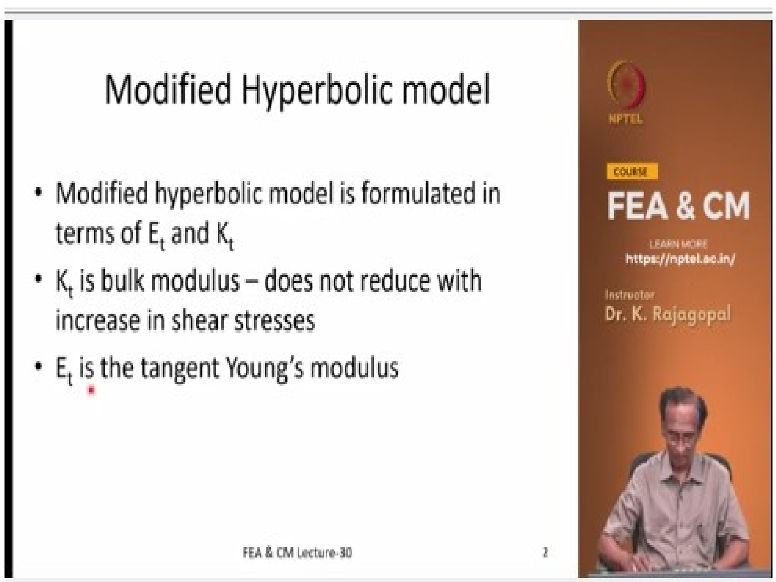
Finite Element Analysis and Constitutive Modelling in Geomechanics
Prof. K. Rajagopal
Department of Civil Engineering
Indian Institute of Technology-Madras

Lecture-34
Numerical Examples on Working with modified hyperbolic models

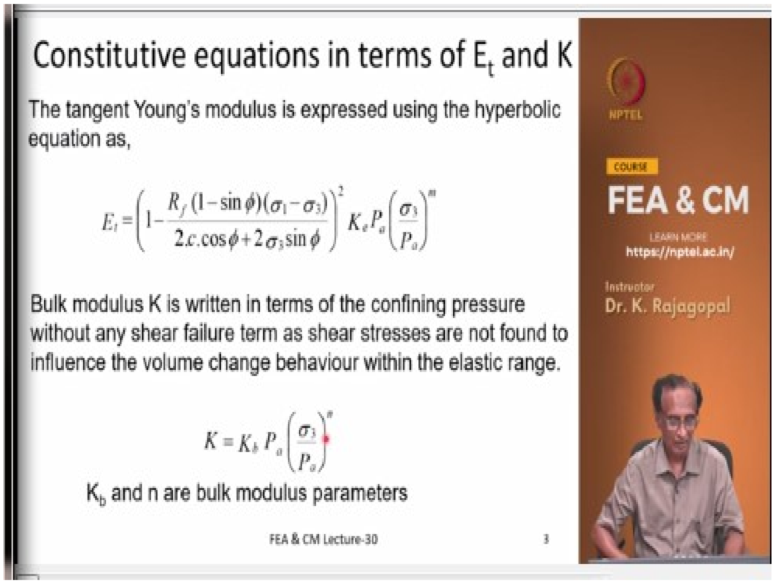
So, hello students let us continue our discussion on the modified hyperbolic model. In the previous classes we had seen some theory behind the modified hyperbolic model which was formulated in terms of tangent Young's modulus and then the bulk modulus. And I have also shown you one excel spreadsheet program. And in today's class let us do step by step calculations and then also try to implement our stress correction procedure that we had learned in the previous class.

(Refer Slide Time: 00:56)



So, just refresh our modified hyperbolic model is written in terms of the tangent Young's modulus E t and tangent bulk modulus K t. And the K t is the bulk modulus and that does not reduce with the shear stresses. Because the bulk modulus is only influencing the bulk stresses and then the bulk behaviour like the volumetric strains that are not affected by the shear stresses. So, the shear stress reduction is not applied on the K t but our Young's modulus E t is affected by the shear stresses.

(Refer Slide Time: 01:40)



And our tangent Young's modulus E t is written in terms of this shear stress basically this R f multiplied based by this term, this term we have seen that it represents the ratio of the mobilized shear strength and at the limit it is going to become 1. And this whole square multiplied by initial modulus that is K eta P a times sigma 3 by P a to the power m and then we have the other bulk modulus K that is K b P a times sigma 3 by P a to the power n.

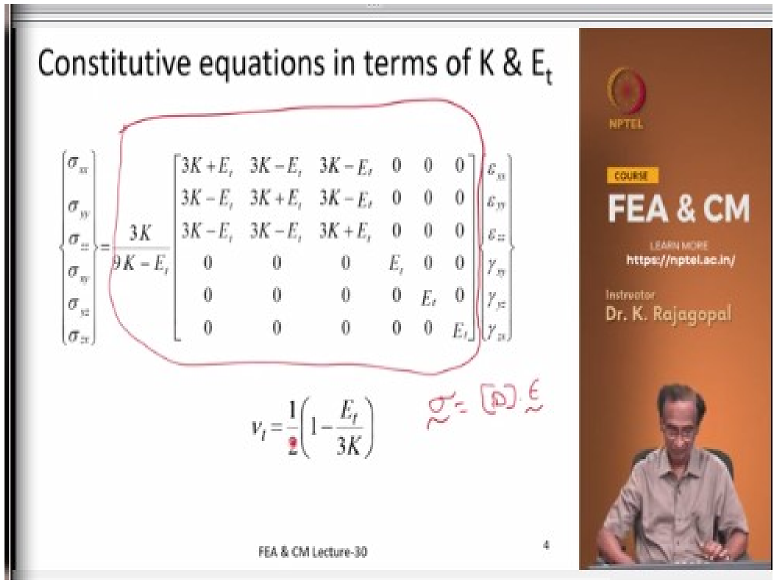
The tangent Young's modulus is expressed using the hyperbolic equation as,

$$E_t = \left(1 - \frac{R_f (1-\sin\phi)(\sigma_1 - \sigma_3)}{2.c.\cos\phi + 2\sigma_3 \sin\phi}\right)^2 K_e P_a \left(\frac{\sigma_3}{P_a}\right)^m$$

Bulk modulus K is written in terms of the confining pressure without any shear failure term as shear stresses are not found to influence the volume change behaviour within the elastic range.

$$K = K_b P_a \left(\frac{\sigma_3}{P_a}\right)^n$$

(Refer Slide Time: 02:30)



And combining these 2 we can write our constitutive matrix, that is the sigma is a D times epsilon the constitutive matrix D is written like this.

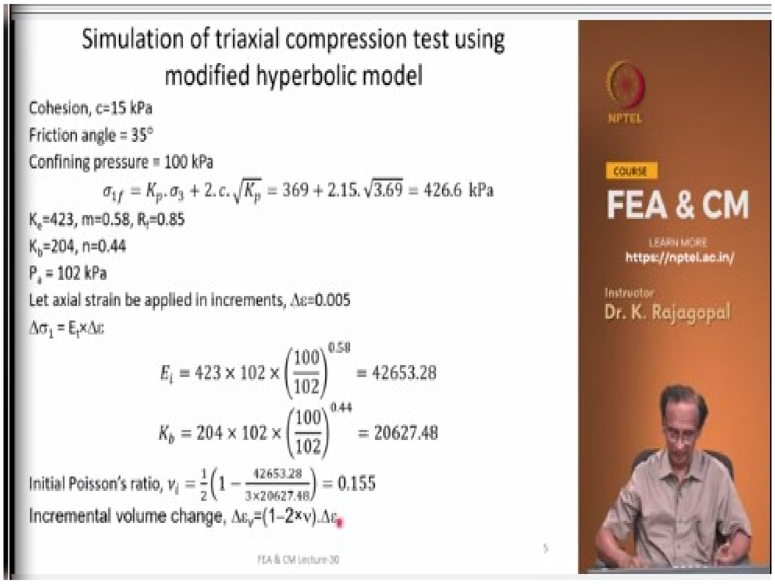
$$\begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{yz} \\ \sigma_{zx} \end{Bmatrix} = \frac{3K}{9K-E_t} \begin{bmatrix} 3K+E_t & 3K-E_t & 3K-E_t & 0 & 0 & 0 \\ 3K-E_t & 3K+E_t & 3K-E_t & 0 & 0 & 0 \\ 3K-E_t & 3K-E_t & 3K+E_t & 0 & 0 & 0 \\ 0 & 0 & 0 & E_t & 0 & 0 \\ 0 & 0 & 0 & 0 & E_t & 0 \\ 0 & 0 & 0 & 0 & 0 & E_t \end{bmatrix} \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix}$$

$$\nu_t = \frac{1}{2}\left(1 - \frac{E_t}{3K}\right)$$

This whole thing is our constitutive matrix, our sigma D times epsilon, D is our constitutive matrix. And the advantage that we can relate the Poisson's ratio nu to our Young's modulus and bulk modulus like this. And as your shear stresses are increasing your Young's modulus is going to reduce as you are reaching the peak stress.

And as the Young's modulus is reducing our Poisson's ratio will tend towards 0.5 and that is the Poisson's ratio of 0.5 at the critical state when the volume remains constant.

(Refer Slide Time: 03:51)



So, let us simulate a test for these properties, let us say that our soil has a cohesive strength of 15 kPa and friction angle of 35 degrees and the confining pressure is 100 kPa and the sigma 1f is K p times sigma 3 + 2 c square root K p and that comes to 426.6. That is the maximum axial stress that you can apply in triaxial compression test. And let us take these hyperbolic properties K is 423 and the exponent for the Young's modulus is 0.58 and the failure ratio R f is 0.85 K b and then and atmospheric pressure 102 kPa.

Simulation of triaxial compression test using modified hyperbolic model

Cohesion, c=15 kPa

Friction angle = 35°

Confining pressure = 100 kPa

$$\sigma_{1f} = K_p.\sigma_3 + 2.c.\sqrt{K_p} = 369 + 2.15.\sqrt{3.69} = 426.6 \text{ kPa}$$

K_e=423, m=0.58, R_f=0.85

K_b=204, n=0.44

P_a = 102 kPa

Let axial strain be applied in increments, $\Delta\varepsilon$=0.005

$\Delta\sigma_1 = E_t \times \Delta\varepsilon$

$$E_i = 423 \times 102 \times \left(\frac{100}{102}\right)^{0.58} = 42653.28$$

$$K_b = 204 \times 102 \times \left(\frac{100}{102}\right)^{0.44} = 20627.48$$

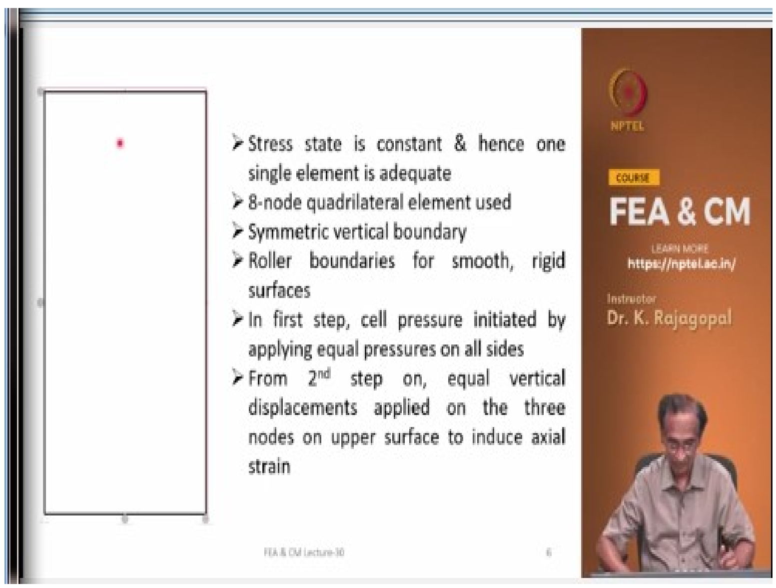
Initial Poisson's ratio, $\nu_i = \frac{1}{2}\left(1 - \frac{42653.28}{3\times20627.48}\right) = 0.155$

Incremental volume change, $\Delta\varepsilon_v=(1-2\times\nu).\Delta\varepsilon_z$

And let us apply axial strain increments in increments of 0.005 and delta sigma 1 is E t times delta epsilon, E t is the tangent Young's modulus multiplied by this strain increment. And our initial modulus is K e times P a times sigma 3 by P a to the power m, that is 42653 and the K b is this. And this E is going to change during the analysis because of our shear stresses whereas the bulk modulus is going to remain constant and the initial Poisson's ratio nu i is 1 half 1 - E i by 3 K b and that is 0.155.

And the incremental volume change at any stage is 1 - 2 nu times delta epsilon z, where delta epsilon z is the axial strain.

(Refer Slide Time: 05:54)



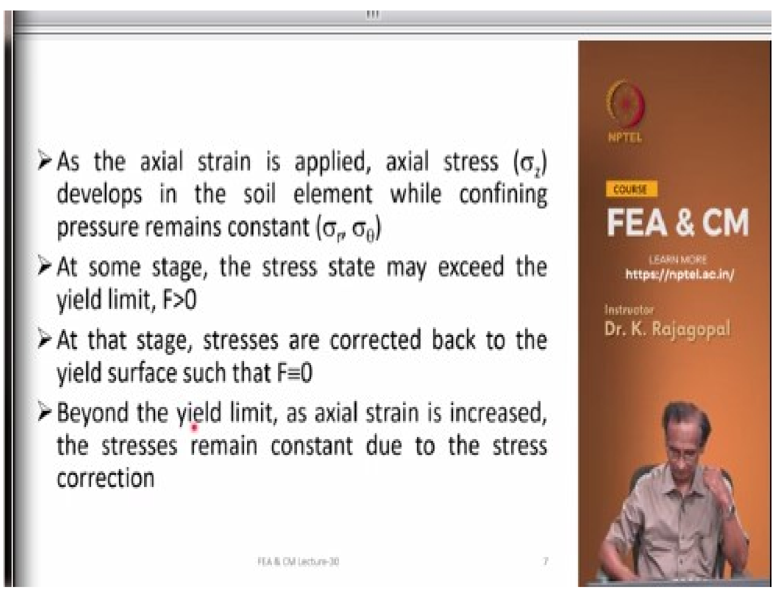
So, how do we simulate this triaxial test in finite element analysis? The first thing is we should realize that the stress state is constant, whether you look at a point here or here the stress state is constant and because of that we can just simply use a one single element for doing the simulation. And in this case I have used an 8-node quadrilateral element and I have used a symmetric boundary condition.

And this analysis is done using axis symmetric idealization because we have a cylindrical sample and then we applied all round pressure on a cylindrical sample. So, we can simulate it as an axis symmetric 1 and you see the symmetry boundary conditions on this vertical surface at half the diameter. And this is the roller and this point is common for both vertical and horizontal and horizontal surface is also supported on roller.

So, the roller boundary conditions are very common in geotechnical engineering wherever you have a smooth rigid surface. And in the first step we apply cell pressure by applying equal pressure in both the horizontal surface and vertical surface we apply the same pressure and initiate our cell pressure and then do the calculations. Say from second step onwards we apply equal vertical displacement at all the nodes on the vertical surface on the horizontal surface to induce the axial strength.

So, we can do this test either with the load control or displacement control but in the laboratory what we are doing is actually it is a displacement control. We are applying some axial strain and then measuring the reaction force through the proving ring and the same thing we can do in the finite element analysis. We can apply axial strain and then look at what happens to the stresses.

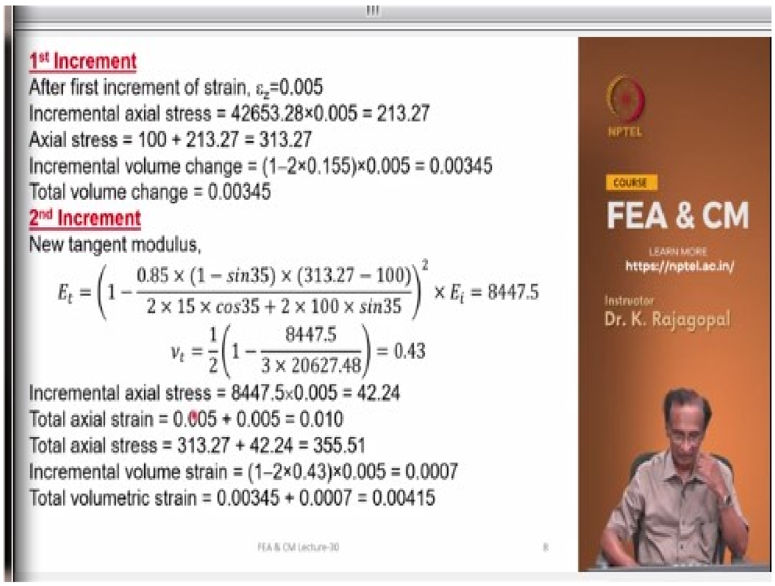
(Refer Slide Time: 08:22)



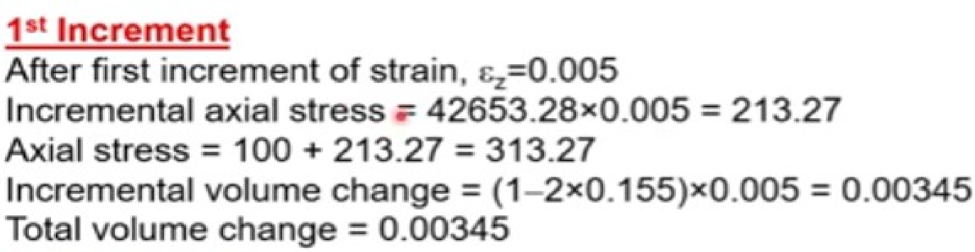
And as the axial strain is applied, the axial stress goes on increasing while the confining pressure remains constant. The confining pressure is the sigma radial and then the circumferential stress they should remain constant and during the analysis. Then at some stage our stress state makes a yield the limit that is when F greater than 0. Then at that state stresses are corrected back to the yield surface such that our F is exactly 0.

And beyond the yield limit as the axial strain is increased the stresses will remain constant are due to the stress correction. So, the lateral pressures will remain constant at the cell pressure whereas the vertical pressure initially they might increase then at some point they will remain constant.

(Refer Slide Time: 09:23)



So, that is what we can see we let us do the hand calculations first, so that we understand what is really happening. And in the first increment our yield, our tangent modulus is just simply E i and we applied an axial strain of 0.005 and the incremental axial stress is 213.27, this is the stress increment under an axial strain of 0.005. And the axial stress now is 100 + 213.27, 100 is the cell pressure and then on top of that we induce it as some additional stress of 213.27.



So, it is the total axial stress is 313.27 and the incremental volume change 1 - 2 nu multiplied by delta epsilon z that is 0.00345 and the total volume change at this stage, it is the same because there is no volume change before this, so it is 0.00345. Then in the second increment we are going to use an updated tangent modulus based on the axial stress of 313.27, so the E t is 1 - R f

0.85 times 1 - sine phi times sigma 1 - sigma 3 divided by 2 **c** cosine phi + 2 sigma 3 sine 5, this whole thing square multiplied by E i that comes to 8447.5.

2nd Increment

New tangent modulus,

$$E_t = \left(1 - \frac{0.85 \times (1 - sin35) \times (313.27 - 100)}{2 \times 15 \times cos35 + 2 \times 100 \times sin35}\right)^2 \times E_i = 8447.5$$

$$\nu_t = \frac{1}{2}\left(1 - \frac{8447.5}{3 \times 20627.48}\right) = 0.43$$

Incremental axial stress = 8447.5×0.005 = 42.24

Total axial strain = 0.005 + 0.005 = 0.010

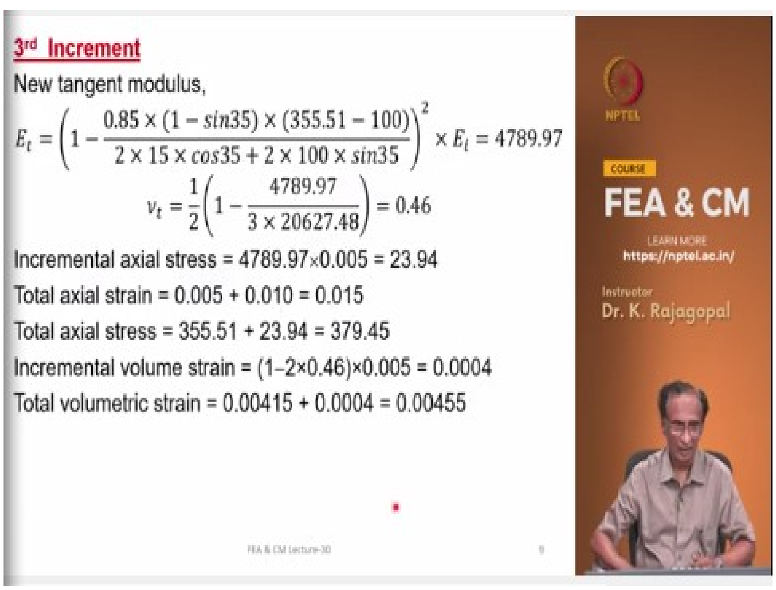
Total axial stress = 313.27 + 42.24 = 355.51

Incremental volume strain = (1–2×0.43)×0.005 = 0.0007

Total volumetric strain = 0.00345 + 0.0007 = 0.00415

And immediately we calculate are the tangent Poisson's ratio based on this and then the K b value and this Poisson's ratio comes to 0.43 and the incremental axial stress is E t multiplied by axial strain that is 42.24 and the total axial strain is 0.005 + 0.005 that is 0.01. And the total axial stress is 313.27 that is from the previous step plus 42.24 that is 355.51 and the incremental volume strain is 1 - 2 times nu t multiplied by axial strain that is 0.007 and the total volumetric strain is this 0.00415.

(Refer Slide Time: 12:18)



And then we can continue, send the third increment we are going to update our tangent modulus once again and it comes to 4789. Initially our tangent modulus was 42653, it has in decreased to

8447 and it has further fallen to 4789. And our Poisson's ratio previously to start with the Poisson's ratio was 0.155, it has increased to 0.43 and in the next step it has further increased to 0.47 and our incremental axial stress is 23.94, so our total axial stress is a 379.45 and this is our total volumetric strain and we can go on continuing.

3rd Increment

New tangent modulus,

$$E_t = \left(1 - \frac{0.85 \times (1 - sin35) \times (355.51 - 100)}{2 \times 15 \times cos35 + 2 \times 100 \times sin35}\right)^2 \times E_i = 4789.97$$

$$\nu_t = \frac{1}{2}\left(1 - \frac{4789.97}{3 \times 20627.48}\right) = 0.46$$

Incremental axial stress = 4789.97×0.005 = 23.94

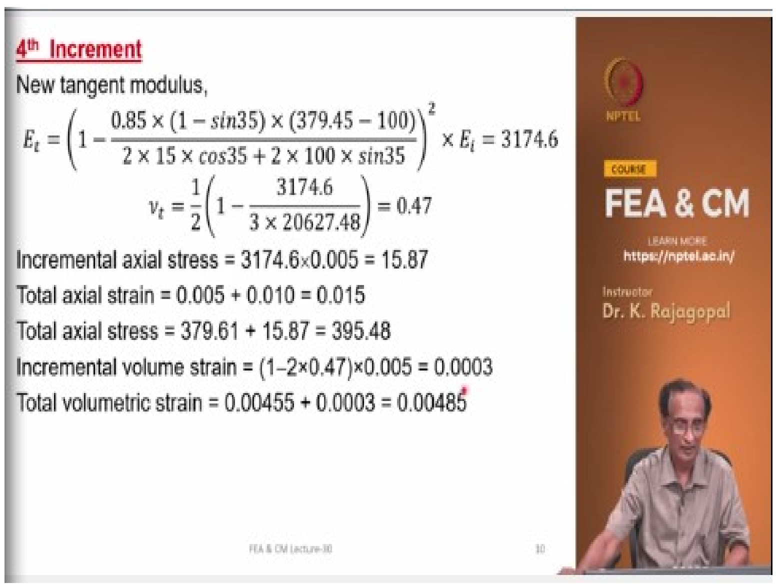
Total axial strain = 0.005 + 0.010 = 0.015

Total axial stress = 355.51 + 23.94 = 379.45

Incremental volume strain = (1−2×0.46)×0.005 = 0.0004

Total volumetric strain = 0.00415 + 0.0004 = 0.00455

(Refer Slide Time: 13:17)



So, our in the next step our tangent modulus has decreased to 3174 and the Poisson's ratio slightly increased to 0.47 and our axial stress is 395.48 and the incremental volume strain is 0.0003 and this is the total volume strain.

4th Increment

New tangent modulus,

$$E_t = \left(1 - \frac{0.85 \times (1 - sin35) \times (379.45 - 100)}{2 \times 15 \times cos35 + 2 \times 100 \times sin35}\right)^2 \times E_i = 3174.6$$

$$\nu_t = \frac{1}{2}\left(1 - \frac{3174.6}{3 \times 20627.48}\right) = 0.47$$

Incremental axial stress = 3174.6×0.005 = 15.87

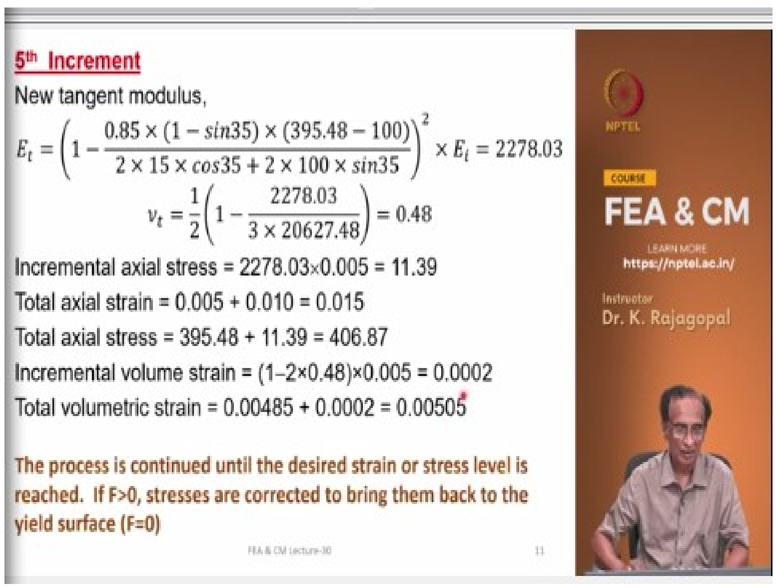
Total axial strain = 0.005 + 0.010 = 0.015

Total axial stress = 379.61 + 15.87 = 395.48

Incremental volume strain = (1−2×0.47)×0.005 = 0.0003

Total volumetric strain = 0.00455 + 0.0003 = 0.00485

(Refer Slide Time: 13:50)



So, in the fifth step our E t is a 2278, it has further reduced and then our axial stress is increased to 406.87 and this is our total volumetric strain. Then we can go on continuing and our axial stress will increase from 406.87 to something else and then it will go on increasing and at some point the yield stress limit of 426.6 is reached. At that stage what we do is we correct the stresses back and to bring them back to the yield surface.

5th Increment

New tangent modulus,

$$E_t = \left(1 - \frac{0.85 \times (1 - sin35) \times (395.48 - 100)}{2 \times 15 \times cos35 + 2 \times 100 \times sin35}\right)^2 \times E_i = 2278.03$$

$$\nu_t = \frac{1}{2}\left(1 - \frac{2278.03}{3 \times 20627.48}\right) = 0.48$$

Incremental axial stress = 2278.03×0.005 = 11.39

Total axial strain = 0.005 + 0.010 = 0.015

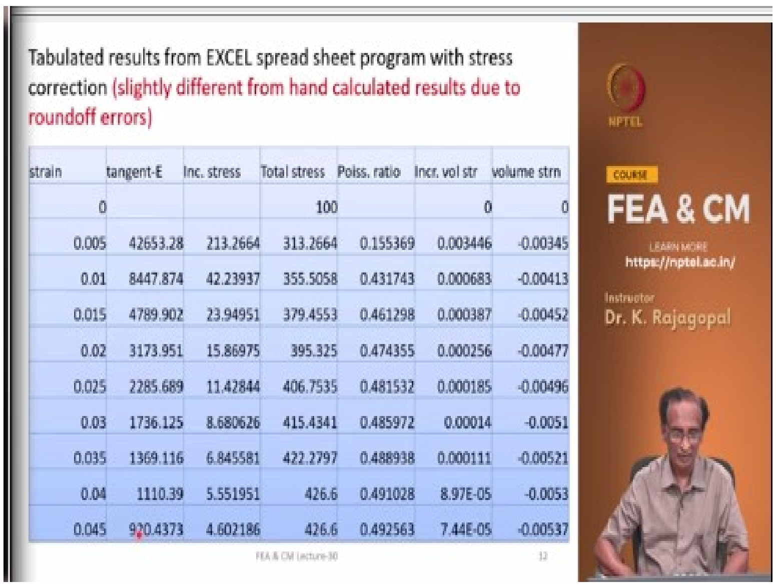
Total axial stress = 395.48 + 11.39 = 406.87

Incremental volume strain = (1–2×0.48)×0.005 = 0.0002

Total volumetric strain = 0.00485 + 0.0002 = 0.00505

The process is continued until the desired strain or stress level is reached. If F>0, stresses are corrected to bring them back to the yield surface (F=0)

(Refer Slide Time: 14:35)

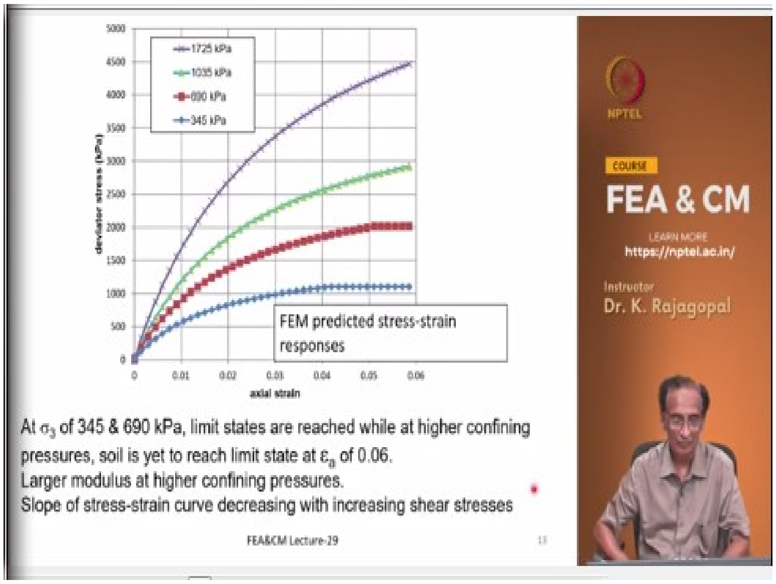


Tabulated results from EXCEL spread sheet program with stress correction (slightly different from hand calculated results due to roundoff errors)

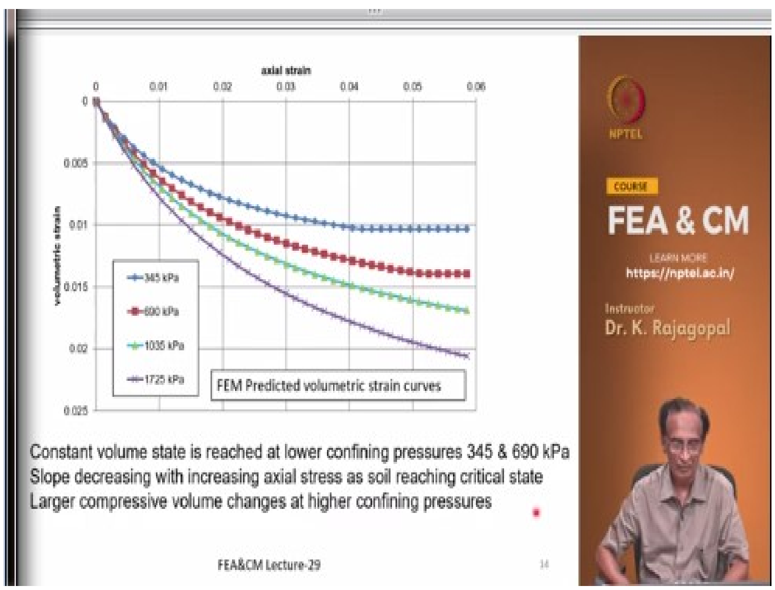
strain	tangent-E	Inc. stress	Total stress	Poiss. ratio	Incr. vol str	volume strn
0			100		0	0
0.005	42653.28	213.2664	313.2664	0.155369	0.003446	-0.00345
0.01	8447.874	42.23937	355.5058	0.431743	0.000683	-0.00413
0.015	4789.902	23.94951	379.4553	0.461298	0.000387	-0.00452
0.02	3173.951	15.86975	395.325	0.474355	0.000256	-0.00477
0.025	2285.689	11.42844	406.7535	0.481532	0.000185	-0.00496
0.03	1736.125	8.680626	415.4341	0.485972	0.00014	-0.0051
0.035	1369.116	6.845581	422.2797	0.488938	0.000111	-0.00521
0.04	1110.39	5.551951	426.6	0.491028	8.97E-05	-0.0053
0.045	920.4373	4.602186	426.6	0.492563	7.44E-05	-0.00537

And then we will see this, this is the result from excel spreadsheet, there is a slight difference between these values that you see here and the hand calculated values. But say the total stress initially it was 100 because that was the cell pressure 313, 355, 379, 395, 406 and so on. This is where we stopped with the hand calculations, then the next step 415, 422 then at 4 percent axial strain your stress exceeds the yield limit then it is corrected back to 426.6 and with any further increase it remains constant. And the Poisson's ratio will go on increasing and as the Poisson's ratio approaches 0.5 your volume strain remains constant at some point.

(Refer Slide Time: 15:38)

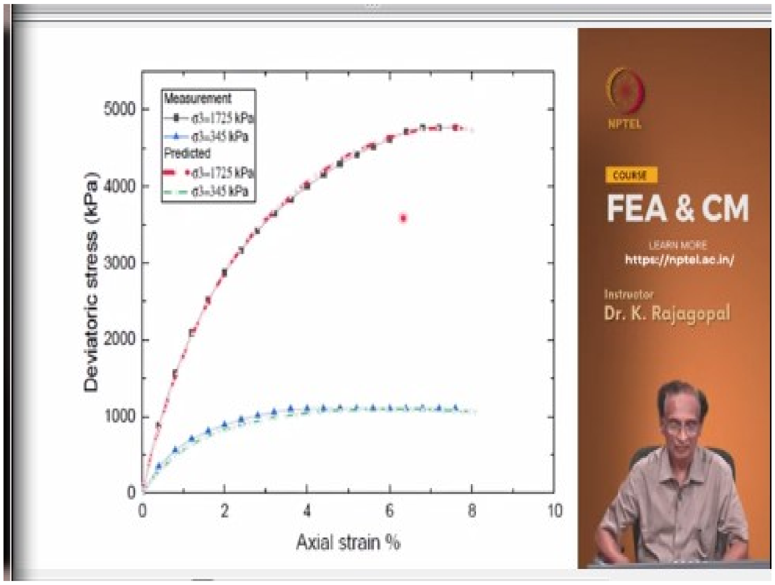


(Refer Slide Time: 15:38)

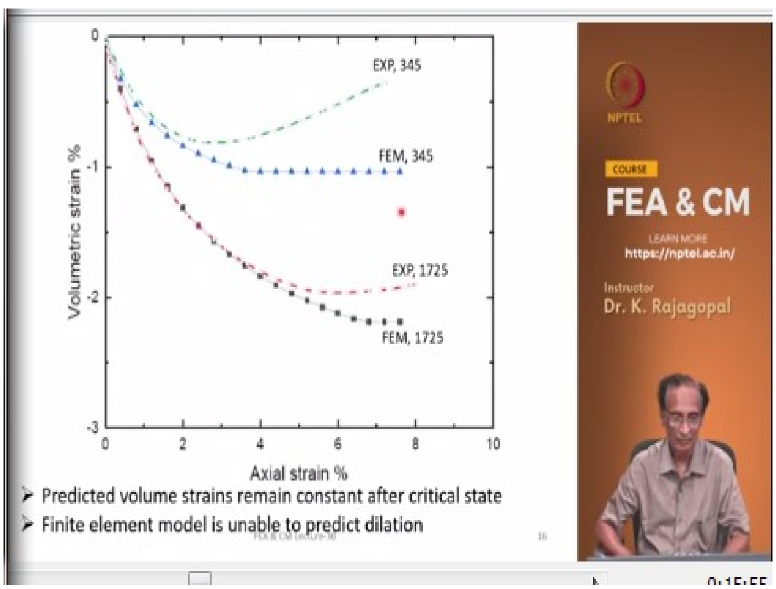


So, this is what we see here after some axial strain level, we reach the constant volume state.

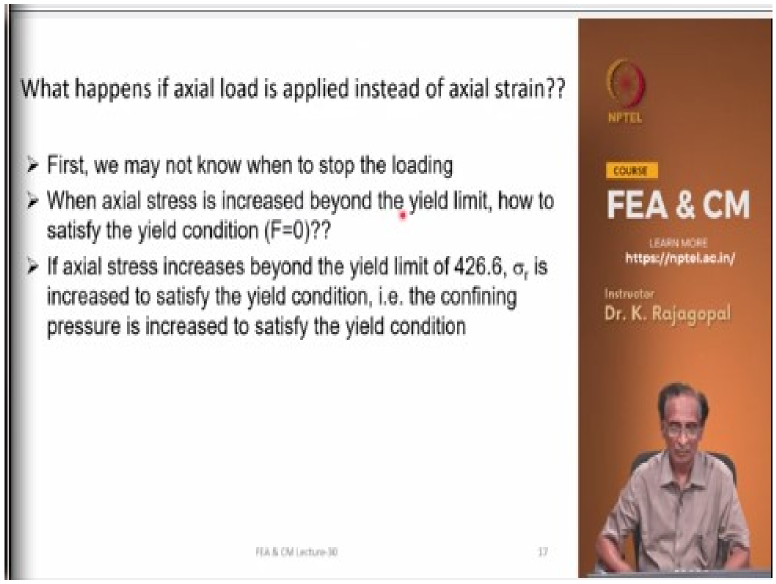
(Refer Slide Time: 15:52)



(Refer Slide Time: 15:53)



(Refer Slide Time: 15:56)



And let us say instead of applying axial strain we apply axial forces, what happens? See if we want to apply load, first of all we do not know how much load we need to apply. Because in some cases like our let us say you have a bearing capacity of a layered soil, we do not know what is the maximum pressure that the soil can take or the footing can take. And in that case we do not know what is the pressure that we need to apply and let us say we go on increasing the axial stress.

And at some point we apply a stress more than the yield limit and then the F value might cross the yield limit like might become greater than 0 and then what happens? So, since you are applying the vertical pressure load the stress, the vertical stress should increase but then if the vertical stress increases your yield function value will exceed 0, F will become greater than 0 and then what do we do?

So, the program or the stress correction method will increase the confining pressure slightly, so that you can satisfy your yield condition. So, actually when you apply the loading up to elastic limit your confining pressure will remain constant but beyond the elastic limit when you reach the limit state we will have a problem your lateral pressure will go on increasing and then the strain that you get could be infinite strain because suddenly you have failure., And then if you are not careful enough the entire program can reach a state where your most of the diagonal terms and the stiffness matrix are 0 because you have reached the plastic yield limit. And in that case your strains will become very, very large and your displacements also will become very, very large. So, let us look at this program, let me show you some**. (Video Starts: 18:53)** This is the one for the result for hyperbolic model, let me just zoom in a bit.

So, there are totally 8 nodes in the mesh. Corresponding to 8 node quadrilateral element that is used and the first 3 nodes are the surface horizontal surface at the y of 2 and 4 and 5 are at mid height and 6, 7 and 8 are at the bottom. And the material properties used are corresponding to hyperbolic model, modified hyperbolic model and the stress condition number is 3 that is 3 corresponding to axis symmetric.

So, the program gives you an option of plane stress, plane strain or axis symmetric and the 3 corresponds to axis symmetric case and your K for Young's modulus K for the bulk modulus and n for the bulk modulus. And there is a facility to fix a minimum Poisson’s ratio. In this case I have just set it to 0 but you can set it to some value 0.25 or 0.3 and it will accordingly adjust the K and b values.

And the failure ratio R f is 0.87, the cohesion is 15 and the friction angle at 1 atmospheric pressure is 35 degrees and there is no change in the friction angle, it is given as 0, dilation angle is 0 and the atmospheric pressure is 102. And then the earth pressure coefficient is 0, like it does not apply for this particular case and we are using a 2 point integration, integration order is 2, it is in the Xi direction and 2 in the eta direction, so we have totally 4 integration points.

And initially we applied a cell pressure of 100, so you see that the sigma x, sigma y and sigma theta they are all 100 and sigma xy is 0 like 10 to the power of -9. Then after you apply the cell pressure, you set all the displacements and then the strains to 0. You say this displacements and in both x and y direction are 0. And then let me show you the second phase of the test, oops! let me zoom it. So, here in the second phase we are applying axial deformations.

So, at nodes 1, 2, 3 at the top I am applying displacements of 0.01 at each step and that divided by 2 is 0.005 that is what we had done in the hand calculations. And our sigma x and sigma t, the sigma theta they are remaining constant at 100 sigma y has increased to 313.27. Let me just pop up, so if you see this at the end of the first step, this is 313.27 and that is what we got from the excel program and also the hand calculations.

Then in the second step sigma y is 355 and that is what we got even by the hand calculations. Actually I can send hand calculation also we got the stress of 355 and that is what we got with the finite element analysis. And then the stress has increased to 379 and then 395, 406 that is exactly the same as how it is increased by hand calculations and then 415, 422. And then at this stage the stresses reach the limit state.

The stress state that F value is almost 0 or slightly increased beyond 0 and then it is a stress state is corrected back. And now our sigma y is 426.65 that is what we calculated from our Mohr–Coulomb theory as sigma 1 F is K p times sigma 3 + 2 c times square root K p and that is what we get. And with further increase in the straining it is the sigma 1 is remaining constant, 426. And if you apply displacements you can calculate reaction forces that is what is done here.

So, at each of these nodes at nodes 1, 2, 3 equal vertical displacement was applied and then the reaction force is developed. And the node 1 is corresponding to symmetric axis, so there is no reaction force that is 10 to the power of -7, then node 2 and node 3 there are some reaction forces and the sum total of the reaction forces is a 213.32. And the corresponding pressure is, this is corresponding to a unit radian that is r square unit radian area.

And the pressure corresponding to this is the force divided by 0.5 r square because pi r square divided by 2 pi is corresponding to 1 unit radian. So, the y direction reaction force is 213.32 the divided by 0.5 r square will give you this sigma y of 426.65. Because r is 1 in this case and this divided by 0.5 is a 426, so let me just show you how the stresses have increased, open. So, these are the node numbers 1, 2, 3, 4, 5, 6, 7, 8 and let me just delete the node numbers.

And on the symmetry plane it is a roller boundary condition and then on this horizontal plane also we have the roller boundary conversion, so that we have the smooth rigid boundaries and then we are applying strain. So, let me just go to the last node step, let me magnify it a little bit, so this yellow line is the original mesh and this black line is the deformed mesh. And you see that let me just magnify it a bit.

And the entire soil element as uniformly expanded laterally and uniformly compressed in the vertical direction and then in the lateral direction there is an expansion. And we can look at how the stresses have changed; we can plot a graph between sigma y and epsilon y. And we are calculating at 4 integration points 1, 2, 3 and 4, we can select any of them by pointing mouse at that and then clicking the button.

And to start with the stress was 100 and then with increasing axial strain the axial stress is increased, pardon me for this actually this plotting program is not very good at plotting and it is not showing in the form that we are used to in the positive quadrant. But anyway you can see the axial stress is going on increasing the negative direction then at some point it has reached 426.6 that is the limiting state and then after that it has remained constant.

And you can also see how the soil is or how the deformation is taking place. So, these arrows they show how each node is deforming and this is a symmetry plane, so all the nodes are acting or deforming only in the vertical direction, whereas these nodes are not only deforming the vertical direction but also in the lateral direction. So, you see here then this particular node is moving only in the horizontal direction.

And let us now see what happens with the load control analysis, let me zoom it a bit. So, for this we used a simple bilinear elastic model. So, our vertical stress is increasing and within the elastic limit your sigma x and sigma t they are remaining constant at 100 because that is the soil is within the elastic limit. And the let us go on increase in the vertical pressure 260, there is no failure because our limit is a 426.6.

So, up to 420 we are within the elastic limit and you can tell that by looking at the norm of outer balance forces it is 10 to power of -28 means we are exactly satisfying the equilibrium. So, you are in the elastic limit and your sigma x and sigma theta they are 100 corresponding to the cell pressure. And now let us see what happens when we increase the pressure? The next step the vertical pressure is increased to 430.98.

Then because that is the applied pressure in the stress correction procedure it cannot reduce them back to 426.6 because then it will violate the equilibrium and then in the horizontal direction it cannot keep the confining pressure at 100 because the vertical stress is 430, so it has slightly increased the confining pressure to 101.18. And when you look at this outer balance force norm, it is not exactly 0, it is a some value 0.76 percent.

Whereas in the elastic limit it is 0, it is a 10 to power of -28 but here it is a 10 to the power of -02. So, if you look at higher stresses let us say let us go to the end of the and towards the end the vertical pressure is increased to 474 and then the confining pressure it has increased to 112.89, that is corresponding to a condition that our yield function value F should be 0. So, for this if we calculate you take the sigma 1 and sigma 3 as 112.89 and 474.22.

And substitute them in the equation for the F that is sigma 1 - sigma 3 minus of sigma 1 + sigma 3 times sine phi - 2 c cosine phi, if you substitute these values you will get exactly 0. So, this is what we see if you go on applying vertical pressure because of the stress correction procedure the lateral pressures will go on increasing. And let me illustrate that through this program. So, there is a data file called triax load that is corresponding to load control analysis.

And let me just set this, let me plot how the stress strain curves are changing. See here at one of the integration points the stress is initial increasing at a fast pace because you are in the elastic limit and then during the plastic state it is still increasing but at a slower pace because now you are in the plastic state. And this is the sigma y and let us look at the sigma x, how the sigma x is changing.

So, initially the sigma x was exactly 100, this is 100 up to some stage. And then beyond that the stress, the confining pressure will go on increasing because your vertical stress is increasing. And then initially the strain that you have achieved is small because you are in the elastic limit then in the plastic limit, the strains have increased a lot. So, that is what we see here the elastic limit is reached this is a corresponding to vertical horizontal strain 10 to the power of -3.

But then in the plastic state it has suddenly increased to about 10 to power of -2, that is because of the plastic straining that you are inducing will have very large strain. Actually let me just zoom it a little bit. So, with the load control analysis you get very, very large displacements because there is no limit on the displacements and then it also depends on the number of iterations.

Here I have limited the number of iterations to only 10 but if you use more number of iterations you will see more displacement. Because there is always some outer balance force because the material is not able to support, a vertical stress more than 426.6 for a confining pressure of 100. So, as you are increasing the stresses, the stress correction is taking place and because of that there is an outer balance force and then the displacements will go on increasing.

If you do more number of iterations you will see more larger displacements. And it is actually these programs and the data files you will get as part of the course material and then I am also going to give you some instructions on how to run this program. You should not have any issues with running this program. So, let us come back to my power point presentation. **(Video Ends: 40:24)**

So, basically when we use the load control analysis we may not be able to get convergence, especially this is we are only doing for a single element to triaxial compression test. But let us say you do for any bearing capacity or for lateral earth pressures, the whole thing might deform a lot without any limit and because you are not constraining the system and interpreting those results could be very difficult.

Instead if you apply displacement control, we are applying some displacements and then we are calculating the reaction forces. That we will see with an example in the next class what happens when you do any bearing capacity problem by applying uniform displacements or uniform pressure? So, that is a brief introduction on how to perform the hand calculations for the modified hyperbolic model.

And I compared the hand calculated results with the finite element results. And the finite element results are obtained exactly the same manner that we have used in the hand calculations. And the advantage with finite element analysis is you can do the analysis for any number of elements or any number of degrees of freedom without any sweat it can do the analysis. But when you are doing with hand calculations cannot solve large matrices or we cannot repeat the calculations large number of times.

So, I hope you understood these calculation methods and if not you please write to me or this email address profkrg@gmail.com and then I will respond back to you. So, thank you very much we will meet next time.