FEM and Constitutive Modelling in Geomechanics

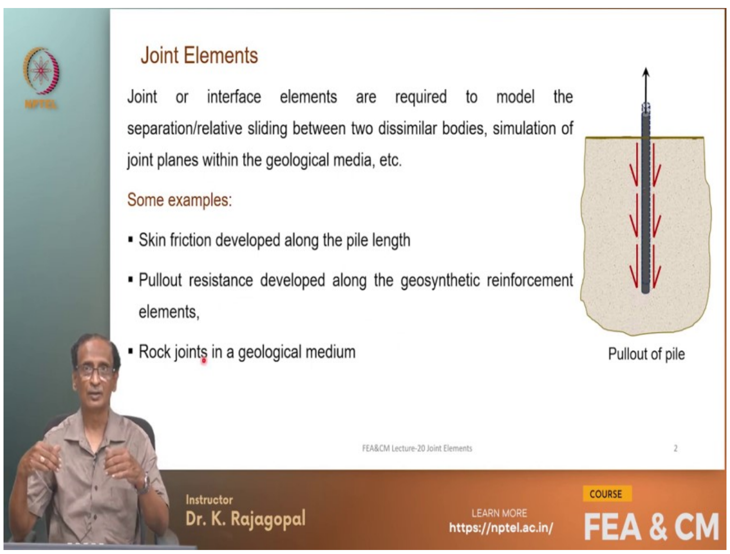
K. Rajagopal
Professor and PK Aravindan Institute Chair
Department of Civil Engineering
Indian Institute of Technology - Madras

Lecture: 24
Modelling of interfaces - Joint Elements

So, hello student let us continue from the previous class and in today's class let us look at a very typical problem that we face in geotechnical engineering that is the modelling of interfaces. Because whenever there is a joint there is a possibility for relative slip and in geological medium it is very common for the occurrence of joints and interfaces because whenever we see any Rock medium.

You will see some joints along with the sliding could take place or within the soil deposit there could be a thin layer of soft clay along with the sliding could take place.

(Refer Slide Time: 01:07)



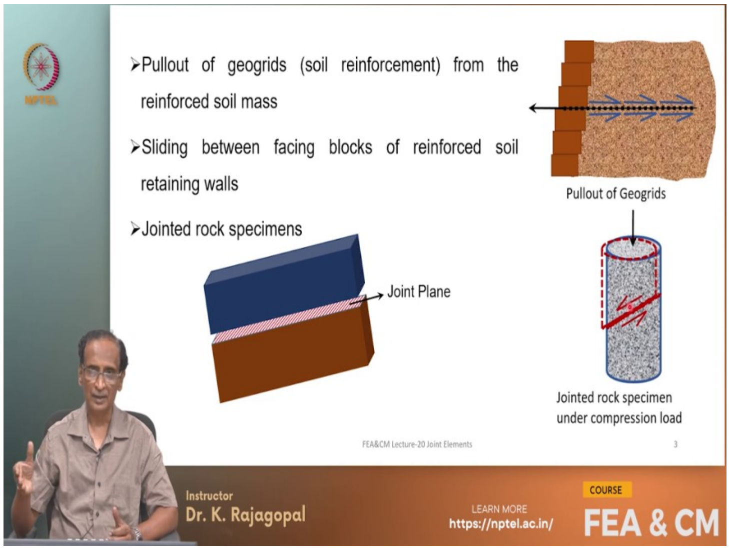
And when we do any finite element modelling we should be able to capture the behaviour. Apart from those natural phenomenons we could have joints in several other cases like for example here we have a pile that is installed in the soil and let us say you are pulling it out. And the predominant resistances from the is from the resistance the skin friction that is acting along the surface apart from the weight of the pile.

And the same thing happens even when you compress the pile some capacities because of the skin friction and some capacities because of the end bearing. And by using the special type of joint or interface elements we will be able to simulate what is happening along the joint between these 2 materials. Like a pile that could be made of reinforced concrete or steel and then the soil and by in the process we should be able to get the interaction that is taking place between the soil.

And then the pile and we would like to know how much of the applied loads are transferred into the into the soil through the pile. And in another case we may have some geosynthetic type reinforcement in reinforced soil embankment and we would like to know how much pull out force that we can apply for a given length of the geosynthetic. And how the loads are transferred from the geosynthetic into the soil and that also requires us to model the interface between the geosynthetic.

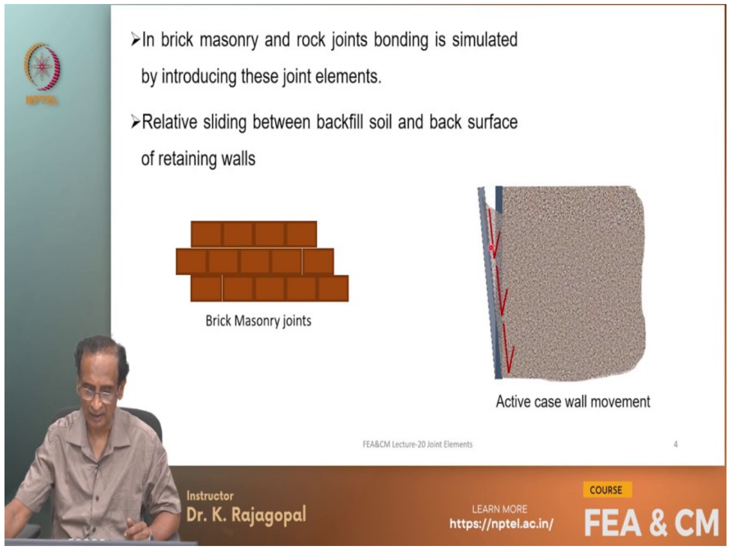
And then the soil then of course as I mentioned earlier the rock joints is a very common occurrence whenever we deal with rock mechanics problems.

(Refer Slide Time: 03:12)



And we could also have the joints between facing blocks within a reinforced soil and back within a reinforced soil wall or let us say rock joint or sometimes we collect a non-disturbed rock sample. Then we see one seam that is there inside embedded as a geological formation and we would like to know its strength and then how it is going to offer the resistance. All these things we can do by modelling with a with an interface element.

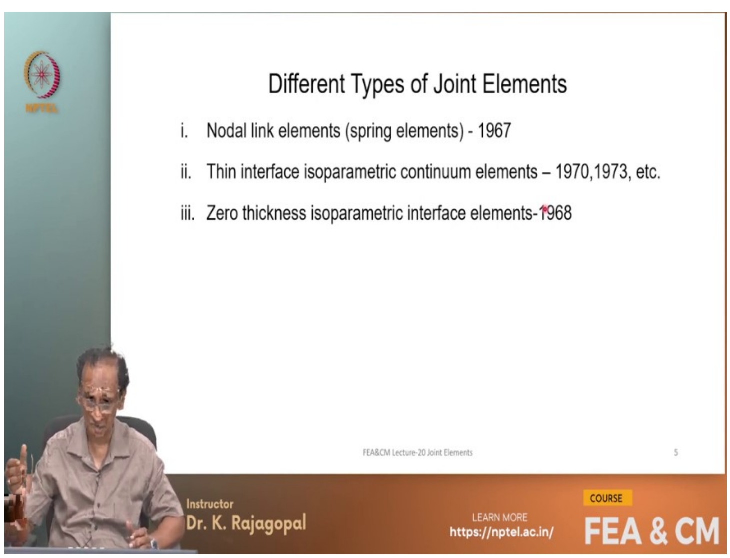
(Refer Slide Time: 03:57)



And even in the construction industry whenever we have a motor wall brick masonary wall we may need to to simulate the joints between these bricks so, that we can estimate the behaviour. Then in another example is between a retaining wall and then the soil. Let us say our retaining wall is deforming away from the soil either by rotation or by bulging out. Then what happens should the soil stick to the retain wall then we will transfer unnecessary magnitudes of the shear stress into the soil.

Like these are not real but because of the numerical improper modelling we could be transferring some additional Shear stresses that might lead to failure. And to prevent that we can we can place an interface between the wall panel and then the soil so, that the sliding relative sliding can take place against the retaining wall.

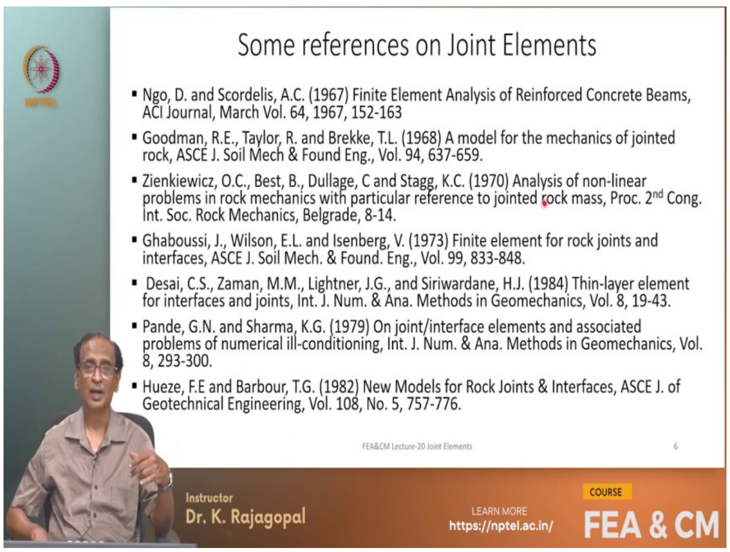
(Refer Slide Time: 05:14)



So, there are different types of joint elements that have been developed in the past 50, 60 years. Originally in 1967 is not exactly the joint element that we know as of now but it was to simulate the slip between reinforced concrete and then the steel reinforcement and they called these elements as nodal link elements way back in 1967. And later on people started using thin isoparametric elements the Continuum elements.

And then later the zero thickness isoparametric elements have come in 1968 and then 1970s and 1981 there were several papers.

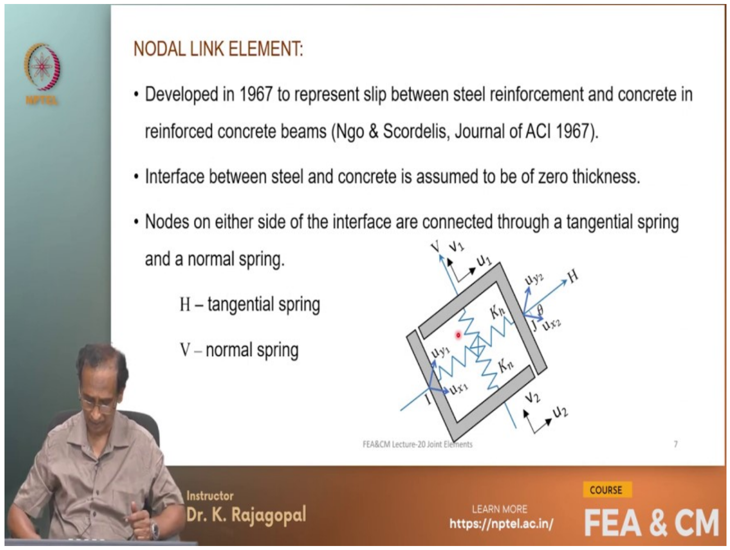
(Refer Slide Time: 06:07)



See these are all some of the papers that were published on the joint elements and the first one that was published in 1967 by Ngo, D and Scordelis that was for finite element analysis of reinforced concrete beams and in that paper they proposed one simple element that can simulate the slip between the steel reinforcement and then the concrete. And then later on Goodman Taylor and Becky and others they started working on zero thickness joint elements.

Initially the normal joint elements but later isoparametric joint Elements which are compatible with our isoparametric continuum elements.

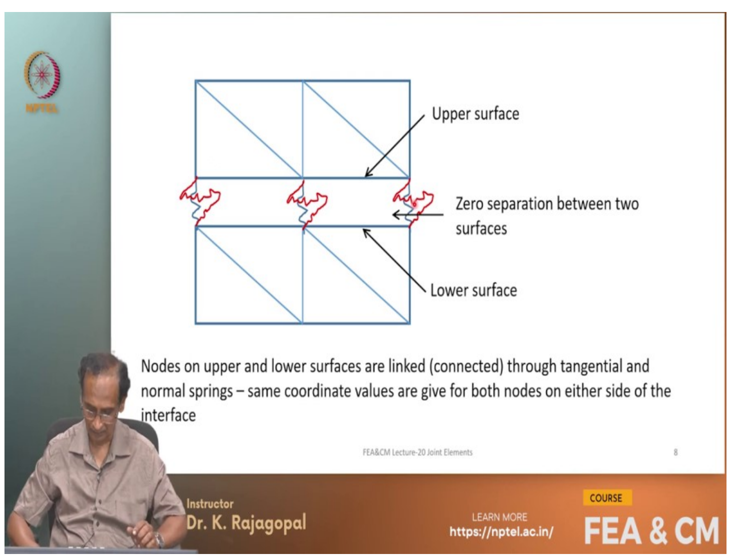
(Refer Slide Time: 07:03)



And the element that was developed in 1967 by Ngo and Scordelis is something like this is actually you connect 2 points one point on the concrete and the other point on the on the steel through 2 springs one is a normal spring and then the other is a tangential spring. Like this say this K n is a normal spring between the concrete and the steel and the K h is the tangential spring between the concrete and the steel.

And the displacements on both sides of the interface are referred as u 1 v 1 u 2 v 2 where u sir the tangential displacements and then the v is your normal displacement with respect to your interface and our H is the tangential spring and v is the normal spring. So, we need to define what is the tangential direction so, that we can assign the stiffness to these elements properly.

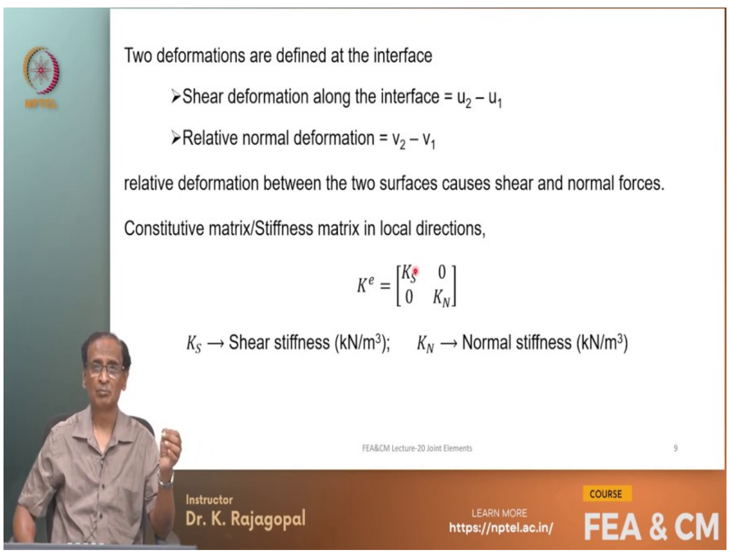
(Refer Slide Time: 08:29)



And here it is say here we have a an upper surface like this. And then a lower surface one could be really one could be related to concrete and the other could be related to steel and we have 2 springs the black coloured ones are the normal springs 1 2 3 and then this red colour ones are the shear springs and there is no separation between these 2 surfaces. In fact the separation that I am showing is only just for illustration purpose but it is not so in reality.

And there are 2 nodes defined with the same coordinates and these 2 nodes are connected with the normal spring and then the tangential spring and we have somehow defined the tangential direction for these spring elements that we do by defining 2 additional nodes. Basically this element has 2 nodes between which these 2 Springs the normal and tangential Springs are connected but then we need 2 other nodes to define the tangential direction.

(Refer Slide Time: 09:40)



And we will have a Shear deformation along the interface that is u 2 minus u 1 where u 2 and u 1 are the shear displacements tangential to the interface and the relative normal deformation is v 2 minus v 1 that is normal to the interface direction. And these are relative to the interface these are not global displacements. See the global displacements could be u x and u y and we need to apply the direction cosines to convert the global displacements to the element directions.

➢Shear deformation along the interface = $u_2 - u_1$

➢Relative normal deformation = $v_2 - v_1$

And we can write the constitutive stiffness as K s and K n basically there are 2 stress components and so, we will have 2 stiffnesses one is in the shear direction and the other is in the normal direction and our there is there are no strains because this being an element of zero thickness we cannot define strain per se because there is no length or there is no thickness we define only the relative displacement between the 2 surfaces.

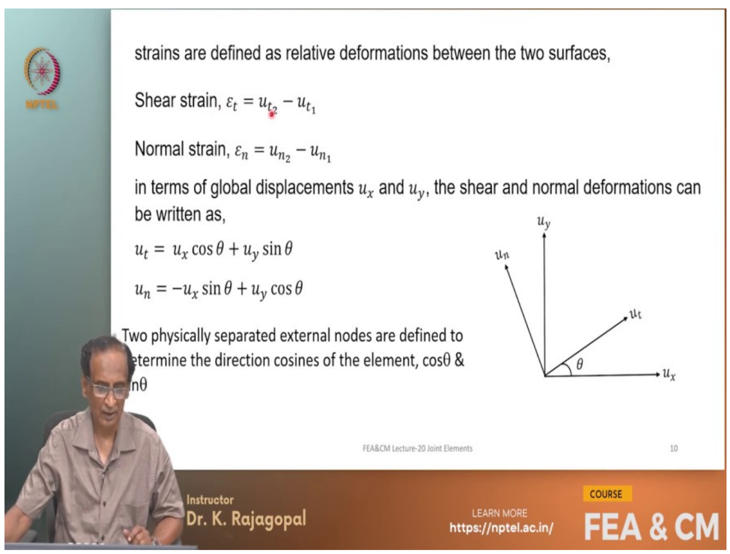
Constitutive matrix/Stiffness matrix in local directions,

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

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

So, if you have 2 surfaces whether they are sliding against each other or they are separating out or they are compressing against each other and that is enough for us to calculate the stresses by multiplying the relative displacement with these stiffness coefficients case and K n and the units for these are the F by L Cube units kilo Newton per cubic meter. So, that this multiplied by our displacement relative displacement will give you the stress either the shear stress or the normal stress.

(Refer Slide Time: 11:36)



And the shear strain is u t 2 minus u t one actually the subscripted t refers to the tangential direction to be more specific Epsilon t is the relative Shear displacement between the 2 nodes node one and node 2 and the normal strain Epsilon n u n 2 minus u n one that is the relative normal deformation.

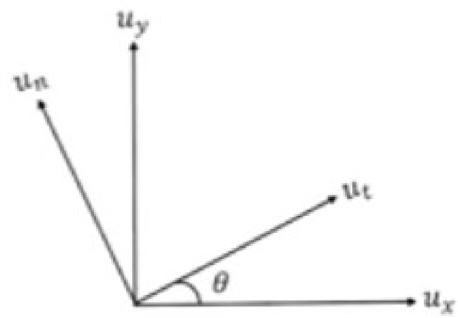
Shear strain, $\varepsilon_t = u_{t_2} - u_{t_1}$

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

And in terms of the global displacements u x and u y we can get the local displacements by applying our direction cosines cosine Theta and the sin Theta like this.

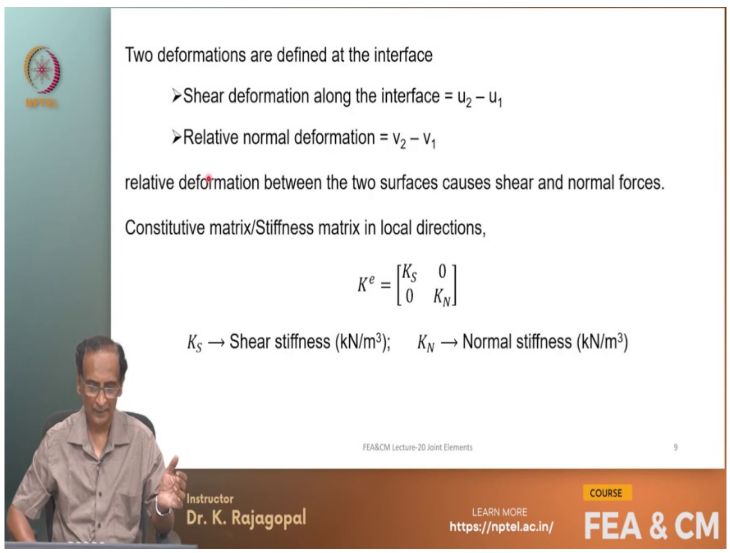
$$u_t = u_x \cos\theta + u_y \sin\theta$$

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



And then apart from these 2 nodes node one and node 2 we define 2 other nodes node 3 and node 4 for getting our tangential direction.

(Refer Slide Time: 12:36)



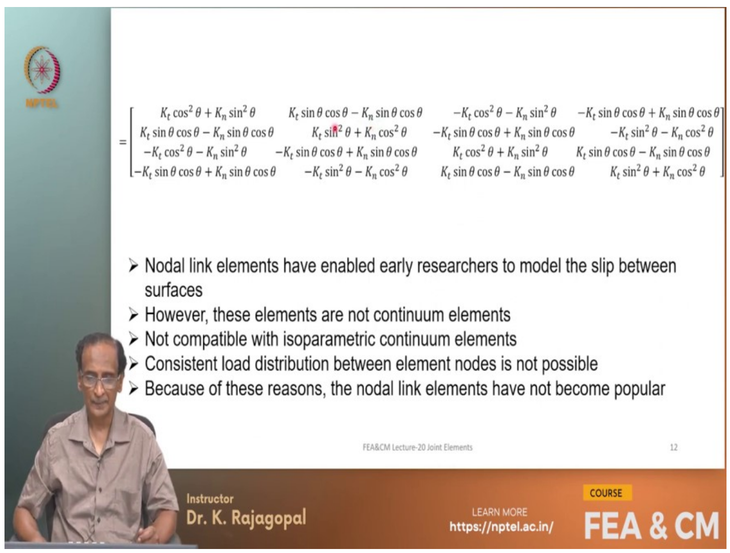
And our Epsilon t and Epsilon n are the shear strain and the normal strain can be obtained in terms of the Cartesian displacements are the 2 nodes 2 and one like this. And then we can get our global stiffness matrix as a transpose C A where C is our K s and K n the constitute matrix. So, the C is this and our direction cosine matrix is this to convert from the global sorry from the local directions to the global directions.

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

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

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

(Refer Slide Time: 13:22)



So, actually this looks very similar to our bar element stiffness matrix because we have the same terms like cosine Square sine square and so on and cosine Theta sin Theta for all the off diagonal terms. And so, these nodal link elements that were developed in 1967 they enabled early researchers to model the slip between the 2 surfaces the concrete and the steel however these are not Continuum elements.

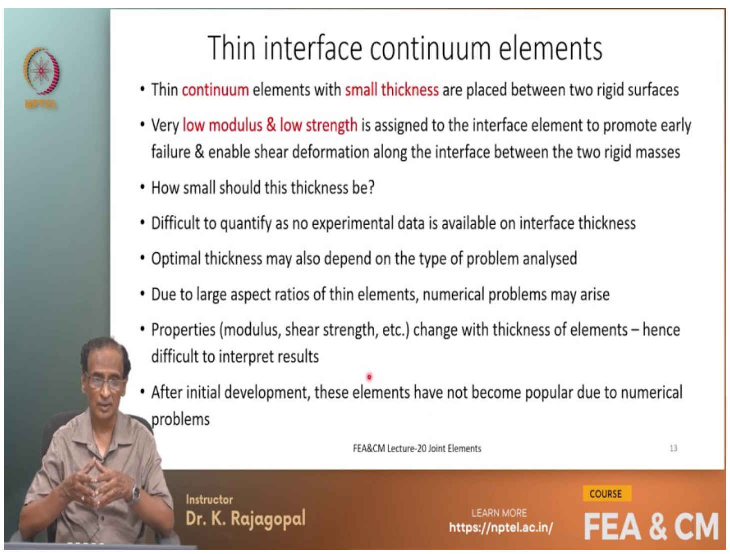
$$= \begin{bmatrix} K_t\cos^2\theta + K_n\sin^2\theta & K_t\sin\theta\cos\theta - K_n\sin\theta\cos\theta & -K_t\cos^2\theta - K_n\sin^2\theta & -K_t\sin\theta\cos\theta + K_n\sin\theta\cos\theta \\ K_t\sin\theta\cos\theta - K_n\sin\theta\cos\theta & K_t\sin^2\theta + K_n\cos^2\theta & -K_t\sin\theta\cos\theta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

So, actually these are a discrete provided at discrete points and these are not really compatible with the isoparametric elements. Because with the in the case of isoparametric elements we have seen that the corner nodes they will have some weight factors and then the mid side nodes they will have some weight factors and in the case of a 9 node quadrilateral the center node will attract more loads and so on we have seen.

But when we use these this nodal spring elements we need to externally compute these stiffness coefficients and that may not be possible especially if you have a curved surface how do we deal with the stiffness because that depends on the curvature and so on and we cannot get a consistent load distribution between the element nodes by using the nodal spring elements along with isoparametric elements.

So, these nodal link elements they have not really become popular because of these reasons. Originally in 1960s the Continuum element that was used was only 3 node triangle in which all the nodes have equal weightage but later we started using the 6 node triangles 8 node quadrilaterals 9 node quadrilaterals and so on where the stiffness contribution at different nodes is different that comes from the shape of the element.

(Refer Slide Time: 15:45)



And later people started using the thin Continuum elements. The idea is that if we place a thin Continuum element and with some reduced properties for the interface you can promote the failure along this preferential directions that is along the along our weak plane and we assign very low modulus and very low strength to these interface elements to promote early failure and enable Shear deformation.

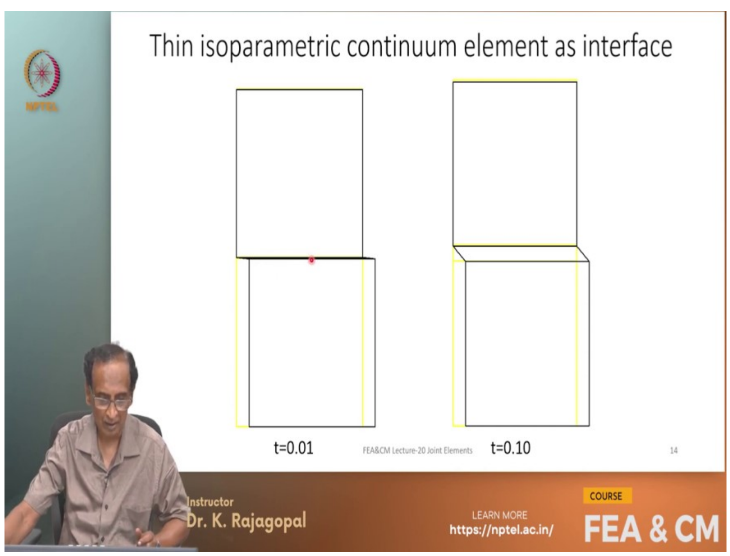
But then small thickness means how small should it be whether it should be a 0.1, 0.01, 0.0001 and so on. And the main problem is we do not have any experiment to quantify the the interface thickness because that are that depends on the type of soil and then the type of interface whether it is a smooth interface or a rough interface and whether your soil is going to dilate in that case the effect of the interface might propagate deep into the soil and so on.

So, the optimal thickness for the interface might change with the type of problem that we have. So, it is and then when use thin elements the aspect ratio could be very long very large say the length to thickness ratio it would could be very high. But ideally the for most accurate results the length to thickness ratio cannot be more than about 2 to 3 or at the limit about maybe 5 to 6 more than that we may start having a numerical issues especially depending on the type of elementary that we have.

And the other problem is with the different thicknesses that use for the interface your properties might change the modulus that use the shear strength properties that use you may have to calibrate a little bit so, that you can get some reasonable results. So, because of these reasons these elements have not become very popular and another reason is that the numerical issues.

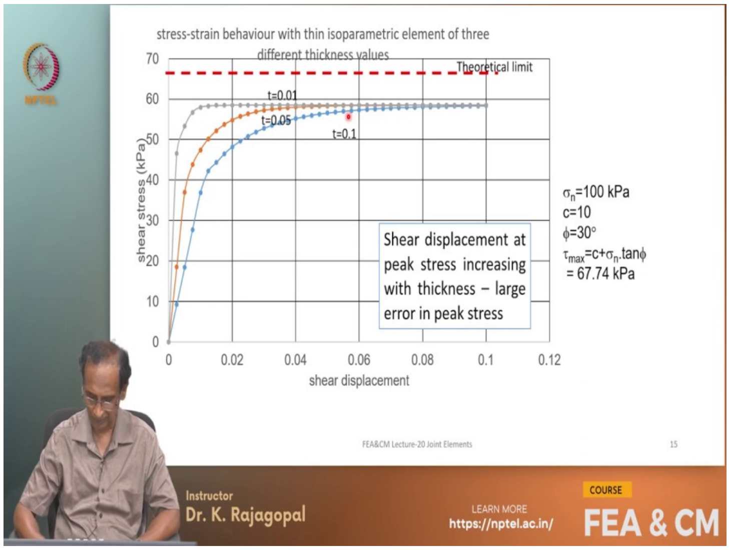
So, we could have like singularity problems or some unnecessary modes of deformation might develop because of these numerical issues.

(Refer Slide Time: 18:31)



So, let me show you 1 example of using the thin interface element. Say here we have 2 blocks the upper block and the lower block and then these 2 are connected with a thin isoparametric element in fact all of them are 8 node quadrilaterals. Even this thin element is an 8 node quadrilateral and the thickness here is 0.01one and here its 0.1. And then is actually it is the shear deformation applied for the lower part against the upper part. And we are going to monitor the shear stresses that are developed here.

(Refer Slide Time: 19:20)

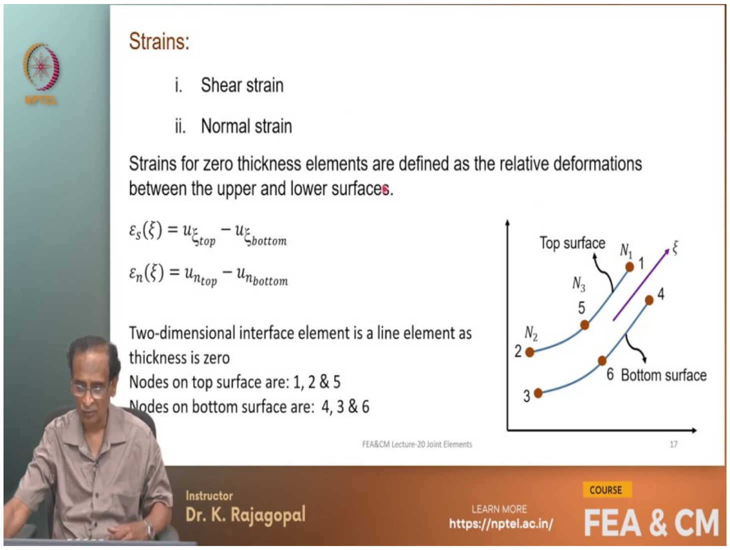


And if you plot a graph between the shear displacement and then the shear stress these are the different graphs and you see with the different thicknesses of 0.01, 0.05, 0.1 we get different result and of course the ultimate stress is the same because that is controlled by the by the plastic limit whereas the actual stress strain behaviour is different with the 0.01. The peak stress is reached at a very low displacement may be about 0.01.

But with point one it is reached almost at about 0.08 or 0.09 and the normal pressure that is applied on the interface is 100 kPa and the C is 10 Phi is 30 degrees and then the Tau Max as per our Mohr Coulomb relation is C plus Sigma and tan Phi that is a 67.74 let me just erase this. So, here our the theoretical limit is 67.74 but the finite element predicted one is less than 60 about 57 or something.

So, there is a huge difference it is actually as a percentage it is about 15 percent difference is there between the theoretical limit and then the finite element limit that is predicted. So, that means that this model is not able to to exactly replicate the strength of the interface. So, we need to go in for some other type of elements.

(Refer Slide Time: 21:15)



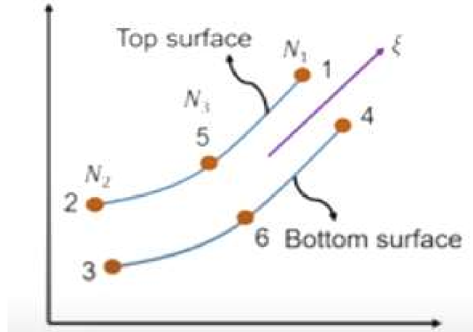
And later more recently thin interface sorry the zero thickness interface elements have become a very common. And initially they were developed in the generalized coordinate method that are compatible with say the the 3 node triangles or 4 node quadrilaterals and they are only suitable for planar surfaces. But now we have the isoparametric elements that we can use these even for a curved surface.

Like let us say you have 2 cylinders one rotating against the other internally even that interface we can simulate by using these isoparametric conjoint elements. And once again we define these elements in terms of 2 strains Shear strain and then a normal strain and we have 2 surfaces upper surface and bottom surface and this interface could be made up of either 4 nodes or 6 nodes.

Four nodes means you can only have a flat a planar surface and with larger number of nodes like 6 or we can define with 8 nodes and so on we can have a curve. And denotes 1 2 5 are in the upper surface and nodes 3, 4 and 6 are in the bottom surface and let us define x i along the length and eta normal to the surface and although I am showing with its separation but in reality the coordinates of one and 4 are the same 5 and 6 are the same and 2 and 3 are the same.

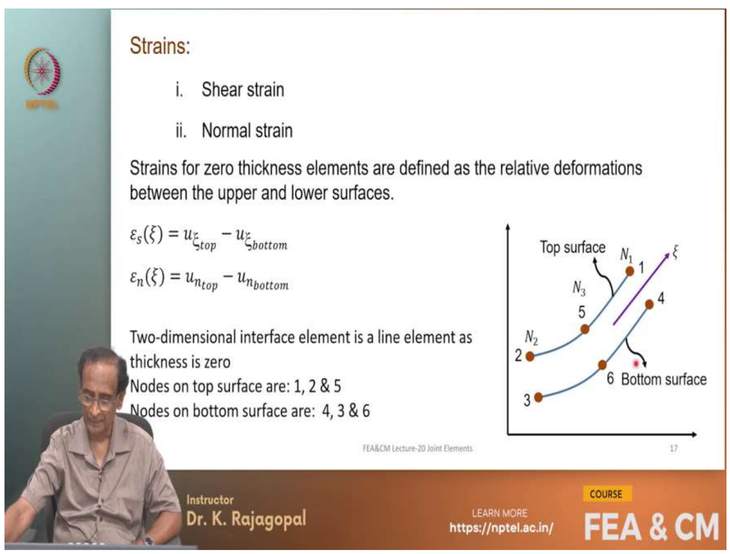
$$\varepsilon_s(\xi) = u_{\xi_{top}} - u_{\xi_{bottom}}$$

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



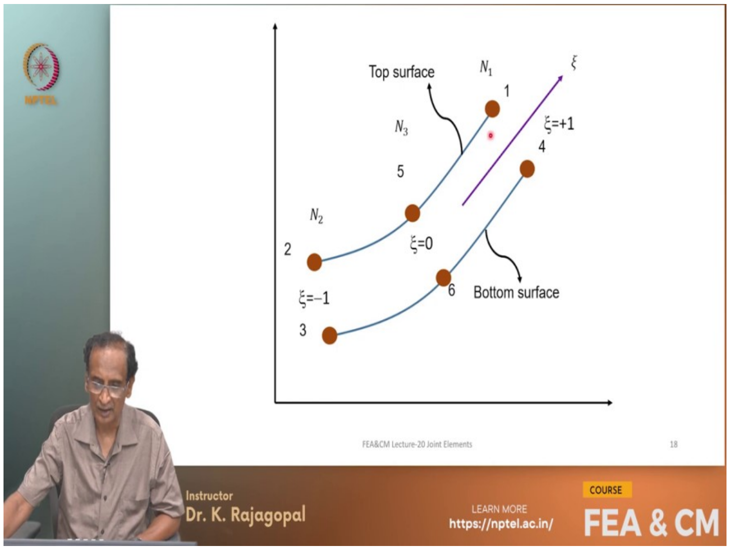
So, basically it is a line element but then we have some normal direction. So, Epsilon Shear is a u xi upper surface minus u xi bottom surface and the normal displacement or the relative normal displacement is the upper surface the normal displacement minus the normal displacement of the bottom surface. And by expressing this in terms of the local coordinates Xi and eta we can work in terms of our natural coordinates that vary from -1 to +1.

(Refer Slide Time: 23:52)



And here is a close-up. So, this Xi of plus 1 refers to nodes one and 4 and Xi of minus one nodes 2 and 3 Xi of zero at nodes 5 and 6.

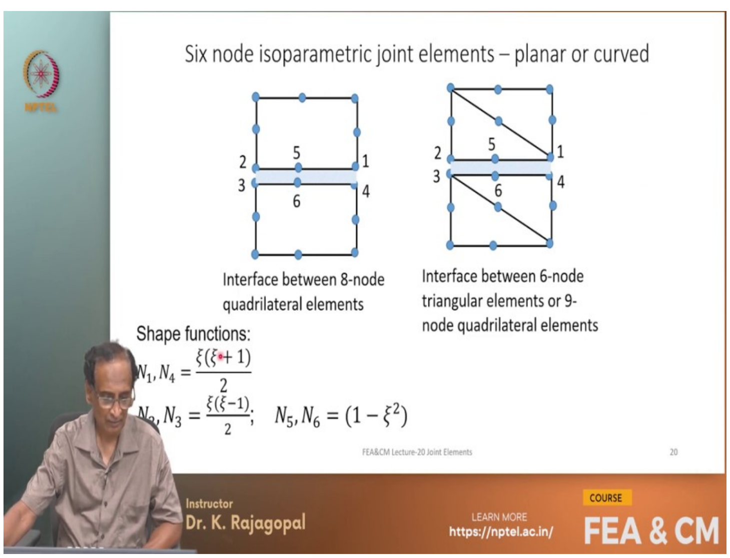
(Refer Slide Time: 24:08)



And when we have a lower order of element like a 4 node quadrilateral or a 3 node triangle we will only have a planar surface planar interface in that case we can have a 4 node interface one 2 3 4 nodes. And the shape functions are the mapping functions for nodes one and 4 or 1 + Xi by 2 because now it is actually it is a line element and n 2 and n 3 are one minus x i by 2 and when you have higher order element like an 8 node quadrilateral or a 6 node triangle we need the 6 node interface element to be compatible.

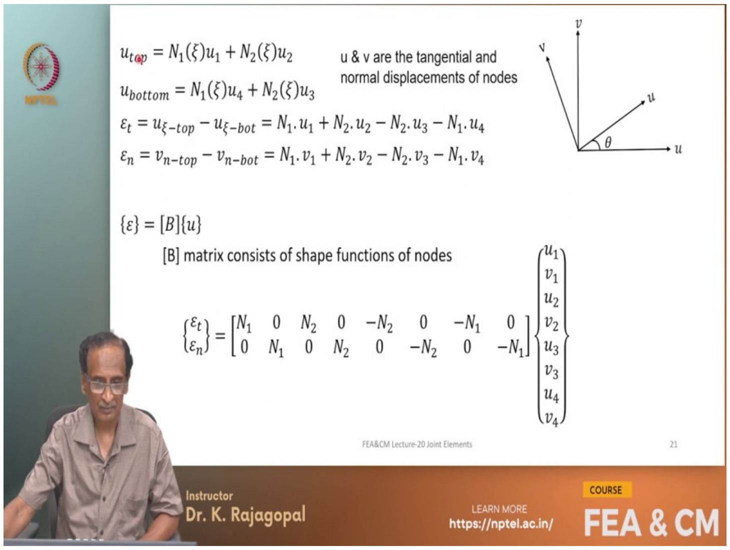
And the shape functions at nodes 1 and 4 are Xi times Xi + 1 by 2 and n 2 n 3 are Xi times Xi - 1 by 2 and at nodes 5 and 6 these are the mid side nodes and the shape functions are 1 minus xi Square. And you see these shape functions are similar to the ones that we had seen for for a one dimensional isoparametric element.

(Refer Slide Time: 25:27)



And the tangential displacement at the top node is n 1 u 1 plus n 2 u 2 or n one u 4 plus n 2 u 3 this is when we have 4 nodes and our Epsilon is BU that is the finite element convention that we had seen earlier with Continuum elements the strain is B times u where your B is the strain displacement matrix. In the case of Continuum the B matrix was a matrix of the shape function derivatives with respect to Cartesian coordinates but for the joint elements this is basically the B matrix is consisting of only the shape functions. Because now our strains are only relative displacements these are not really strains.

(Refer Slide Time: 26:20)



And in the case of 6 node element we have 3 nodes at the top and 3 notes at the bottom and our tangential displacement and then the normal displacement we can write in terms of the global displacements like this.

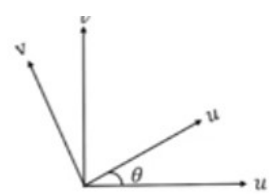
$u_{top} = N_1(\xi)u_1 + N_2(\xi)u_2$

u & v are the tangential and normal displacements of nodes

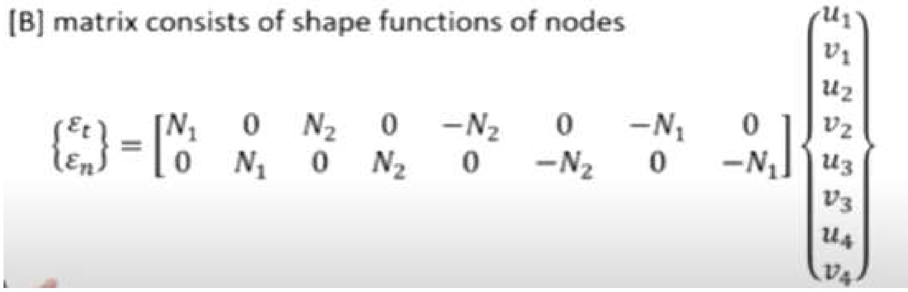
$u_{bottom} = N_1(\xi)u_4 + N_2(\xi)u_3$

$\varepsilon_t = u_{\xi-top} - u_{\xi-bot} = N_1.u_1 + N_2.u_2 - N_2.u_3 - N_1.u_4$

$\varepsilon_n = v_{n-top} - v_{n-bot} = N_1.v_1 + N_2.v_2 - N_2.v_3 - N_1.v_4$

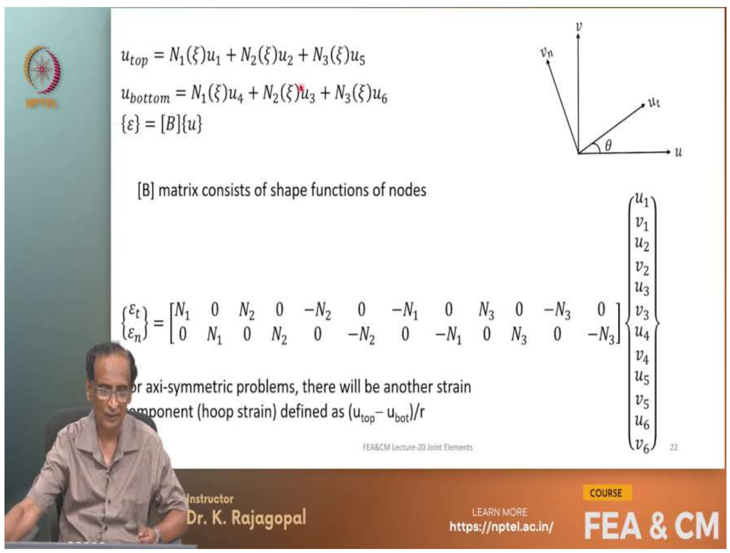


$$\{\varepsilon\} = [B]\{u\}$$



And the case of axis symmetric problems will have one more strain component that is the epsilon theta that is the circumferential or hoop strain. And u top minus u bottom divided by radius corresponding radial distance can be your hoop strain.

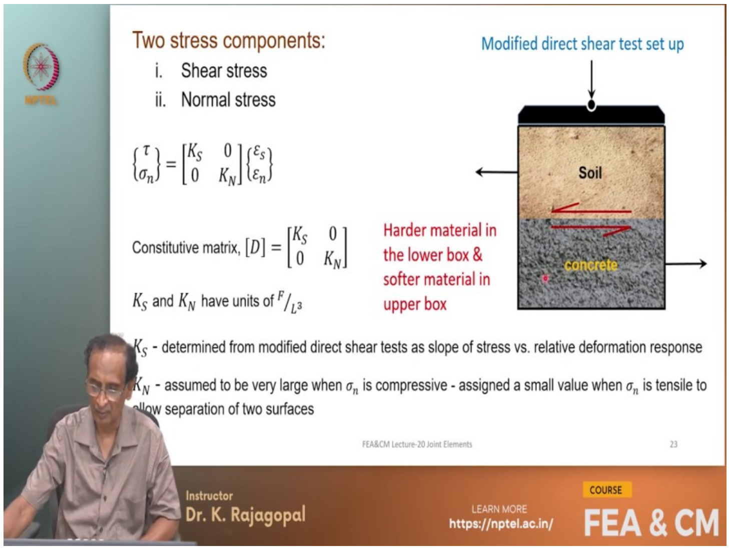
(Refer Slide Time: 26:59)



Then our stresses are shear stress and normal stress and the Tau we can obtain as K s times the relative shear displacement and the sigma n is K n times the relative normal displacement and our constituted matrix is this and our units are F by L Cube units. And the K s we can determine from the modified directional test that we perform between the 2 surfaces and the K n actually there is a no test that we can perform for determining the normal stiffness.

Because we do not really measure anything at the interface level we only measure at the top surface. And the K n is assumed to be very large when Sigma n is compressive and when Sigma n is tensile we set the K n to a small value not exactly zero but some small value so, that separation can take place between the 2 surfaces.

(Refer Slide Time: 28:13)



So, the modified direct shear box test is like this. So, all of you know how we perform the direct Shear test. So, in the lower box and upper box we fill with the soil and then do some compaction so, that the density of the soil is representative of the in situ density and then we move we move the lower box to cause some Shear deformation at the interface and then the corresponding stress we call it the shear stress.

Two stress components:

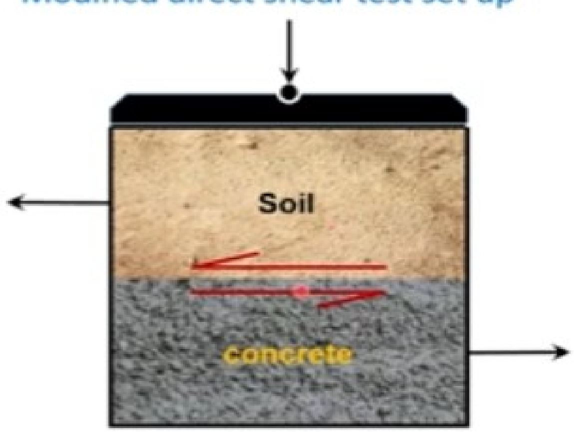
i. Shear stress
ii. Normal stress

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

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

K_S and K_N have units of F/L^3

Modified direct shear test set up



And then we can we can calculate the interface strength and here when we do the modified directional test we place the harder material in the lower box and then the softer material in the upper box. We do not do it the reverse way because if you place soft soil or a soft material here then hard material here and then if you apply the pressure loading your hard material might punch into the soft material.

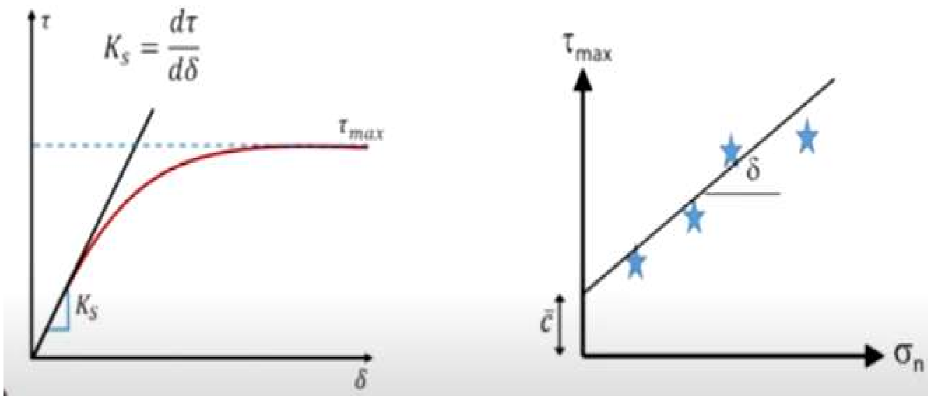
And then your hard material might be interfering with the shear movements because our Shear plane is a predefined as you know in the direction box it is predefined and we like our unless your Shear plane is on is on that surface will not be able to get the proper result. Suppose you place the concrete block here and it penetrates into the soil in the bottom box when we perform the test instead of measuring the shear strength of the interface we will be measuring the shear strength of the concrete.

So, we should be careful. So, we always place hard material in the bottom box and then the soft material in the upper box. So, from the direct Shear test we will get a graph between the shear displacement and then the Tau and then the slope of this will be your K s the shear stiffness is d Tau by d Delta and the Tau Max of the interface we can express as a c bar plus Sigma n Tan Delta where c bar and the Delta are the interface Shear strength properties.

$$\tau_{max} = \bar{c} + \sigma_n \tan\delta$$

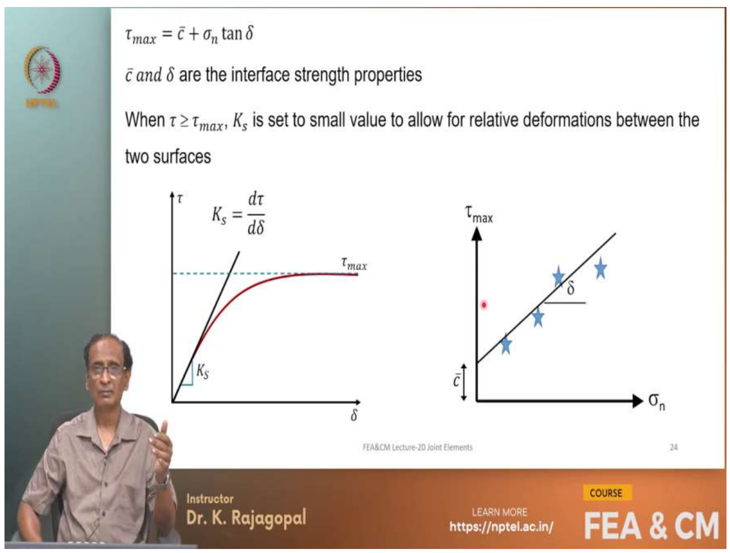
$\bar{c}$ *and* δ are the interface strength properties

And we can determine them from our modified direction test by performing tests are different normal pressures like this here. Let us say that we have performed 4 different tests or different normal pressures and then we can we can do some regression analysis and plot the best fitting line at the intercept on the y axis is your interface cohesion c bar and then the slope is your Delta.



So, it is actually it is very similar to how we perform how we perform the normal Direct shear test then how we determine the C and Phi of the soil.

(Refer Slide Time: 31:30)



And the stiffness matrix the local coordinates we can get in terms of our B and D matrices as B transpose D B although it is written as volume integrated over the volume but ours is a line element. So, we are going to do this integration from minus 1 to plus 1 in the iso parametric space multiplied by some Factor that relates the outer plane direction either the thickness or unit value or the radius in the case of axis symmetric analysis and because it is a line element will have only one dimensional integration that is in the Xi direction.

So, we have one weight factor and then the Jacobian matrix determinant of the Jacobian matrix J. Then once we get the stiffness matrix in the local directions we can get it in the global directions as Lambda transpose K Prime Lambda that is our target and transformation that we have seen earlier. And our Lambda is a similar to our transformation matrix that we use it in the case of bar elements.

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

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

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

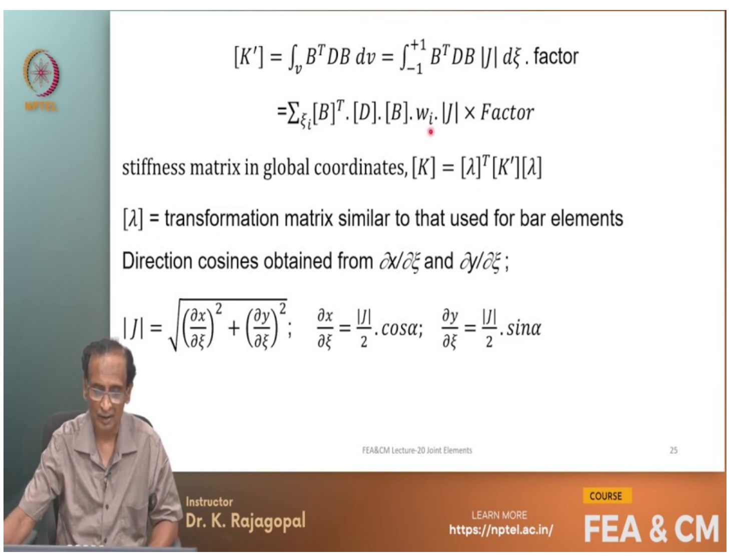
$[\lambda]$ = transformation matrix similar to that used for bar elements

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

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

And for these surfaces we can get our direction cosines Theta and the sine Theta from our dou Xi by dou Xi and dou y by dou xi and our determinant of the Jacobian matrix we can get as square root of dou x by dou xi whole square plus dou y by dou dou Xi whole Square and we have already seen in the context of the pressure loading that our dou X by dou XI is mod J by 2 times cosine Alpha and dou Y by dou Xi is mod J by 2 times sine Alpha and this cosine Alpha and sine Alpha are over direction cosines.

(Refer Slide Time: 33:33)

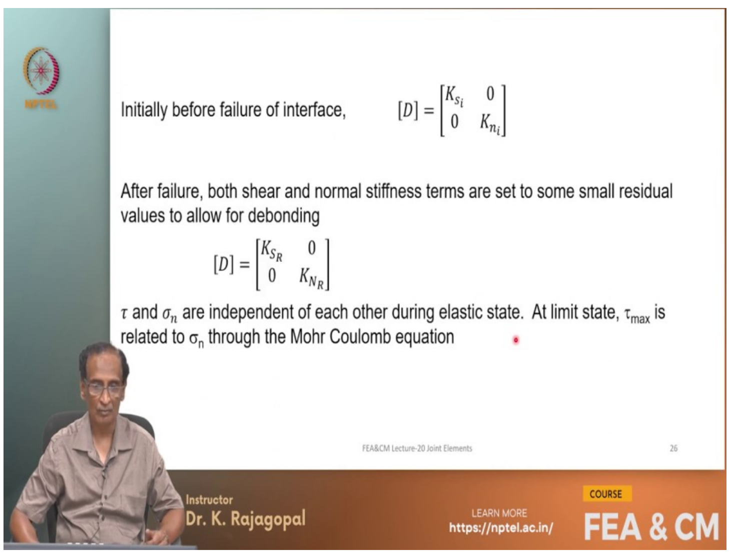


And how we operate with these interface elements is that before the failure we will assign some initial values K si and K ni a where K si is the one that we determined from the modified direction test and the K n is our normal stiffness usually we set it to a very high value and after we reach the limit state we can relax our K s we can set it to some small value and as long as our interface is in compression our 2 surfaces are going to be in contact.

So, we can give a very large value for this K n and only when we see some tensile stress like for example you are playing a tensile loading on the interface we would like the 2 surfaces to be separated out without offering any resistance in that case the K n can be set to a small value. So, we reformulate our constitute matrix D in terms of K sr and K nr. And so our actual this is more we will see more of it when we discuss the constitute of modelling.

But for now in the interface we strictly enforce some limit on the shears on the shear stresses that we generate in terms of the normal stress and then the shear strength properties of the interface. And so, in the elastic limit the Tau and sigma n could be independent of each other but after the failure are at the limit state the Tau max is related to Sigma n through the Mohr Coulomb parameters the as a C plus Sigma and tan Phi.

(Refer Slide Time: 35:42)



Now let us perform the same Shear test that we had done with the thin layer isoparametric element. So, here we have the interface here the upper box is fixed against lateral deformation similar to our direct shear box test and the bottom box is moved horizontally and the shear zone between these 2 boxes is connected by a 6 node joint element as we had seen earlier.

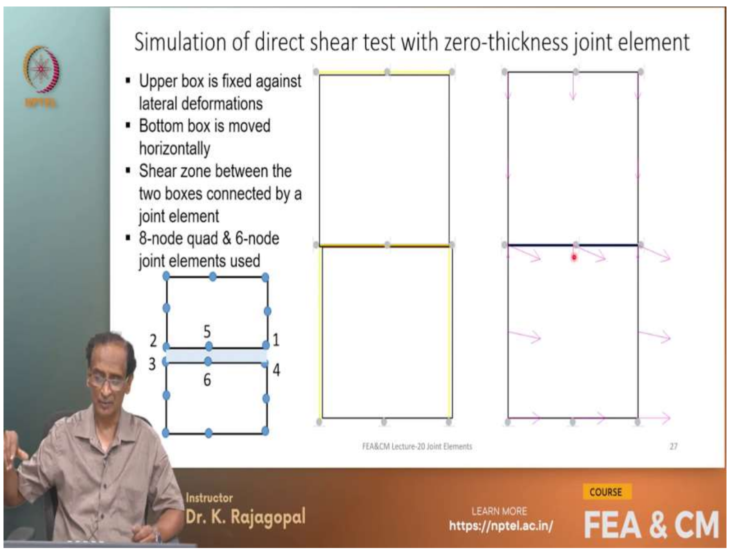
Initially before failure of interface, $[D] = \begin{bmatrix} K_{s_i}^{*} & 0 \\ 0 & K_{n_i} \end{bmatrix}$

After failure, both shear and normal stiffness terms are set to some small residual values to allow for debonding

$$[D] = \begin{bmatrix} K_{S_R} & 0 \\ 0 & K_{N_R} \end{bmatrix}$$

And this is how the deformed mesh looks like and then these are the the displacement vectors. Upper node is moving down whereas the bottom node is undergoing lateral deformations combined vertical and lateral deformations. So, we see some vectors like this.

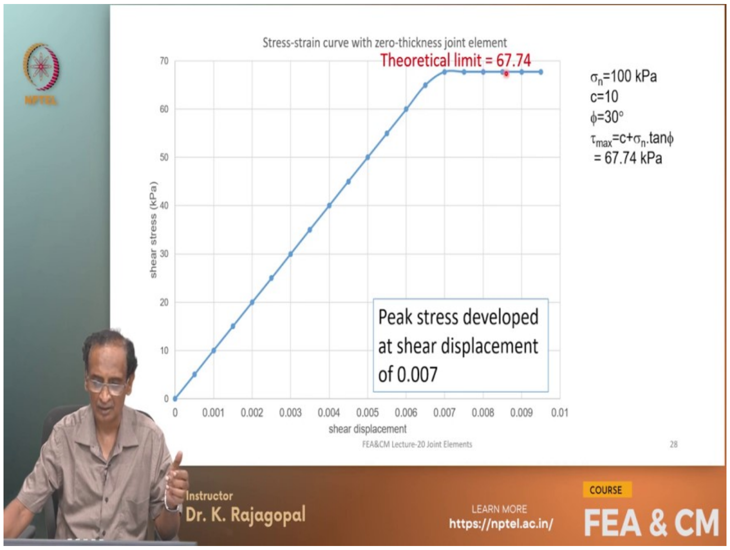
(Refer Slide Time: 13:22)



And this is our stress strain curve that we get on the x axis we have the shear displacement and the y axis we have the shear stress and is actually the theoretical limit is a C plus Sigma and tan Phi that is 67.74 and what is predicted by the joint element is also the same thing 67.74 its exactly equal to the theoretical limit. And the peak stress is developed at a displacement of 0.007 at a relative displacement of 0.007.

Whereas in the previous case when we model the interface through thin layer elements that deformation was varying a lot depending on the thickness feels a very thick element the peak stress was happening at a very large deformation and with a very small thickness its happening faster and so on. But then in none of the 3 cases that we had seen earlier we were able to reach this theoretical limit. See previously it stopped at about 57 or 58 whereas the theoretical limit is 67.74.

(Refer Slide Time: 38:06)



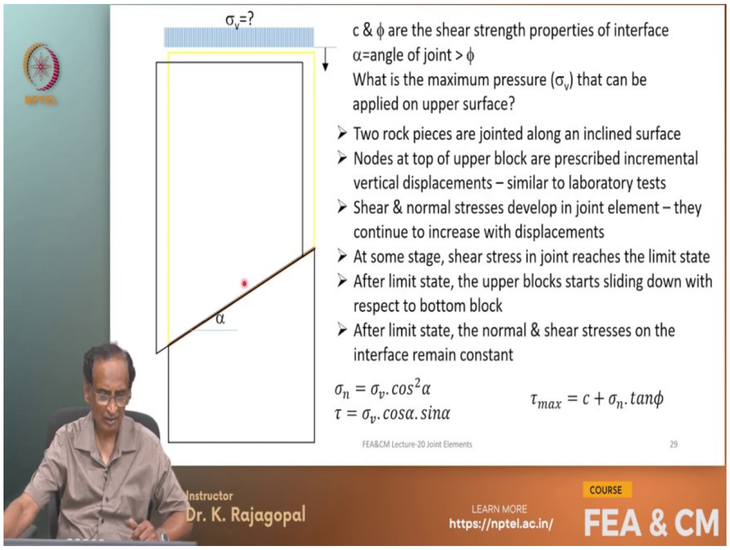
Now let us apply this joint element to some practical problem. Let us say that we are doing a triaxial compression test on a cylindrical rock sample with a joint like this. And this joint is at an angle of alpha and we would like to know what is the maximum pressure that we can apply on the surface Sigma v. The C and Phi are the shear strength properties of the interface and Alpha is the angle of the joint and that is greater than Phi.

And we would like to know what is the maximum pressure that we can apply without failing this interface and see how we can perform is we do not know what is the what is the maximum pressure. So, if we do not know we cannot really apply any pressure. And see if you look back on how we perform this Laboratory test all these strength tests are mostly displacement control test in our direct shear test or in the triaxial compression test.

We deform the sample and then and then find the response or how much force is developed and the same thing we can do we can apply equal vertical displacements or the upper node upper surface and then see what is the reaction force developed. And as we are moving the upper surface the stresses will be transferred to the interface element and then it will develop some normal stress and then the shear stress they will continue to increase with the displacement.

Then at some stage the shear stress on the interface might reach a limit state right. And then after that at the upper block will simply slide relative to the lower box. In fact this is what you see here after some deformation the shear strength of the of the joint is reached and then the upper block will just simply slide. And so, after the failure or the limit state the normal and Shear stresses on the interface will remain constant.

(Refer Slide Time: 40:56)



So, we can define some theoretical values by giving some numerical values. So, on the shear plane the normal stress is Sigma v times cosine square alpha in fact these are easily derived by drawing the Mohr Circle and then you know that any line drawn from the pole will intersect the Mohr Circle and that gives the normal and shear stress and a plane parallel to that to that line.

And if you apply that you can easily derive that the normal stress on the interface is Sigma v times cosine Square Alpha and the shear stress is Sigma v times cosine Alpha sine Alpha right. And our the maximum stress on the interface Tau Max is C plus Sigma and tan Phi the C and tan Phi are the shear strength properties of the interface and the sigma n is the normal stress on the on the on the interface.

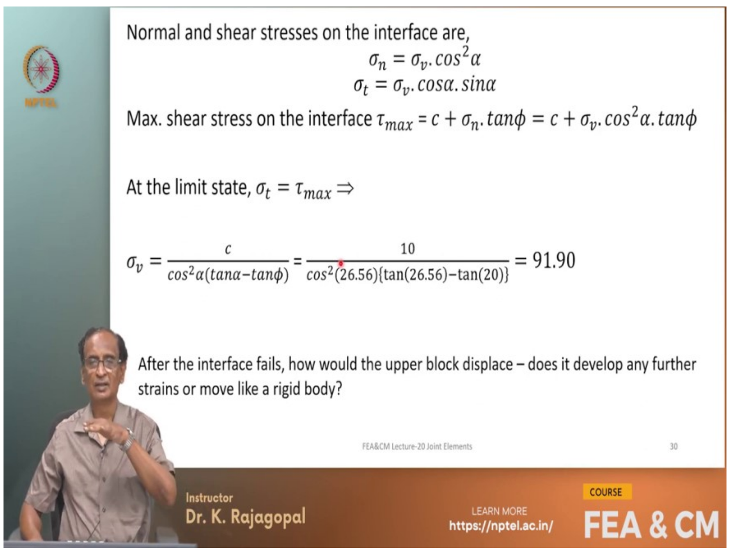
And this we can write as C + Sigma v cosine square alpha times tan Phi because our the maximum shear strength is C plus Sigma and tan Phi over Sigma n is Sigma v times cosine Square Alpha and so, we can get the maximum shear stress like this and at the limit state the sigma t should be equal to Tau Max and by equating these 2 we can get an value get an equation for Sigma v.

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

Like this Sigma v is a C by cosine Square Alpha times tan Alpha minus tan Phi. So, the C is given as 10 and then the alpha is a 26.56 and Phi is at 20 degrees and so, this is 91.90 actually if the alpha is less than Phi they will not be any failure you can apply any amount of Sigma V. So, that is the reason why I said that our Alpha is greater than Phi that is because of this theoretical limit. So, actually we are applying vertical deformations are the upper surface and then the upper block is moving.

So, after the interface reaches the limit state what would happen to the strains within the upper box will the strains continue to increase or will it move like a rigid body.

(Refer Slide Time: 44:04)



That we can we can see it by performing the finite element analysis and looking at the results. So, here this is our graph between the vertical displacement and then the vertical pressure and this is about 91.9 it is increasing the vertical pressure goes on increasing and at some stage it will reach the limit when your interface fails. And these dots are the locations where our stresses are computed in the Continuum element.

$$\sigma_v = \frac{c}{cos^2\alpha(tan\alpha - tan\phi)} = \frac{10}{cos^2(26.56)\{tan(26.56) - tan(20)\}} = 91.90$$

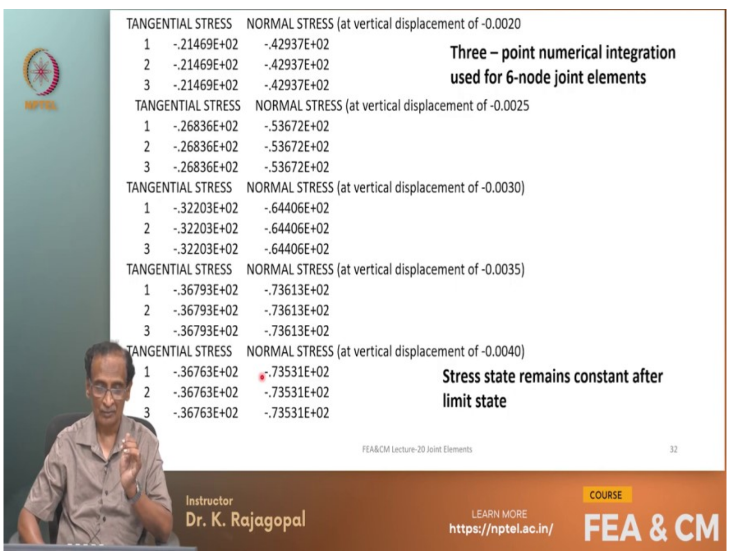
And the continuum stiffness matrix of the Continuum is formulated in terms of 2 point integration 2 points in each direction and then for the interface element it is done with the 3 points because it is a 6 node element and if you look at the stresses and strains in the in the upper block. So, I am looking at this point any point like you can see at any point see our stress vertical stress is going on increasing with the strain 6.7, 13.4, 20.1, 26 and so on.

And its increasing up to about is increase slightly to 92 because at that point the shear limit comes into picture initially slight increase has happened because before only after the stresses are calculated we check for the yields limit and then correct. So, the maximum stress is 91.906. And you see here until that until this limit your stress vertical stress is increasing and then even your vertical strain is increasing.

But beyond this your vertical stress is remaining constant and then even your strain is remaining constant that means that beyond that point the entire body is sliding like a rigid body. So, it is from this point onwards our upper block is moving like a rigid body and our strains and the vertical stresses are remaining constant. So, in fact this is one of the requirements when we develop our shape functions.

If there is any constant strain state that happens we should be able to represent and that is exactly what we are able to represent through this example.

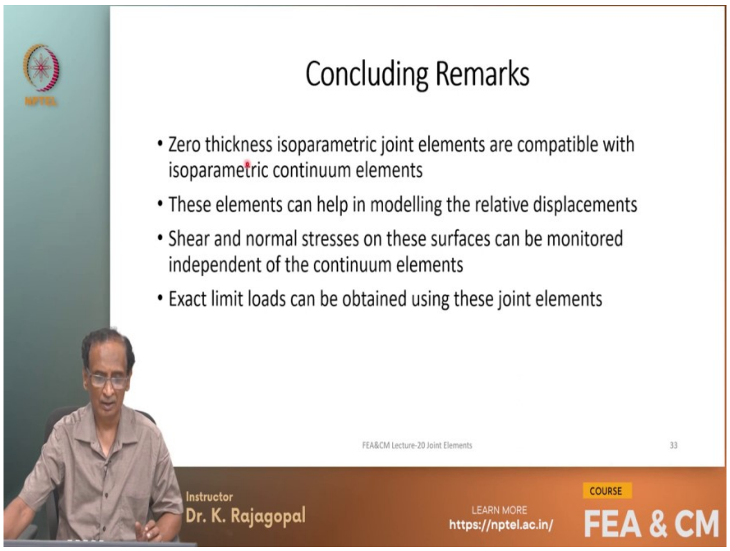
(Refer Slide Time: 46:59)



And let us look at the stresses that are developed in within the interface element. the This interface element is formulated in terms of 3 point integration 1 2 3 these are these look these refer to the in integration points and this is the shear stress and then the normal stress under different displacements. You see they are increasing up to up to this point the normal stress of 73.661 and then the shear stress of 36.79.

And beyond this point the normal and the shear stress they remain constant. So, after the limit state they do not change. So, here through this simple example we are able to demonstrate the use of interface element for for doing the they come modelling of the rock joints are some other weak planes like for example. So, you have 2 layers of soil and we can place an interface so, that we can capture the stresses that are that are active at the interface between the 2 layer between the 2 soil layers or between a retaining wall. And then soil we can place a the interface layer and see what is happening.

(Refer Slide Time: 48:40)



So, I think these just to conclude see we have seen zero thickness isoparametric elements which are compatible but then the main problem is we can have numerical issues. We could have especially if you have elements with very very large aspect ratio say a length of one and the thickness of point zero one means. So, the aspect ratio is 100. So, we are not sure whether the results that we get with that type of element are reliable or not.

So, we can go in for our zero thickness joint elements and these are versatile and we will see later on how we can apply them for modelling the joints between the retaining walls and then the soil are between geosynthetic reinforcement on the soil and so on. So, that we will see after we deal with constitute to modelling. So, this is the end of today's lecture and if you have any questions please write to me.