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

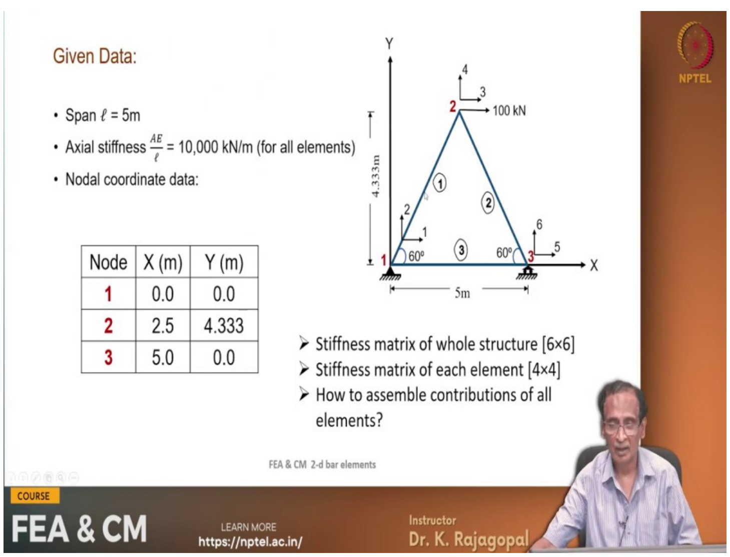
Lecture: 8
Use of GEOFEM finite element program_ Part - I

So, hello students welcome back and let me introduce you to my finite element program geofem this lecture is slightly different from the previous ones we are not going to do any mathematical work or anything we are only going to see how we can apply the finite element program for our Solutions. And this particular one Geofem is developed at IIT Madras as part of several students project works and PhD thesis.

And all the constituted models that are going to discuss they are implemented here along with number of other options and till now we have solved some problems the problem of the truss element and then the soil structure interaction of a combined footing. And I will just demonstrate how we can do this analysis using this program. And what I will do is I will give you the data files. So, that you do not need to recreate the data.

And what you can do is you can modify the properties material properties like your young's modulus or the spring stiffness or something and run the program for getting your results.

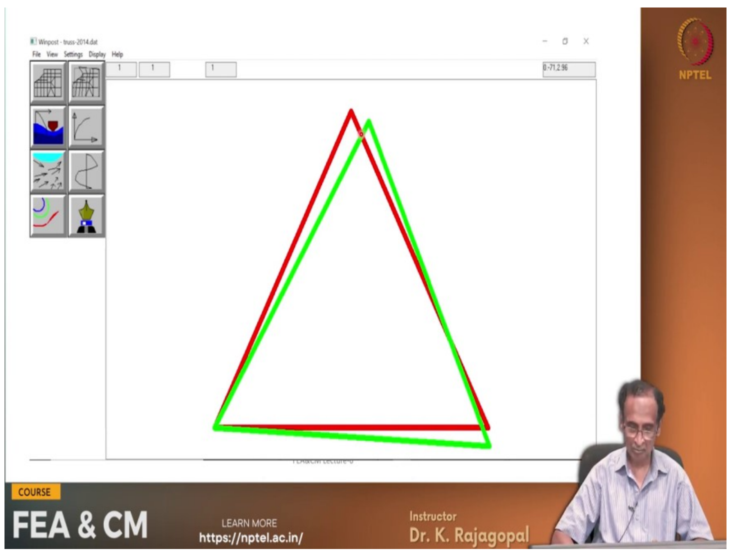
(Refer Slide Time: 01:50)



And so, the we are going to look at the same problems that we had solved earlier see one is this a truss element with the 3 nodes and the 3 elements and the AE by l for each element is ten thousand and it is applied with a 100 kilo Newton load are the at the tip at this point 2 in the x direction and we had considered 2 cases one is a the left side support on a hinge on the right hand side and a roller.

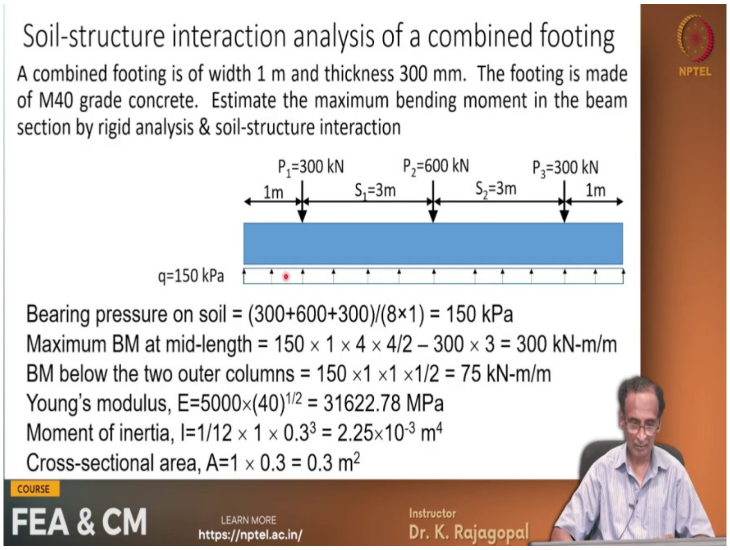
And then we have also considered the case of settlement of this of this right hand side support by 50 millimeters. And we will see 2 methods for simulating this one is by exact method and by the boundary spring method and by the boundary spring method we can even vary the spring stiffness and see what is the effect on the on the displacements that we get and then the results.

(Refer Slide Time: 02:59)



And so, this is what we had seen in one of the lectures the right hand side support settles by 50 millimeters and then the entire structure is just simply rotated and that is one problem that we are going to consider.

(Refer Slide Time: 03:13)



Then the other one is the combined footing with a supported on soil and the soil itself is modelled using the Winkler Springs.

(Video Start: 03:29)

Let me show you the finite Elementary equivalent.. So, here we have this beam and it is supported by a series of Springs and this is the loading that we have given. And so, we get different solutions and that we will try to see how we can do this in the finite element program and before that let me show you this program and this is the user manual for the program Geofem geotechnical finite element modelling.

Say it is a simple enough program and it requires all the finite element analysis they require a lot of data. We need to define the nodal points and then the different elements their connectivity their material properties and so on. And for doing that we have to go through systematic procedure for giving this input to the program and the first line of this data file for any problem is the title of the analysis just for your own information analysis of truss or analysis of truss with settlement or something.

And then the next line is some basic data number of nodal points and then the number of element types like you may have different types of elements like bar element beam element and then a continuum or anything. And depending on the type of elements that you have you can choose it and there are totally five different types of elements in this program a bar beam and then a nodal link element between 2 points and then the joint elements and then the Continuum five types.

And then you can consider the gravity forces and then you can create a post processor file for interpreting your results and I think I will skip all these because they may not mean much to you. And then for each node we need to give the x and y coordinates and then we have to also tell the program whether it is an active node or a fixed node and these are the boundary condition codes corresponding to each node 0 means it is an active node and one means it is deleted it is a fixed degree of freedom and then x and y coordinates.

And we can actually give you know the coordinate values either in the Cartesian system or in the polar coordinates and that we can we can prescribe by some other input. And the program itself does got the bar elements the bar element is an axial element and you can define this with the 3 nodes actually I have defined only with the 2 notes in till now but later on we will see 3 node bar elements.

And these need not be straight these could be curved for representing say rope or a cable or something. Then we have a beam elements 2 node beam elements and these beam elements that defined with axial degree of freedom Shear deformation and then and then the rotational degree of freedom and it will ask for moment of inertia I. And if you are the beam has a width of b and a depth of d we need to externally calculate this is 1 12th b d Cube and give that as an input.

And then the area cross-sectional area also b times d and this particular beam element it has the shear induced deformations also and that also needs to be given as an input.. So, for each beam element we need to define the node 1 and node 2 and then the material property set number like you need not have the same material all through your length of the beam could be made of different materials.

And if you want you can define and then the cross sectional area also we need to give and then the moment of inertia and then this particular beam element it can include the shear deformations but if you do not want to include the shear deformations we can set the shear area to a very very large area large value.. So, that we can we can ignore the shear Def deformations and the element becomes a pure flexural member.

Then the other type of element that we had seen till now is the spring element.. So, the spring element is actually it is a spring between 2 nodal points and we can define the stiffness and then we need to define the normal and the tangential directions and because these nodes node 1 and node to a spring element they are having the same coordinates because basically this is a contact element contact element with the zero thickness.

So, there is no separation and for defining its tangential in the normal direction we have 2 other nodes node 3 and node 4 along this line we have the shear direction and normal to this is your normal spring direction and when we use them say for a combined footing I will explain that we can connect one of these nodes say node 2 to the beam or to the footing and node 1 can be your fixed end node that is corresponding to the soil right.

And so, let me now open and all the results once we run the program the results can be saved in a in a text file let me just zoom it a little bit.. So, here this is our results file for the example of the truss that we have done in the class. And there are 3 nodal points and then there is only one element type because all the elements are only bar elements and then the coordinates x coordinate of node 1 is 0, 0.

And I have made all the equations are active.. So, that we can apply the boundary conditions and then see how the the boundary forces are developing. And then the node 3 is at a distance of 5 the x direction and the y is 0. And node 2 is x is a 2.5 and y is 4.3301 as we had seen in the lecture and there is no rotational degree of freedom for the bar elements.. So, this is made as this is zero.

And then I have given the Young's modulus of 50000 and then the area of the element as one.. So, that your AE by l see the length of each of these elements is 5.. So, A E by l is 10000 that is what we use it in the class. So, I am giving the Young's modulus as 50000 and the area of the element is 1 and the length is 5 units.. So, A E by l is a 10000. And so, we have 3 elements element one is connected between node 1 and node 2.

And element 2 is connected between nodes 2 and 3 and element 3 is connected between 1 and 3 that is what we had seen here let me just go back.. So, this is a node 1 at 0 0 node 2 a 2.5 and 4.33 and node 3 is at distance of 5 and 0 x is 5 and y is 0. And element one is between nodes one and the 2 element 2 is between 2 and 3 element 3 is between 1 and 3 right. Then we have applied a load of 100 kilo newtons at at node 2 in direction one.

So, we can apply the load in different directions direction 1 refers to x direction 2 is refers to y and l code 3 means a rotational or moment then actually this particular one is corresponding to settlement of 50 millimeters and the right hand side support. And these are the results that we get see these are the displacements at node one the displacement x and y are 0 because these are this is a fixed end node and node 2 it has deformed by by this much in the x direction 0.65 and y direction minus 0.0264. And the node 3 it has moved in the horizontal direction by 0.005 and the settlement is 0.05 that is what we imposed. And then these are the the forces and the bar elements.

Actually the area that is specified is one so whether it is a 4 force or stress it is the same.. So, in element 1 it is a tensile force of 100 say the sign convention in this program is that elasticity sign convention tension is positive compression is negative. So, element one has a force of 100 kilonewtons and element 2 is a force of minus 100 and element 3 it is it is a tensile force of 50. And actually you might wonder why it is calculating at 2 points that I will explain later because these correspond to numerical integration points that we will see.

And the reaction forces at node 1 there is a reaction force of minus 100 to compensate for the right hand side force of 100 and then at node 1 in the direction 2 it is minus 86.6 and node 3 it is 86.6 it is basically it is counteracting the applied moment.. So, actually we are applying a force of here 100 here and the reaction force as we have seen it is basically 100 times 4.33 by sorry 100 times 5 is 0 is your external moment.

And the couple is the reaction force multiplied by 4.33. So, you can easily calculate the moment and that is what we get here. So, the reaction forces are 86.6 acting up on the right hand side and 86.6 acting down in the in the left hand side. So, if you look at the let us see the data file. So, this is the data file for this problem for the truss element. So, this is the actually all the results are stored in a binary file for our post processing.

And there is one file name given here trust dot bin then as the program is running we can save all the intermediate data in some files and this is required. So, if you want to stop the program at after some time and then re-run we can do this. Because most of these finite element programs they can run for a long time for sometimes 2 to 3 days and if you want to see how the progress is going on like you can stop after say five six hours look at the results.

And then restart and for restarting the finite element program has to read some files. So, it is going to read from these 2 files and if you put a star in the name it will be opened as a temporary file. So, you will not be able to restart if you restart it will start from the beginning. And so, if you want to restart any analysis we need to give a name without a star and your Young's these are the element coordinates and then the boundary conditions that we give for the degrees of freedom and your Young's modulus is 50000.

And then the cross-sectional area is one and then the loading is applied at node 2 in direction one of 100 and these are the boundary conditions at node one we have applied a displacement of 0 in the x direction and at node 1 the displacement y direction also we have applied the zero displacement and then in node 3. We have actually applied zero displacement. So, and running the program is very simple.

So, you just type I am going to give you this program Geofem it is a run unit and it will it is actually it is going to open a Dos window and then ask for the input file you can. So, this is the input data and then and then I am giving dot out for sorting out these files that is the output file. Then I just it will go through and then finish the run and then you can open the file for looking at your results where is the truss.

So, this is your results file. So, in this particular run that we have done right hand side support is not allowed to settle down and these are the displacements that we got in the class we have seen at node 1 there are no deformations and node 2 these are the deformations and at node 3 there is a roller. So, it will deform only in the horizontal direction but not in the vertical direction and these are your axial forces node element one has a tensile force of 100.

Element 2 has a compression force of 100 and element 3 has a tensile force of 50 right. And so, it is actually very simple to run and when I give you these programs I will put some comments I will show you how with this comments look like. So, that it is on each line you can put some comment. So, that you know what you are doing. And now let us look at this program or this analysis of beam and elastic foundation.

And this is your so, this particular analysis was done with the 16 beam elements our total length of the footing is eight meters and we have taken we have divided that 8 meters into 16 elements each of 0.5 meters length. So, if we look at the our x coordinates x coordinate one is at 0 0 second one is at 0.510 and actually it is this the entire beam is along the x axis. So, there is no y coordinate and then in our analysis we have neglected the axial deformations.

So, the there are no equation numbers in the index direction everything is zero and there is only displacements in the y direction and we have totally 17 nodes along the beam and all of them they have the rotational degree of freedom. Like these beam elements they have axial shear and rotational degree of freedom but I have deleted the axial degree of freedom because we are not interested.

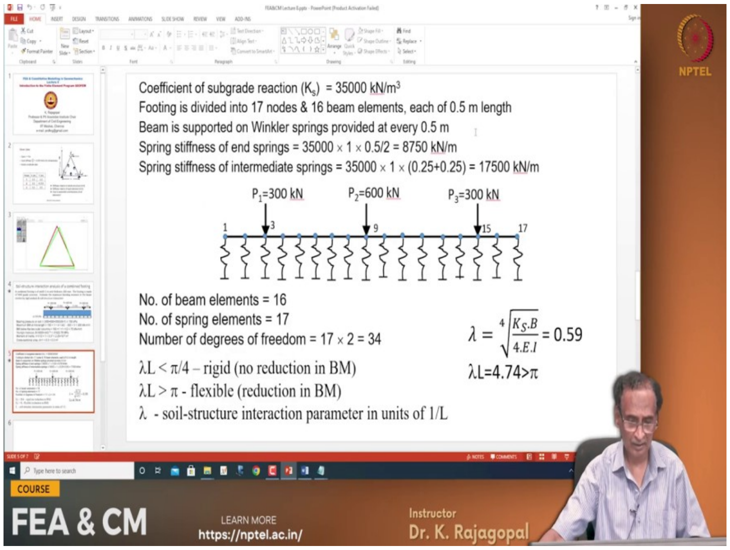
So, the number of degrees of freedom are 17 times 2 that is 34 and then below this we have the the nodes corresponding to the soil and they are all fixed they are not allowed to deform. So, that our soil has a firm support and then in between the beam and the soil at the fixed end we are going to provide our spring elements and the beam elements. So, our young's modulus is 31.622 times 10 to power of 8 kPa and the cross-sectional area is 0.3.

And then the moment of inertia that we had calculated as a 2.25 times 10 to the power of minus 3 I think I may not have copied all the data oh yeah here. So, our Young's modulus is 31 622.78 mPa and in kilopascals 31.62 times 10 to power of 6 and the moment of inertia is a 2.25 times 10 to the power minus 3 cross sectional area is 0.3 And so, that is what we have here 0.3 is the cross-sectional area and 2.25 times 10 to the power of minus 3 and then the Young's modulus.

And the shear area is set to a very large value so, that we do not include the shear induce the deformations in our calculations. And then these are the spring elements and there are 2 stiffnesses one is 8750 and then 17500.

(Video End: 25:30)

(Refer Slide Time: 25:31)



See the 2 Springs are the at the edge of the footing they will have a stiffness of 8750 see they contributing area for the 2 end Springs is only half length of the of element one and element one is of length 0.5 and half length is a 0.25. So, the spring stiffness for these 2 end ones is 35000 times 1 times 0.5 by 2 that is 8750 then for the intermediate springs it is 17500 right. So, we define 2 material properties one with 8750 and the other with 17500.

Then when we define our spring elements see for these are 2 end springs we have material property of one that means that it has a stiffness of 8750 and then all the intermediate ones are linked with the second material that has 17500 and then there are 3 applied loads 300 600 and 300 at different points at nodes 3 9 and 15. say node 3 is at a distance of 1 meter that is what we have in the actual one and node 9 is corresponding to 4 meters that is the mid length and then node 15 is at a distance of 7 meters from the left side.

These are these are at the at one meter 4 meters and seven meters from the left side node and these are the displacements that we have and set the mid length your rotation should be zero because it is a symmetry point and that is one indication of whether you got the solution right or wrong. And here if you see a rotation at the ninth node that is at the mid length it is 10 to the power of minus sixteen which is negligible that is a very very small value.

Then at the 2 ends we get a displacement of 0.0032 and it is it is symmetric and then the maximum settlement happens at the midpoint because that is affected by the entire loading whereas away from it the load intensity is a smaller. So, you have a smaller displacement.

Then the results in terms of the bending moment and other things say the maximum bending moment that we have is 187.7.

And then at the below its its outer columns your bending moment is a 58.89 that is what we had seen in the in the classroom lecture. So, we can I will give you these um these data files and then the program and you can run. And just to illustrate what I am going to do is I will just take this case of the truss and the truss example. Just one minute well anyway I think I do not have the data file.

But I will show you the results. See this is the truss example with the right hand side supported on a boundary spring and we have seen that I am sorry I think this is not the correct one see this is with a truss with a boundary spring and I am giving a very small stiffness like a stiffness for the boundary spring of only 1000. And so, if you want to get a compression of 50 millimeters we need to apply a force of 50 kilonewtons 50 divided by 1000 is 0.05.

And let us see whether we will be able to get it or not and all the other properties are the same as we had earlier and let us see truss. So, here server displacements they are totally different. So, I have actually I wanted a displacement of 0.05 but what we got is minus 0.1366 I think let me just zoom it a little bit. Say although we applied we wanted to apply a settlement of 0.05 but we got 0.1366 that is because our boundary spring stiffness value was chosen as 1000 very very low compared to the compared to the modulus of your other elements.

And we can improvise the solution by using a larger value. Let us let us go back and change the some property. Let me just it now I am going to change the boundary spring by another thousand times and correspondingly we need to apply much larger much larger force. And now I am going to rerun the same the same data file and let us see what happens? There is something wrong with this just one minute I think I made a mistake.

So, that is why it is right very stress now it has run and see now you see the result that we have. So, I have given a very very large stiffness and then applied the force corresponding to that stiffness and this node has settled by the desired amount of 0.05 millimeters that is what previously we got some 0.136 and these displacements are also they are corresponding to what we had got in the classroom example.

And the settlement is not going to affect our element forces they are still the same because this is a determinate structure. So, like this you should be able to use this program geofem for all your classroom problems and we will get an executable version of this program along with all the data files I will give you sufficient instruction so, that you can run this program get a result and interpret.

So, that your you will be able to check your answers because we will not be able to do all the hand calculations especially for larger problems and the small problems you can do by hand and when we have very large structure let us say some thousand degrees of freedom obviously you cannot solve by hand you can use this computer program. And especially as we move into the higher order elements like continuum we will be dealing with a very very large number of degrees of freedom.

There you need to use the program and this particular program has got a reasonably good post processor but unfortunately that is only for continuum elements. So, when we go into the Continuum analysis I will show you that part of the program I cannot show much with the beam elements and bar elements. So, thank you very much. So, we will meet in the next class and this is a brief introduction to this use of geofem finite element program.

And I have shown you 2 examples one is the truss example and the other is the combined footing analysis. So, we will meet next time thank you very much bye.