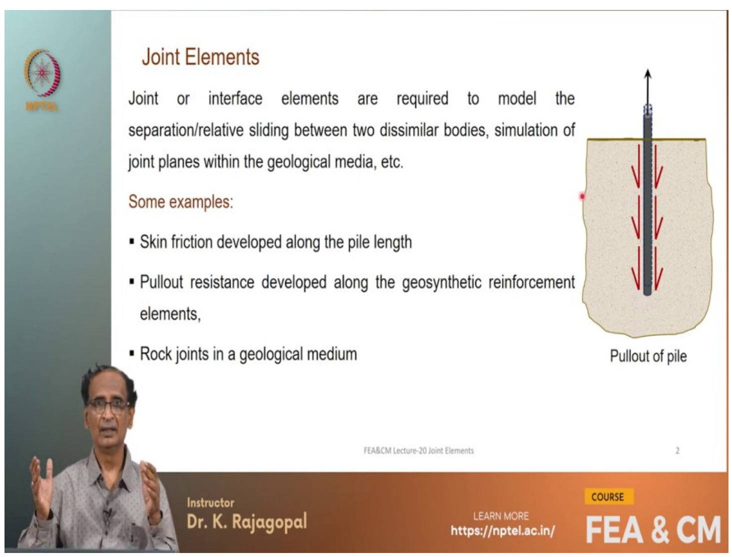
-FEM and Constitutive Modeling in Geomechanics
Prof. K. Rajagopal
Department of Civil Engineering
Indian Institute of Technology – Madras

Lecture – 23
Joint and Interface Element Modelling

So, hello students let us see some other aspect of geotechnical engineering that is the modeling of the joints and then interfaces. So, within a soil medium there may be some say fault plane or some area with very weak soil along some direction or say you may have some rigid object like a pile foundation inside a soil and the pile is very stiff comparative soil. And so, along that interface the shear stresses may generate because of the incompatibility or the difference in the stiffness.

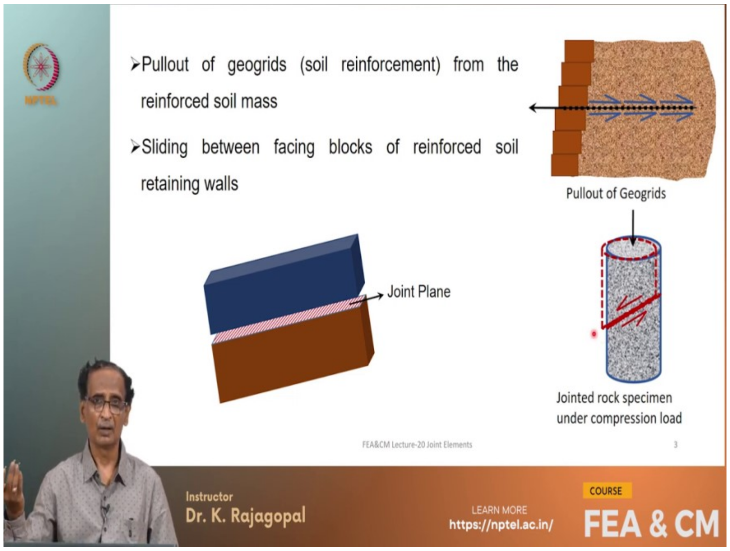
Or say you have a reinforcement a geosynthetic reinforcement and that is under safe pull out there will be some interfacial stress developed between the reinforcement. And the soil and all these things can be solved by modeling the interfaces through the joint elements that we are going to study in this lecture.

(Refer Slide Time: 00:51)



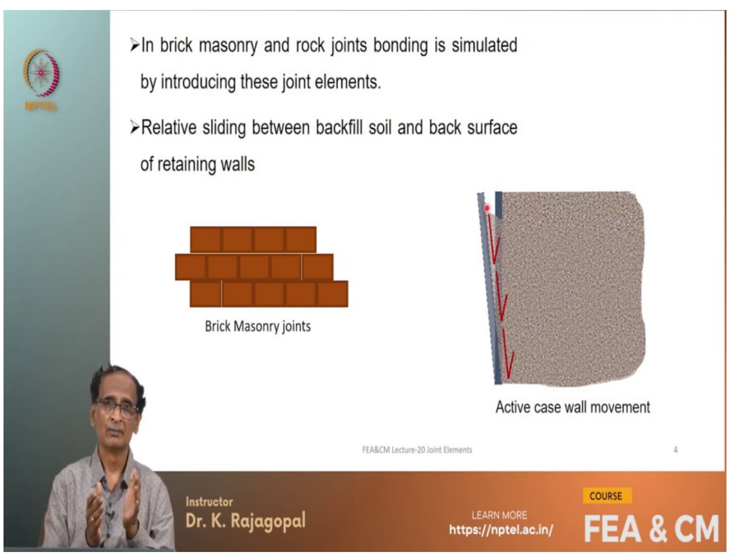
See 1 of the most common thing is the pile foundation and if you subject any pile foundation to an uplift what is the pull out resistance that the pile will have because of the soil support all around or you may have some things like geosynthetic reinforcement or the rock joints within a geological medium. And so, all these things we need to model by using an appropriate type of a joint element.

(Refer Slide Time: 02:13)



And we may have like these are all some cases like the pull out of a geosynthetic reinforcement or a joint plane between rocks or say within a rock medium there could be some joint planes.

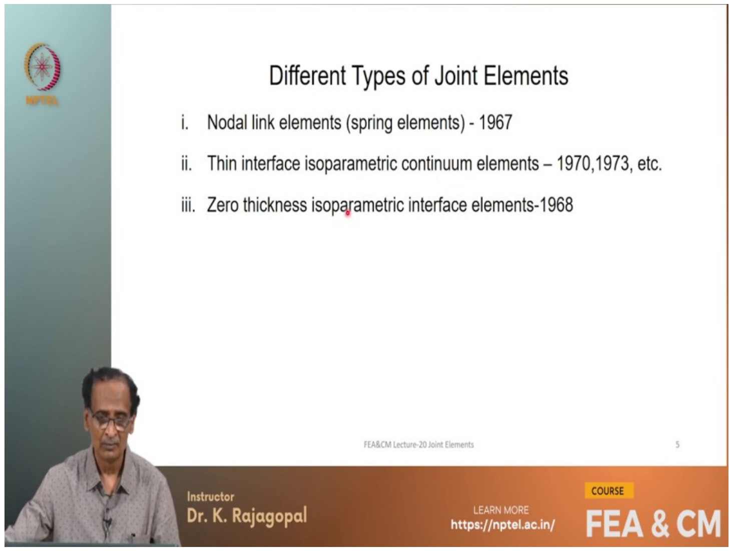
(Refer Slide Time: 02:33)



Or within a block Masonary wall there will be number of joints both horizontal and vertical and if you are interested in simulating that how do we simulate. And here is another example let us say have a retaining wall and then behind that you have a granular fill and as the retaining wall is moving away or rotating the soil is going to move relative to the to the surface and that will cause some Shear stresses.

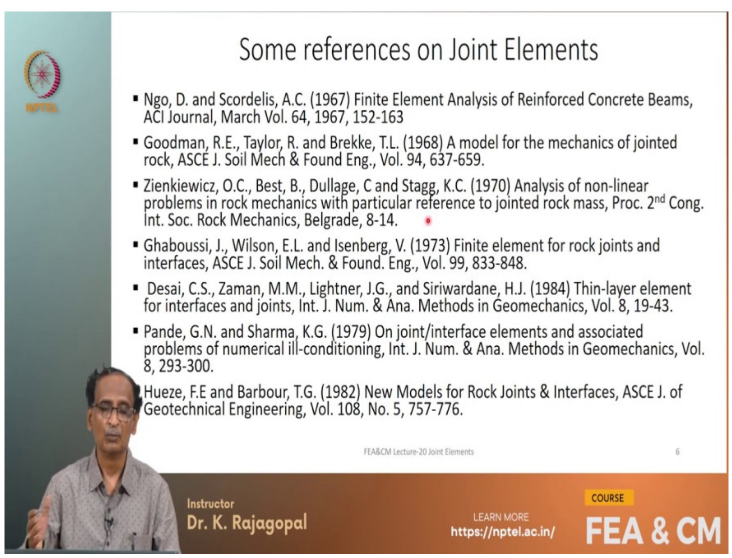
And by placing joint elements we will be able to simulate the development of the shear stress between the wall panel and then the soil.

(Refer Slide Time: 03:23)



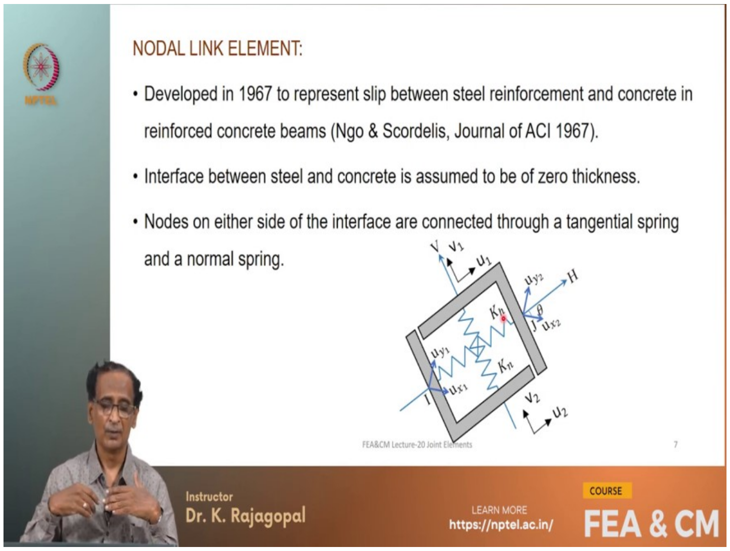
And there are different types of joint elements that have been developed in the recent past. So, the 1 of the first attempt at modeling the joints was done in 1967 in the context of reinforcement concrete and the joint element was in the form of a nodal link element. These are basically spring elements that we have seen earlier. And then in the late 60s and 70s people started developing isoparametric Continuum elements or 0 thickness isoparametric elements for interface and so on.

(Refer Slide Time: 04:05)



And these are all the different references published papers on joint elements. Actually these are all the major papers that are most referred but there are several other papers and you can look in the library for more number of papers.

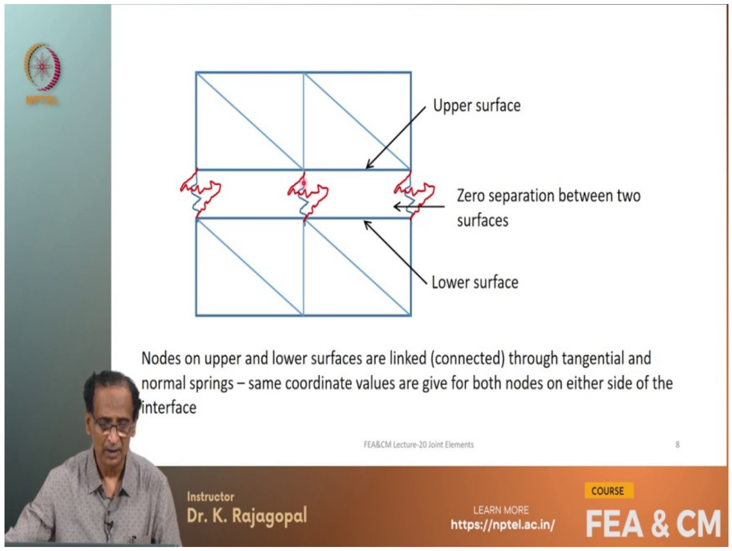
(Refer Slide Time: 04:26)



And so, the first joint element was an nodal link element this was developed in 1967 to represent the slip between the steel reinforcement and concrete in the reinforced concrete beams. And the interface between the steel and concrete was assumed to be of 0 thickness and we connect the surface of Steel and concrete through 2 Springs 1 is a vertical spring and the other is a horizontal spring.

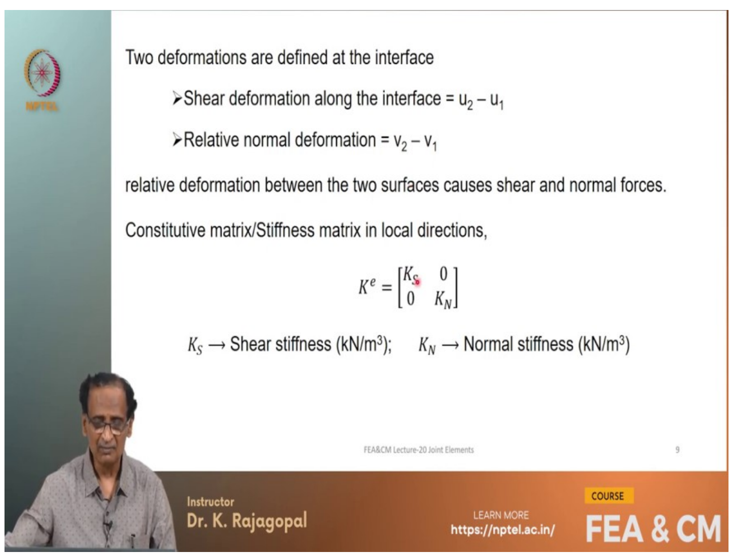
The horizontal spring is K h vertical spring is K n and then by giving some modulus values for K h and K n we can simulate the contact of the slip. So, if you give a lower value for K h there can be more relative deformation and so on.

(Refer Slide Time: 05:35)



And here is another picture of the same thing let us say you have an upper surface and then a lower surface and we connect them by using set of Springs 1 is a normal spring and then there is a tangential spring. And although I am showing with this gap there is no gap between these 2 surfaces they have the same coordinates and just for illustration purpose I have separated them out and in fact we give the same coordinate values for both upper and lower surface. So, that they are they are joined together.

(Refer Slide Time: 06:27)



And we can define 2 deformations 1 is in the tangential direction and the other is in the normal direction and the shear deformation along the length or along the interface is U 2 minus U 1. And the relative normal deformation is V 2 minus V 1 U 2 and U 1 are along the interface V 2 V 2 are the normal direction and the relative deformation between the 2 surfaces will cause some shear and normal forces.

Two deformations are defined at the interface

- Shear deformation along the interface = $u_2 - u_1$
- Relative normal deformation = $v_2 - v_1$

relative deformation between the two surfaces causes shear and normal forces.

Constitutive matrix/Stiffness matrix in local directions,

$$K^e = \begin{bmatrix} K_S & 0 \\ 0 & K_N \end{bmatrix}$$

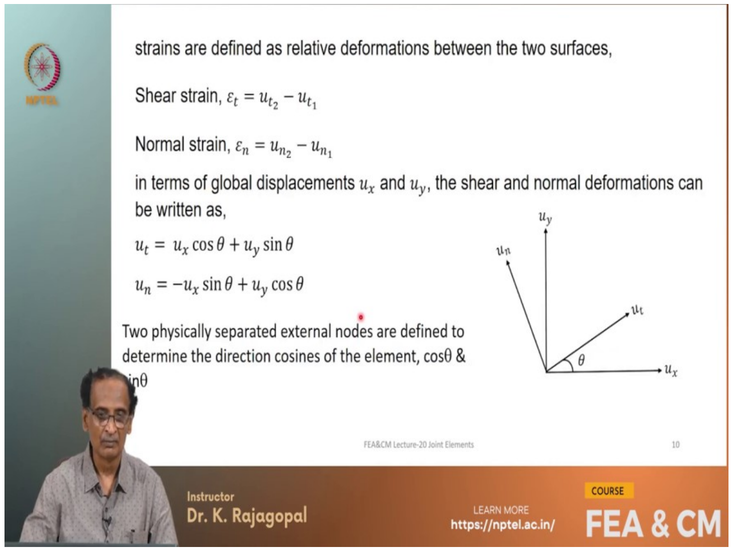
$K_S \longrightarrow$ Shear stiffness (kN/m^3); $K_N \longrightarrow$ Normal stiffness (kN/m^3)

So, when there is a deformation like this along the length there is shear deformation and then shear stress and if there is a separation that is a tensile strain then there will be tensile stress in the in your normal spring or you might have a tensile separation or if you have a compressive force then the joint will be in compression and both the surfaces are pressed together. And corresponding to these deformations 1 is a Shear deformation and the other is a normal deformation.

We can have the the stiffness and then the stresses we can have a normal stress and then a shear stress and correspondingly we will have this stiffness coefficients K S and K N. K S is the shear stiffness K N is the normal stiffness and these are in the units of F by L Cube units because the with deformation we should develop some stress. So, K S times the relative deformation and here we are not defining strains but we are actually taking the relative deformation as the as our strain.

So, that is why we have these 2 stiffness values K S and K N in the units of F by L Cube units.

(Refer Slide Time: 08:30)



And the shear strain is u t 2 minus u t 1 that is along the along the interfacial tangent direction and normal strain Epsilon n is u n 2 minus u n 1 that is in the normal direction and we can resolve them into Cartesian system Cartesian coordinates u x and u i through our direction cosines. And to get you a Theta what we do is see if because both these 2 surfaces they have the same coordinates we do not know how to define the direction.

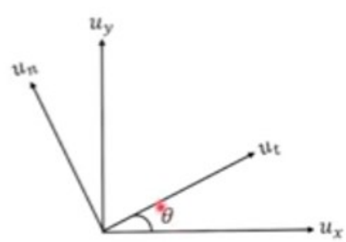
$$\text{Shear strain, } \varepsilon_t = u_{t_2} - u_{t_1}$$

$$\text{Normal strain, } \varepsilon_n = u_{n_2} - u_{n_1}$$

So, what we do is corresponding to each spring element we define 2 external nodes that are along the tangential direction along the shear direction. So, for example for this spring elements we can give additional 2 nodes like this and this as the as the tangential direction or if your spring is in the other direction we can give 2 nodes along the vertical surface to define the shear direction.

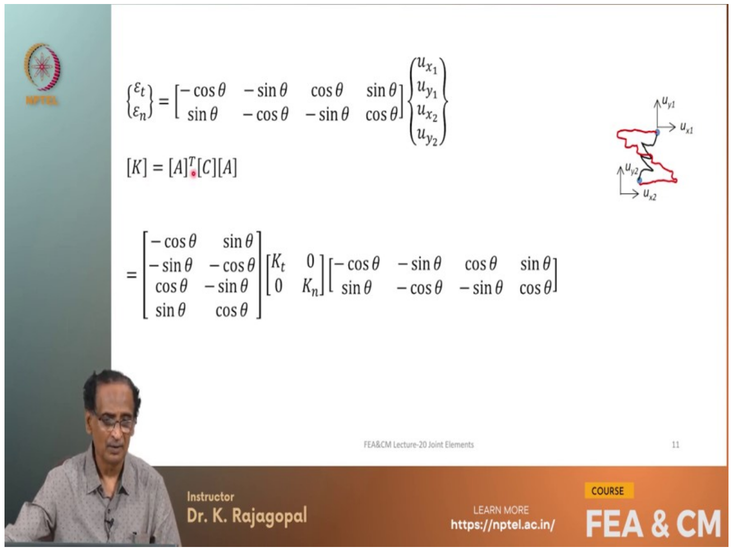
$$u_t = u_x \cos\theta + u_y \sin\theta$$

$$u_n = -u_x \sin\theta + u_y \cos\theta$$



And then we can define this cosine Theta and sine Theta based on this angle and then we can convert the Cartesian displacements to the local displacements in the tangent direction and normal direction.

(Refer Slide Time: 10:13)



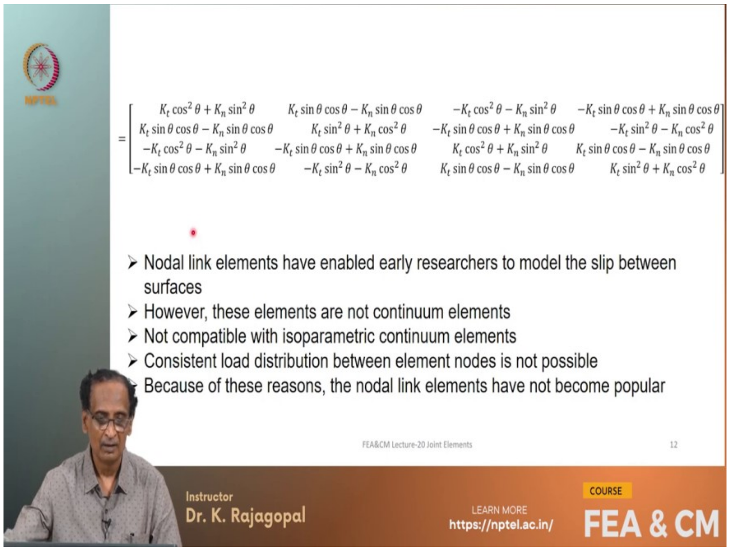
And we can get the stiffness Matrix and the global coordinates as Lambda transpose C A where C is the constituent Matrix in the local direction the K t and K n and our a is the transmission Matrix and this is similar to the matrix that we had seen long back in the context of the bar elements.

$$\begin{Bmatrix} \varepsilon_t \\ \varepsilon_n \end{Bmatrix} = \begin{bmatrix} -\cos\theta & -\sin\theta & \cos\theta & \sin\theta \\ \sin\theta & -\cos\theta & -\sin\theta & \cos\theta \end{bmatrix} \begin{Bmatrix} u_{x_1} \\ u_{y_1} \\ u_{x_2} \\ u_{y_2} \end{Bmatrix}$$

$$[K] = [A]^T[C][A]$$

$$= \begin{bmatrix} -\cos\theta & \sin\theta \\ -\sin\theta & -\cos\theta \\ \cos\theta & -\sin\theta \\ \sin\theta & \cos\theta \end{bmatrix} \begin{bmatrix} K_t & 0 \\ 0 & K_n \end{bmatrix} \begin{bmatrix} -\cos\theta & -\sin\theta & \cos\theta & \sin\theta \\ \sin\theta & -\cos\theta & -\sin\theta & \cos\theta \end{bmatrix}$$

(Refer Slide Time: 10:47)



And once you go through this orthogonal transmission and this is your stiffness Matrix and the global coordinates. And these nodal link elements they have enabled early researchers to model the slip between the surfaces between the concrete and the steel or between any masonry blocks and so on. But these nodal link elements there is these are not continuum these are at particular points at the nodal points.

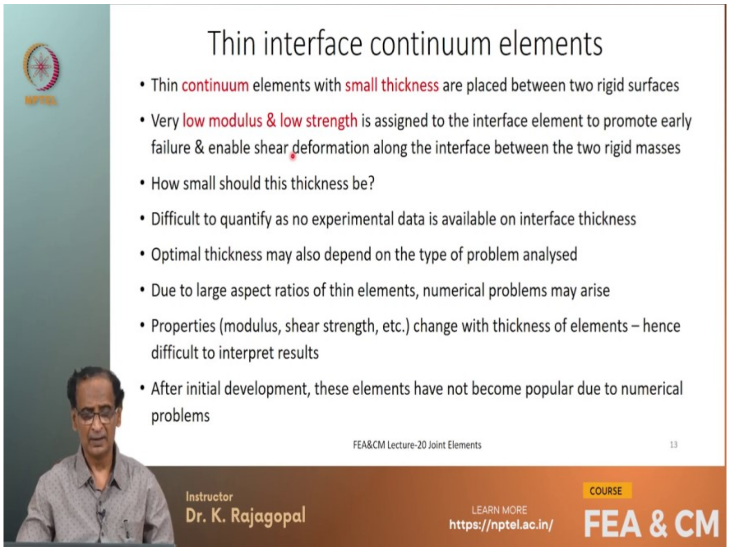
And so, these are not compatible with isoparametric elements because the isoparametric elements we have seen when we did the calculation the load calculations. Say the the loading is distributed in a consistent manner in which the stiffness is distributed. For example when we had the uniform pressure the mid side nodes they attract four times higher load than the corner nodes or when we did the load calculation because of initial stresses at the interior point in the nine node quadrilateral it has attracted 16 times the load corresponding to the end points are the corner points and so on.

$$= \begin{bmatrix} K_t\cos^2\theta + K_n\sin^2\theta & K_t\sin\theta\cos\theta - K_n\sin\theta\cos\theta & -K_t\cos^2\theta - K_n\sin^2\theta & -K_t\sin\theta\cos\theta + K_n\sin\theta\cos\theta \\ K_t\sin\theta\cos\theta - K_n\sin\theta\cos\theta & K_t\sin^2\theta + K_n\cos^2\theta & -K_t\sin\theta\cos\theta + K_n\sin\theta\cos\theta & -K_t\sin^2\theta - K_n\cos^2\theta \\ -K_t\cos^2\theta - K_n\sin^2\theta & -K_t\sin\theta\cos\theta + K_n\sin\theta\cos\theta & K_t\cos^2\theta + K_n\sin^2\theta & K_t\sin\theta\cos\theta - K_n\sin\theta\cos\theta \\ -K_t\sin\theta\cos\theta + K_n\sin\theta\cos\theta & -K_t\sin^2\theta - K_n\cos^2\theta & K_t\sin\theta\cos\theta - K_n\sin\theta\cos\theta & K_t\sin^2\theta + K_n\cos^2\theta \end{bmatrix}$$

These are all unique parametric elements and depending on the shape the distribution may change. So, it is very difficult to link these nodal link elements to isoparametric elements because we do not know how best to represent the relation between the spring element and then the isoparametric element especially with respect to load sharing. And because of these reasons the nodal link elements have not become very popular.

So, in those days in 1967 there was only 1 finite element finite element that is the three note triangle which is very very simple and the contribution in a three node triangle is equal between the nodes. So, there was no problem but let us say you take a distorted quadrilateral we do not know how to distribute the load between the different nodes unless you perform your calculations N transpose V dV R integral V transpose Sigma dV and so on.

(Refer Slide Time: 13:29)



So, that way this nodal link element we do not know how to how to link it to regular finite elements and later what some people have done is they have used a regular Continuum element but very thin 1 very small thickness and with the lower modulus and the lower strength so, that you promote the failure along some preferential direction. Let us say within a foundation bed you have a seam of soil that is of low strength.

And you suspect that the soil is going to slide along that line and you can make it do that by assigning the lower properties to the into the joint element. So, that is what is done by some researchers but then how small should the thickness be whether it should be 0.1 or 0.01 or 0.001 and so on like very difficult to quantify because there is no experimental data on measuring the thickness of the interfaces whether you have a steel plate and then soil or a Geo grid under the soil or a geotextile in the soil.

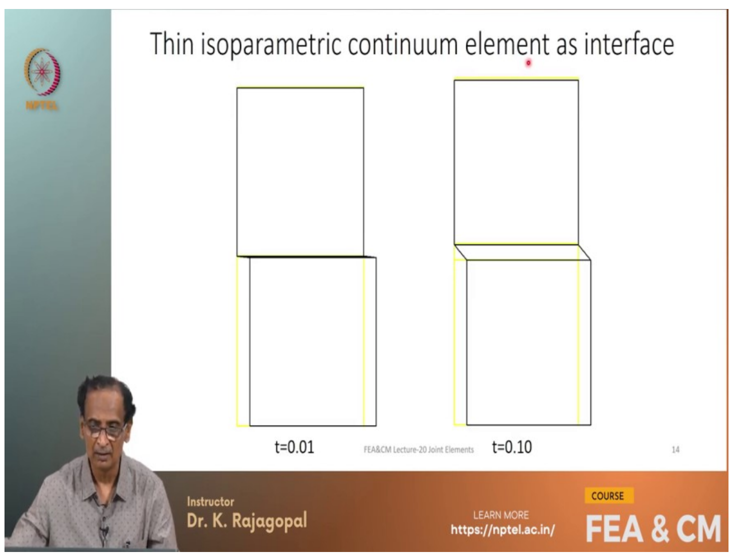
The thickness is very difficult to to imagine because there is no measurement so, we do not have an idea of how thick this interface should be but in the thin layer interface elements depending on the thickness that you use your result might change that I will illustrate later. And so the optimal thickness that we need to use also may change depending on the problem.

So, if you have Rock joints your optimal thickness may be something but if you have a smooth geotextile your thickness may be may be negligible might be just 0.

But if you have a Geogrid the bonding is very good between the open apertures and then the soil. So, the stiffness and then the strength of the interface could be very high. See the other problem with these thin interface elements is their aspect ratio is very very large say their length may be something but their thickness could be very small. Say if they are of length say 10 units long but the thickness is only 0.1.

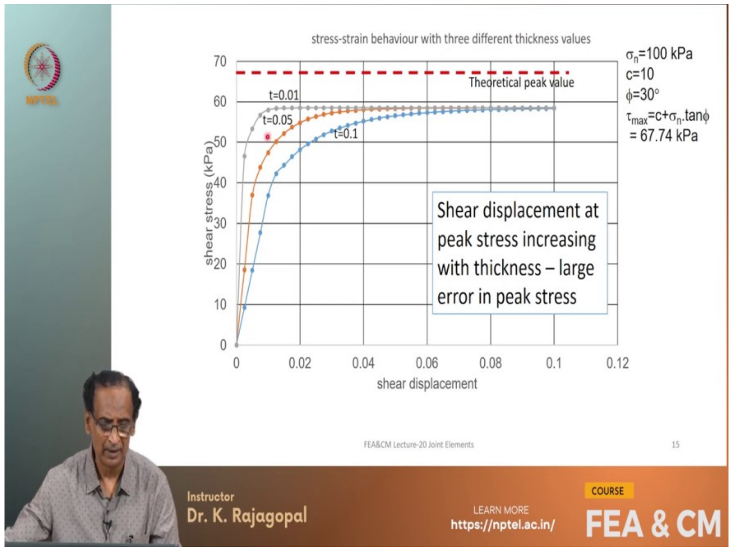
So, the aspect ratio is 10 by 0.1. So, that is 100 but then when our finite elements are too oblong like very very large aspect ratio they may not perform they may not be able to give the result that that is close to the accurate result. Then the properties of the interface may might change with the thickness that we do not know how to change that thing. So, because of these reasons these elements have not become very popular although people use.

(Refer Slide Time: 16:46)

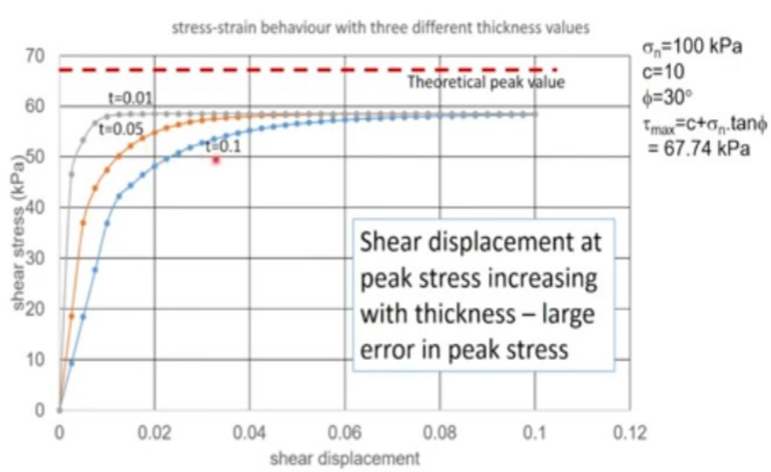


So, here I am simulating the direct shear box test by assuming the 2 thicknesses at this at this Junction 1 is with a thickness of 0.01 and the other is with the 0.1 and there was another test with the 0.05 and this was done for properties of C is 10 Phi of 30 degrees and the applied pressure of 100 kPa it is not shown here but at the top there is a pressure loading of 100 kPa.

(Refer Slide Time: 17:28)

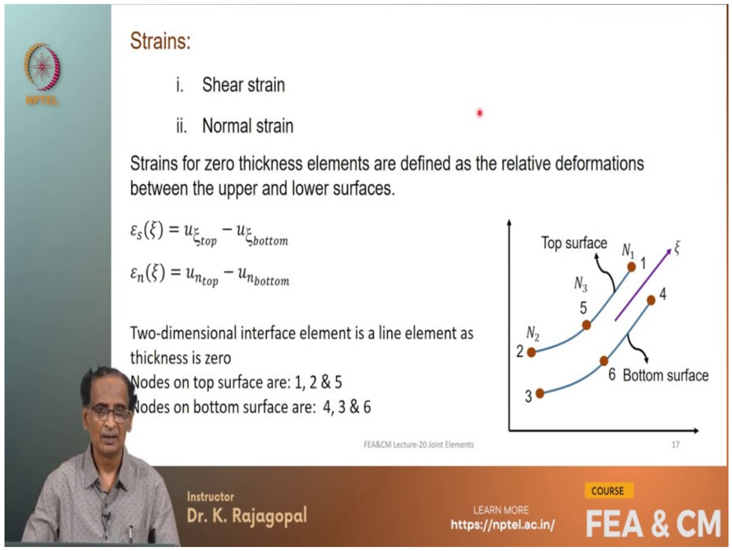


And the stress strain responses predicted by these three models is like this the thickness with the 0.01, 0.05 orange color and blue color is 0.1 and everywhere the maximum stress is stopping at about 59 or something whereas the theoretical limit of the interface stress is about 67.7 kPa. And we also notice that although the predicted shear stress is the same but the nature of the variation is different say when you have very small thickness 0.01 the peak stress is more or less reached at strain of about 0.015 or something.



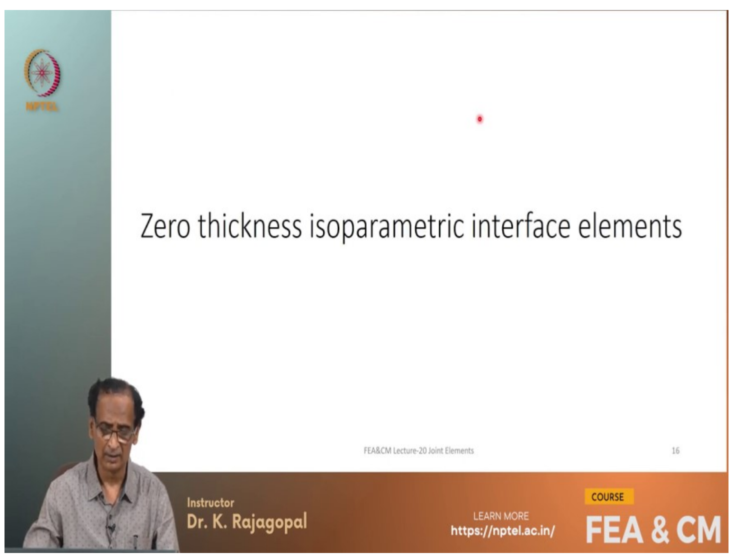
Before maybe after this point the stress is constant whereas with a t of 0.05 this transition point is different and with the t of 0.1 its even different and in all the cases although the theoretical peak stress is the 67.74 but these models that predicting only 59 kPa.

(Refer Slide Time: 19:00)



And later in the actually this development started from late 70s but it picked up momentum towards the late 87, I said the initial development was in the late 60s but in the 70s also there was lot of development and by 78 there were some very good papers.

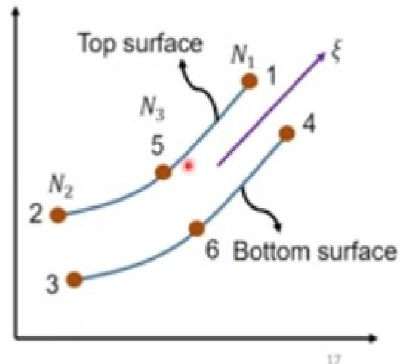
(Refer Slide Time: 19:27)



And it is the 0 thickness ISO parametric interface elements and these interfaces are of six nodes or four nodes and 1 2 3 4 5 6 these are the nodes are the interface and these surface need not be a straight surface it could be even a curvy linear surface because that is how the formulation is done. And the u Xi the at the between the 2 surfaces is a u Xi top minus u X i bottom.

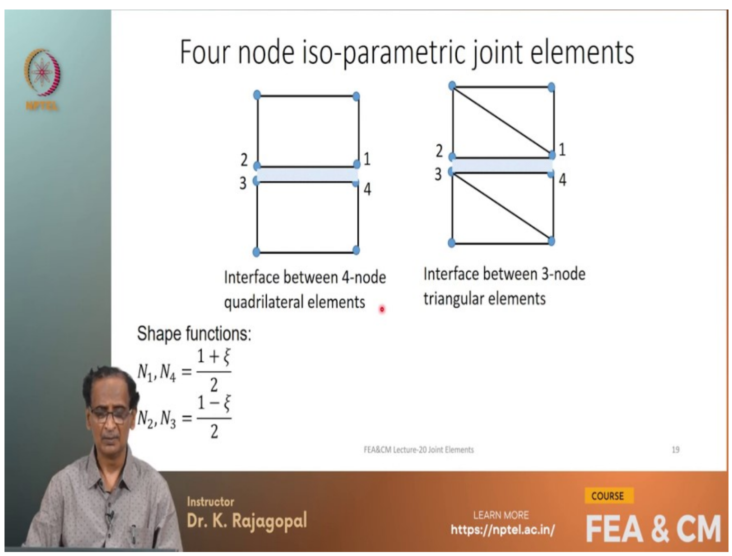
$$\varepsilon_s(\xi) = u_{\xi_{top}} - u_{\xi_{bottom}}$$

$$\varepsilon_n(\xi) = u_{n_{top}} - u_{n_{bottom}}$$



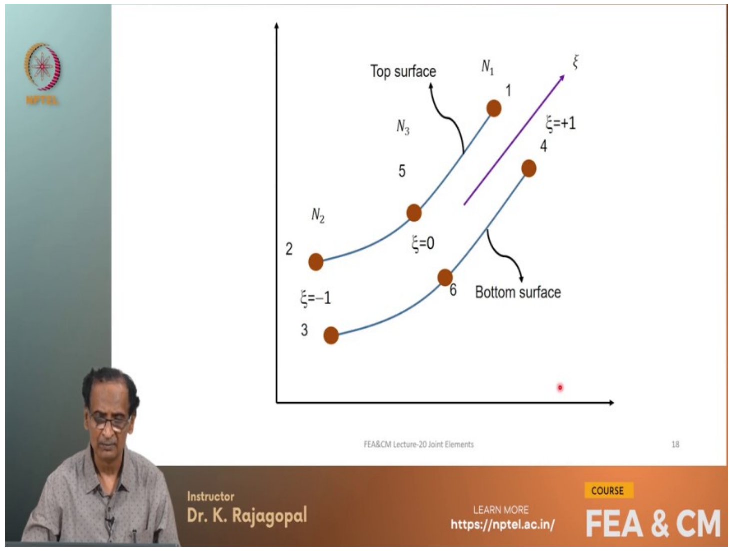
Say upper surface might have received or moved in some direction whereas the bottom surface might move in some other direction and then the relative deformation is your Shear strain. And then the normal strain is the normal deformation between the 2. And on the upper surface the nodes are 1 2 and 5 and the bottom surface 4 3 and 6.

(Refer Slide Time: 20:40)



And this is how we have and we can define the interface between either four node elements or eight node elements eight and nine node.

(Refer Slide Time: 20:55)



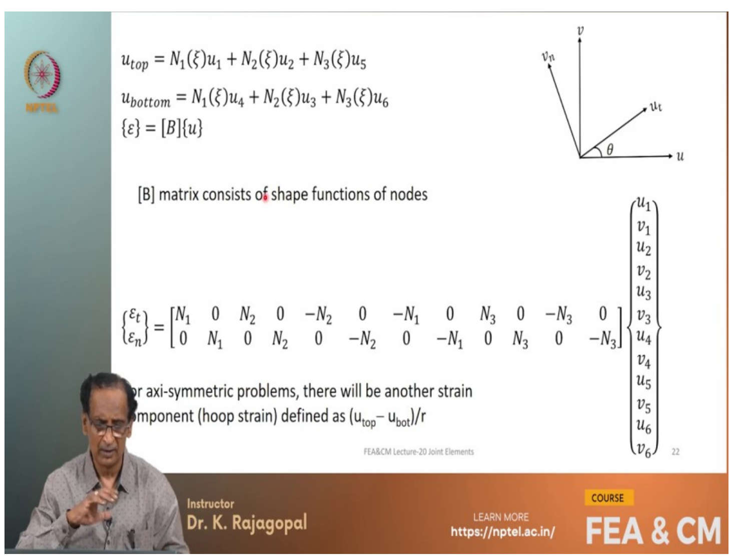
And the sixth node element is good for higher order elements like six node triangle or eight node quadrilateral nine node quadrilateral and so on. But if we have very simple elements like three node triangle and a four node quadrilateral we can use a another equivalent interface element that is formulated in four nodes in this. And the shape functions at this end at the right hand is 1 + Xi by 2 whereas at the negative end it is 1 - Xi by 2.

Shape functions:

$$N_1, N_4 = \frac{1+\xi}{2}$$

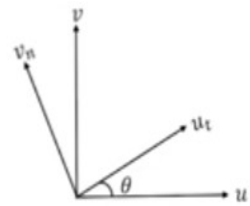
$$N_2, N_3 = \frac{1-\xi}{2}$$

(Refer Slide Time: 03:23)



So, the displacement the shear displacement at the top we can write in terms of the displacements at nodes 1 2 and 5 whereas bottom is with respect to 4 3 and 6 and our epsilon that is the strain matrix we can write like this and this is for a four node plane strain element.

$$u_{top} = N_1(\xi)u_1 + N_2(\xi)u_2 + N_3(\xi)u_5$$
$$u_{bottom} = N_1(\xi)u_4 + N_2(\xi)u_3 + N_3(\xi)u_6$$
$$\{\varepsilon\} = [B]\{u\}$$

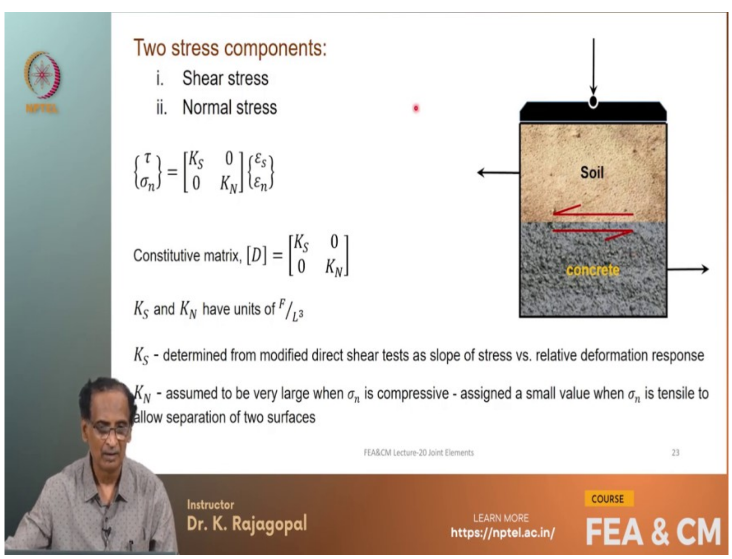


[B] matrix consists of shape functions of nodes

$$\begin{Bmatrix} \varepsilon_t \\ \varepsilon_n \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & -N_2 & 0 & -N_1 & 0 & N_3 & 0 & -N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & -N_2 & 0 & -N_1 & 0 & N_3 & 0 & -N_3 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \\ u_5 \\ v_5 \\ u_6 \\ v_6 \end{Bmatrix}$$

So, four nodes means N 1 0 N 1 N 2 N 2 N 1 N 3 N 6 this is the sixth node element. And if you know these displacements u 1 u 2 and so on you can convert them to the local coordinates with respect to the Joint elements and then do the calculations.

(Refer Slide Time: 22:56)



And the stresses are shear stress and normal stress K S and K N and the constitute Matrix is the same K S 0 0 K N and they are in the units of force per L Cube units. And the K S is the shear modulus or the shear stiffness that is ideally determined from direct shear test. So, here let us say you are interested in determining the friction between a concrete block and then the soil.

Two stress components:

i. Shear stress
ii. Normal stress

$$\begin{Bmatrix} \tau \\ \sigma_n \end{Bmatrix} = \begin{bmatrix} K_S & 0 \\ 0 & K_N \end{bmatrix} \begin{Bmatrix} \varepsilon_s \\ \varepsilon_n \end{Bmatrix}$$

Constitutive matrix, $[D] = \begin{bmatrix} K_S & 0 \\ 0 & K_N \end{bmatrix}$

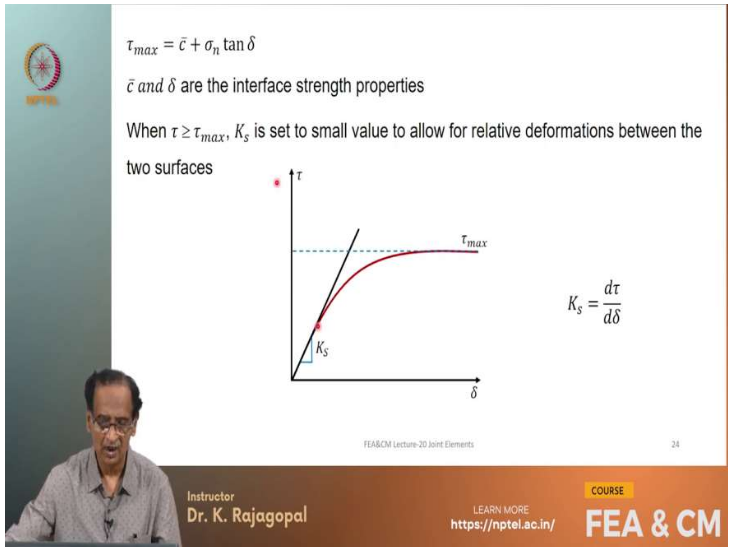
K_S and K_N have units of $^F/_{L^3}$

What we do is we place the concrete block in the lower box and then place the soil and do some compaction to compact the soil and then we can determine the stresses and then calculate our interfacial this coefficient or the friction angle. And this test is called as a modified direction test because it is similar to direct shear test but it is not exactly not exactly the sorry I think. And this K S we can determine from the slope of the shear stress versus shear strain.

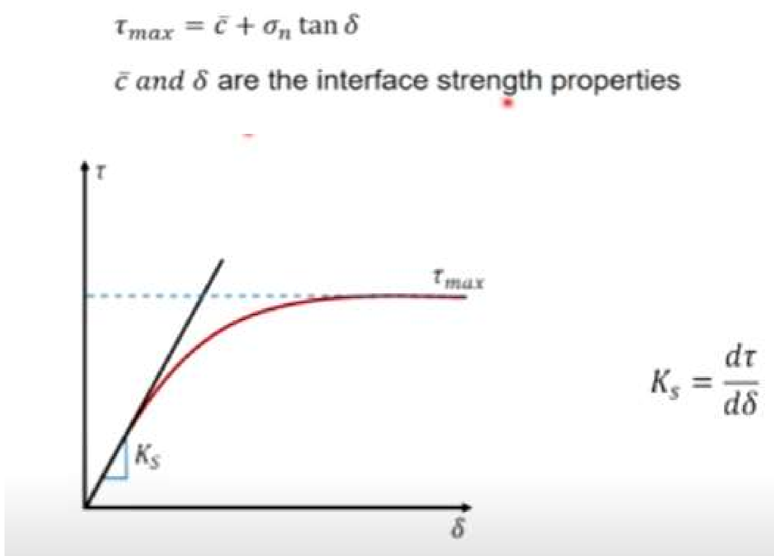
And the shear strain is whatever strain that is happening at the interface we can take as the shear strain because 1 box is kept stationary and the other box is moving. And so we can consider that deformation as the as the shear deformation and then the K N is taken as a very large value when the interface is in compression but when it is in tension we can relax the K N value to a small value not exactly 0 because 0 means we may have some 0 diagonal terms.

So, we do not make it as 0 but we reduce it to a small value so, that the tensile separation can happen without developing too much of tensile stresses.

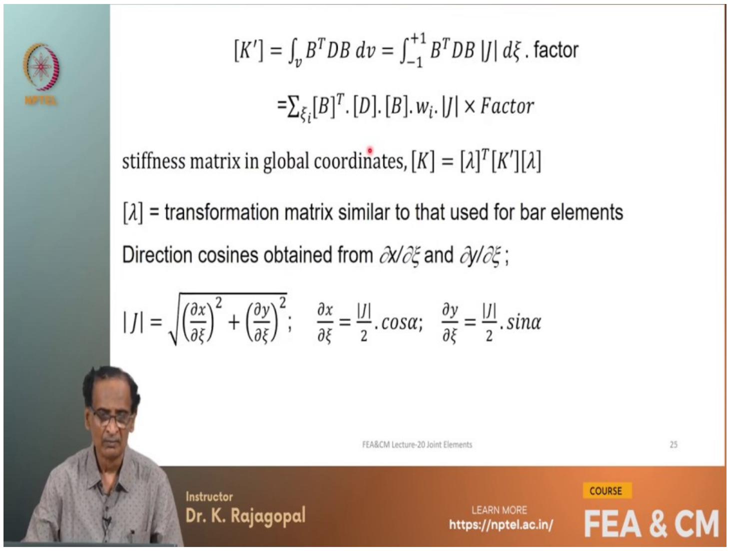
(Refer Slide Time: 25:32)



And so, the slope of this tau delta graph will give you the K S and the Tau Max the maximum shear stress and the interface is the C plus Sigma and tan Delta where C and Delta are the interface Shear strength properties that we get from the modified directial test. And when the shear stress is more than they are lower shear stress the K S is set to a small value to allow for relative deformations between the 2 surfaces.



(Refer Slide Time: 26:08)



And our stiffness matrices we can formulate in the same manner B transpose DB K Prime that is the stiffness Matrix and the local coordinates can be formulated as B transpose B DV integrated of the volume and its actually it is a line integral because our thickness is very small and very 0. So, there is no integration in the eta direction there is only integration on the length direction Xi. Then once you get your stiffness matrix in the local coordinates we can transform the that to the global coordinates by Lambda transpose K Prime Lambda.

$$[K'] = \int_v B^T DB\, dv = \int_{-1}^{+1} B^T DB\, |J|\, d\xi \text{ . factor}$$

$$= \sum_{\xi_i} [B]^T.[D].[B].w_i.|J| \times Factor$$

stiffness matrix in global coordinates, $[K] = [\lambda]^T[K'][\lambda]$

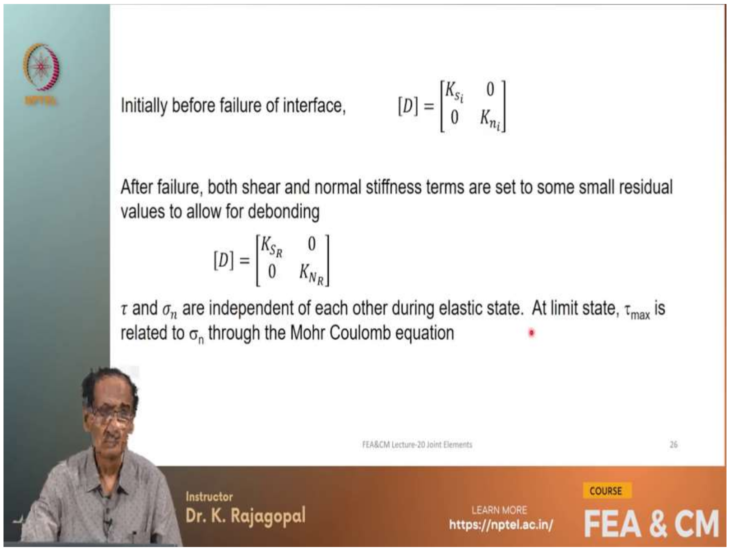
$[\lambda]$ = transformation matrix similar to that used for bar elements

Direction cosines obtained from $\partial x/\partial\xi$ and $\partial y/\partial\xi$;

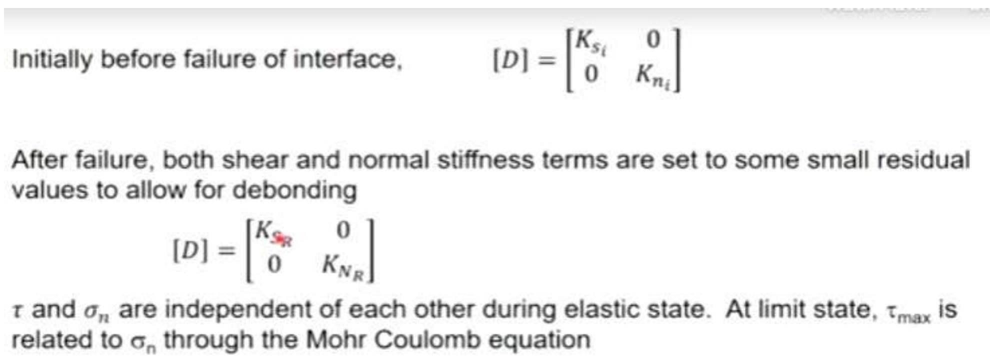
That the orthogonal transformation the Lambda is similar to the transformation matrix that we have for the bar elements. And the direction cosines dou x by dou Xi dou y by dou Xi are directly related to the to the Jacobian and then the direction cosines. So, we have seen as today that dou x by dou Xi is determinant by 2 times cosine Alpha and dou Y by dou Xi is the determinant by 2 times sine Alpha.

$$|J| = \sqrt{\left(\frac{\partial x}{\partial \xi}\right)^2 + \left(\frac{\partial y}{\partial \xi}\right)^2}; \quad \frac{\partial x}{\partial \xi} = \frac{|J|}{2}.cos\alpha; \quad \frac{\partial y}{\partial \xi} = \frac{|J|}{2}.sin\alpha$$

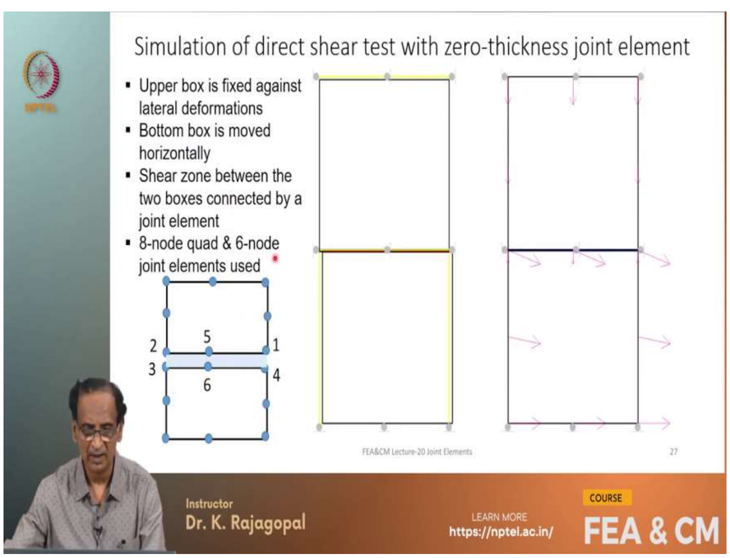
(Refer Slide Time: 27:39)



And initially before the failure will have very large values for shear stiffness and the normal stiffness and then after the failure we can relax these values like relax them to a small value. So, that there will not be any further development of the of the shear stresses or normal stress.

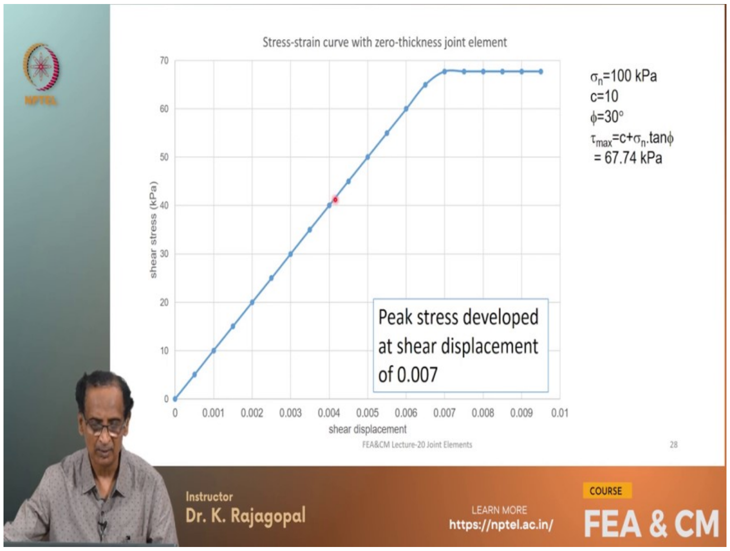


(Refer Slide Time: 28:05)



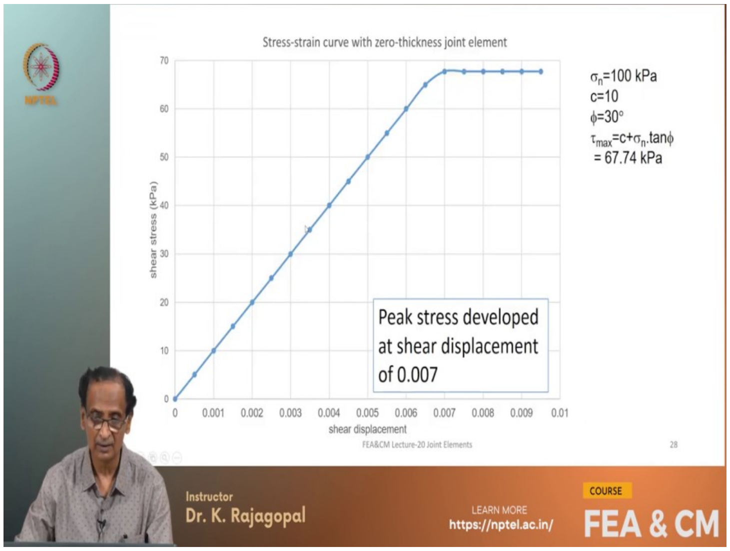
So, let us simulate the same directial test that we had done earlier with the thin isoparametric element. So, here the upper and the bottom box are connected through an interface element of 0 thickness and the upper box is restrained from moving laterally whereas the bottom box is moved horizontally.

(Refer Slide Time: 28:34)



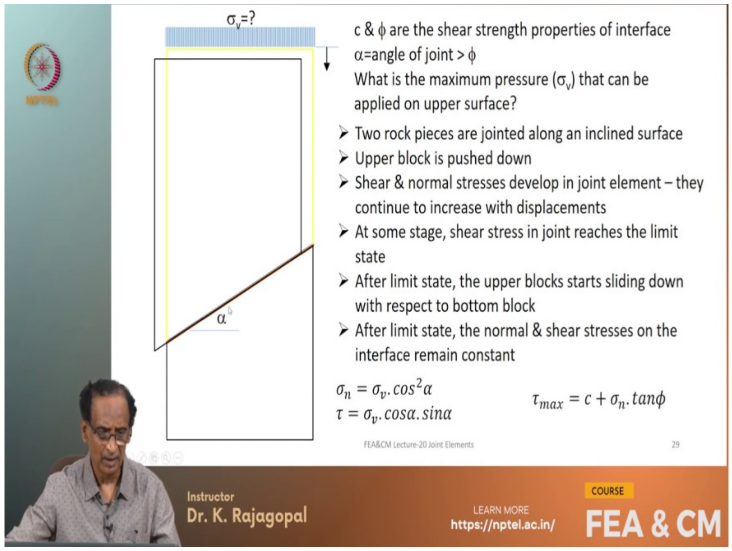
And this is the response that we get and this is for C of 10 and Phi of 30 degrees sigma is 100 and this is about 67 like actually we can see that see here 67.735 that is the theoretical strength of the of the interface tau max C plus Sigma and tan Phi.

(Refer Slide Time: 29:07)



You get the same the same values.

(Refer Slide Time: 29:12)

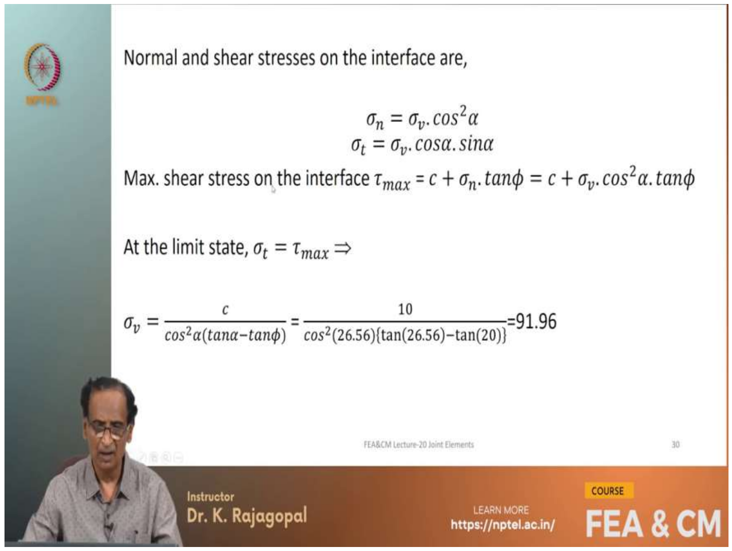


And let us apply this interface element to another problem this is a problem with the let us say you collect a rock sample a cylindrical rock sample and then there is a joint plane at an angle of alpha. Let us say that we know the shear strength properties of this interface C and the Phi and we want to know what is the maximum vertical stress that we can apply before this the interface fails.

So, how do we do that is actually we can easily simulate this through finite element analysis and since we do not know how much pressure we can apply what we do is we apply deformations. In fact most of our laboratory tests like our direction test are the triaxial compression test they are all stress con sorry the displacement control test where we are applying some displacement and then measuring the reaction.

Whereas the consolidation test is a load control test we are applying the pressure and measuring the deformations but here we are applying deformations and then measuring the stresses. So, here we are applying some displacement and then we can measure the stresses that are acting along the interface.

(Refer Slide Time: 30:48)



So, our the normal and shear stress on the interface can be related to Sigma V by drawing more Circle this is if this is Alpha Sigma n is Sigma V times cosine Square Alpha and then tau is Sigma V times sine of sine Alpha cosine Alpha and. So, if this is the case the maximum shear stress on the interface is tau max C plus Sigma and tan Phi and sigma n is Sigma V times cosine Square Alpha.

$$\sigma_n = \sigma_v . cos^2\alpha \qquad \tau_{max} = c + \sigma_n . tan\phi$$
$$\tau = \sigma_v . cos\alpha . sin\alpha$$

Normal and shear stresses on the interface are,

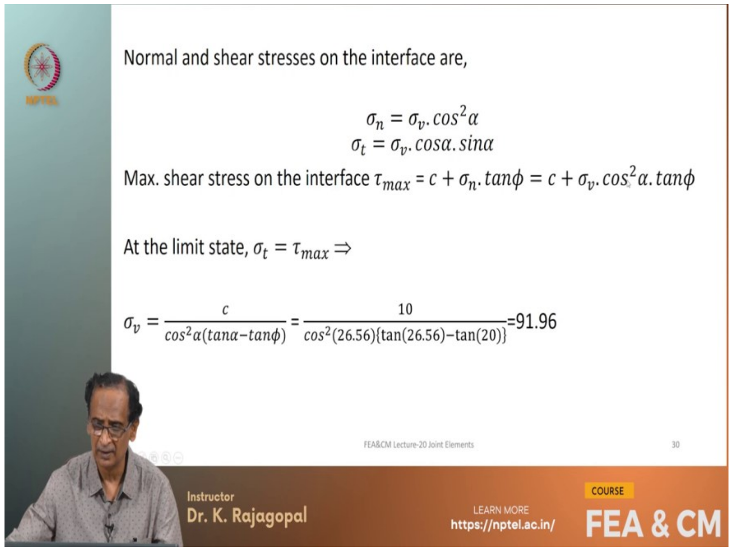
$$\sigma_n = \sigma_v . cos^2\alpha$$
$$\sigma_t = \sigma_v . cos\alpha . sin\alpha$$

Max. shear stress on the interface $\tau_{max} = c + \sigma_n . tan\phi = c + \sigma_v . cos^2\alpha . tan\phi$

At the limit state, $\sigma_t = \tau_{max} \Rightarrow$

$$\sigma_v = \frac{c}{cos^2\alpha(tan\alpha - tan\phi)} = \frac{10}{cos^2(26.56)\{tan(26.56) - tan(20)\}} = 91.96$$

(Refer Slide Time: 31:35)



So, we can write this as C plus Sigma V cosine Square Alpha tan Phi and at the limit state the tau max should be exactly equal to Sigma t right and by sorry and by doing that we get an equation for Sigma V as C by cos n Square Alpha tan Alpha minus tan Phi and for some numbers C of 10 and Phi of 20 degrees and this slope angle is 26.56 the sigma V comes to 91.9.

Normal and shear stresses on the interface are,

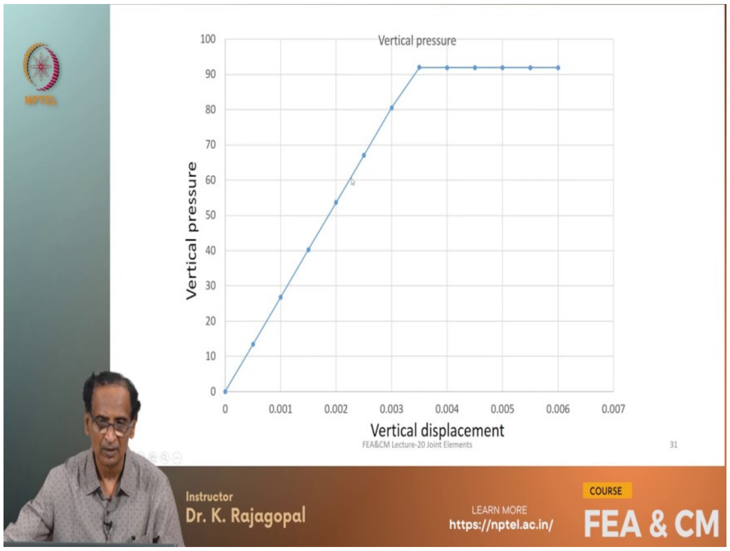
$$\sigma_n = \sigma_v . cos^2\alpha$$
$$\sigma_t = \sigma_v . cos\alpha . sin\alpha$$

Max. shear stress on the interface $\tau_{max} = c + \sigma_n . tan\phi = c + \sigma_v . cos^2\alpha . tan\phi$

At the limit state, $\sigma_t = \tau_{max} \Rightarrow$

$$\sigma_v = \frac{c}{cos^2\alpha(tan\alpha - tan\phi)} = \frac{10}{cos^2(26.56)\{tan(26.56) - tan(20)\}} = 91.96$$

(Refer Slide Time: 32:22)



And so, this is actually the plot between the vertical stress and then the vertical displacement and this is from our finite element analysis and you see this is a 91.90 whatever stress that we can that we have determined through the theoretically. So, I think that may be the last slide. And let me just open this data file and show you what we have done there is format.

(Video Starts: 33:22)

So, this is our interface and we are dealing with plane strain problem and initially the tangent modulus and the normal stiffness are given very high and then the residual tangent stiffness is very small it is taken as 100 and the residual normal stiffness I am not allowing it to fail it will remain like very high in contact perfect contact. The cohesive strength is a 10 and the interface friction angle is a 20 degrees.

And these soil elements are actually the rock are taken as elastic because there is no there is not going to be any failure and this is our joint element connected between six nodes. And these are the 2 the Continuum elements 2 of them and they are defined with eight nodes, material type is 1. And then the IRCT is 1 because they have eight nodes integration order is a 2 stress group is one.

So, here what I am doing is I am monitoring the reaction force that is applied at the top and the width is taken as one. So, the reaction force divided by the width will be your stress Sigma V and with first increment the vertical pressure developed is sorry 13.418 then it has become 26, 40.25 then 53 then 67, 80 then 92, 91 actually initially it might overshoot because when we are applying displacement the previous displacement increment might be in the elastic State and then suddenly might have reached the plastic state.

So, the predicted stress might be slightly more than what is allowed but if you are if you continue the analysis that stresses are decreasing. See once the peak stress is reached it is exactly 91.90. See here it is now it is showing the in the elements that the Continuum elements also the compression stress is exactly 91.90 and if you look at the joint elements the tangential stress is 36.

And the normal stresses are 73.5 and you can check whether the tangential stress is allowed or not as 10 plus Sigma and tan Phi. So, if you do that you will get exactly the same tangential stress. So, that means that that is the joint element is at the limit state.
(Video Ends: 37:55)

So, this is the result that we get from the inclined joint plane. So, we can place these joints wherever there is a discontinuity between a pile and the soil or between a geosynthetic and the soil and so on. And we will see some example analysis through the tutorial. So, that you can you can understand better and the application of these joint elements. So, thank you very much I think that is the end of the slide end of my presentation.