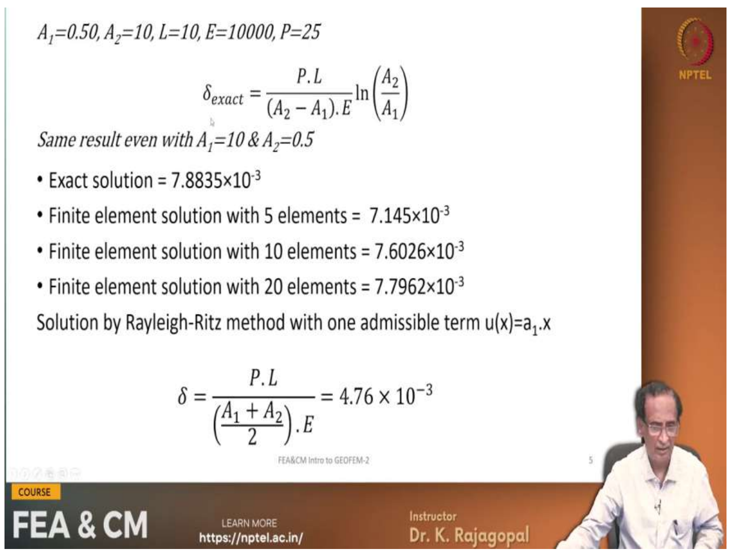
But then it is actually the taper is much larger because the area at the top is only 0.5 whereas the area at the bottom is 10. So, you can imagine how much should be the diameter at the top and bottom. And just for simplicity I have given straight lines but it is not really a straight line and then it is actually it is a simple one dimensional problem with the loading at the top and how do we do it?

Because in our stiffness formulations we did not consider the varying cross sectional area of the bar elements we assume that the bar elements have the constant cross sectional area and then the constant properties e. So, that the stiffness of the bar elements can be written can be written as AE by l. And what we can do is we can split this entire length into some convenient intervals like some number of elements and calculate or the average area from the areas at the 2 ends and then I use that as the area for that particular element.

So, here element one is at the top area is 0.975 2 is the 1.925 and so, on. And so, on the left hand side you see another mesh but with smaller number of elements see this mesh has 10 elements and then 11 nodes whereas this mesh has 5 elements under six nodes. So, here you see this jump between the 2 elements is larger compared to here. And the tapered cross section we can simulate by having a series of elements each having a different area.

And as we are approximating this taper section with the uniform cross sections within elements the more number of elements that we consider the more accurate is going to be the result and finally we can compare our finite element results against the exact theoretical result just for comparison purpose and here on the left hand side see this total length of 10 is divided into 5 elements with a Delta L of 2 and E is a 10000 and the right hand side. We have a mesh with 10 elements and Delta L of one. And so, let us look at what happens.

(Refer Slide Time: 16:18)



$$\delta_{exact} = \frac{P.L}{(A_2 - A_1).E} \ln\left(\frac{A_2}{A_1}\right)$$

And the theoretical solution is given like this P l by a 2 minus a 1 e Logn of a 2 by a 1 and for particular properties of a 1 of 0.5 a 2 of 10 and l of 10 E of 10000 the applied load of 25 the exact solution is is this 7.8835 times 10 to the power of minus 3. And you might wonder what happens if you flip the the areas like let us say we put a 2 there and a one here. But we get the same solution like you can try it out you can change a 2 and a 1.

Exact solution = 7.8835×10^{-3}

Finite element solution with 5 elements = 7.145×10^{-3}

Finite element solution with 10 elements = 7.6026×10^{-3}

Finite element solution with 20 elements = 7.7962×10^{-3}

But you will get the same result and the corresponding solutions with finite element analysis with 5 elements it is 7.145 10 times 10 to the power of minus 3 and with the 10 elements it is getting closer to the theoretical solution 7.60 times 10 to the power of minus 3 and then with the 20 elements 7.79 times 10 to the power of minus 3 whereas the exact solution is a 7.88 times 10 to power of minus 3.

And actually as you see here as we increase the number of elements in the mesh we are moving closer to the to the exact result and and with the smaller number of elements we get a lesser displacement. So, we can in other words we can say that our response is a stiffer see with the lower number of elements we are predicting a lesser displacements. So, that means that our response is stiffer.

And as we are increasing the number of nodes and number of elements we are approaching towards the the exact solution. And just for comparison purpose I have given you the solution with the Rayleigh-Ritz method by considering only one admissible term u of x is a 1 x and the solution is like this the average area a 1 Plus a 2 by 2 is considered for in your Delta of P l by A E equation and that is that is way different from the the exact solution.

Solution by Rayleigh-Ritz method with one admissible term $u(x)=a_1.x$

$$\delta = \frac{P.L}{\left(\frac{A_1 + A_2}{2}\right).E} = 4.76 \times 10^{-3}$$

(Video start: 19:01)

Let us look at the data files for this. So, actually this is our data file for this problem. So, actually when I give you these data files for your running I will put in some comments. So, that is more easy for you to understand and then you also please refer to the geofem manual it is a very short manual it is only some 20 pages. And so, here we have 11 nodes 1 to 11. And then I have deleted it is a one dimensional problem with the displacements only along the y direction.

So, the boundary conditions itself I have eliminated all the other degrees of freedom at node one we are allowing the vertical displacement at node 11 it is a fixed node. And then node 1 is at height of 10 y of ten and the node 11 is a 0 then automatically in between the nodes 1 and 11 the intermediate nodes are generated because we have put a 1 here that is the that is the increment for generating the missing nodes.

And we have 10 elements and E is ten thousand and element 1 is connected between nodes one and 2 and the area is 0.975 and the element 10 is connected between nodes 10 and 11 and the area is 9.525 and then once again the elements between 1 and 10 are automatically generated. So, if we give 1 then the next element it generates is a 2 3 4 5 and so, on. But if you give let us say 2 then it will generate the 3 5 7 9 and so, on.

And then at node one; we have applied a load of 25. So, it is just to compare this with. So, this is the mesh that we are considering there are 11 nodes node one is at a height of 10 and node 11 is at at the height of 0 and then node one 2 3 4 and so, on. And element one has an area of 0.975 where is element 10 has an area of 9.525 right. So, that is; 0.975 and 9.525 and now let us look at the result right.

So, here we gave the data only for nodes 1 and 11 and all the intermediate points are generated automatically and you see that no degree of no equation number is assigned for displacements in x direction or the rotation because this is a bar element and only y direction displacements are defined and the element modulus is 10000 and then element one is connected between nodes one and 2; 2 is between 2 and 3 and so, on.

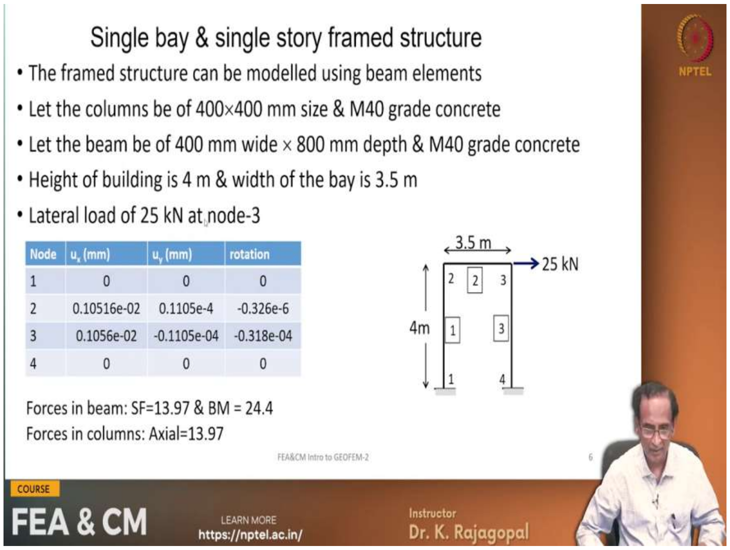
And you see here our area is also proportionally increased 0.975 1.925 and so, on. And element 10 has an area of 9.525 right and then these are the displacements and different nodes at the top is a 0.0076 7.6 that is what we saw times 10 to power -3 and these are the axial stresses. So, it is actually these one dimensional elements they do not have the pre-processor but then it gives an option for generating the missing elements and nodes.

If you just give; for node 1 and node the last node and in between we can generate and the same for elements. So, now let us continue.

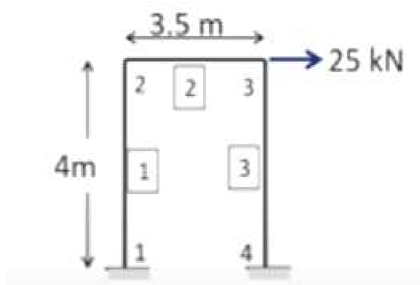
(Video Ends: 24:03)

So, we see that this approximation with finite element analysis tends towards the exact solution as you increase the number of nodes and number of elements in the mesh right.

(Refer Slide Time: 24:18)



So, now let us move on to to beam elements the use of beam elements and let us consider a very very simple single story frame like this and it has 4 nodes node 1 2 3 and 4 and we have 2 columns and then one beam. Let us say that the columns are of dimensions 400 by 400 millimeters and these columns are made of M40 grade concrete and the beam there is a width of 400 MM and a depth of 800 mm.



And it is also made of M40 grade concrete and the height of the building is a 4 and the width is 3.5 and then there is a lateral load of 25 applied at this at this point and these are the these are the displacements and the rotations and then the forces in the beam Shear force and bending moment and then forces in the columns. The column will have predominantly axial and of course there will be some shear and bending moment but have not shown here.

And what happens see when one of your supports undergoes some relative settlement because previously when we applied the some settlements for our truss structure the settlements had no effect on the element forces because all the joints are hinged connection. So, we can yeah the structure can freely rotate and then accommodate the deformations without increasing the forces.

But then here we have a is a statically indeterminate structure and then all the node points they are rigidly connected. So, if you and if you subject support to some deformations then it will lead to some additional Shear force and bending moment in the in the elements and that is what we have seen. See here the right hand side support is assumed to undergo a settlement of 25 millimeters and then when you calculate the shear force and the beam is 53.3.

The blue one is without the settlements it was only 13.97 and bending moment now is 93.4 and without settlements it was only 24 and the axial Force and the columns is also very high 53.3 versus 13.97 without the settlements. So, that shows the effect of settlements on any indeterminate structure like our building frames and also one important point or one important lesson that we learn is if your structure is going to undergo any relative deformations.

We have to take care of the additional forces that are generated additional Shear force and then additional bending moment. So, that we can provide sufficient steel reinforcement to prevent the cracking cutting that may be the last one. And let me show you these data files.

(Video Starts: 27:59)

So, this is our data file see there are 4 nodes and are purposely did not constrain any of these nodes at the element level are at the assembly level. So, all the degrees of freedom are given as 0. So, that it will number them node 1 at x of 0 and y of 0 node 2 is your text is 0 y f 4 node 3 is at 3.514 and node 4 is at x of 3.5 and the y of 0. Then we have 3 elements then Young's modulus is calculated as 5000 times square root of F ck and this is our modulus.

And the cross-sectional area for the columns is a 0.16; 4 times 0.4 and then the moment of inertia 1 12 b d Cube that comes to this much. Then for the beam the cross sectional area is point 4 times point eight that is 0.32 and then this is our moment of inertia 1 12 b d Cube then a load of 25 is applied at node 3 along x direction. So, this one corresponds to x direction node 3 is this and then all the the degrees of freedom and the 2 supports node 1 and node 4 they are constrained.

And let us look at the results. So, this is the result file these are all the this is the nodal coordinate data there are totally 12 degrees of freedom and then the beam element data cross-sectional area moment of inertia or Shear area is actually as I mentioned the shear area is taken as very very large value. So, that these elements become pure flexural members and these are the results of the displacements and then the beam element forces axial Force Shear force and moment at the 2 nodes.

Because each element has node 1 and node 2 and these are the axial and the shear in moment then these are the reaction forces and since we applied a force of 25 in the positive x direction the reaction force the x direction is also coming out as minus 25 for equilibrium then in the vertical direction there was no force applied. So, the sum total of the reaction force is 0 but then if you see the node one has minus 13.97 whereas node 4 that is at the right hand side it has an equal and opposite force plus 13.97.

So, that our net y direction reaction force is 0 and I think the same thing with settlements. So, this problem is the same as the earlier one except that we specified a settlement of 25 millimeters at at node 4 at the right hand side support and these are the settlements like at node 4 y direction displacement is a minus 0.0025 and then these are the shear forces and bending moments and so, on.

These are much higher than what we had when we did not have any sub any settlements. So, this example it shows you the use of beam elements and when we see these files I am going to put some comments so, that you understand what each line of the data means. So, this is how we can utilize the spring elements bar elements and then the beam elements for some practical analysis.

And in your tutorials you will have some problems that you need to solve by hand and then you can compare your hand calculations with these analyses through the finite element program so, that you can check whether your answer is right or wrong. So, thank you very much I think this may be the last slide and if you have any questions please contact me at this email address and I will respond back to you at the earliest. So, thank you very much.