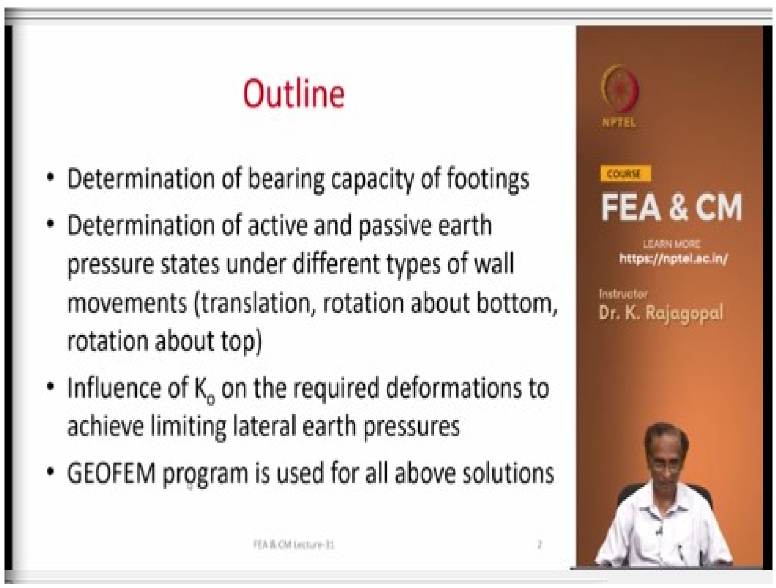
Finite Element Analysis and Constitutive Modelling in Geomechanics
Prof. K. Rajagopal
Department of Civil Engineering
Indian Institute of Technology-Madras

Lecture-35
Some Limit Solutions in Geotechnical Engineering

In the previous classes we had looked at working with the hyperbolic model and how we can perform some numerical analysis. And we have also seen how to do the stress correction to bring the stress state back to the yield surface. And in today's class let us look at some more practical problems that is how to obtain the limit solutions using finite element approach. I will focus only on the bearing capacity and lateral earth pressures because for which we have theoretical solutions.

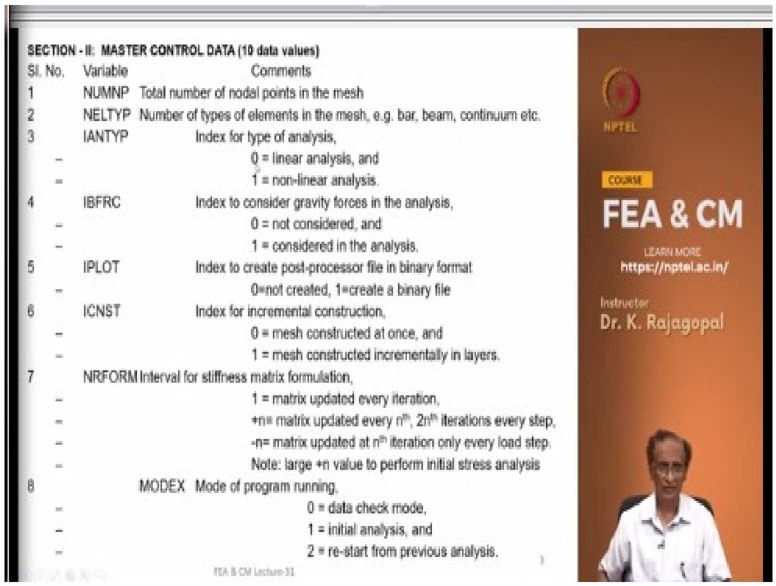
(Refer Slide Time: 00:54)



And the brief outline is determination of bearing capacity factors and then determination of the active and the passive earth pressures. And we will see how to incorporate the effect of different types of wall movement that is lateral translation, that is corresponding to Rankine's active earth pressure theory or passive earth pressure theory where he assumed that the entire wall deforms laterally either away from the soil or into the soil or we can consider rotation about the bottom and rotation about the top.

And then we will also look at the influence of K naught that is the initial at pressure state on the magnitudes of deformation that we require for achieving our active and passive limit states. And in all these I will demonstrate the use of the GEOFEM program that is going to come as part of this course that you can use for doing all these analysis.

(Refer Slide Time: 02:09)



And before we going I just want to highlight some data that we give for the GEOFEM program, this is called as the master control data. This comes on the second line, the first line is the title of the work that you are doing and this is the second line it has a very important data. And number of nodal points is the total number of nodes that you have in the mesh and then the number of element types and then the analysis type 0 and 1, 0 is for linear elastic analysis and 1 for non-linear analysis and non-linear including the elastic plastic and all these things.

SECTION - II: MASTER CONTROL DATA (10 data values)

Sl. No.	Variable	Comments
1	NUMNP	Total number of nodal points in the mesh
2	NELTYP	Number of types of elements in the mesh, e.g. bar, beam, continuum etc.
3	IANTYP	Index for type of analysis,
–		0 = linear analysis, and
–		1 = non-linear analysis.
4	IBFRC	Index to consider gravity forces in the analysis,
–		0 = not considered, and
–		1 = considered in the analysis.
5	IPLOT	Index to create post-processor file in binary format
–		0=not created, 1=create a binary file
6	ICNST	Index for incremental construction,
–		0 = mesh constructed at once, and
–		1 = mesh constructed incrementally in layers.
7	NRFORM	Interval for stiffness matrix formulation,
–		1 = matrix updated every iteration,
–		+n= matrix updated every n^{th}, $2n^{th}$ iterations every step,
–		-n= matrix updated at n^{th} iteration only every load step.
–		Note: large +n value to perform initial stress analysis
8	MODEX	Mode of program running,
–		0 = data check mode,
–		1 = initial analysis, and
–		2 = re-start from previous analysis.

3

This particular index is important because if you place it as 0 then the program will not calculate the reaction forces and it will assume that you are satisfying the equilibrium at all the times and it will not calculate the out of balance forces norm of outer balance forces and so on. And if you give it as 1 then the program will calculate the reaction forces and this value is also automatically chosen by the program in case you have any incremental construction or x equation this has to be 1.

Because you require your b transpose, sigma or n transpose, b vectors and then if any of your materials has elastic plastic constitute to matrix that also requires. And then IBFRC is for considering the gravity forces, if it is 0 you do not consider any gravity forces and if it is 1 then it considers the gravity forces. And IPLOT is to create a post processor file in the binary format, 0 means it will not create the file and 1 means it will create a file and you need to give a name for this file.

ICNST is also very important parameter that we will see in the next class a demonstration of how to do the incremental construction and incremental x equation. 0 means the mesh is constructed at 1 stretch and 1 means the mesh is constructed incrementally that is simulate incremental construction. And then NRFORM is actually it is a very important parameter that tells the program how to update the stiffness matrix.

If you give it as 1 then the stiffness matrix is updated at every iteration, updated means it has to go through the calculation b transpose db and then assemble it and then even the triangulation. Then +n means the matrix is updated at every n to nth and so on, like if you give it as 10 then the stiffness matrix are updated at the end of 10th iteration, 20th iteration and so on. And then -n means it gets updated only once within a load step, say if you give it as -1 then the stiffness matrix is updated at the first iteration in every load step.

Say load step 1 it will get updated at the first iteration load step 2 till once again get updated at the first load step. And you can simulate the initial stress analysis where we do not update the stiffness matrix by just simply giving this value as some very large value say some 10,000 or 100,000 then this condition of +n will never happen because you are not going to have that many number of iterations.

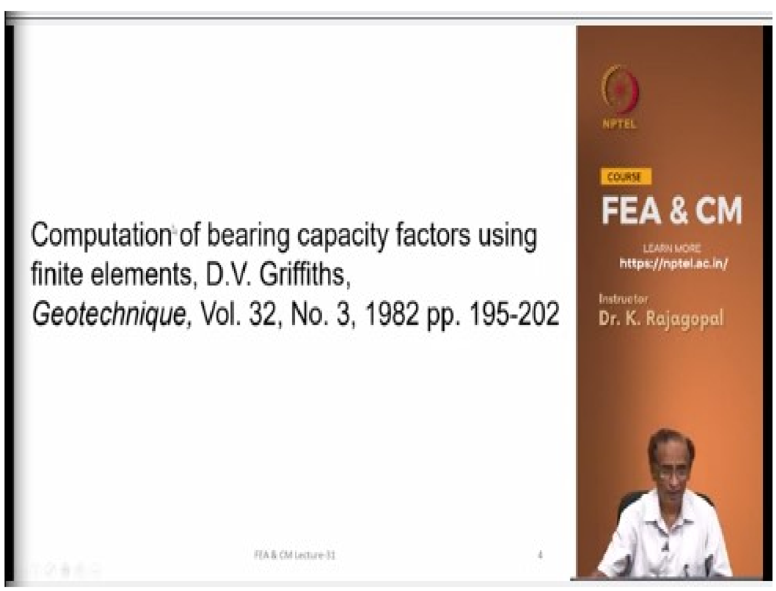
And in all these things you may need to run the program repeatedly. Say if this MODEX is the mode of program execution, 0 means it is only a data check run; the program will run through and tell you whether the entire data is good or bad. Like if some data is inconsistent then it **it** will prompt you, 1 for initial analysis and as I mentioned earlier all these finite element programs they can take a very long CPU time, sometimes the CPU time could be even 1 week.

And if you want to examine the results say every day you can run the program up to certain stage, stop it, examine the results are then restart. And the MODEX of 2 means you can restart the program from where you stopped previously. And especially when we are doing the incremental construction or the change of boundary type like sometimes we change the boundary from load control boundary to a displacement control boundary, in that case also this option becomes quite useful.

And let us say you apply the K naught with a very, very large value like K naught of 3, means the program will apply the self weight with the dummy Poisson's ratio of K naught by 1 + K naught that is 0.75. And then at the end of the analysis anyway the program will reset all the displacements and then the strains to 0, you will generate the required at the pressure state say the vertical pressure will be gamma z and the lateral pressure will be K naught times gamma z.

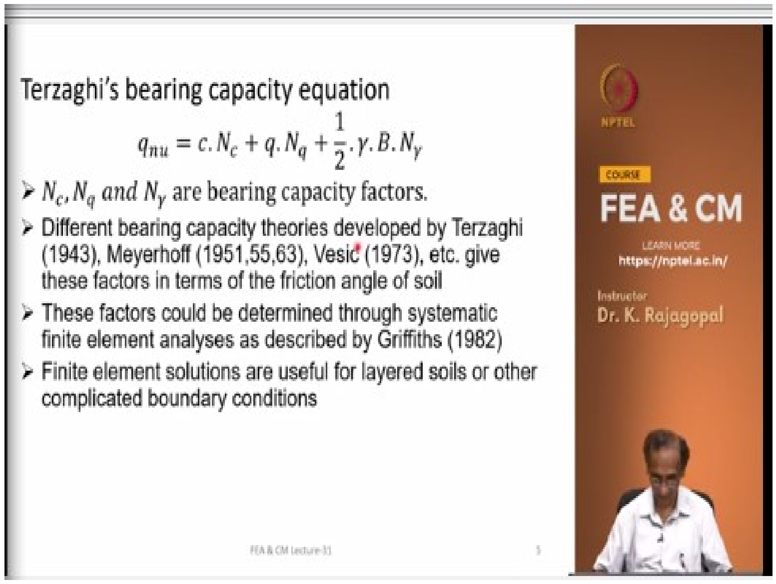
And in this case because your Poisson's ratio is beyond your elastic limits of -1 to 0.5, that Poisson's ratio needs to be reset before you do the actual analysis. So, in that case what we do is we play the K naught with MODEX of 1 and then restart the analysis with the MODEX of 2 later. That is one way of overcoming this problem of incompatible Poisson's ratio that also I will demonstrate.

(Refer Slide Time: 09:20)



And in this lecture I am going to follow this paper that was published long time back by professor Griffiths, he is now at university of Colorado. He published a paper on competition of bearing capacity factors using finite elements and if you are interested you can refer to this paper.

(Refer Slide Time: 09:44)



And all of us know the Terzaghi's bearing capacity equation, the q nu c N c + q N q + 1 half gamma B and gamma, where N c, N q and N gamma are the bearing capacity factors and which are related to the friction angle. And there are different theories that describe how to determine these N c, N q and N gamma in terms of the friction angle phi. Now starting from Terzaghi in 1943 and Meyerhof in a series of papers he published the number of them to estimate these bearing capacity factors.

Terzaghi's bearing capacity equation

$$q_{nu} = c.N_c + q.N_q + \frac{1}{2}.\gamma.B.N_\gamma$$

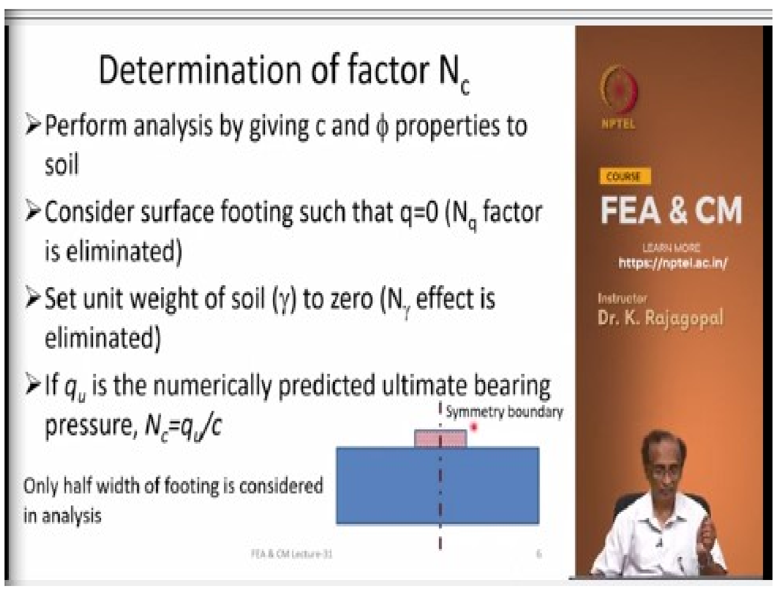
And in between we have several other theories by Hansen and then Bala and so on. And then we have the Vesic's theory in 1973 in fact our IS 6403 is based on the Vesic's theory. And Vesic also has proposed bearing capacity factors N c, N q and N gamma and then also the shape factors for representing the shape of the footing and so on. And in this lecture I am going to only demonstrate the bearing capacity determination for a slip footing and for a circular footing.

And these factors N c, N q and N gamma they can be determined through systematic finite element analysis as described by Griffiths and I am going to describe now for all of you. And this finite element analysis becomes necessary if you want to analyze for a layered soil profile. Or let us say your soil profile is highly complicated you have inclined soil layers and then some joints and so on and you want to know what is the effect of any footing constructed on such a soil.

You cannot use any of the existing theories because all these theories by Terzaghi, Meyerhof, Vesic and so on. They only assumed homogeneous soil conditions and they assume that the boundary is far away. But in some cases your heart stata may be at a shallow depth compared to your foundation width. In that case your actual bearing capacity may be much more than what these theories predict, let me just highlight my laser pointer.

So, these bearing capacity theories they assume homogeneous soil and then the boundary at a large distance from the footing but for other cases we need to use some other methods like finite element analysis.

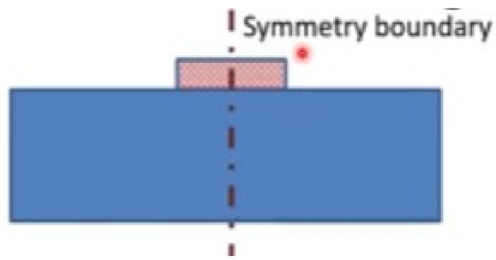
(Refer Slide Time: 12:50)



Let us first look at the factor N c, we can perform analysis by giving both c and phi properties to the soil. And we can consider a surface footing such that our gamma df that is the q is 0 and the effect of N q factor is eliminated. And we can set the unit weight gamma to 0, so that your N gamma effect is eliminated and then we can estimate N c as the q u that is the numerically predicted the ultimate bearing pressure that divided by c will be your N c, just to highlight.

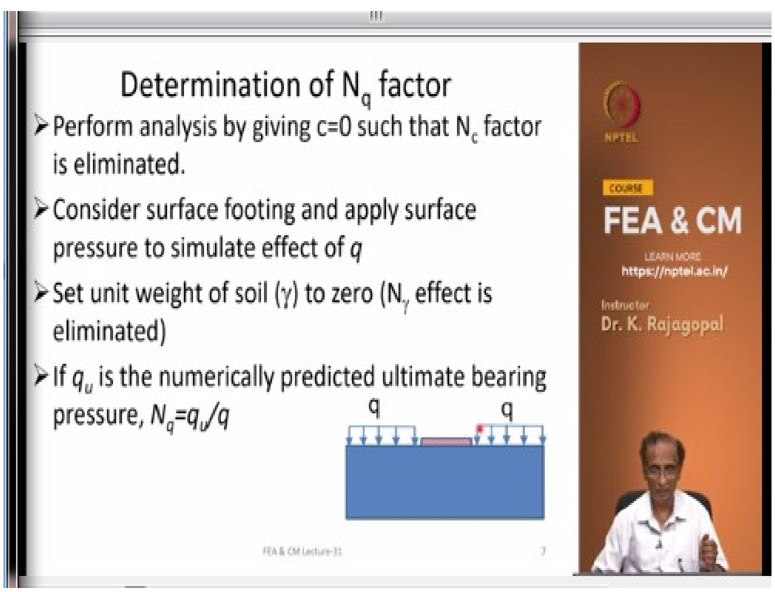
$$N_c = q_u/c$$

So, if you see this equation c N c + q N q + 1 half gamma B N gamma, the q is the overburden pressure at the footing level, that is gamma df or gamma prime df. And if you take the footing at the surface your q is 0 and if you give N gamma of 0 then the N gamma effect is not there and your bearing capacities only because of this c N c. So, this is what we are going to do. And in all these cases if you have a symmetric loading we consider only half the footing by considering the symmetry boundary conditions.

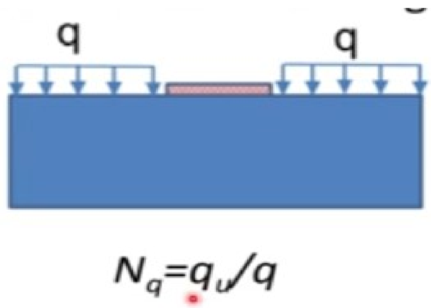


So, at the center of the footing or at along the mid section your lateral deformations are 0 because you have a uniform pressure and it is symmetric. So, this condition that all the lateral deformations are 0 along the mid section that itself is our symmetry boundary condition and by doing that we can consider only half the footing.

(Refer Slide Time: 14:56)

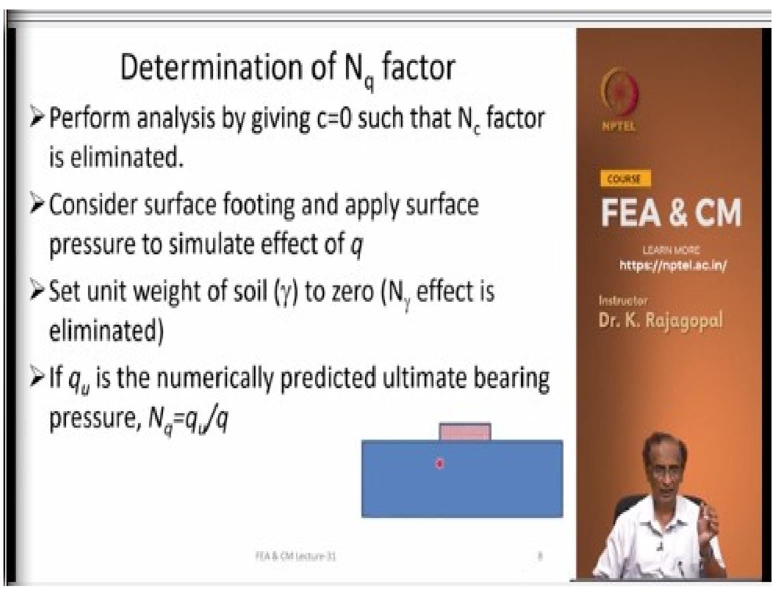


And we can determine N q by performing the analysis by setting the c to 0 such that the effect of N c factor is eliminated. And we consider surface footing but then apply some overburden pressure the q on the surface on both the sides of the footing. And then we set the gamma the unit weight of the soil to 0, so that the effect of N gamma is eliminated. And then once you get your q u that is the numerically predicted ultimate bearing pressure.



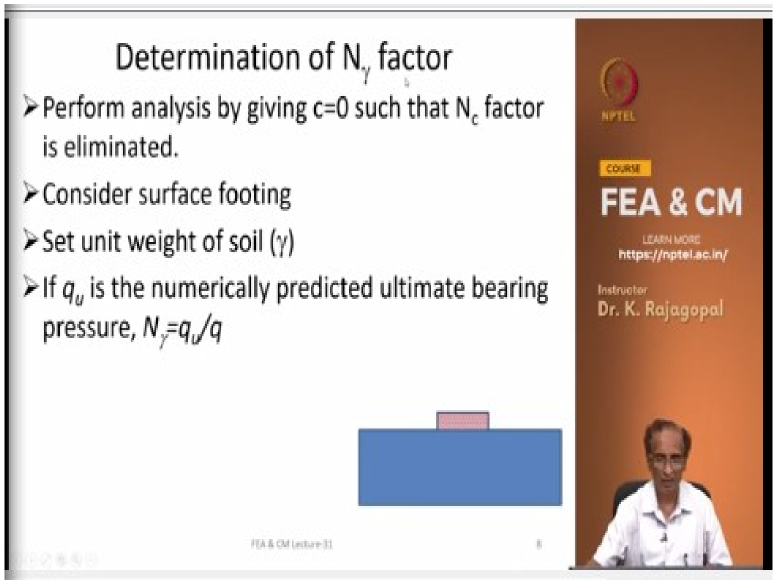
N q is q u by q where q is the applied pressure and the q u is the bearing pressure that we determined from the analysis. And once again we consider only half the footing by considering the symmetry.

(Refer Slide Time: 15:57)

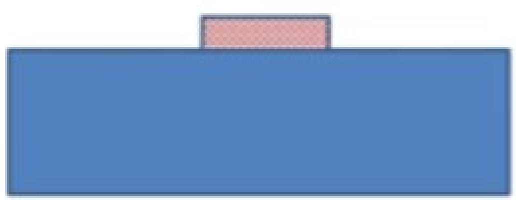


Then this is N q.

(Refer Slide Time: 15:59)

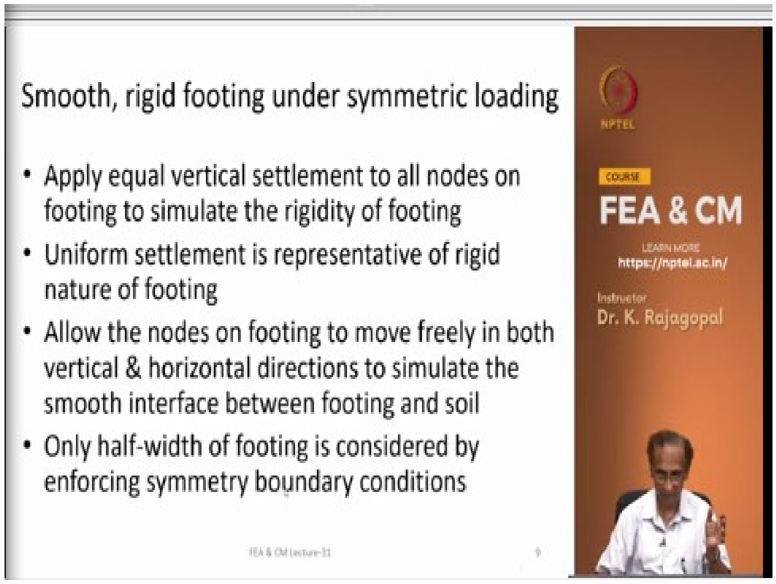


And then N gamma, I am sorry about that, I think I cut and pasted but. For determining the N gamma we can set c of 0, so that the N c factor is eliminated and then we place the footing at the surface, so that our q is 0 and our N q effect is also eliminated and we can set the unit weight of the soil to gamma and then perform the bearing capacity analysis. And our N gamma is a q u by q, so separately we can determine N c, N q and N gamma and then we can superimpose them or you can also do that through the finite element analysis, we will see that. We can get the bearing capacity because of all the contributions because of the cohesive strength and because of the overburden pressure and then because of the unit weight of the soil.



$N_{\gamma}=q_u/q$

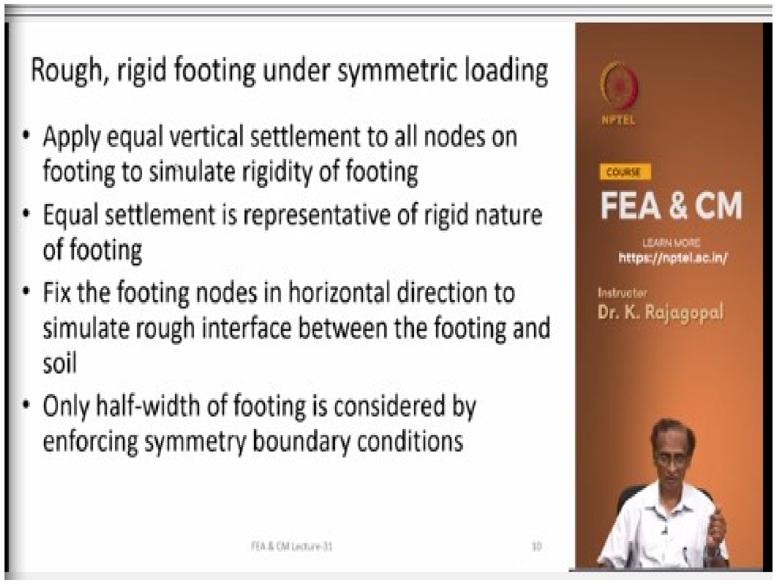
(Refer Slide Time: 17:04)



And we can actually have different variations of the type of footings. Let us say we have a smooth and rigid footing under symmetric loading and the rigidity of the footing we can simulate by applying equal vertical settlement to all the nodes on the footing. So, that the rigidity of the footing is simulated. And the uniform settlement is representative of the rigid nature of the footing.

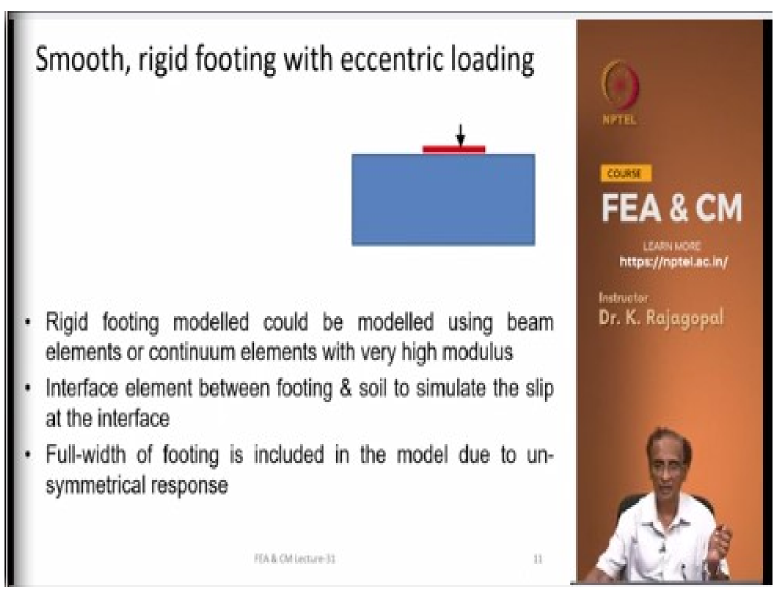
By rigid what we mean is the settlement is uniform over the entire footing provided your loading is symmetric, if you have an unsymmetric loading that you will not have it that we will see later. And then we allow the nodes on the footing to move freely in both vertical and lateral directions to simulate the smooth interface between the footing and then the soil. And we consider only half width of the footing by enforcing the symmetry boundary conditions.

(Refer Slide Time: 18:13)

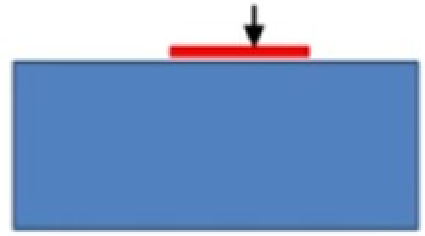


And then we can also have a rough rigid footing wherein our footing is perfectly bonded with the soil. And once again the rigidity of the footing is simulated by applying equal vertical displacements to all the nodes on the footing. And then we fix the nodes on the footing in the horizontal direction, so that the nodes are forced to move only in the vertical direction and this is to simulate the rough interface between the footing and then the soil. And once again we consider only half width of the footing by enforcing symmetry boundary conditions.

(Refer Slide Time: 18:58)



And sometimes we may have an eccentrically loaded footing.



Let us say you have a strip footing with an eccentric loading and let us say that our footing is smooth. And if you have this type of loading, we do not really know how the settlements change, so we cannot really apply any displacements because we do not know how much displacement we need to apply and in what proportion. So, we can simulate this in an indirect manner, we can simulate the footing through either a continuum element or a beam element.

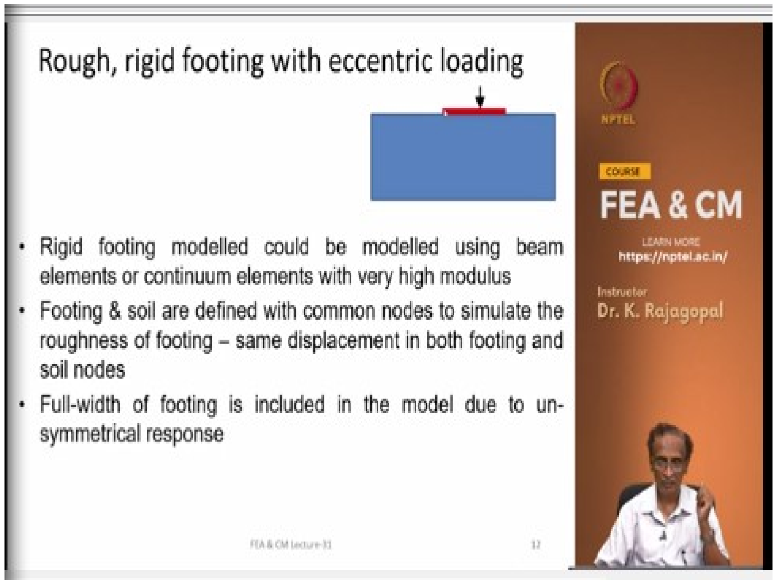
Usually we use a beam element because it is more easy to control the modulus, you can control the Young's modulus and then the moment of inertia and then the section. And then we can apply the loading eccentrically and we provide an interface between the footing and then the soil, so that we can simulate the slip. We can give the exact properties of the interface between the footing and then the soil.

And if the footing is perfectly smooth what we do is we give the shear modulus of the interface element to we set it to a very small value, not exactly 0 but some small value, so that the soil nodes can move freely laterally. And here we are forced to consider the full width of the footing because there is no symmetry, so we need to consider the full width and then the soil on both to the right hand side and the left hand side and we can apply the load.

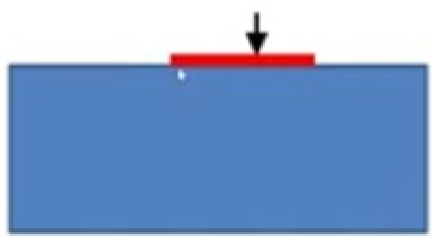
And in the GEOFEM program there is an option for using the constraints we can relate the displacements along the footing through some linear equations, so that we can simulate the rigidity of the footing. But I do not suggest using that because that will complicate the analysis. And that will actually spoil the banded nature of the stiffness matrix and then you end up spending lot of time in doing the computations.

Only for small problems you can do the constraint option but for a large problem you better use a rigid element like either a beam element or a continuum.

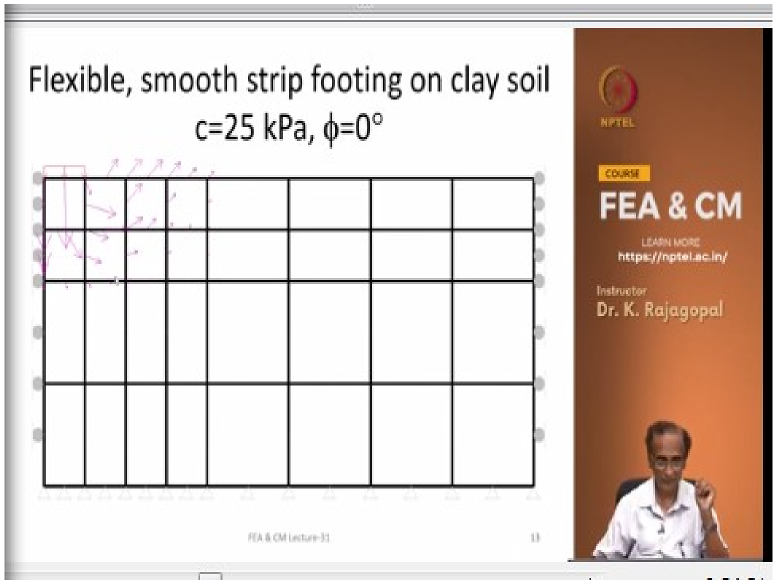
(Refer Slide Time: 21:54)



Then if you have a rough, rigid footing with eccentric loading, we do not provide any interface element we directly connect the beam element with the soil elements. So, that the soil loads will move along with the beam and this is to simulate the roughness of the interface between the footing and then the soil.



(Refer Slide Time: 22:26)

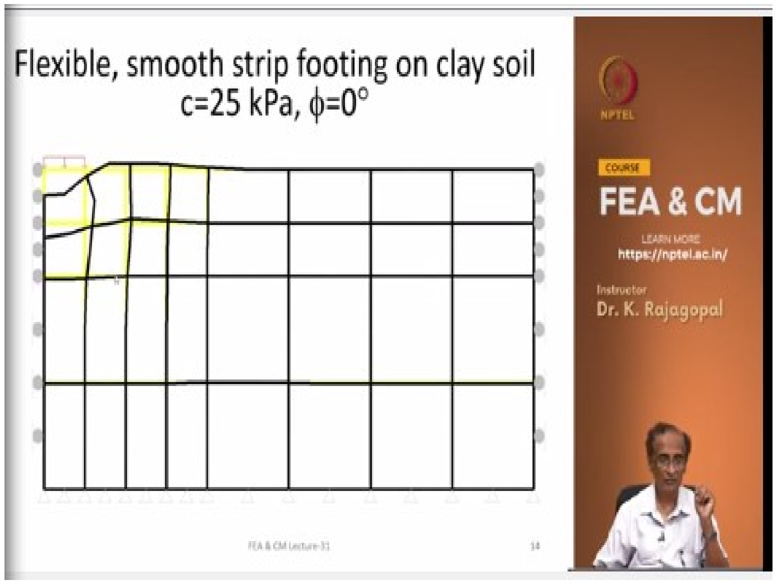


And here you see the deformation of the pattern of a flexibly flexible strip footing is actually it is a smooth footing. And if it is a flexible footing we assume that the pressure is constant all through the footing and what we do is we apply equal pressure. And here I want to illustrate we are considering only half the width of the footing and you see the roller boundary conditions along the mid section of the footing, this is to represent the symmetry in the response.

And on the right hand side also we have provided the roller boundary condition, so that you do not generate some spurious shear stresses. So, if you fix all these nodes because of this supplied loading the soil naturally would like to settle down or deform or instead of settling down it might even heave up. And if you fix these boundaries there might be some spurious shear stresses that could affect our results.

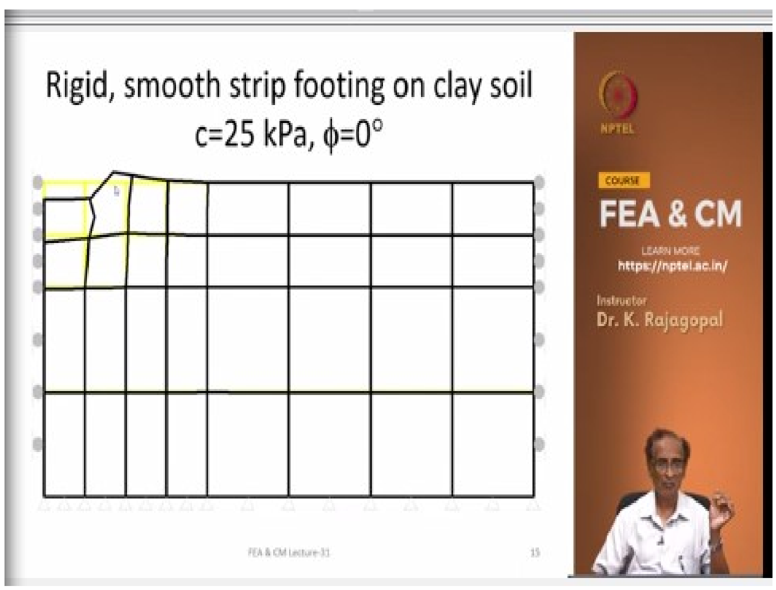
Then at the bottom now we assume that we have a rough rigid surface, so all the displacements on these nodes are fixed, so that is shown by a hinge. This is a typical roller at all these and the elements that are used are 8 node quadrilaterals and this particular soil is a clay soil with a c of 25 and phi of 0. And you can actually see the pattern of the circular slip surface, if you are phi is 0 your slip surface will be more like a circular slip surface.

(Refer Slide Time: 24:32)



And this is your flow vector, these are the flow vectors and this is your deformation pattern. The yellow line is your original mesh and then this black line is the deformed mesh.

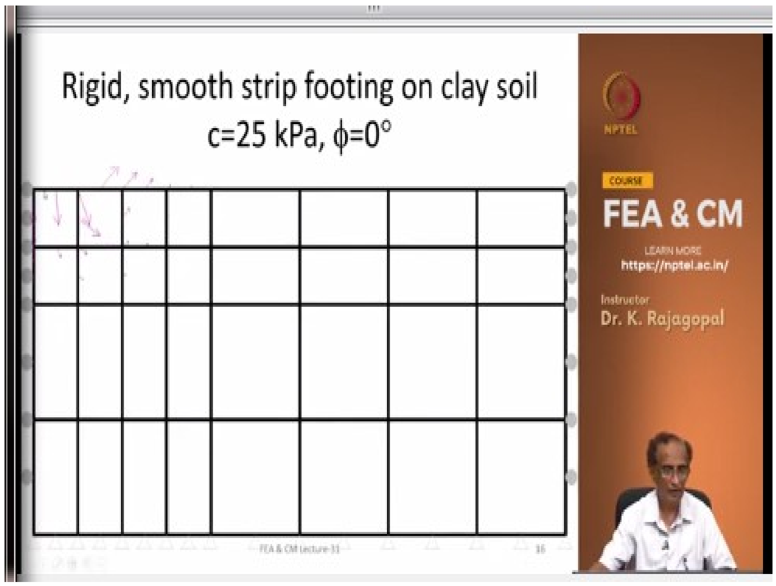
(Refer Slide Time: 24:49)



And this is the rigid footing, when we have a rigid footing we apply equal vertical displacements. There are 3 nodes on the surface, so we have applied equal vertical displacement. And because it is a smooth footing we are allowing these nodes to deform laterally, so this particular node what was there has deformed this position. Because we are allowing the nodes to move freely as you are compressing in one direction the material will expand in the other direction that is what we see here.

In fact through these vectors you can see this particular node, it is not only moving vertically but also moving laterally, so this is the resultant direction. And all these vectors they represent the direction of the displacement and they are all scaled vectors. So, the maximum length is representing some length that I think I did not plot here but when we see the post processor I will show you that.

(Refer Slide Time: 26:12)

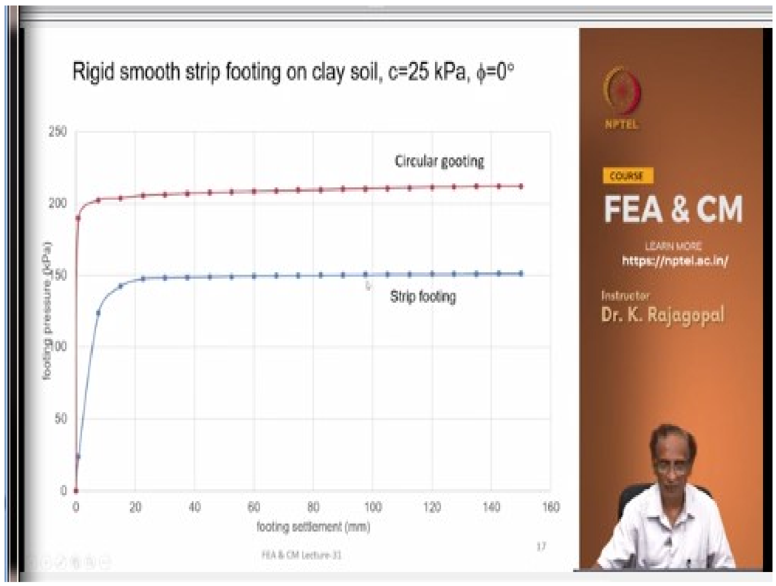


And these are the flow vectors for a rigid footing. And you can see under flexible footing you have one deformation pattern and then under a rigid footing you have different pattern. Because actually when we are deforming the soil this is the deformation pattern, we get a similar pattern but the effect of load is distributed for a longer distance because once you apply the pressure it has to disperse.

But if you apply displacement the effect may not disperse deep because if the soil elements here they enter the plasticity the effect of that is not seen here. So, that is why you see most of the deformations are confined to only the soil around the footing. And just for the purpose of quick calculations I have used a very coarse mesh and the results that you get with this may not be very accurate. And since I am going to run this program in front of you and I want this to run quickly, I used only a small mesh.

But we might get slightly different results if you take a much finer mesh. So, we see the effect of a load control analysis, with the load control once you apply the pressure on the surface, so it has to naturally spread to a longer depth and width because the pressure has to disperse. Whereas with the displacements we represent a better distribution because of the soil elements around the footing fail then you get a localized slip surface that is what we see here.

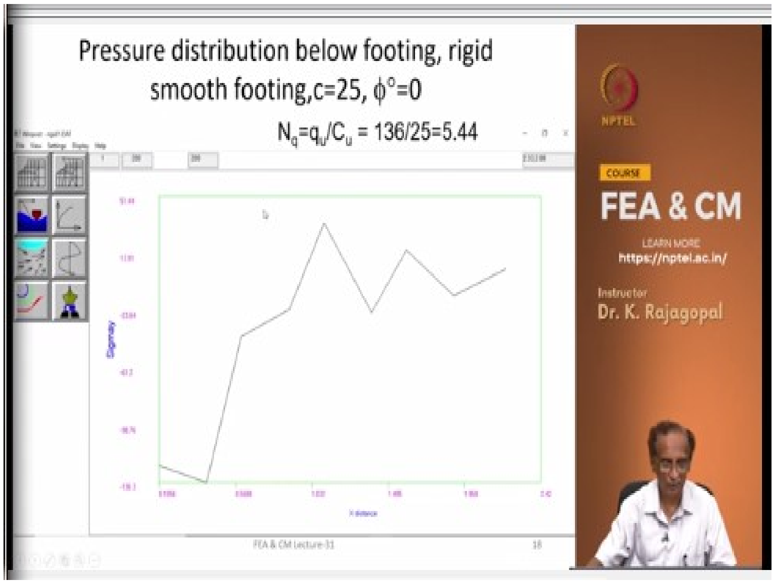
(Refer Slide Time: 28:17)



And these are the pressure settlement graphs for a strip footing and a circular footing. And for a strip footing it is predicting about 150 kPa and so your N c factor for strip footings about 6 is actually as per our Terzaghi's theory, it is 5.7 whereas as per Vesic's theory it is 5.14. So, it is not bad like actually we have used a very, very coarse mesh but still we got value N c very close to the to about 5.7.

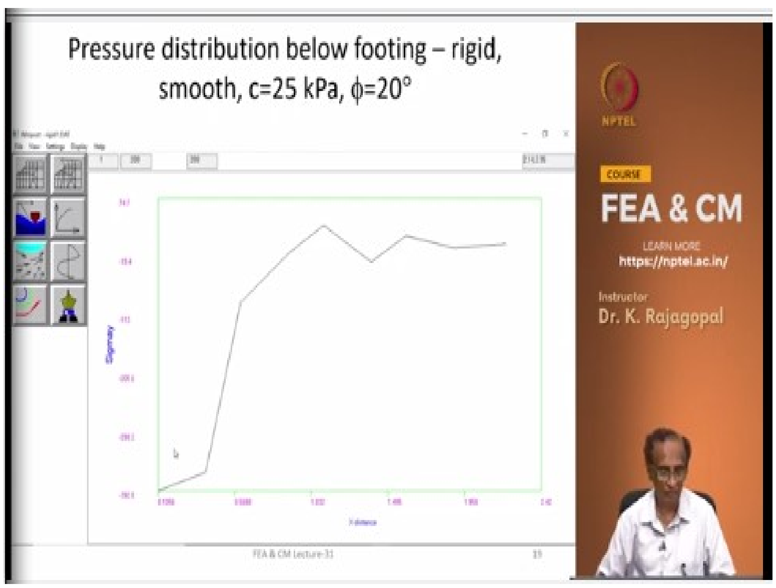
And we also know that circular footing will have a higher bearing capacity by about 30 percent. So, if you see here this is the bearing pressure graph or a circular footing for the same properties of c of 25 and phi of 0. It has reached an ultimate pressure of about 210, so your N c factor is more and that shape factor of about 1.3 for circular footing is because of the higher bearing pressure that you get because of this 3 dimensional load distribution.

(Refer Slide Time: 29:46)



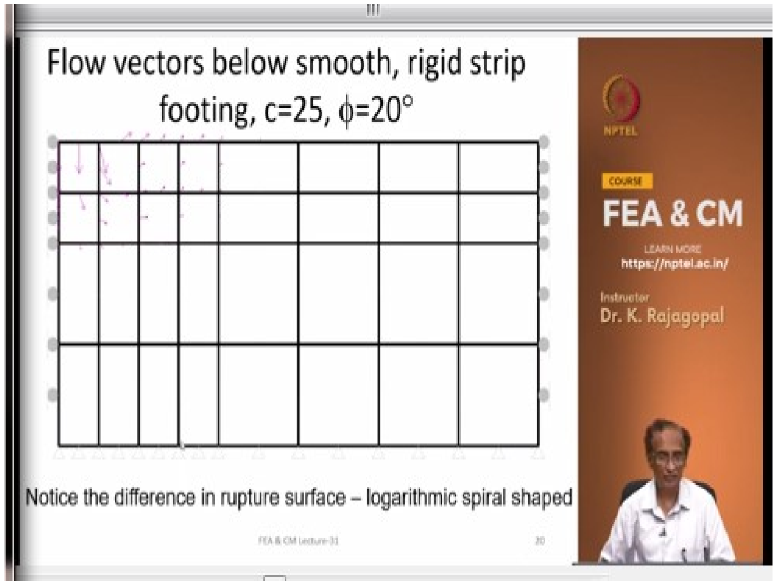
And this is actually, I will show you this graph, actually within a mesh we can choose any line and plot how the pressures change that I will demonstrate when we go to the post processor.

(Refer Slide Time: 30:10)

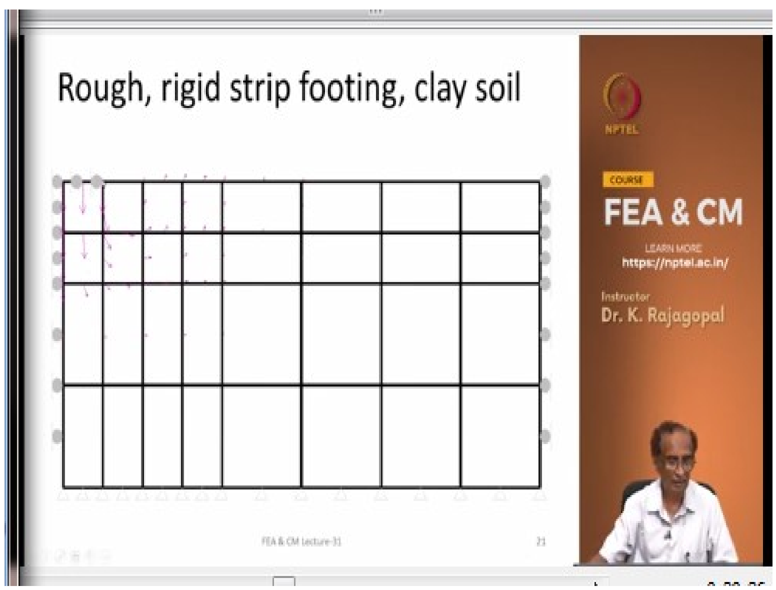


And this is the rigid footing is actually what has shown is for a actually this is for a rigid footing also and this is for a rigid the smooth footing.

(Refer Slide Time: 30:26)

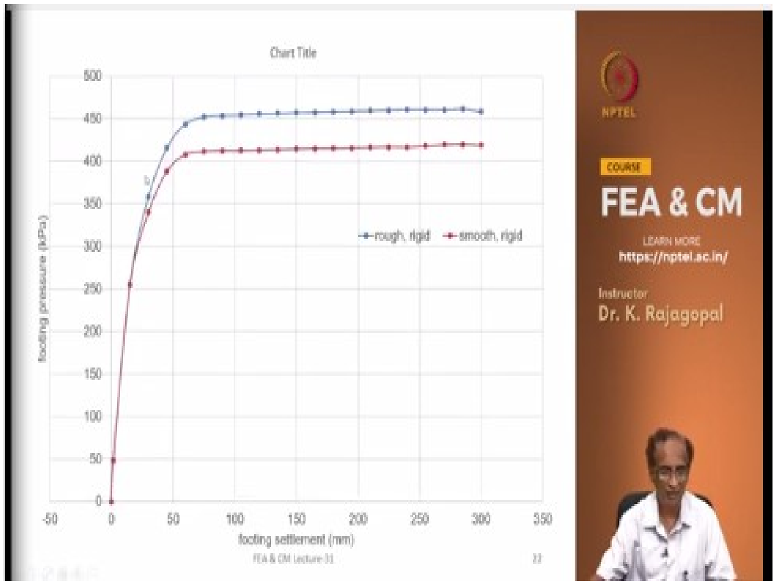


(Refer Slide Time: 30:32)



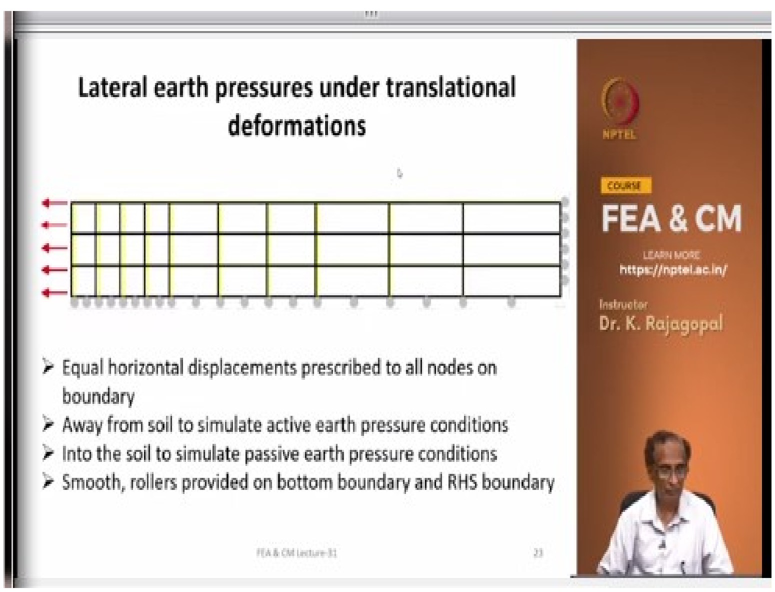
And this is for a rough, rigid footing, actually these 3 nodes they correspond to the footing and we have applied equal vertical displacement to all these 3 nodes. And then we constrained all these 3 nodes from moving in the horizontal direction and that is what we see here. It is actually we see the roller here, so that means that this particular node will move only in the vertical direction and not in the lateral direction, that is why you see this vector is going down.

(Refer Slide Time: 31:09)



And the effect of smooth and rough footings, this is the rough footing and this is the smooth footing. And our Terzaghi’s theory and Meyerhof they all consider a rough rigid footing, so the finite element results that we get with roughness they are closer to the reality that is our bearing capacity theories. And with the smooth footings you get a slightly lesser pressure.

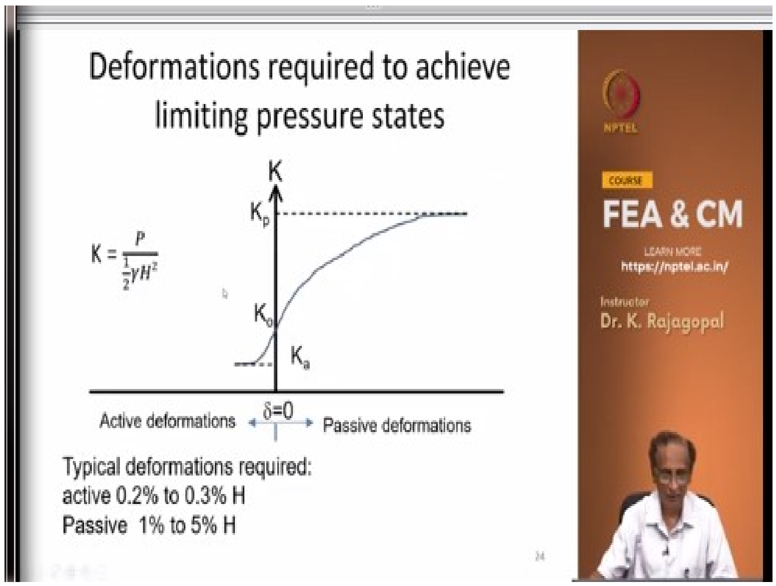
(Refer Slide Time: 31:43)



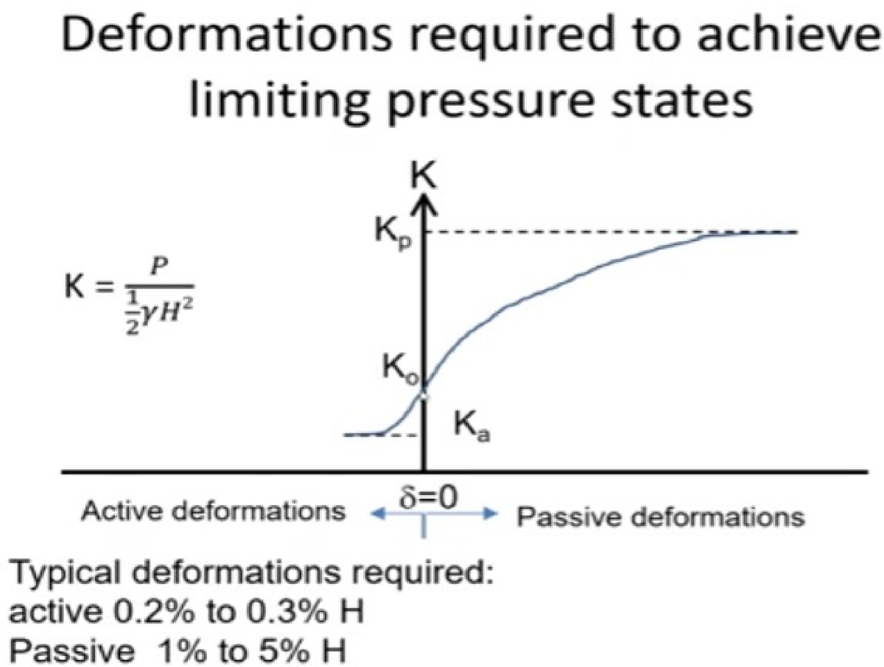
And before we go into the finite element demonstrations let us also look at the lateral earth pressures and we will simulate only the Rankine's values both active and passive. And Rankine has assumed the lateral translational type deformations and we can simulate this by placing some depth of soil and then we place the rollers all along the horizontal surface, so that we have only a smooth lateral deformation.

And then we take the boundary at a far off place and then place rollers on them and we can apply equal horizontal displacements along the surface. And if you apply these displacements away from the soil we are going to generate active pressures and if you apply these displacements into the soil we develop passive pressures. And the smooth rollers are for simulating the smooth boundaries.

(Refer Slide Time: 33:01)



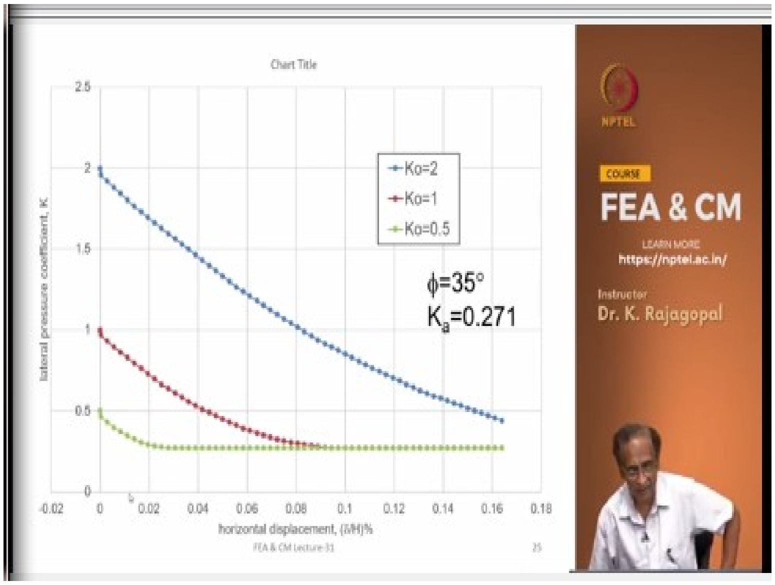
And from our earth pressure theory we know that if you deform the wall away from the soil you develop some active pressure state. And if you push the wall into the soil you develop passive earth pressure state and the magnitudes of deformation that we require for active state are very small whereas for passive pressure they are very large of the order of about 1 to 5 percent depending on the type of soil.



And when we simulate this through finite element analysis we can estimate the lateral pressure constant K as the P is the sum total of the reaction forces on the nodes where we applied the deformations. The sum total divided by 1 half gamma H square will give you the K. And this is a simple thing because in all these programs there will be a post processor that will let you do these calculations very easily.

And typical deformations for generating the active earth pressure state are about 0.2 to 0.3 percent of the wall height, very small deformations whereas passive state is achieved at a very large deformation 1 to 5 percent depending on the friction angle and so on. And also even the K naught value could affect the required deformations.

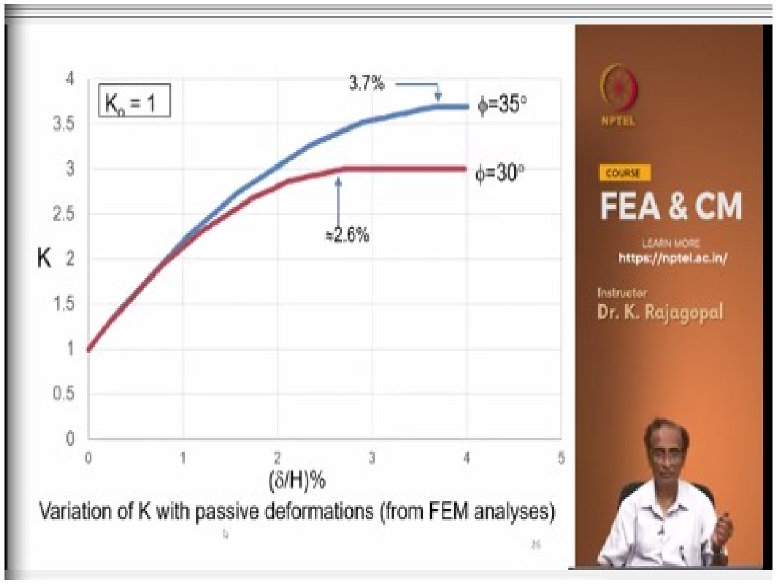
(Refer Slide Time: 34:47)



And here I am showing you the variation of K with lateral translational deformation for 3 different cases. And all these cases they correspond to a phi of 35 degrees and the c of 0 and the K a is 0.271. And this is the result, this green line is the result with a K naught of 0.5 and this red line is with a K naught of 1 and in both the cases we see that with the K naught of 0.5 we reads this limit state of 0.271, the K of 0.271 at lateral deformation of about 0.04 percent of the wall height.

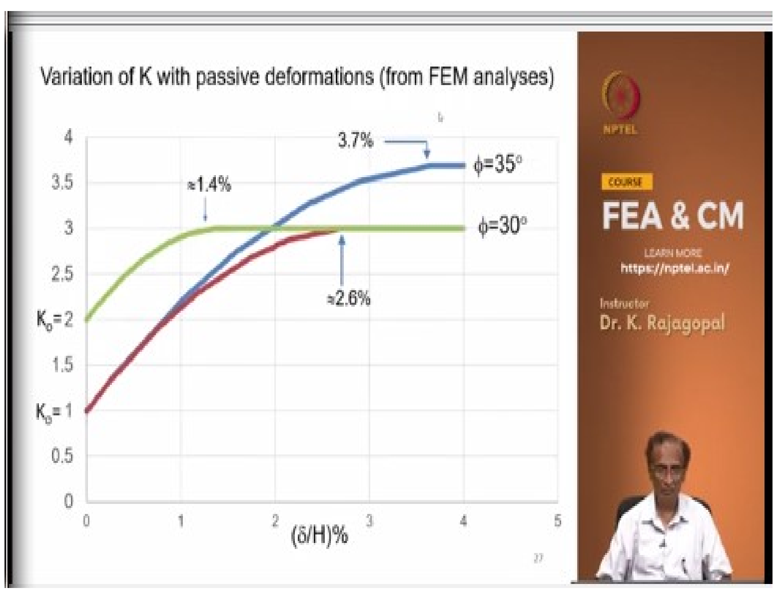
Whereas when K naught is 1, you require a much larger deformations of the order of about 0.09. And if we take K naught of 2 even after about 0.16 percent we have not still reached the limit state, it is still far away from the limit state. But if you continue further then we might reach the limit state at a very large lateral deformation. And we see that all the deformations that we are dealing with for the active state are very small about 0.1 percent, 0.0 5 percent, that also depends on the Young's modulus and then the K naught.

(Refer Slide Time: 36:25)



And this is you know the plot for passive pressure and for 2 different friction angles phi of 30 degrees and 35 degrees. And when phi is 30 degrees the K p is 3 and when phi is 35 degrees K p is 3.69 and the initial starting point K naught is 1 for both the cases. And this red line is for friction angle of 30 degrees, it has reached at the limit state of K of 3 at a deformation of about 2.6 percent whereas when friction angle is 35 degrees we require about 3.7 percent. Because with a higher friction angle we generate more pressure, so that requires more deformation.

(Refer Slide Time: 37:34)



And here we see the effect of the K naught, this green line is corresponding to 30 degrees of friction angle and this red line is also corresponding to 30 degrees friction angle with a K naught of 1 we have seen earlier that you require almost 2.6 percent of the wall height deformation to reach the K a state. But if your K naught is 2 initial K naught itself is high closer to 3, you required only 1.4 percent wall deformation.

Because already the K naught of 3, so you might think if what if K naught is 3? Then initial itself for the soil will be in passive state and with further increase of the lateral deformations the pressure state will not change, it will remain at the K naught of 3. And this is for 35 degrees with the K naught of 1, we require about 3.7 percent wall deformation. So, these are some limit solutions that we can easily derive based on our finite element analysis.

And let me now open the finite element program and the data files and show you what exactly we are going to do. This is the end of my power point presentation but we will come back to here. **(Video Starts: 39:18)** See this GEOFEM is the program that you will get this is the geotechnical finite element modeling and this is only a 2D program and there is another version GEOFEM 3D.

Mind if you are comfortable using it I do not mind sharing that program also for three dimensional analyses. And let me just show you one, this is the data file for generating active pressure state and I have taken a very small mesh of 170 nodes. And here when you give the material point data there is the last value that you give is the K naught. So, this is the K naught of 2.

And then we are going to demonstrate the active pressure case and let me just run this for one case. And I will demonstrate the run with mod x of 1 that is we are going to run with the K naught of 2 and that will give us a Poisson's ratio 2 by 3 that is more than 0.5. So, let me initially the lateral deformations are 0 that is what the boundary condition value we are prescribing the negative 0.01 that is about 0.3 percent of the wall height.

And this is an active deformation it is moving away from the soil, so let me run this. So, I am just generating the result and this is the result that we have let me zoom in. So, this particular one was this stress condition is 2 that is the plane strain case and our internal friction angle is 35 degrees and dilation angle is neglected 0 and then the earth pressure constant is 2. So, if we look at and at the end of the first stage all the displacements are set to 0.

I have not printed all the displacements but you can see here say the displacement x and y everything is 0. And here if you say sigma X is -111.55 whereas sigma YY is 55.77 and this multiplied by 2 is your 111.55. And we can actually see this so this is our mesh and we can display the boundary conditions, these are the boundary conditions and then let me show you. So, these are the vectors of the principal stresses.

So, you see the length of this horizontal vector is long compared to the vertical one is almost double because our K naught is 2. And you see the maximum compression stress is 111.5 and this tension it is printing but there is no tension in the soil because it is under K naught conditions. But then this is a spurious value that it has to show something and our displacements are 0 because we have reset all the displacements to 0.

And now let me do this, let me restart the program with a mod X of 2 and let me just run for about 0.05 deformation. And this will take some time and I am using another form of 1000, so that the stiffness matrix is not reformulated. And the analysis will go very, very fast, so that is what we see here because the stiffness matrix is not updated. So, it is actually on my laptop it will run much faster but since I am running on a vacuum tablet, it is not so fast.

So, I have totally 1200 steps, so it will take some time maybe another 1 minute, it is 700, 800, it will run fast 900, it is almost there, here 1200, okay, right it is finished. So, now let me show you active hits, we need to first set the load step at least once you have to do, so that you will be able to generate the or show the mesh and so on. And let me just set the current load step to the last value 1200 and then this is the deformed mesh and let me just magnify the deformations a bit.

See this the yellow line is the original one, the black one is the deformed mesh and we can see the vectors. So, we have applied horizontal deformations, equal horizontal deformations to all the nodes on this boundary. And let us look at one of the stress components, see these are all the integration points then I am going to select one at this point and it is going to plot the lateral earth pressures at that particular point. And as the wall is deforming the pressure has reduced initially it is about 111 but as the wall is deforming this is the x axis you have the epsilon X and it is a tensile strain because we are extending.

And then at some point it has reached the constant stress we do not know what is the K because this divided by sigma Y will be your K. And if you look at this is the sigma X and I am going to plot sigma Y and the sigma Y should remain constant. So, that is what we see here, the sigma Y is remaining constant at the gamma z. And although we are applying lateral strain the sigma z is the remaining constant.

And I want to show you one feature of the program, let us say we have plotted this and we can actually generate a PDF file of this deformation like you can go to print. And this particular program is windows compatible, so if you connect any printer to your computer you can print it or you can generate the PDF file. Oh God! I think not sure whether let me try PDF annotator. And then it will be sent to a PDF file I am not sure where it is?

I think it is still working. Here is the PDF file of your deformed plot and you can also export it as a meta file, export meta file and then you can give some file name, so all these are meta files that are created later you can import them into MS word or into power point for preparing your slides. You can give any name that you want and you can save it and so on you can do all these things.

And there is another feature that helps us in quickly summarizing the results, this is you press on this pen and then you can create a summary file with only the selected results. See we have done the analysis and we want to know how the earth pressure constant is changing with the lateral deformations. And here it is prompting you to all the data that you generate through this will be saved in a file and let us say I want to save it as an active and say translation dot, dot, dot.

And then the search radius, see the program will prompt you to select some nodes and select some displacements. And when you float the cursor you may not be able to exactly match the node coordinates, so within a tolerance of 0.1 it will search the nearest node. So, that is such radius 0.1, I am saying yes. And then the width of the footing this option is actually you can enter 90 1 half gamma H square our gamma is 20 and the height of the wall is a 3 that is 1 half 20 times 9 is 90.

So, if you give this as 90 and then the direction for the reaction forces we are applying the displacements in the horizontal direction that is the direction 1, so you choose it as 1. And then it will sum up all the reaction forces at different steps and then divide by 90 to calculate our earth pressure constant, so okay. And then we are interested in knowing how much was the deformation, so I am floating the cursor and then sticking.

And if you have any interface at the interface you could have possibly 3 nodes with the same coordinates. So, the program is asking which of these nodes you want? Node 1, first node, second node, third node I am just saying 1. And then the degree of freedom of that particular displacement we are interested in the horizontal deformation, so the degree of freedom is 1. And then it is asking whether you want to save any other nodal point data and let us say we select another node and once again 1.

And then let us say we want to store the vertical deformation, so I am saying it as 2. And then we can say no, and then there is another option for contour plotting. So, if you press this you can give some file name and then it will ask you what is the range of the contours that you want to generate and then what is the X and Y axis range and then let us say it is giving a minimum X of 0.21 let us give it as 0.2 and maximum X is 19, let us say 20 and minimum Y is 0.2 and say I give maximum Y as 3 and then number of grid points.

And then actually this program is going to generate the data in the XYZ format, X and Y are the nodal coordinates and Z is the contour value. And it can interpolate by providing a grid of values. Let us say along the x axis I wanted 20 points and then along y axis I need at 10 points, okay you say yes. And then the contour lines let us say 10 and I think this is not working I do not know, maybe I do not know how to use this.

I am just giving contour values, so let us open our file that we have generated, this is the summary file that we have generated, let me zoom it a bit. See these are the number of load steps that we have run up to 1200 steps and then our initial K naught is close to, so it is 1.98. And our lateral deformation at node 1 at X coordinate of 0 and Y of 3, this is the X direction displacement.

And then at some other point we have selected Y direction displacement, then these are the displacements with the load step. We have applied lateral displacement of up to 0.50mm. And the X direction the K factor that is the P divided by 1 half gamma H square to start with it is close to 2 because we applied some lateral deformation this is not exactly 2, so it is closed to 1.98, it has reduced 1.8, 1.63 and so on.

And then finally at some point it has reached a 0.274 and for a friction angle of 35 degrees the K should be 0.2714. So, if you deform a little more we might reach the active limit state? But in this particular case because the K naught is very high, you require very large deformation. So, by doing this we can save some important data and then we can import this into an excel file. Let me show you how to do that.

These columns can be directly transferred into the excel file, not sure whether we have excel here. I think the excel program is not here, what we do is we can take this and separate all the columns by tabs or there is a tab in between these 2 columns, so you can use the paste special and then use the tab as the separator between the columns and then the program the excel will be able to easily download this into the excel and then you can select the columns and plot graphs similar to what I have shown here. **(Video Ends: 1:00:08)**

So, all these graphs they are plotted by importing the data from the finite element program into the excel program and then after that we can manipulate in this usual manner that we do and we can plot the graphs. So, I think I hope you appreciate how easy it is to generate limit solutions. And in the next class I am going to demonstrate some more cases like the case of eccentrically loaded the footing and so on. And there is another program called as a preprocessor that will let you generate the meshes.

And I may not be able to demonstrate but I will give you the instructions on how to generate the data files. So, that you do not need to go through the user manual and then enter the data manually. So, that I will do that through some instructions and you just follow the instructions and it is very easy, all these programs are very easy to run. And if you have any doubts or if you encounter any problems in the process you send an email to this address profkrz@gmail.com then I will respond back to you. So, thank you very much.