**Finite Element Analysis**

**Prof. Dr. B. N. Rao**

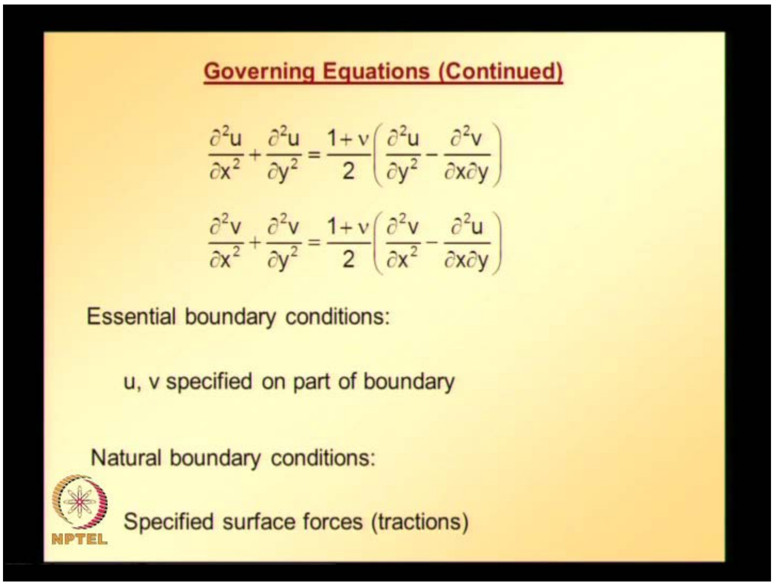
**Department of Civil Engineering**

**Indian Institute of Technology, Madras**

**Lecture No. # 35**

In the last class, we have seen the derivation of governing equations for elasticity problems under plane stress and plane strain conditions. For completeness, let us look those, what we did in the last class once again. We derived this second order differential equation starting with equilibrium equations in x in x direction.

(Refer Slide Time: 00:40)

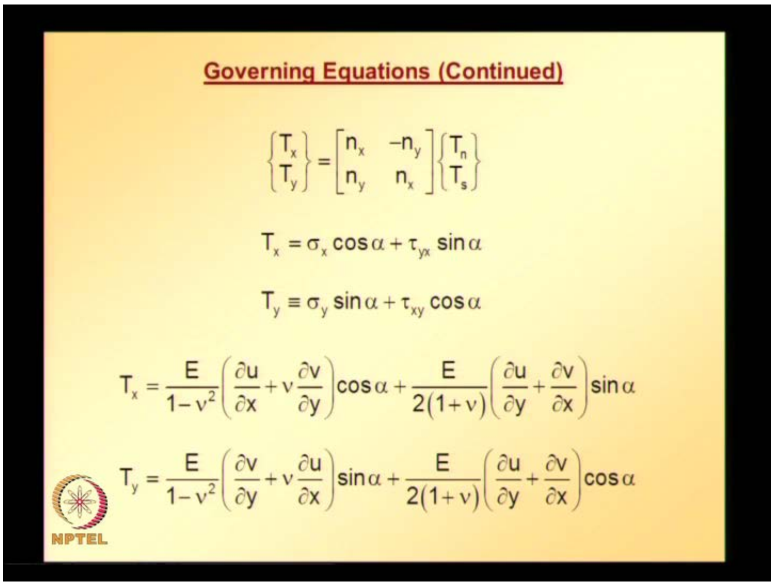



Substituting stresses in terms of strains using stress strain relation, and then substituting strains in terms of displacements. Using strain displacement relations, and after doing couple of mathematical manipulations, we finally arrived at this equation. And similarly, starting with second differential equation sorry second equilibrium condition, which is equilibrium condition in the y direction. Sum of all forces in the y direction is equal to zero. Applying that condition, we get the second equilibrium equation. And following similar steps, as we did for getting the first equation shown here that is expressing stresses in terms of strain; strains in terms of displacements.

And doing couple of mathematical manipulations finally, we arrive at the second differential equation, which is also second order differential equations. And so, solution for a three dimensional elasticity problem is basically solving these two coupled second order differential equations subjected to boundary conditions. u and v specified on part of boundary and you can notice here, these two are second order differential equations. So, those boundary conditions of order 0 are essential boundary conditions and those boundary conditions of order one are natural boundary conditions.

And natural boundary conditions are specified surface tractions and when I made this statement that, the specified surface forces are first order equations. That is based on what we did similar to what we did in last class, that is expressed tractions in terms of stresses and stresses in terms of displacements via stress strain and strain displacement relations. Finally, we can see tractions are related to first derivative of displacements, which are natural boundary conditions.

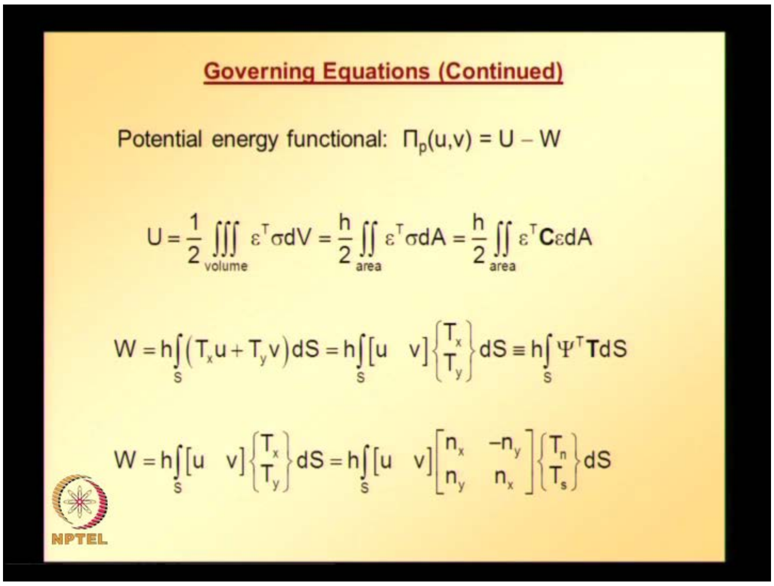
(Refer Slide Time: 03:23)




And usually, these tractions the prescribed surface forces in practical problems, we usually specify normal and tangential components. So, if one is interested in getting traction component in x and y direction form, the components which are specified normal to the surface and tangential to the surface. Then we can use this relation, which we also we have seen this in the last class and also tractions can be expressed in terms of stresses internal stresses; writing equilibrium equation for a triangle showing the traction surface tractions and internal stress components. In the last class, we have seen how to get this equation.

Similarly, tractions in the y directions can also be expressed in terms of stresses and now, replacing stresses in terms of strains and strains in terms of displacements. We can see that, traction in x direction is indeed related to displacements derivative of displacements. Similarly, traction in the y direction is related to derivative of displacements. So, natural boundary conditions are indeed first order equations. So, solving an elasticity problem is basically solving coupled coupled differential second order differential equations that we have seen. In the previous line subjected to the boundary conditions that u and v are specified on a part of boundary or tractions are specified on a part of boundary.

(Refer Slide Time: 05:29)

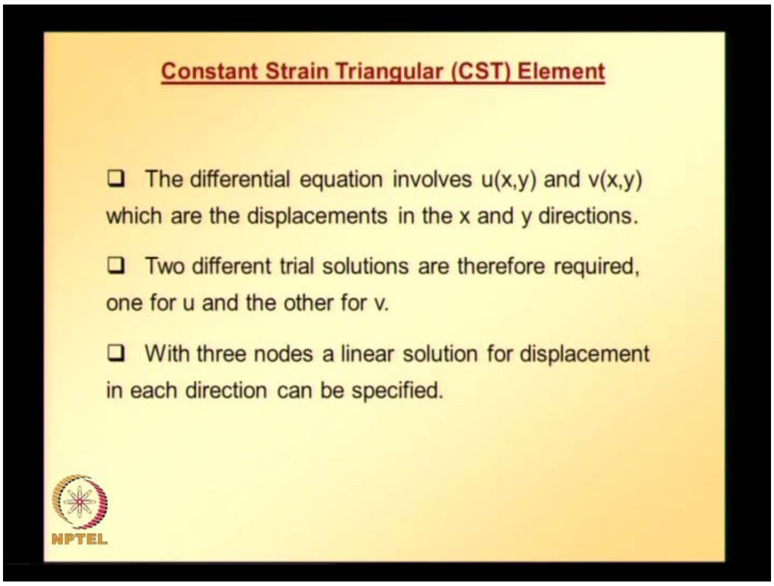



So, now before we proceed to derive the finite element equations for a specific element, we need to know, what is the equivalent potential equivalent energy potential for plane stress; plane strain problems are basically for elasticity problems is given by this. Potential energy functional is equal to phi is equal to U minus W, where U is nothing but strain energy and which is given by this equation and W is nothing but work done by the applied forces.

And if applied forces are point forces or concentrated forces, then work done by the concentrated forces is simply the product of applied forces and corresponding displacements at the point of application of load. But if work is if applied applied forces are distributed forces then, work done due to applied forces can be calculated using this relation. In both these relations U and W, h thickness is assumed to be constant. So, it is pulled out of the integral and if it is not constant, then we need to take inside the integral and then do the integration.

And if these tractions T x T y basically, if they are given in terms of T n and T s that is tangential and normal components, in that case work done by the distributed forces can be calculated using this relation. So, this is how, we can calculate U and W. Once we get U and W, we can plug in into phi; phi is equal to U minus W. And apply the condition that variation of phi is equal to 0 to get the final element equations. So, now let us start with by taking a triangular element three node linear triangular element and derive the element equations for that particular element.

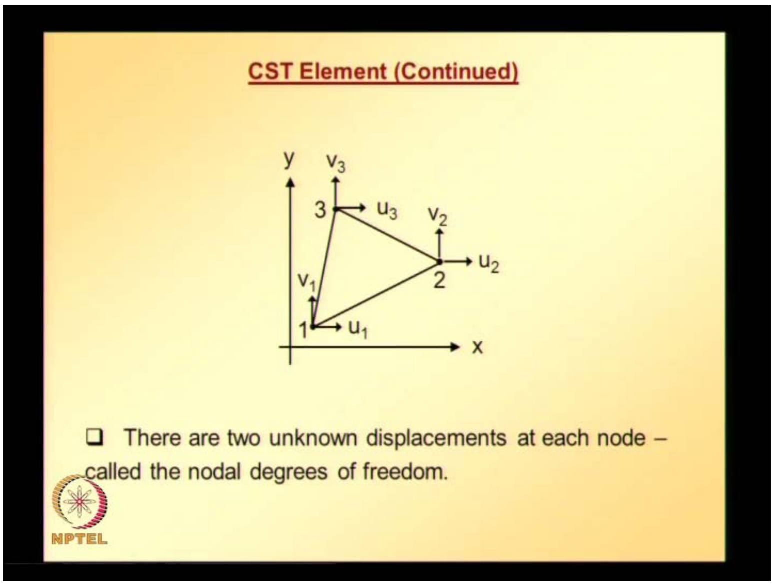
(Refer Slide Time: 07:49)




So, what we the element that we have taken is constant triangular element, because the three node triangular element strain is going to be constant over the entire element, which we have seen earlier and so it is called constant strain triangular element. So, the differential equations involve u and v, which are displacements in x and y direction that is what, you have seen a few minutes back. We need to solve coupled second order differential equations, which are in terms of u and v.

u and v are nothing but displacements in the x and y directions, which are functions of going to be functions of x and y. So, we require two different trial solutions; one for displacement in the x direction; the other for displacement in the y direction. With three nodes a linear solution for displacements in each direction for displacements in each direction can be specified. So, now let us consider a typical three node element and x try to express this displacement at any point in the element in terms of nodal values and shape functions.

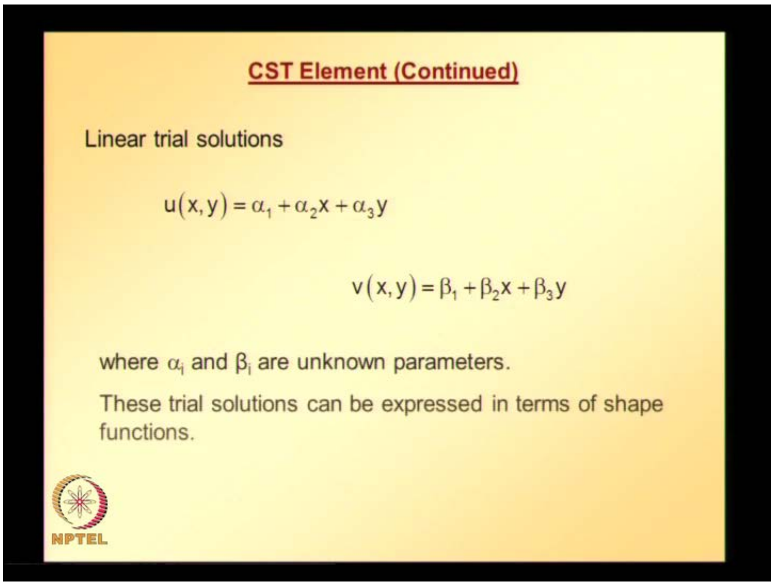
(Refer Slide Time: 09:28)




This is a typical three node element and as you can see from the figure, there are two unknown displacements at each node and they are called nodal degrees of freedom. And we can use linear trial solution for u displacements, because we have three nodes. So, when we are deriving based on polynomial based trial solution, the shape functions if we are deriving based on polynomial based trial solution then, we usually start out with a polynomial, which is having number of coefficients is equal to number of nodes.

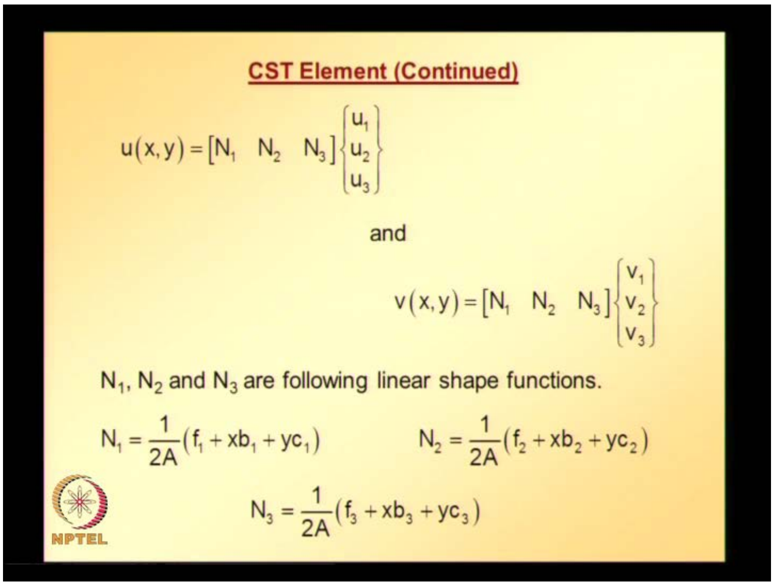
So, displacement in the x direction or displacement in the y direction consists of or the trial solution for displacement in x direction, displacement in y direction consists of a polynomial, which is going to have three coefficients. And that polynomial since we are dealing with two dimensional case here, that polynomial is going to be a linear polynomial in x and y directions. So, there are two unknown displacements at each node called nodal degrees of freedom.

(Refer Slide Time: 11:06)




Linear trial solutions, u is equal to alpha 1 plus alpha 2 x plus alpha 3 y. This is similar to what we did when we derived shape function for three node triangular element. If you recall, we started out with a polynomial something like t is equal to a naught plus a 1 x plus a 2 y. So, following similar logic we can express u, which is displacement in the x direction like this. Similarly, displacements in the y direction like this, where alphas alphas and betas are unknown parameters or unknown coefficients. Similar to a naught, a 1, a 2 you have seen earlier when we are deriving shape functions for linear triangular element starting with a linear polynomial. These trial solutions can be expressed in terms of shape functions using methods that we discussed earlier.

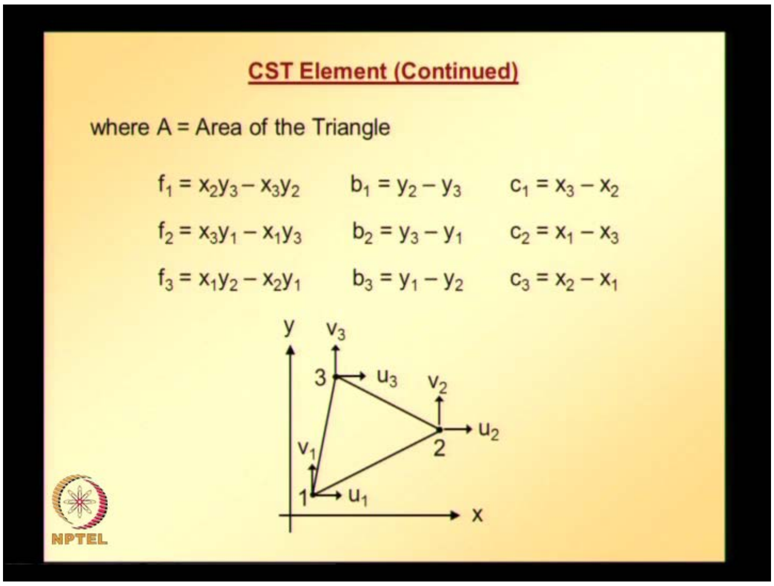
(Refer Slide Time: 12:36)




So, u can be written as N 1 u 1 plus N 2 u 2 plus N 3 u 3. Similarly, v can be written as N 1 v 1 plus N 2 v 2 plus N 3 v 3, which can be expressed in matrix and vector form in this manner. And if you carry out the multiplication, we indeed get what I mentioned just a while ago that is, u is equal to N 1 u 1 plus N 2 u 2 plus N 3 u 3; v is equal to N 1 v 1 plus N 2 v 2 plus N 3 v 3. Here N 1, N 2, N 3 are shape functions, which are going to be linear in x and y and the derivation of the shape functions is same as the procedure for derivation of the shape functional is same as what we have seen earlier.

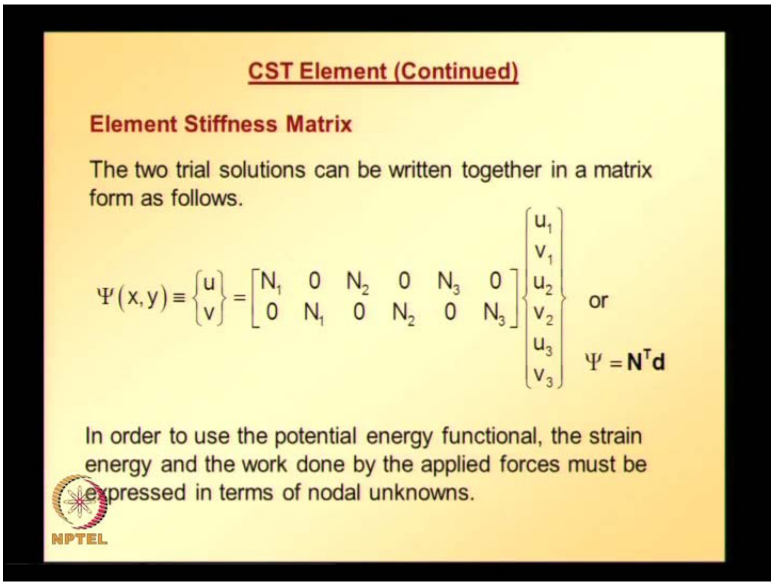
So, this N 1, N 2, N 3 shape functions can be calculated, once we know the nodal coordinates of all the nodes. So, N 1 is given by this, N 2 is given by this, and N 3 is given by this. Here, you can see some intermediate coefficients are defined f, b and c's. So, this f 1, f 2, f 3, b 1, b 2, b 3, c 1, c 2, c 3 these can be calculated based on information of nodal coordinates of that particular element, where here also there is another quantity capital A. It is nothing but area of triangle, which also can be found, once we know the nodal coordinates.

(Refer Slide Time: 14:17)




So, if this is a triangular element that we are looking at, then area can be calculated based on the nodal coordinates. And f 1, f 2, f 3, b 1, b 2, b 3, c 1, c 2, c 3 can be calculated based on the information of nodal coordinates. And this is how, we can express displacement in x direction and y direction in terms of finite element shape functions for that particular element and the nodal values. And this two trial solution that is displacement in the x direction and displacement in the y direction can be written together in one equation.

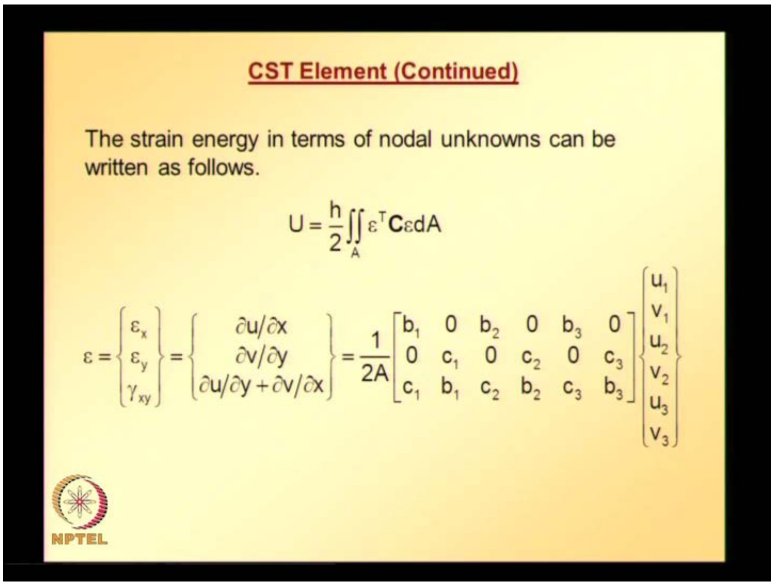
(Refer Slide Time: 15:29)



So, we will be looking at how to derive element stiffness matrix, because we have seen how to express displacement in x direction y direction in terms of finite element shape functions and nodal values. Now, we are ready to derive element stiffness matrix. As I just mentioned, the trial solution x and y direction can be written together in a matrix form. Or compactly, it can be written as psi is equal to N transpose d, where n comprises of all the or N is the vector or matrix consisting of all the shape function expressions.

And d is nothing but, vector of nodal parameters or nodal unknowns and here, this is how, displacement in x direction and y direction can be interpolated using finite element shape functions and nodal values. But if you go back and see, the strain energy equation u; it consists of strains. So, in order to use potential energy functional, the strain energy and work done by the applied forces must be expressed in terms of displacements or in terms of nodal unknowns. Strain energy in terms of nodal unknowns can be written can be written as follows.

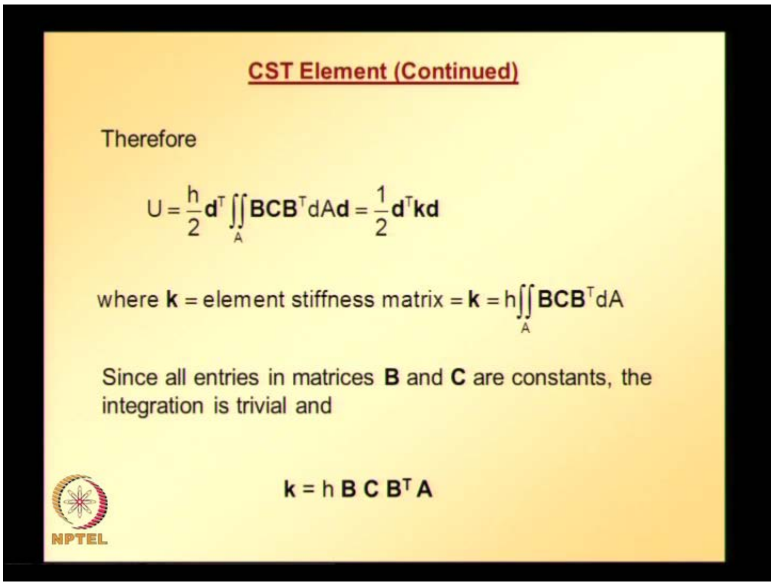
(Refer Slide Time: 16:54)




This is strain energy definition and strain is defined like this. derivatives It consists of derivatives of displacements with respect to x and y and just now, we have seen displacement u and v in terms of finite element shape functions and nodal parameters. So, we can take derivative of those and we can write it in a matrix form. Then, this is what we are going to get. Strain in terms of finite element shape function derivatives and nodal parameters, which can be compactly written as epsilon is equal to b transpose d.

So, now with this, we are able to express strain epsilon in terms of nodal parameters and b matrix consists of derivatives of shape functions. So, substituting this epsilon into the equation for u, we get this one.

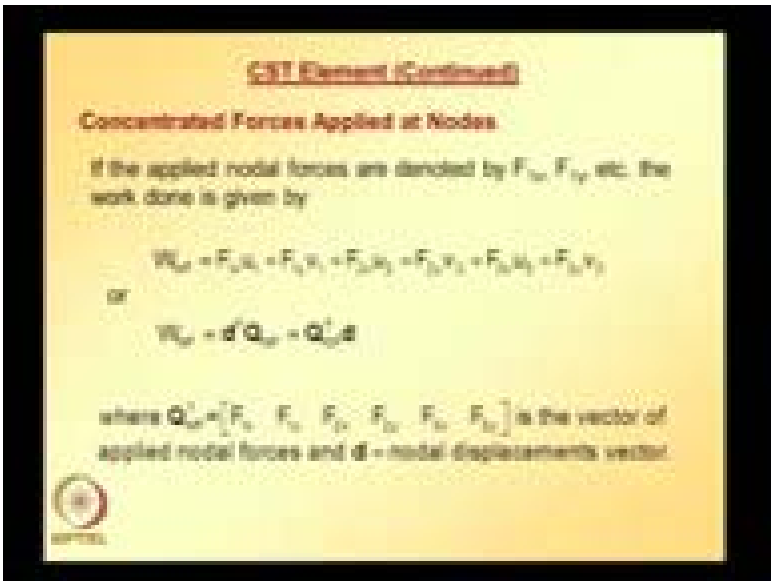
(Refer Slide Time: 18:29)




Where because nodal parameters are independent of are not are not functions of x, they can be pulled of the pulled out of the integral and stiffness matrix element stiffness matrix, k is defined like this. And if you notice B matrix that is strain displacement matrix for this particular element three node linear triangular element, it is a constant. It is not a function of spatial coordinates and if material that we are dealing with is also not a… material properties are also not functions of spatial coordinates. Then, C matrix can also be pulled out to the integral. Finally, this entire thing reduces to this.

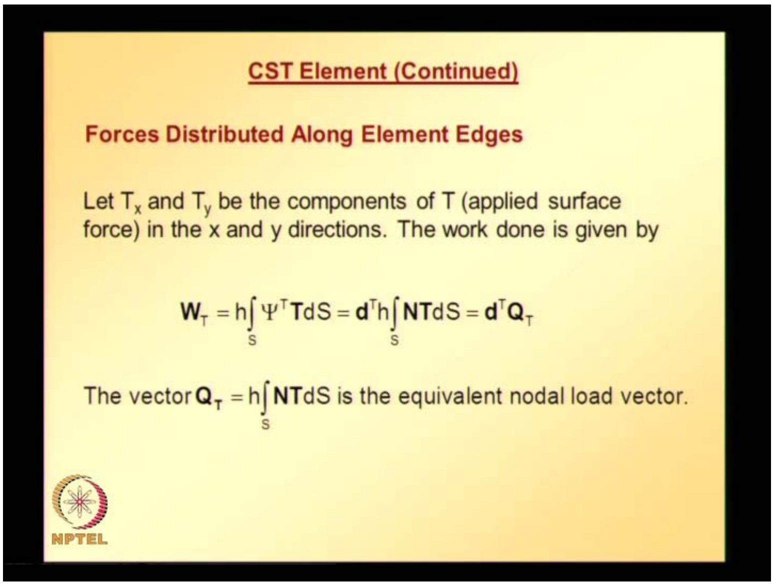
Please note that, this is valid only under the conditions that you are using three node linear triangular elements, which is constant strain triangular element. And if material properties are not functions of spatial coordinates x and y, only under that case stiffness matrix can be directly expressed like this; h times B times C times B transpose times area of triangle, that you are dealing with. So, this if not we need to perform this or we need to carry out integration to get the element stiffness matrix, for that we can you adopt some numerical integration techniques for corresponding to triangular elements, which we have seen earlier. So, now this is about strain energy and now, we need to look at work done by the applied forces.

(Refer Slide Time: 20:44)




Concentrated forces applied at nodes: It is very simple. If applied nodal forces are denoted by F 1 x F 1 y etcetera; that is nodal force applied at node 1 in x direction; nodal force applied at node 1 in the y direction, then the work done is given by the simply product of force at that particular node in the corresponding direction as the displacement and sum it up. We are going to get work done by the applied nodal forces concentrated forces. And this can be compactly written like this, where a vector consisting of nodal forces is defined and d as usual is nodal displacement vector, but if the forces are not concentrated forces, then forces distributed along element edges needs to be evaluated.

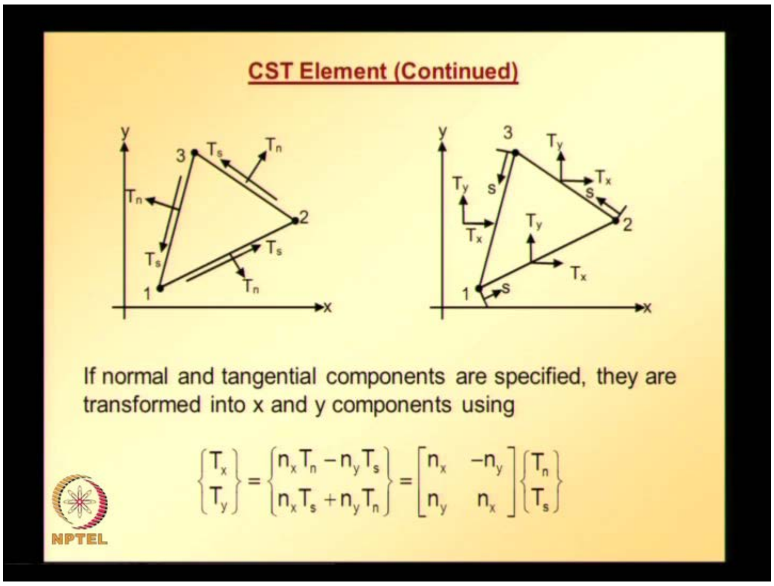
(Refer Slide Time: 21:55)




Let T x, T y be the components of traction in x and y directions. Then, work done is given by, logic is similar; force times force in a particular direction times displacement in the same direction is we what we have to do to get work done. So, it is given by this; work done due to traction. That is why, W subscript T is written is given by and again h thickness is h; thickness is assumed to be constant. So, h is pulled out of the integral and psi is nothing but, it is displacement vector in u direction consisting of displacements in the u direction, in the x direction and y direction. So, psi transpose T gives us displacement in x direction times T x displacement plus displacement in the y direction times T y.

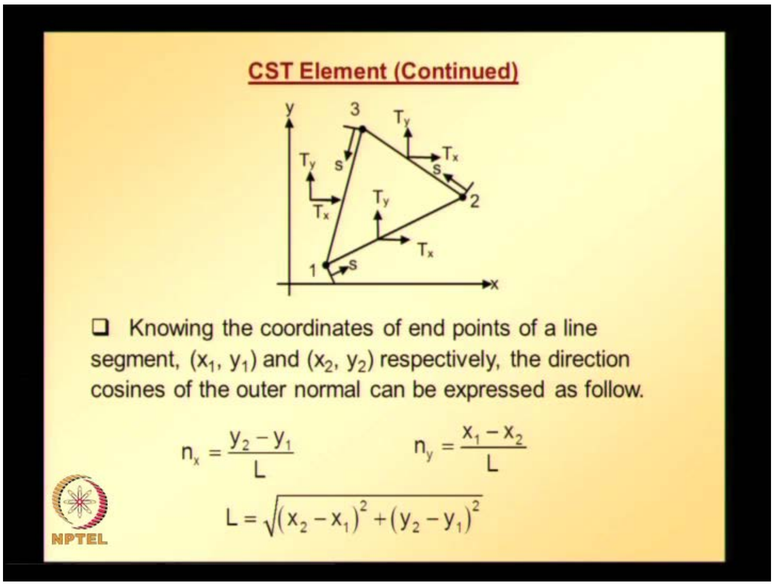
So, psi transpose T is that and substituting psi in terms of nodal parameters and finite element shape functions. We can get the we can further write this W as d transpose h integral N T d S, which can be compactly further written by defining a vector Q T, which is nothing but equivalent nodal vector. Because we have distributed load and that distributed load, we are converting into equivalent nodal loads. So, that is why Q is equivalent node nodal load vector. But, as we discussed if T instead of T x T y, normally we express or when we prescribe surface traction, it is easy to find what are the normal tractions and surface normal tractions and tangential components of traction.

(Refer Slide Time: 24:14)



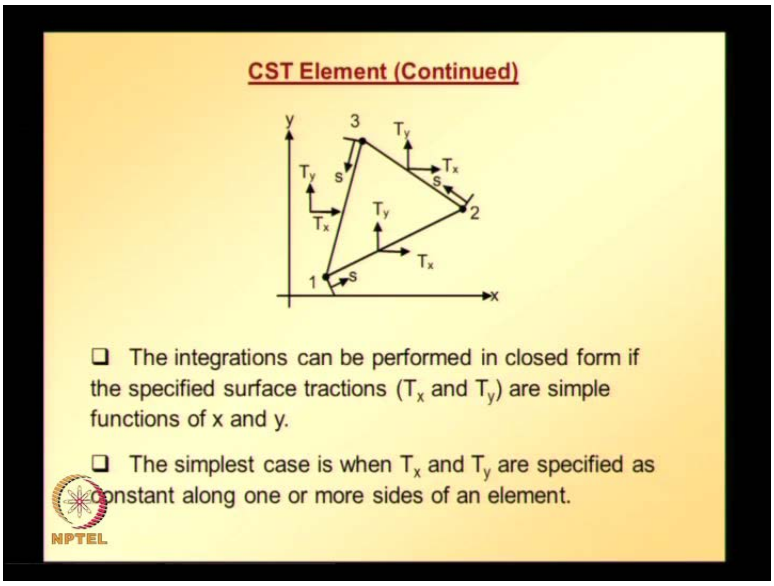

So, in that case if normal and tangential components are specified, then we can get T x T y based on direction cosines of the outer normal for that particular edge. So, how to calculate n x n y? Knowing the coordinates of end points of a line segment, the direction cosines of outer normal can be calculated. Please note that, we are carrying out this integration along each of the element edges. So, we have to take one by one each of these edges. So, once we know the nodal coordinates of the line joining or line passing through that edge, then we can easily calculate n x n y of outer normal of that particular edge.

(Refer Slide Time: 25:19)



So, this is how we can calculate. Suppose, we are interested along edge 1-2; so, what we need to do is we need to know, what are the coordinates of the end points of the line segment joining 1 to 2? x 1 y 1 are the nodal coordinates of node 1; x 2 y 2 are the nodal coordinates of node 2. So, with that information, we can easily calculate what are the direction cosines of outer normal to this particular edge 1-2? Using this relation, n x can be calculated using this relation; n y can be calculated using this relation, where L is length of this edge 1-2; length of the line segment joining nodes 1 and 2, which can also be calculated based on the information of nodal coordinates. And we need to carry out integration to calculate equivalent nodal vector.

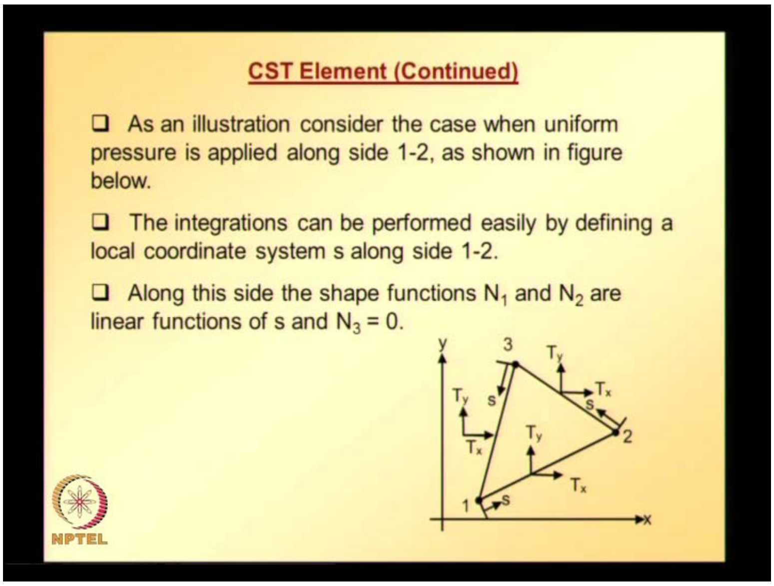
(Refer Slide Time: 27:01)




These integrations can be performed in a closed form, if specified tractions surface tractions T x, T y are simple functions of x and y. And when I say simple functions, for that matter if it is T x, T y are constant, then it is much easier. So, the simplest case is when T x, T y are specified as constant along one or more sides of element. So, for illustration purpose, let us take this triangle and assume that uniform pressure is applied alongside 1-2.

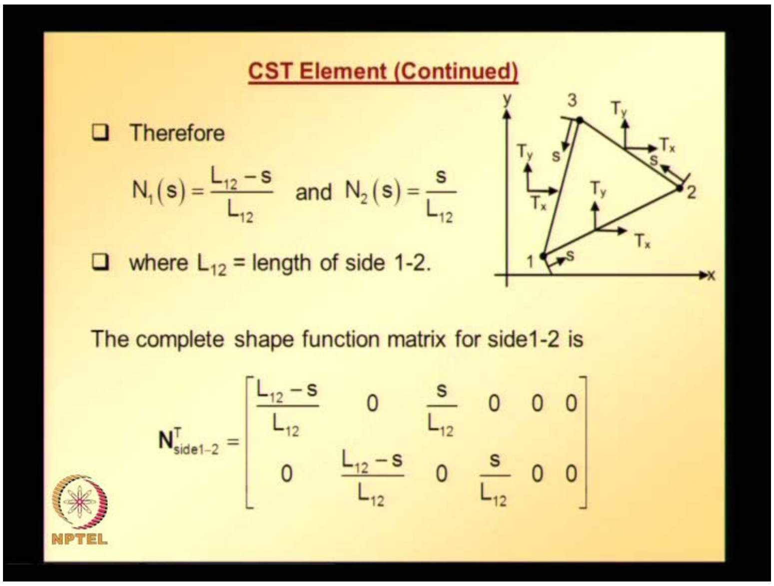
(No audio from 27:18 to 27:36)

(Refer Slide Time: 27:32)



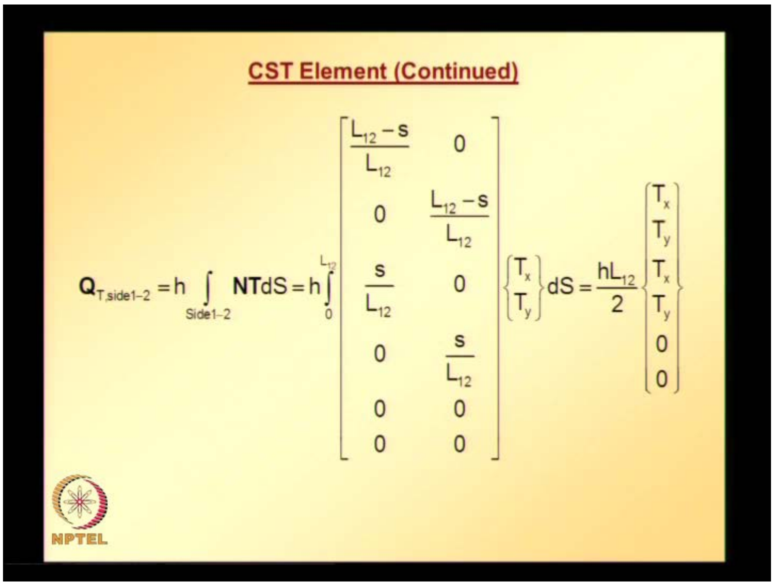

So, we need to know, what are the shape functions along side 1-2? Please note that, alongside 1-2, shape function of node 3 is going to be zero; because node 3 is not part of side 1-2. Along this side, only non 0 shape functions are alongside 1-2 only non 0 shape functions are n 1 and n 2. And the equations or expressions for these shape functions alongside 1-2 can easily be obtained, using Lagrange interpolation formula knowing the nodal coordinates of nodes 1 and 2.

(Refer Slide Time: 28:25)



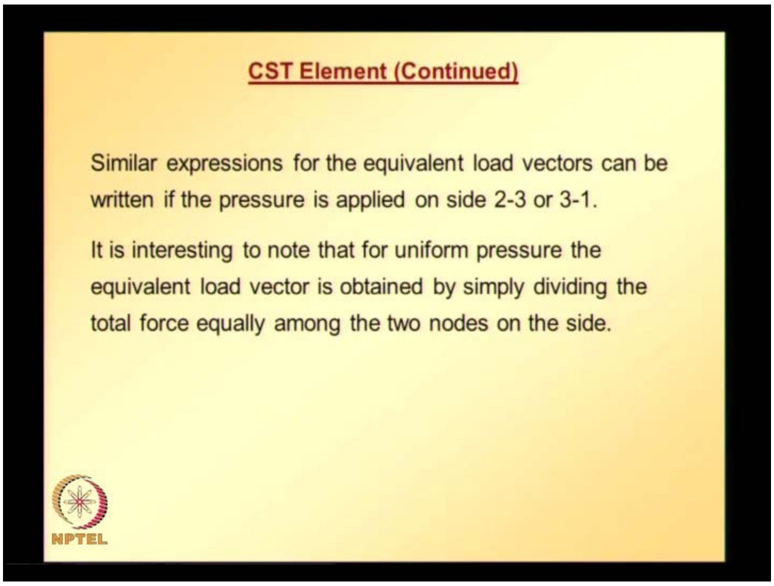
Based on this equation, these equations are obtained using Lagrange interpolation formula that we discussed many times earlier. So, and here L 1-2 is nothing but length of side 1-2 or element or line segment length of line segment joining nodes 1 and 2 and again shape function of node 3 along this side is 0. So, complete shape function matrix for side 1-2 can be written like this; n 1 0 first row consists of n 1 0, n 2 0, n 3 0 where n 3 is zero. So, we are going to get at that location 0 value and the second row consists of 0 n 1, 0 n 2, 0 n 3; again n 3 is equal to 0. So, complete shape function matrix looks like this for side 1-2. So, we got what we want that is we obtained the shape function vector alongside 1-2. So, using this, we can easily calculate equivalent nodal vector.

(Refer Slide Time: 29:55)



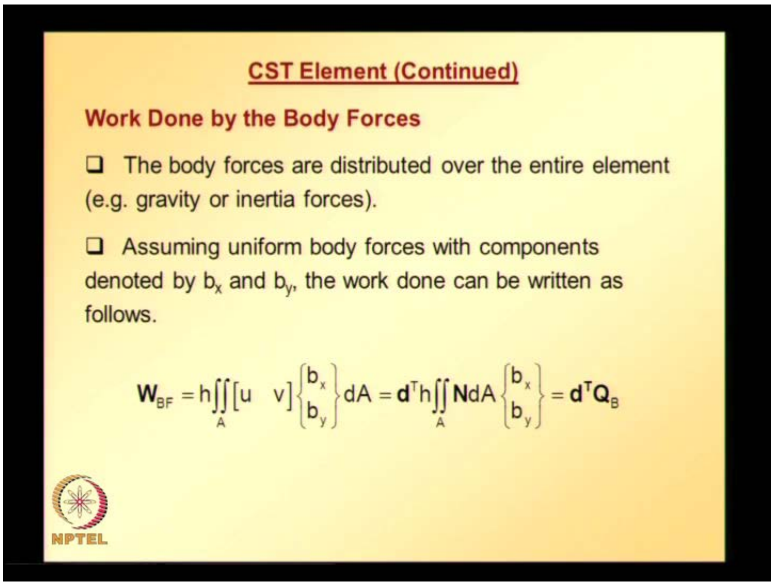

And we assumed alongside 1-2 or we are considering here a case, where tractions are uniform; they are constant. So, T x T y components are not functions of x. So, integration can easily be integration can easily be carried out; because the functions, which are appearing inside the integral are fairly simple. So, finally simplifying this, we get Q as h L over 2 sorry L 1 2 over 2 and at times vector consisting of T x T y, T x T y 0 0 and this is equivalent nodal vector alongside 1-2.

(Refer Slide Time: 31:02)




Similar expressions for equivalent nodal vector equivalent nodal load vector can be written for other sides, side 2-3, and 3-1. And it is interesting to note that, for uniform pressure the equivalent nodal load vector is obtained by simply dividing the total force equally among two nodes on the side; because traction is uniform. So, total load can be divided into two parts and can be assigned to each of the nodes. Instead of carrying out all this integration, because anyway the traction is uniform; it is not a function of x and y. So, this is how equivalent nodal load vector can be evaluated, if tractions are specified and now, let us discuss how to calculate work done by the body forces.

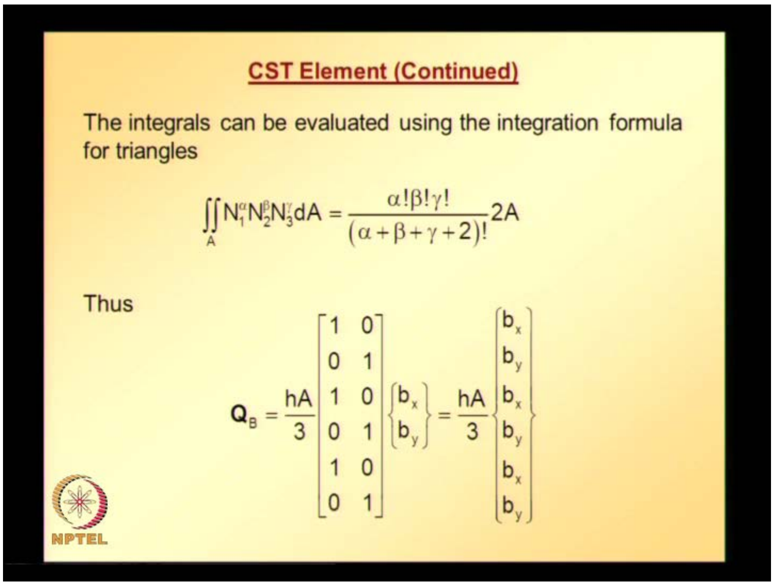
(Refer Slide Time: 32:21)




Please note that, body forces are also distributed; but they are distributed over entire element. For example, gravity and inertial forces again for illustration purpose, let us assume uniform body force with components b x b y. Since we are dealing with two dimensional problems plane stress, plane strain, we are assuming body force components or only we are considering only force component in x and y directions. Work done is given by, work done due to the body forces; that is why, it is written as W subscript BF, BF stands for body forces. It is again same logic; displacement times force acting in that particular direction; displacement component times force acting in that direction.

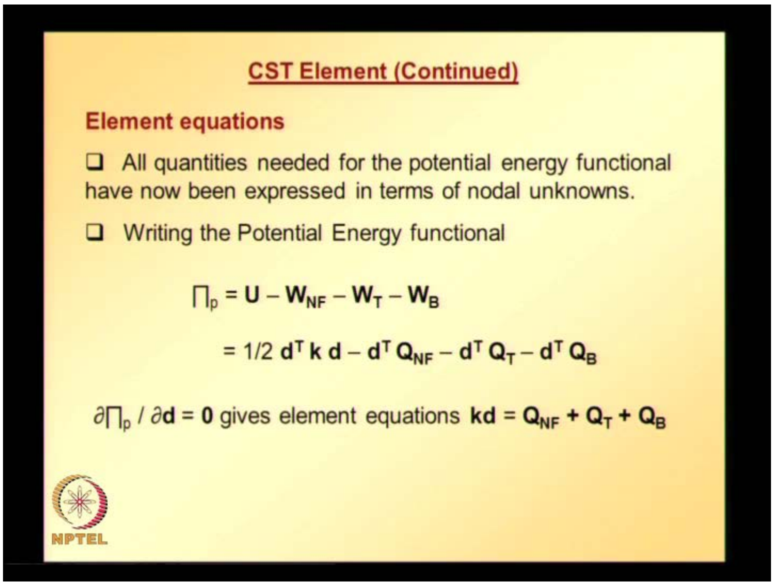
Based on that, we get we can calculate work done by the body forces, using this formula. Or it can be compactly written as W is equal to d transpose Q B, where Q B is defined as h times integral N d A times body force. A vector consisting of body force components and you can notice here to carry out this integration. If the expressions for shape functions are fairly simple, we can easily calculate this; or if the expressions for shape functions are complicated, in that case we can use a formula, which we already seen earlier. But for completeness I am reproducing that formula here.

(Refer Slide Time: 34:42)




These integrals can be evaluated by using the following formula, which we have seen earlier. So, depending on the integrant that you have and if it consists of only the shape function expressions or if it consists of only shape functions, then we can use this. Otherwise, we need to use numerical integration that we discussed earlier. So, adopting either of this; we can get equivalent nodal vector due to the body forces. So, we discussed how to calculate strain energy and how to calculate work done by various forces.

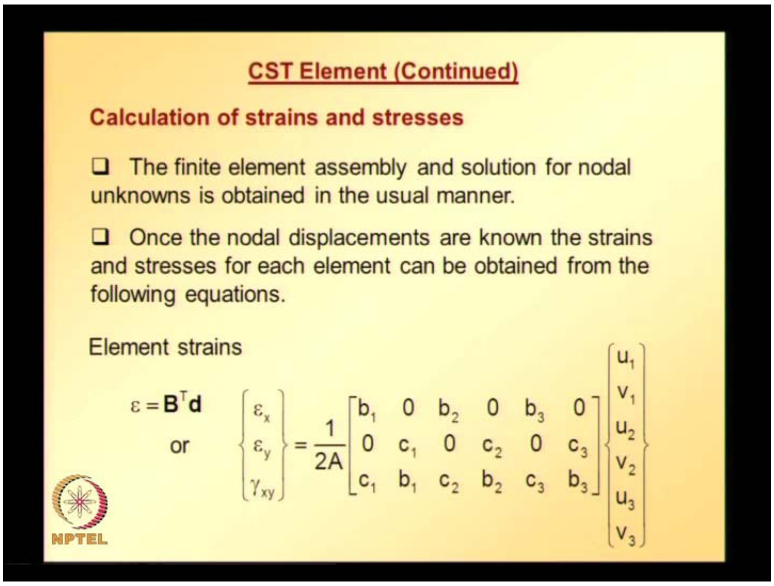
(Refer Slide Time: 35:41)



So, we are ready to get element equations. All quantities needed for potential energy functional have now been expressed in terms of nodal unknowns. So, writing potential energy functional that is U times W, W consists of… it can be the contribution can come from concentrated forces, tractions body forces. So, we are including all that, when we are writing potential energy functional and applying stationarity condition on this potential energy functional, we get the element equations. k d equal to Q NF Q plus Q T plus Q B. Q NF is a vector consisting of concentrated forces; Q T is equivalent nodal load vector due to tractions and Q B is equivalent nodal vector due to distributed forces.

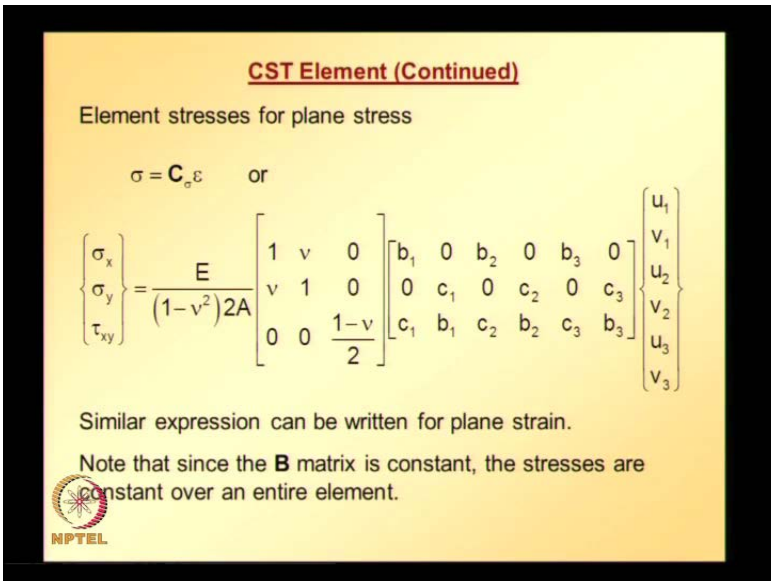
So, solving this, we get nodal unknowns or nodal displacements. So, once we have nodal displacement information, we can easily calculate strains. And from there, we can calculate stresses and do all kinds of post processing and as far as, assembly this whatever we discussed here so far is for one element. So, we need to write this kind of equations for all elements. Once we get equations for all elements, the assembly process, and applying essential boundary condition and solution procedure is similar to what we discussed or what we have been discussing in the last few classes.

(Refer Slide Time: 37:44)




Once the nodal displacements are known, strains and stresses for each element can be obtained from the following equations. Strains can be calculated using this strain displacement relation. And once we know strains, we can calculate stresses which depend on whether you are you started out with plane stress assumptions or plane strain assumptions, you need to use corresponding constitutive matrix. So, if it is for plane stress, this is how you can calculate.

(Refer Slide Time: 38:25)

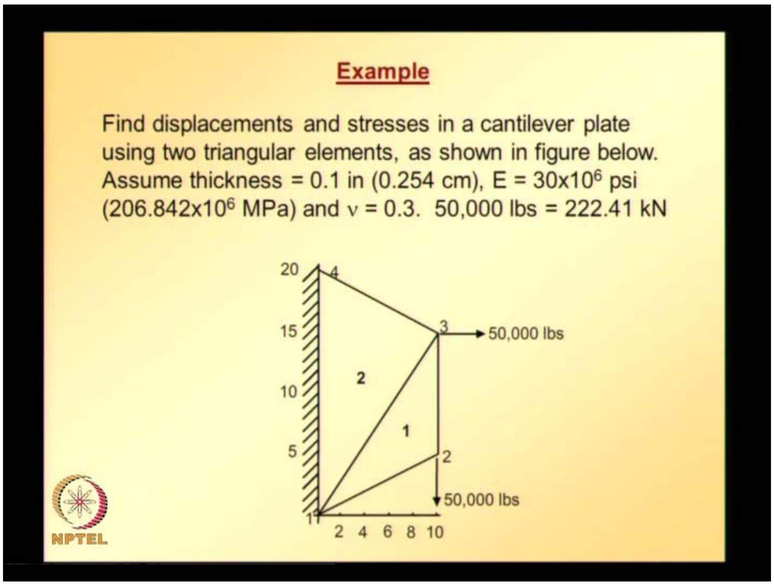



$$\sigma = C_\sigma \varepsilon \quad \text{or}$$

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E}{(1-\nu^2)2A} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} \begin{bmatrix} b_1 & 0 & b_2 & 0 & b_3 & 0 \\ 0 & c_1 & 0 & c_2 & 0 & c_3 \\ c_1 & b_1 & c_2 & b_2 & c_3 & b_3 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix}$$

Similar expression can be written for plane strain case. And note that, since B matrix is constant, stresses or going to be constant over entire element. So, to illustrate these points much better way, let us take an example.

(Refer Slide Time: 38:54)



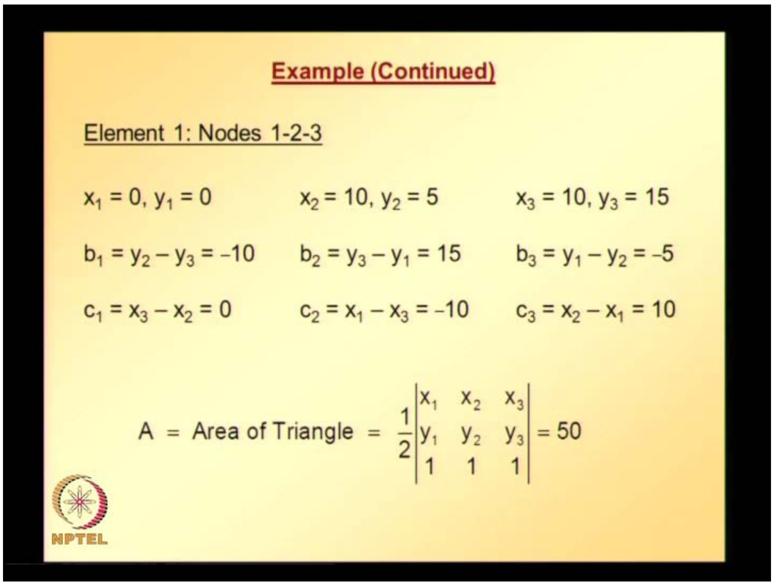
Find displacements and stresses in a cantilever plate using two triangular elements. This is a cantilever plate. From the information that is given in the figure, we can easily figure out what are the nodal coordinates. And the geometric geometrical details and also material property details and load details are given both in FPS units and SI units and since the figures for each of these quantities are round figures in FPS units. We will be doing this problem in FPS units for but just for convenience.

Other than that, procedure it is going to be exactly same irrespective of whatever units are being adopted and the figure shows figure is drawn in FPS units. So, whatever coordinates information that we get from the figure, please read it as in inches corresponding to the nodal coordinates. So, the cantilever plate is discretized using two triangular elements and all the nodes are numbered and the two triangular elements are also numbered. Node element 1 comprises of nodes 1, 2, 3 taken in counter clockwise direction.

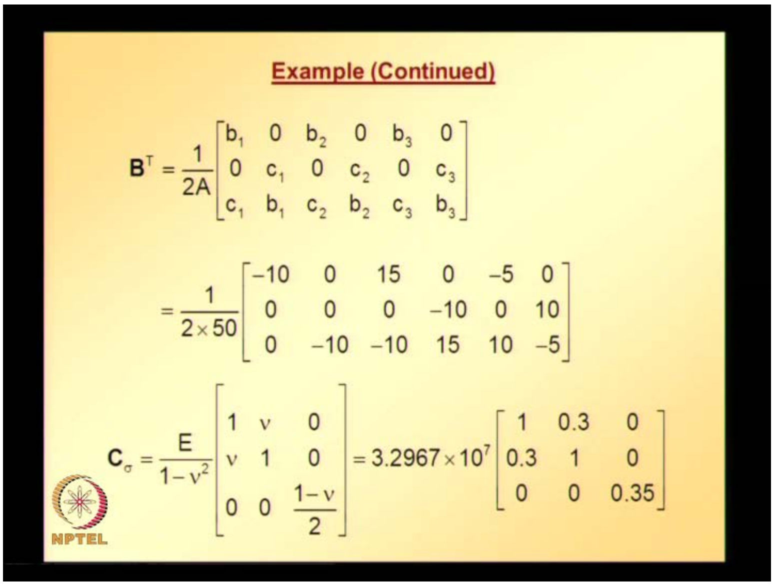
Similarly, element 2 comprises of nodes 1, 3, 4 again when we look in the counter clockwise direction. And we also have the nodal coordinate information given and all the material properties and thickness is also given and the load value is also given. And please note that, since we are using three node triangular element, p matrix is going to be constant for each of these elements. So, now noting all these we can write element equations for element 1, element 2 separately. And then finally, we can assemble global equation system based on the nodal connectivity.

Element 1 is connecting nodes 1, 2, 3; element 2 is connecting nodes 1, 3, 4 and at each node since we are dealing with elasticity problem 2 D elasticity problem, at each node we are going to get two nodal degrees of freedom. So, total final equation system is going to be 8 by 8 and element 1 contribution goes into 1 to 6 rows and columns and element 2 contributions goes into 1 to 5 to 8 rows and columns. So, it goes into 1, 2, 5, 6, 7, 8 rows and columns from element 2. So, noting all this information, we can quickly assemble the element equations. These are corresponding to element 1. Area of triangle can be calculated based on the formula that we discussed earlier.

(Refer Slide Time: 42:49)

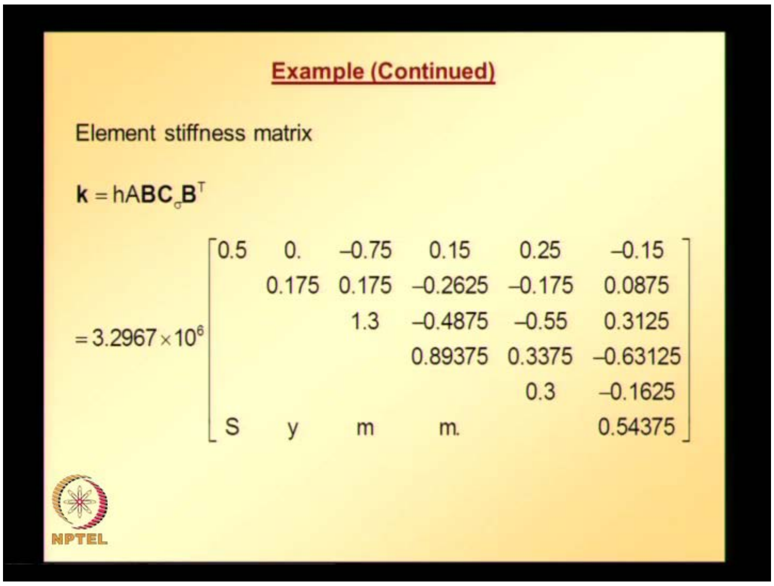



(Refer Slide Time: 42:58)



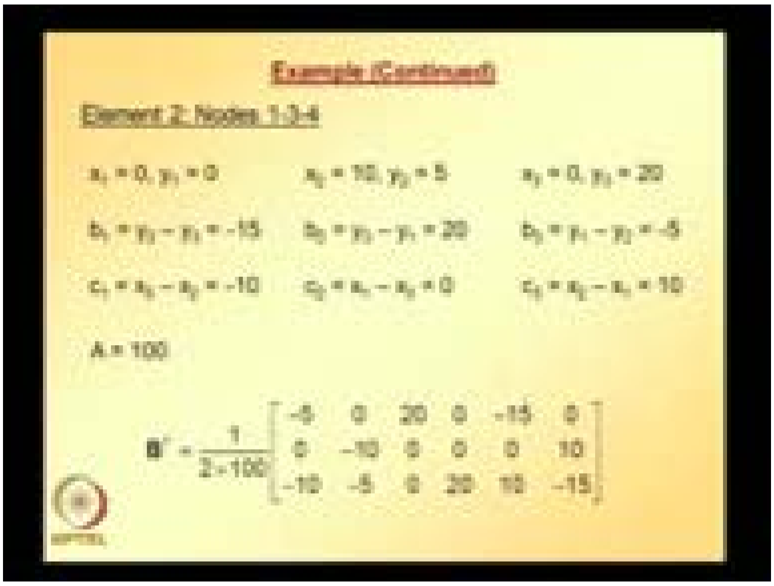

B matrix for element 1. Assuming plane stress condition, C matrix is given by this. Substituting young’s modulus poisons poisons ratio values which are given. And since we are dealing with a three node triangular elements, since B matrix is constant, we do not need to do numerical integration to get element stiffness equation or element stiffness matrix. We can directly carryout multiplication h times A times B times C times B transpose.

(Refer Slide Time: 43:55)



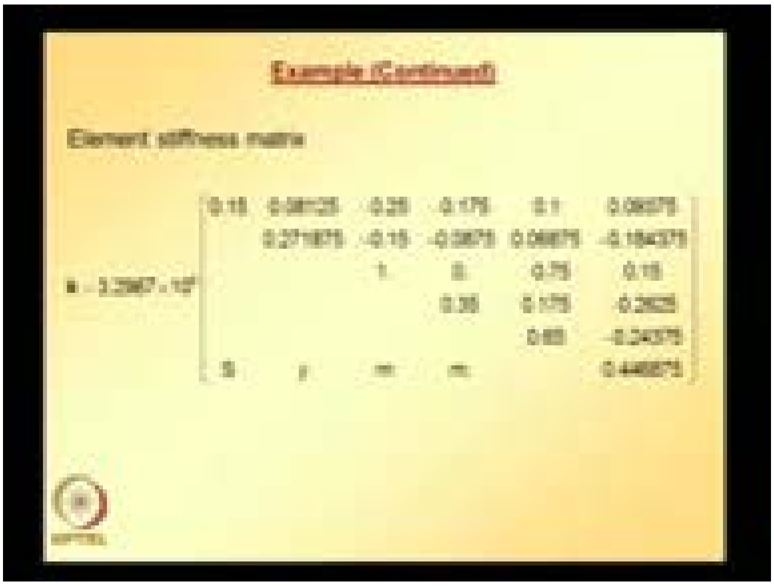

So, carrying out those multiplications of matrices, we get this; element stiffness matrix for element 1.

(Refer Slide Time: 44:03)



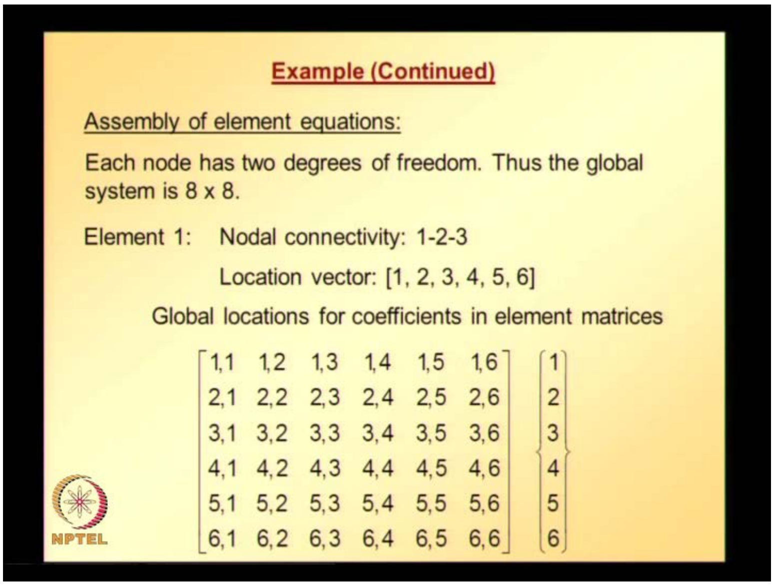

Similar operations can be carried out by noting the nodal coordinate information for element 2. It turns out that, B matrix is as shown here for element 2.

(Refer Slide Time: 44:22)



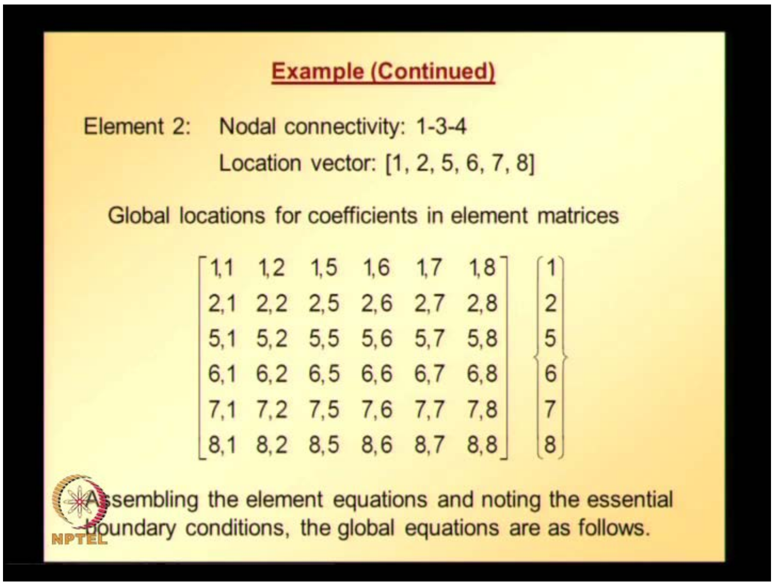

And similarly, element stiffness matrix can be calculated. And now, we got element stiffness matrix for element 1 and element 2.

(Refer Slide Time: 44:41)



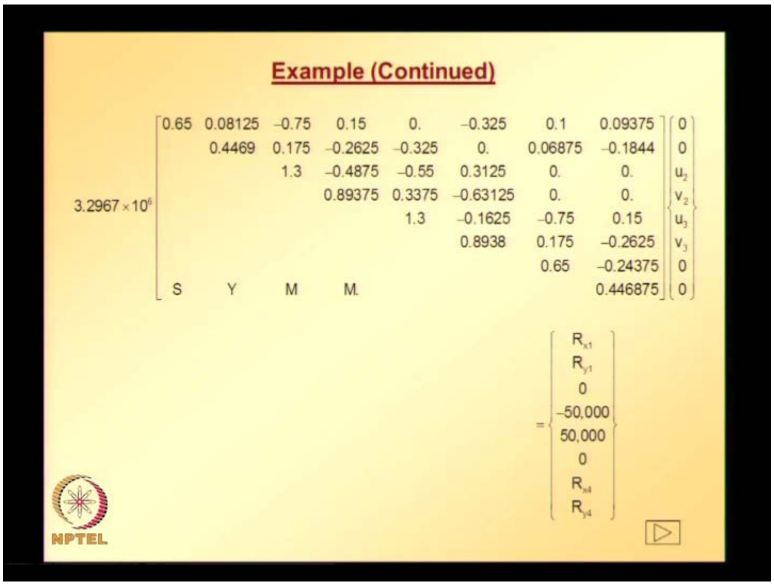

So, now we are ready to assemble element equations. As I mentioned, each node has two degrees of freedom. So, total global equation system is going to be 8 by 8. So, we need to clearly note down, where the contribution from element 1 goes in and where the contribution from element 2 goes in to the global equation system. So, noting nodal connectivity, we can easily figure out exactly at what locations element 1 contribution goes in, which is shown in location vector. Once we have this information, global locations for coefficients in element matrices can be written or can be noted down in this manner. This information helps us to assemble the global equation system. So, this is for element 1.

(Refer Slide Time: 45:42)



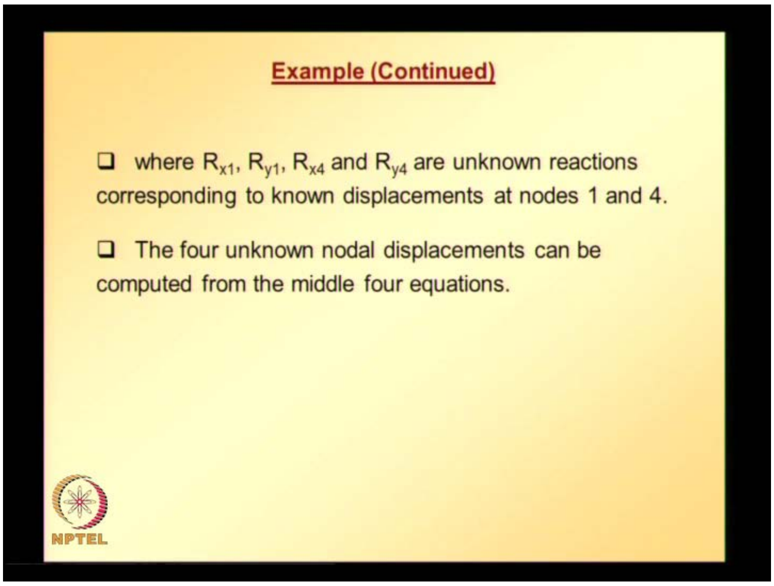

Similarly, for element 2 nodal connectivity, corresponding location, vector global locations for coefficients in element matrices. So, this matrix gives us global locations for element 2. So, using this information, we can assemble assembling element equations and noting essential boundary condition, the global equations are as follows.

(Refer Slide Time: 46:14)




Essential boundary conditions are also specified here and the equations system that is shown. Please note that, u node 1 and node 4 are fixed. So, displacement in x direction, displacement in y direction at these locations; that is, at node 1 and node 4 are zero. And also note that, wherever displacements are specified as zero, at those locations unknown reactions are going to be there; that we need to calculate. So, reactions wherever displacements are specified as zero, reactions are going to be developed, which are unknown.

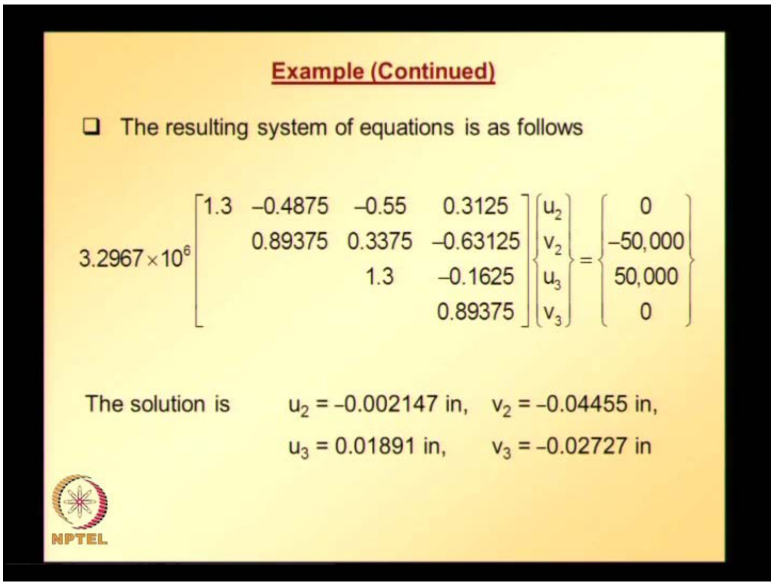
(Refer Slide Time: 47:19)



The corresponding locations in the force vector are replaced with R x 1, R y 1, R x 4, R y 4 respectively. So, in this equation R x 1, R y 1, R x 4 and R x R x 4 and R y 4 are unknown reactions corresponding to known displacements at nodes 1 and 4. Known displacements here are 0 displacements. And if you see that equation global equation system, four unknown nodal displacements are required to be calculated; u 2, v 2, u 3, v 3 and these can be calculated from middle four equations.

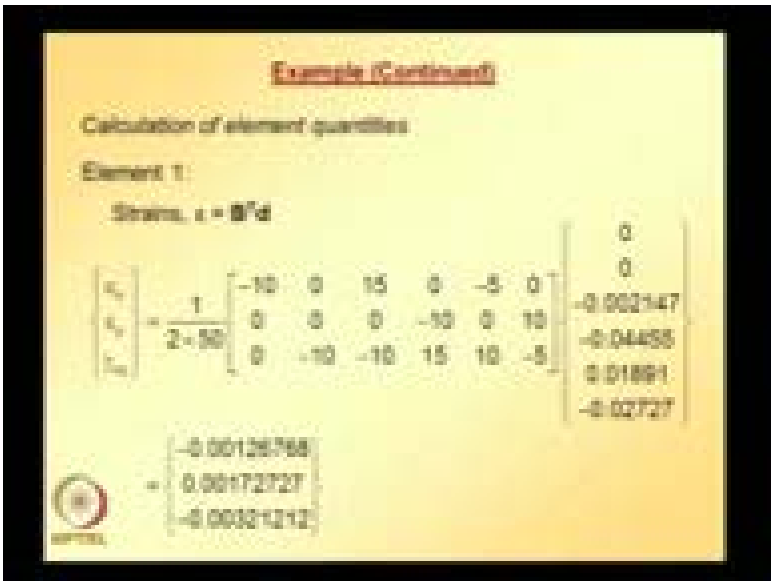
And eliminating the rows and columns in the global equation system corresponding to 0 degrees of freedom, we get reduced equation system. Since the specified displacements are 0 at node 1 and 4, the first two and last two columns will not contribute anything. So, the resulting reduced equation system is given by this, which can easily be solved for the unknowns u 2, v 2, u 3, v 3. Again solution is given in FPS units.

(Refer Slide Time: 48:38)



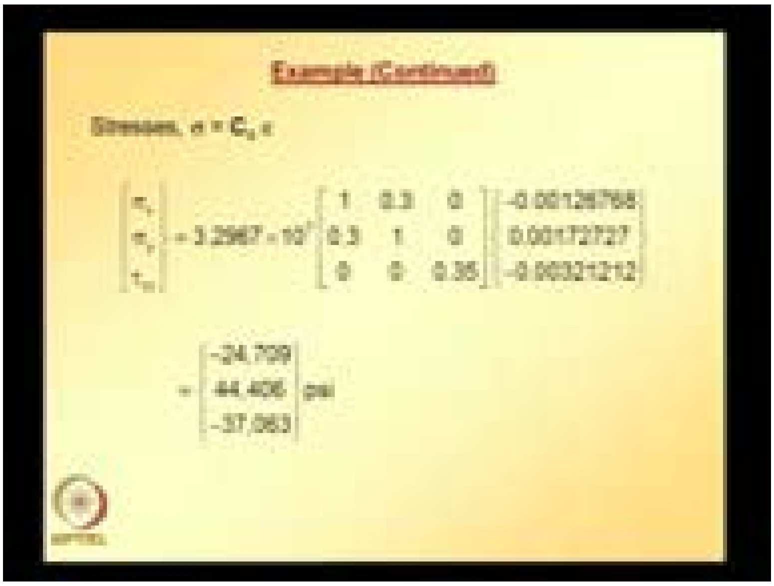

But procedure wise, it is not going to be anything different and now, we got the displacements.

(Refer Slide Time: 48:48)



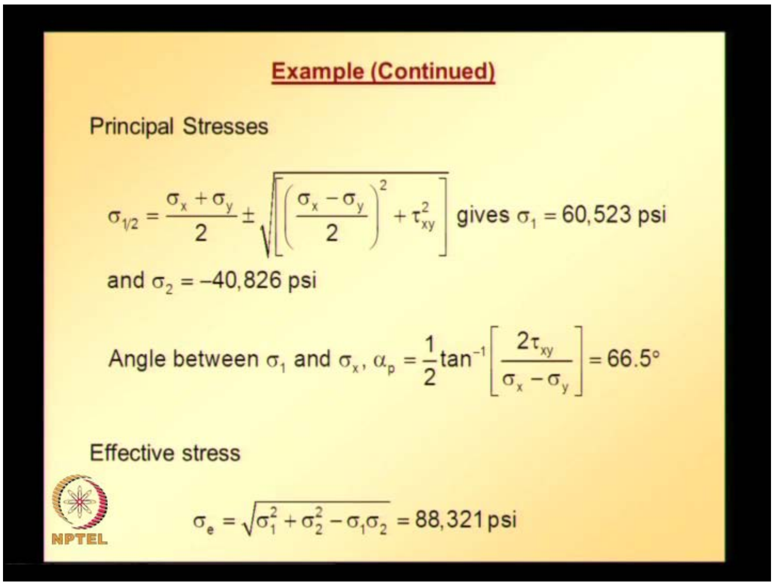

We can easily calculate element quantities corresponding to element 1; strains and then stresses. Strains can be obtained using this relation and please note that, strains are going to be constant over the entire element.

(Refer Slide Time: 49:09)



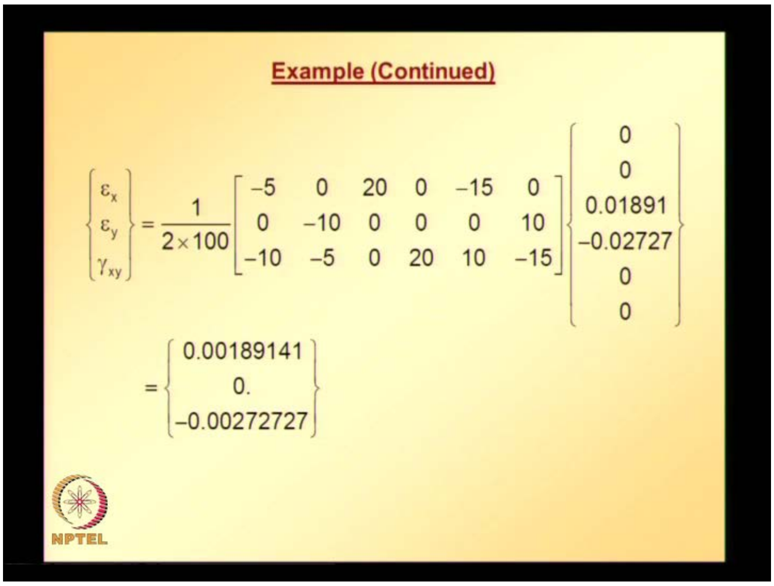

Hence, once we got strains, we can calculate stresses using constitutive matrix corresponding to plane stress condition.

(Refer Slide Time: 49:25)



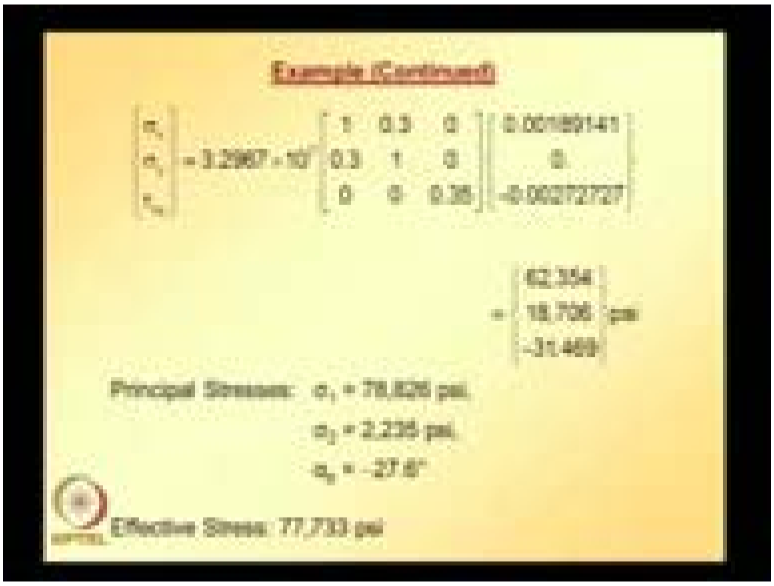

And once we got stresses, we can calculate using these quantities; principal stresses for subsequent usage in failure criteria and all other details required; angle between sigma 1 and x sigma x and effective stress.

(Refer Slide Time: 49:56)




And these calculations can even be repeated for element 2. These are the strains for element 2; stresses for element 2.

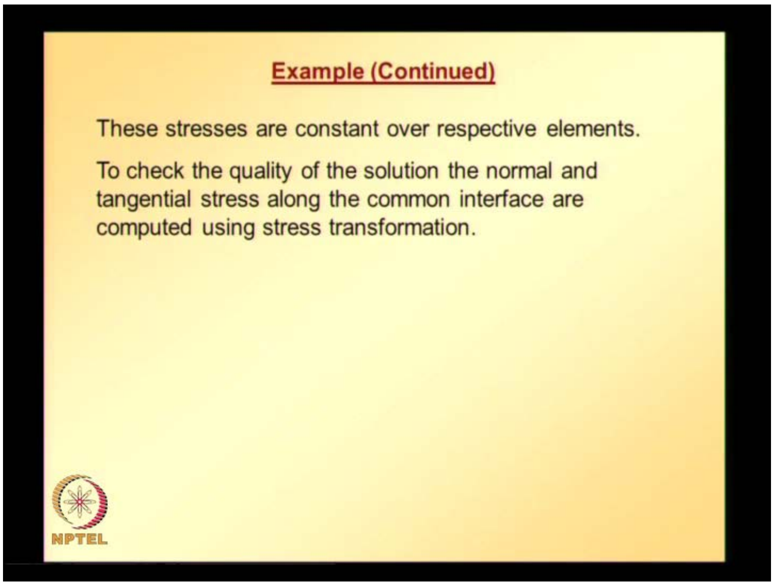
(Refer Slide Time: 50:06)




(No audio from 50:08 to 50:15)

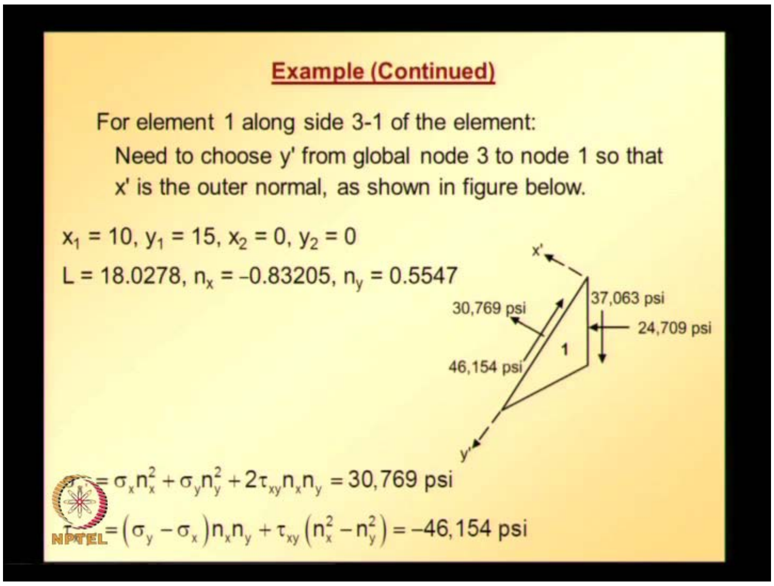
Similar quantities reported as for element 1; that is, principal stresses, sigma 1, sigma 2 and the angle between sigma 1 and sigma x and effective stress.

(Refer Slide Time: 50:56)




Please note that, strains are constant over each element and stresses are also constant over each element. And to actually to really check the quality of solution, the normal and tangential stresses along common interface needs to be computed and see, whether they are matching or not. What I mean by that is we need to calculate these components using stress transformation.

(Refer Slide Time: 51:08)



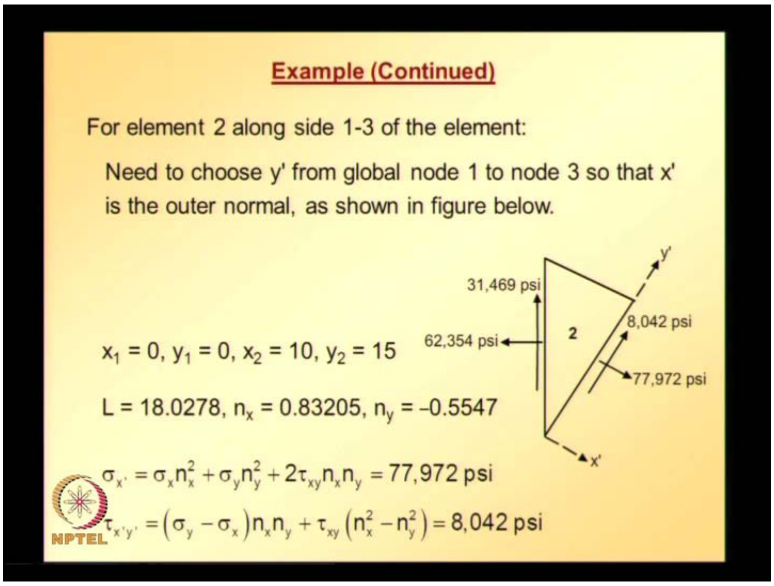

So, what I mean by that is for element 1, please note that; for both elements, side 1-3 is common. So, along this interface 1-3, we can calculate normal stresses and tangential stresses from element 1 and from element 2 and see, whether they are matching or not. If the discretization is enough, then normal and tangential component of stresses along this interface calculated from element 1 and the element 2 are going to match. So, now let us do the calculation for element 1; this is element 1. Actually in this figure, even the values are shown.

But how to calculate those values will be shown shortly. So, just ignore those values for a while and concentrate only on the figure without those values. So, need to choose y prime from global node 3 to global node 1. So, that x prime is outer normal as shown in figure. So, that is how x prime y prime are defined and we know how x y are oriented. So, once we know how x y are oriented and how the new coordinate system x prime y prime are oriented, then we can easily find what is the angle between these two coordinate system.

What is the angle between x axis and x prime axis? Using that information, we can easily calculate what are the normal and shear components along the interface 1-3 or 3-1. And for that, we require to note down what are the coordinates of… we need to note down or we need to know, what is the element length, line segment length connecting nodes 1 and 3. So, for that we required to note down, what are the coordinates of node 3. Node 3 coordinates are denoted with x 1, y 1 and node 1 coordinates are denoted with x 2, y 2.

And here for calculation purpose, node 3 is taken as first node and node 1 is taken as second node. And from that, we can easily calculate what is the length and also we can calculate, what are the direction cosines of outer normal alongside 3-1. And once we have that information, we can easily calculate sigma x prime and tau x prime y prime using these relations. So, this is what I am emphasizing that, these values of sigma x prime tau x prime y prime values calculated from element 1 should match with element 2. Let us see, whether they match or not.

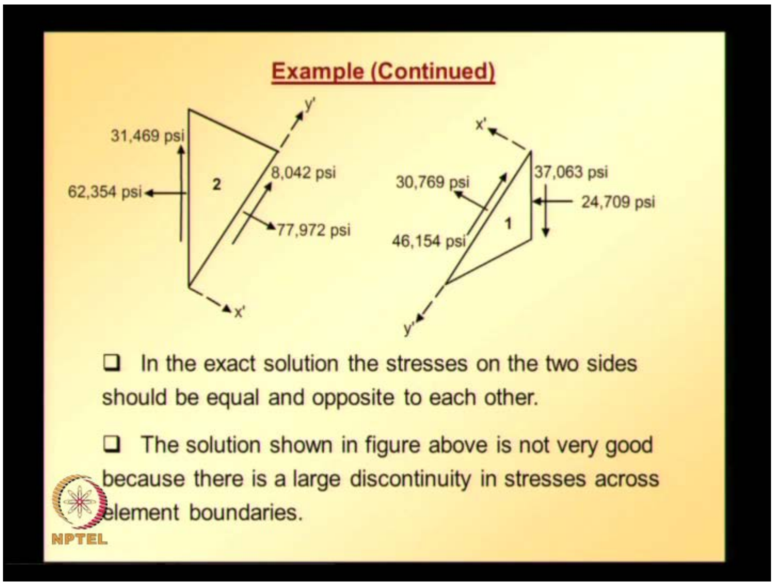
(Refer Slide Time: 54:25)




So, from element 2 for element 2 alongside 1-3, let us define a coordinate system like this. Choose y prime from global node 1 to 3 and x prime, then x prime is automatically going to be outer normal as shown in figure. From this understanding, so here node 1 along line segment 1-3 corresponds to node 1 of the global coordinate system and node 2 corresponds to node 3 of the global coordinate system. So, the corresponding nodal coordinates are noted and also length is calculated.

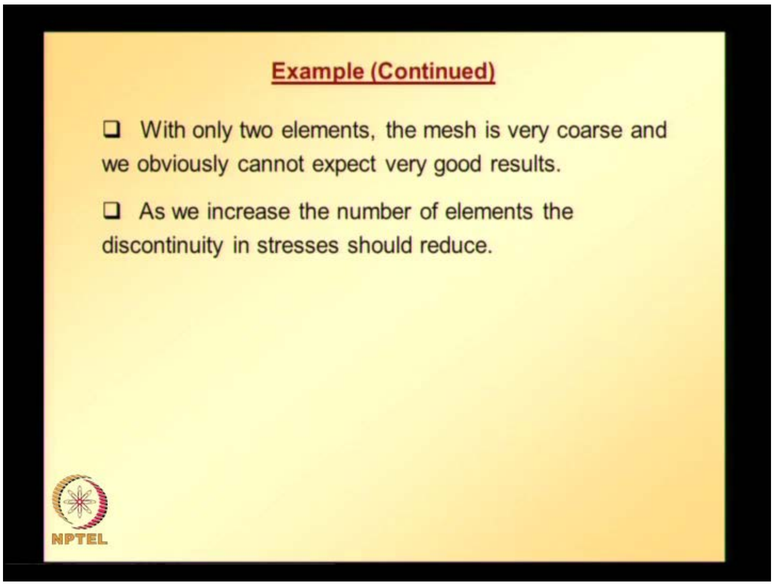
Once we have this, we can calculate, what are the outer normal or direction cosines of outer normal for this particular edge 1-3. Once we have this information, we can easily calculate sigma x prime tau x prime y prime using these relations. So, we calculated sigma x prime tau x prime y prime for both elements along the interface 1-3. So, we need to see whether these values match or not. So, these all these values are put side by side to see whether they are matching or not.

(Refer Slide Time: 55:42)




In the exact solution, stresses on two sides should be equal and opposite to each other. If the solution is exact and the solution shown or solution that we obtained is not very good. Because there is a large discontinuity in stresses across element boundaries; that is, normal stress values are not matching; also tangential stress components are not matching. So, that means the discretization that is adopted for solving this cantilever plate is not enough.

(Refer Slide Time: 56:24)




With only two elements, mesh is very coarse, and we obviously cannot expect very good results. As we increase the number of elements, the discontinuity in stresses should reduce. So and when if we really want increase the number of elements, then we cannot do that using hand calculations; we need to automate this. So, this is about a three node triangular element for solving plane stress, plane strain problems. And in the next class, we will be looking at four node quadrilateral element.