Finite Element Method

Prof. C. S. Upadhyay

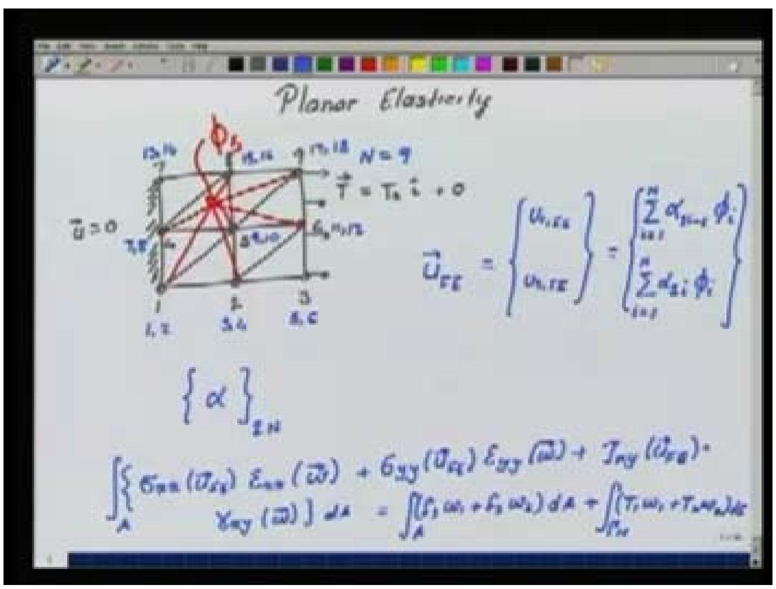
Department of Mechanical Engineering

Indian Institute of Technology, Kanpur

Module 9 Lecture 3

In this lecture we are going to continue from where we left in the previous lecture. We were doing the planar elasticity problem. For that problem, we had derived the weak formulation. Let us put everything into perspective again, the weak formulation corresponds to what. I make a very simple mesh here. Let traction be specified on this edge. This is traction vector T is equal to T_x into i plus 0, because it is the extraction. I could have displacement here fixed to 0, that is, u here is equal to 0.

(Refer Slide Time: 00:24)



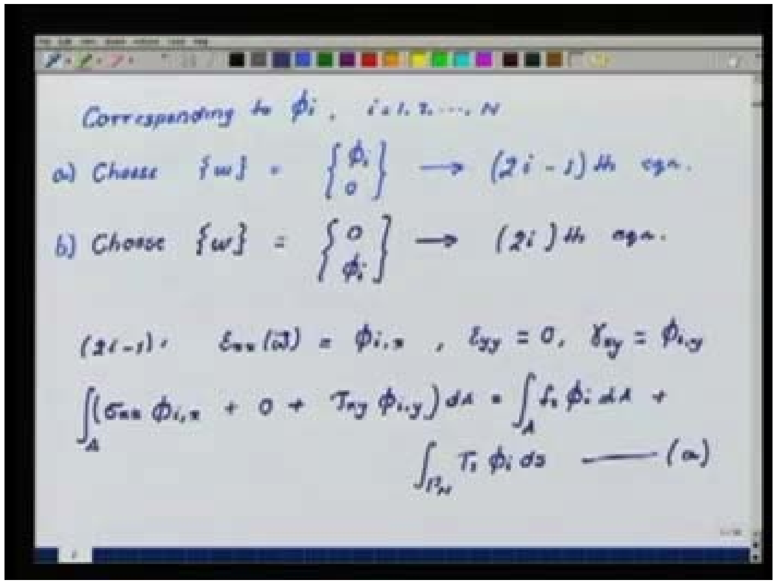
Let us say, in a very simplistic situation, I am using linear elements to solve this problem. That is, piecewise linear approximation is used to solve this problem. These would be the nodes 1, 2, 3, 4, 5, 6, 7, 8 and 9. And we had said, just like we had done in the one-D problem or in the single variable problem, I will have the basis function defined such: phi_i is such that it is 1 at a node and it falls up to 0 at the neighboring nodes.

This for example, is phi_5. With this understanding, we had said that we are going to define our finite element solution, $u_{FE,}$ of which the components are: u_{1FE} and u_{2FE} and they are given by $sigma_i$ is equal to 1 to n. I may have n nodes in the domain and n is equal to 9 in this case. The way we have defined them is, $alpha_{2i}$ minus 1 phi_i was $u_{1FE.}$ And, u_{2FE} was $alpha_{2i}$ $phi_{i,}$ where we had said, corresponding to every node, there are two degrees of freedom. That is, alpha 1, 3, 5, 7, 9 and so on correspond to u_{1FE} and alpha 2, 4, 6, 8 and so on correspond to u_{2FE}.

For example, the total number of nodes n is equal to 9. I will have the degrees of freedom 1 and 2, 3 and 4, 5 and 6, 7 and 8, 9 and 10, 11 and 12, 13 and 14, 15 and 16 and 17 and 18. The total number of unknowns becomes twice the number of nodes, which is 2i, because, we are talking of a problem which has two unknown functions. Here is 2N number of unknowns. So our job is to try to setup the 2N equations that we have to solve in order to get the 2n coefficients $alpha_i$. This is of size 2N.

This is what we had started off with and we said that in order to construct this equation, we look at the equation corresponding to phi_5 or phi_i in the general case. How did we do it? We chose the w that is a test function in a weak formulation. Our weak formulation was integral over the area $sigma_{xx}$ (due to the finite element solution) into u_{FE} into E_{xx} due to the test function w plus $sigma_{yy}$ due to u_{FE} into E_{yy} due to w plus tau_{xy} (due to the finite element solution) into $gamma_{xy}$, due to w, the whole thing integrated over the area is equal to integral over the area of f_1w_1 plus f_2w_2 dA plus, on the Norman boundary $gamma_N$, T_1w_1 plus T_2w_2 ds. We are going to choose w of two types.

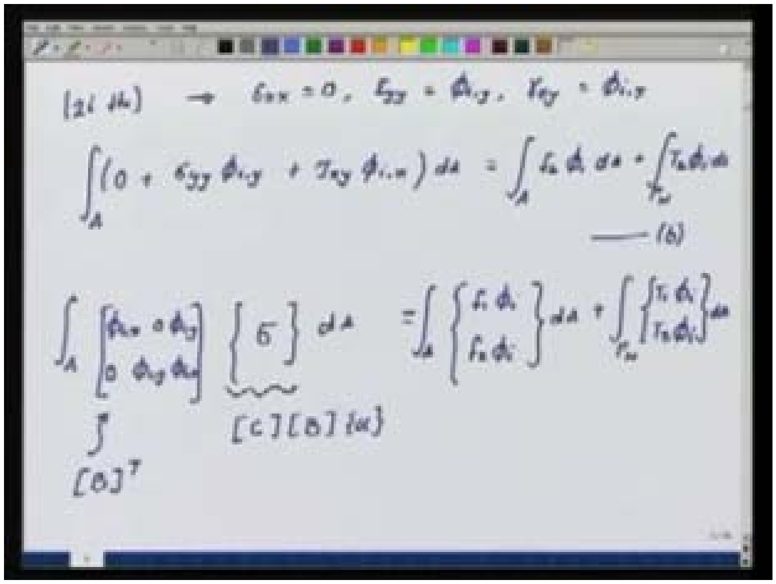
(Refer Slide Time: 06:15)



Corresponding to phi_i, i equal to 1 to N, choose vector w is equal to phi_i, 0. We claimed that this w satisfies all the geometric constraints that are required by the w to be satisfied. I am taking the w_2 component to be 0, w_1 component to be phi_i and put this in the weak form that we have obtained to get the equation, which we had stopped at, as the 2i minus 1th equation. We said this was first choice. Choice b was: choose w equal to 0, phi_i and when I put this, this is also an admissible w, when I put this in the weak formulation I will get the 2ith equation. In this way, we construct all the 2N equations, which are required in order to solve the problem.

What were our 2i equations? The equation 2i minus one was given by the strain due to the choice of w is equal to $phi_{i,x}$, E_{yy} will be equal to 0 (because here I have taken w_2 to be 0) and $gamma_{xy}$, due to this w, is going to be equal to $phi_{i},_{y}$. That is, del wy del y. This is the state of strain. I put it in the weak form. So I will get integral over the area $sigma_{xx}$ due to the finite element solution into the strain due to this, which is, $phi_{i},_{x}$ plus $sigma_{yy}$ into the strain due to this displacement which is 0, plus tau_{xy} into the strain due to this w which is $phi_{i,y}$ into dA. This is equal to integral over area of f_1 into phi_i into dA. F_2 will do no work because we have taken w_2 to be 0, plus integral over the boundary, T_1 phi_i ds. Let me call this equation a. This is the first equation. We said that 2i minus 1th equation is set, where $sigma_{xx}$ and so on will come out of the representation of u_{FE}.

(Refer Slide Time: 10:11)



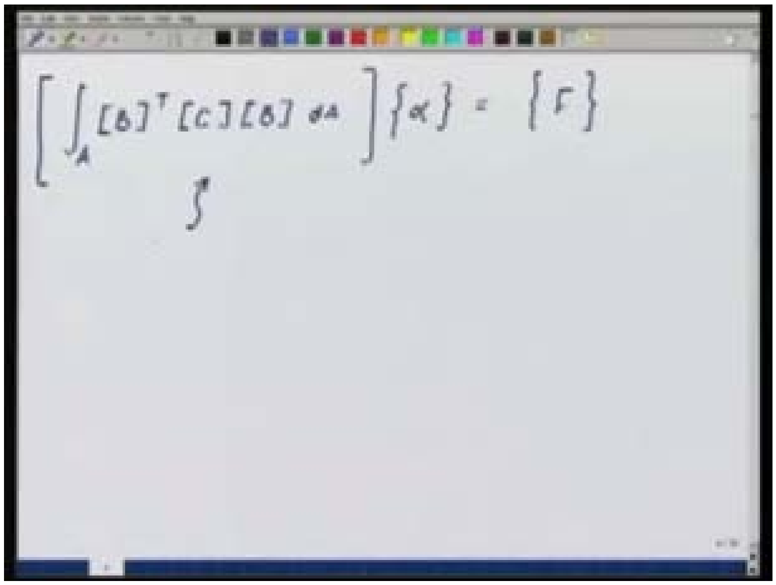
Let us now talk of the 2ith equation. I will get integral over the area. For the 2ith equation, I have E_{xx} is equal to 0, because we have taken w is equal to 0, phi_i. E_{yy} is equal to $phi_{i,y}$ and $gamma_{xy}$ is equal to $phi_{i,x}$. When I put this in the weak form, this will give me $sigma_{xx}$ into E_{xx}, which is 0 plus $sigma_{yy}$ into E_{yy} due to w which is $phi_{i,y}$ plus tau_{xy} into $gamma_{xy}$ which is due to this choice of w, $phi_{i,x}$. This dA is equal to integral over the area of f_2 phi_i dA plus integral over $gamma_N$, T_2 phi_i ds. So this is going to be the 2ith equation. Let me call it b.

If I write a and b together, that is, the two equations corresponding to phi_i, what will I get? Then I will get integral over the area, the first equation will be given by something here into the stress vector sigma dA. This is equal to integral over the area f_1 phi_i f_2 phi_i dA plus integral over $gamma_N$, T_1 phi_i T_2 phi_i. Let us look at the first equation. What will I have? $Sigma_{xx}$ into $phi_{i,x,}$ $sigma_{yy}$ into 0, $sigma_{xy}$ into $phi_{i,y}$.

Similarly, as far as this one is concerned for the second equation, I will have 0 into $sigma_{xx}$ $phi_{i,y}$ and $phi_{i,x}$. So these are the two equations I have written in matrix form. I write all these equations corresponding to the various 'i's. What do I have for sigma here? Sigma, if I go back to what I have done in the previous lecture, is given as C B alpha. Alpha was the vector of the 2N unknown coefficients. This is what I will get. Now I start writing these equations one below the other, corresponding to i equal to 1, 2, 3, 4, 5, and 6. Here, I will have $phi_{1,x}$ $phi_{1,y}$ $phi_{1,y}$

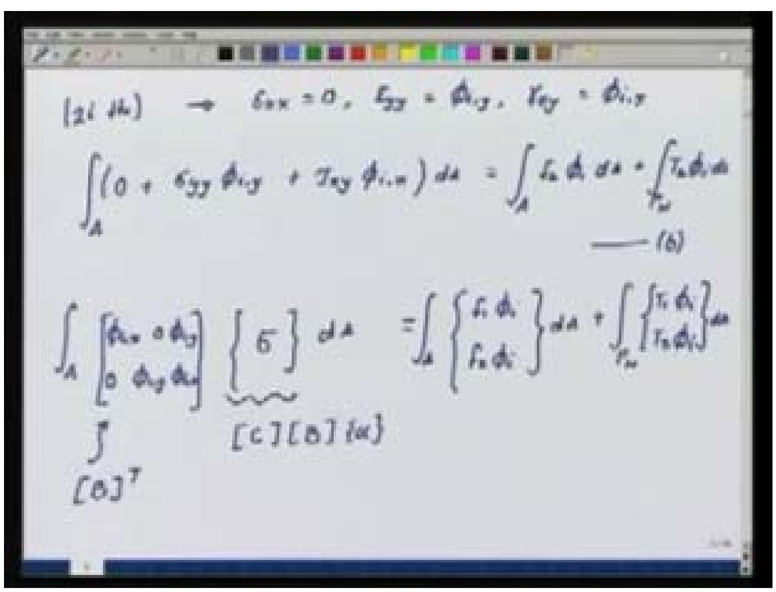
$phi_{1,x}$ and then I will have $phi_{2,x}$ and so on. If I write all of this one below the other, if I do the whole job all the way down to the 2N equations, this will be nothing but B transpose. This is exactly B transpose. One can work it out. It is going to be easy, the algebra will not be tedious.

(Refer Slide Time: 15:03)



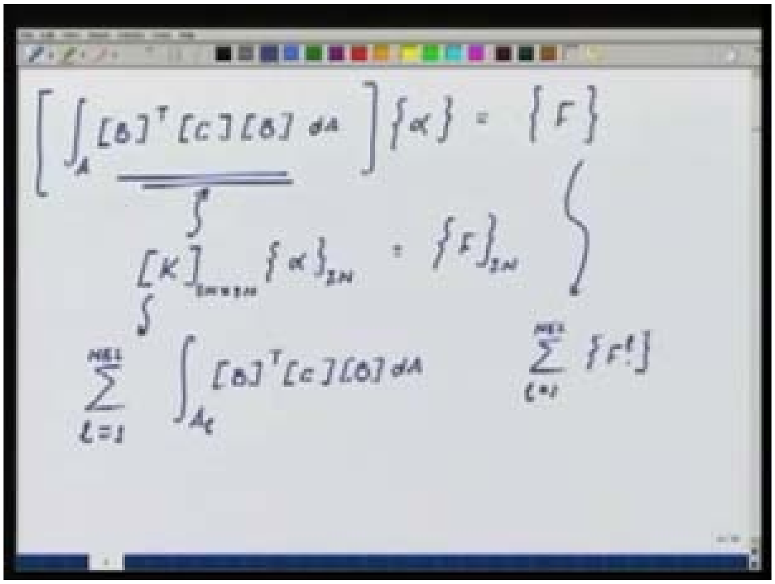
I will end up getting integral over the domain B transpose C B dA. This is a matrix into the vector alpha is equal to vector F.

(Refer Slide Time: 15:37)



What is F? The first entry of f_2 i minus 1 will be f_1 integral phi_i plus integral of T_1 phi_i and f_2 i will be integral over the area of f_2 phi_i plus integral over the Norman boundary of T_2 phi_i. So this is the generic representation that we are going to get if we write the whole equation.

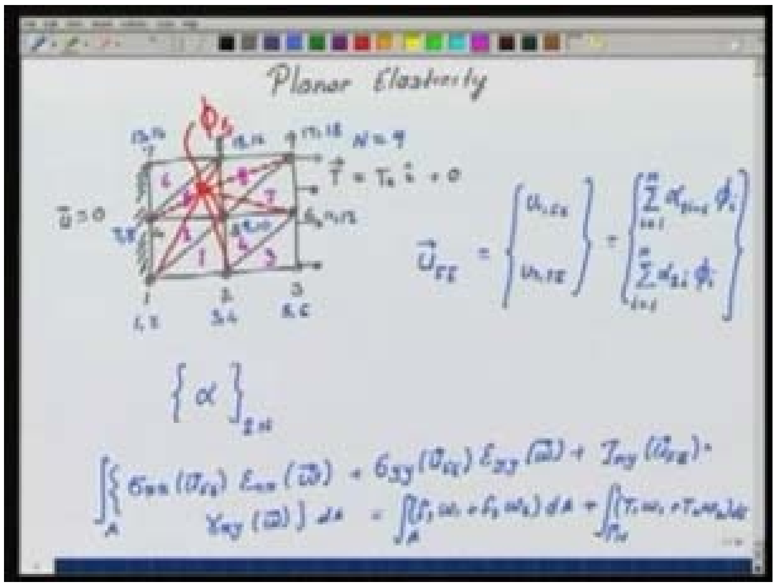
(Refer Slide Time: 16:08)



What is this then? This is now the global matrix K, which is of size 2N by 2N. This is into $alpha_{2N}$ and this is F, which is of size 2N. So the global stiffness matrix comes out of the integral over the area of what we call as the B T D V matrix. D is generally used as the material matrix instead of the C that we have used. That is just a naming convention. So I could do this integral and then I am done. That is, I have the global stiffness matrix.

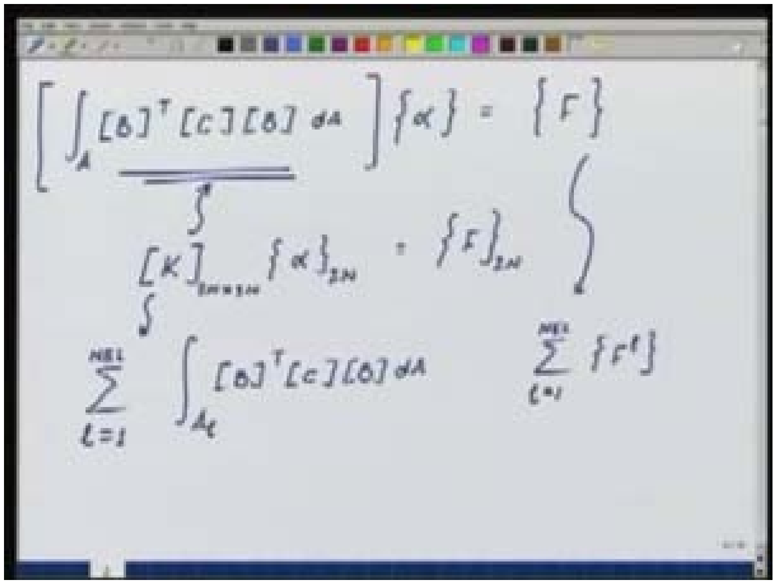
It should be relatively easy to convert this to the element pieces, which could be assembled back to get the global equations. I can now write this as integral over elements 1 to number of elements over the area of the element L of this piece. So this will be this term. Similarly, I can write F as summation over the elements of F due to the element l. Now it is just a matter of obtaining these integrals. Obviously, in order to obtain these integrals, we convert it to the element notation. That is, we write everything in terms of the element shape functions instead of the global basis functions because we are looking at the restriction of the basis functions to the element. Similarly, the load vector will also be written in terms of the element shape functions. The integrals will be done as we have done till now for the one-D and the single variable problem in two-D. Integrals will be converted to an integral over master element because that is where we can do the numerical integration and then evaluate it on the computer. So as far as numerical integration and those aspects are concerned, we are not going to get into that now because we have done enough of that part. Let us now go and look at the element calculations. How will I do this? In order to do this, let us go back to our original figure.

(Refer Slide Time: 19:06)



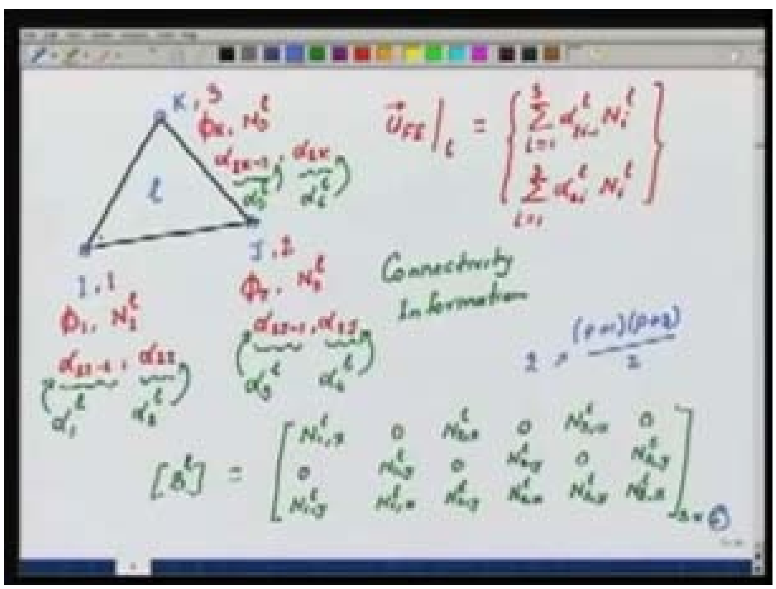
Let us give some names to these elements. Let me give this as element 1, element 2, this is element 3, 4, 5, 6, 7 and 8. So these are our 8 elements that we have taken for this simple problem that we are considering here.

(Refer Slide Time: 19:36)



The number of elements for the example problem is 4. Now let us go and do the following.

(Refer Slide Time: 19:49)



I have a generic element. I will do it with the piecewise linear only. This can be extended to the higher order ones, where these are the global nodes I, J, K. This is the generic element l. So this is the global node I, J, K. Similarly, if we are talking of the higher order element approximations, then we will have global interior nodes, the mid side nodes, volume nodes and the area nodes, depending on the order of approximation. So these are the global nodes I J K.

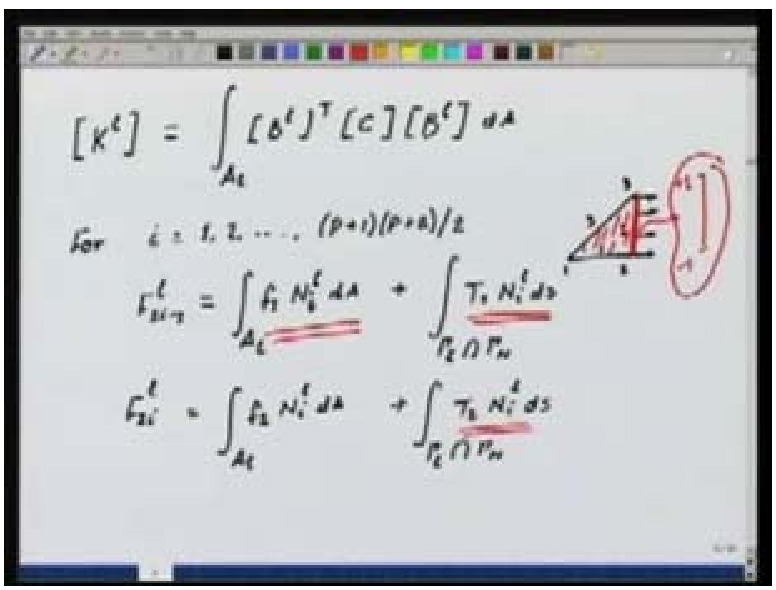
When we go to the element convention, this I will correspond to 1. This J will correspond to 2, this will correspond to 3. So in the element, I am giving a new naming convention. This is the node 1, 2 and 3. Corresponding to this node, I had phi_i globally. In the element, I will call it N_1 of the element l. Here I had phi_j globally and here I will call it as N_2 of element l and here I had phi_k globally, which is the same as N_3 of the element l. So what would the finite element solution in the element look like? Vector u_{FE} in the element l would be given by sum i is equal to 1 to 3 $alpha_{2i \text{ minus } 1}$ of the element l, N_i of the element l.

Similarly, the second component of the displacement or the v component will be $alpha_{2i}$ of the element l, N_i of the element l. Corresponding to $alpha_{2i}$ I will have the global degrees of freedom, $alpha_{2I \text{ minus } 1}$ and $alpha_{2I}$. Here, it will be $alpha_{2J \text{ minus } 1}$ and $alpha_{2J}$ and here it will be $alpha_{2K \text{ minus } 1}$ and $alpha_{2K}$. This will correspond to $alpha_1$ of the element l. This will correspond to $alpha_2$ of the element l. This will correspond to $alpha_3$ of the element l, $alpha_4$ and $alpha_5$ of the element l and $alpha_6$ of the element l. We have done a counter-clockwise enumeration of the degrees of freedom, that is, we went from here to here to here and not from here to here to here. (Refer Slide Time: 23:24) This is our choice. It is not that I have to make this node 1. I could have made this one 1, but it is naming convention that we choose at the element level. The bottom line is, it should be counter clockwise. The node should be numbered in a counter clockwise manner.

Once we have this, we know we have to establish this local to global enumeration. How will we establish this? When we do the degrees of freedom creation and numbering, that is where we establish this relationship. It is called the connectivity matrix, connectivity information. It is not difficult to create this for the mesh. Once I have this, what it tells us is that, the finite element solution in the element is given in terms of these. When we talk of B in the element or the matrix B that we have created, B will have non-zero entries only corresponding to the basis functions, which are non-zero in the element. B will have a non-zero entry for the element corresponding to only phi_i, phi_I, phi_J and phi_K. All other phi will have zero entries. But now we write these in terms of N_1, N_2, N_3 of the element. We can write the B transpose CB matrix at the element level and understand where to put them in the global matrix. That will do the job.

What is the B matrix at the element level? The B matrix at the element level for this case, a piecewise linear problem, will be N_1 of the element, x, 0, N_2 of the element, x, 0, N_3 of the element, x, 0. Similarly, I will have 0, N_1 of the element, y, 0, N_2 of the element, y, 0, N_3 of the element, y. Here I will have N_1 of the element, y, N_1 of the element, x, N_2 of the element, y, N_2 of the element, x, N_3 of the element, y and N_3 of the element, x. This is B for the element l. This is now, in the case of linear approximation, 3 by 6 because there are 3 basis functions. By the same token, if we have elements of order P, then how many unknowns will I have in the element? I will have P plus 1 into P plus 2 divided by 2 numbers of nodes into the number of degrees of freedom per node. That is, it will be P plus 1 into P plus 2. So this quantity would be P plus 1 into P plus 2. If we see for the linear, we had P plus 1 is 2. This one is 3. 2 into 3 is 6. This is going to be the B to the power l matrix that we have created at the element level.

(Refer Slide Time: 27:53)

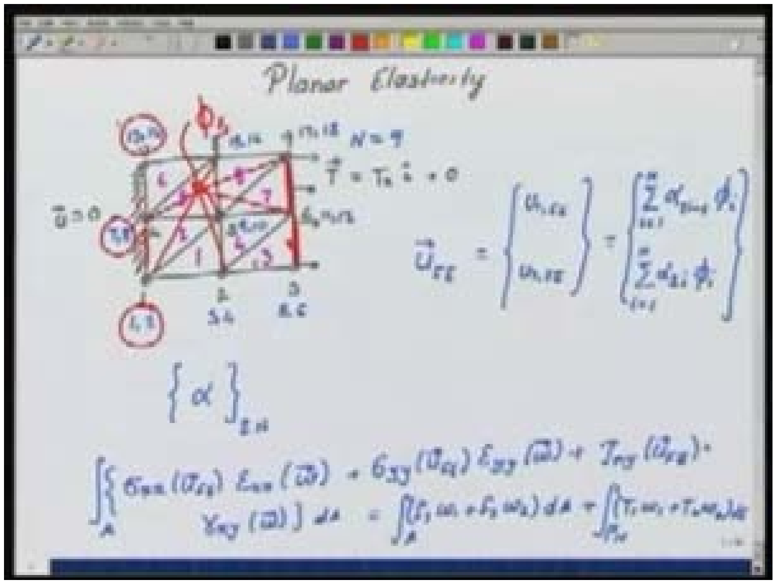


Similarly, it is rather easy to define the element stiffness matrix, that is, a part of B transpose CA, that is going to come out of the element. This is integral B power l transpose C B power l dA. What will happen to the load vector? The load vector will be: for i is equal to 1, 2, … (P plus 1) into P plus 2 divided by 2, I will have F from the element 2i minus 1 that is, it corresponds to the choice w is equal to phi_I, 0. This will be equal to integral over the area of the element, integral f_1 in the element into N_i in the element dA plus; if the element shares an edge with the Norman boundary, that is, let this element have an edge with its Norman boundary, it could be that this is the node 1 of the element, node 2 and node 3, this is going to be the edge 1, this is edge 2 and this is edge 3, this edge is common with the Norman boundary; we will denote the boundary of the element as $gamma_l$ intersection the Norman boundary of the domain, T_1N_il ds.

Similarly, F_{2i} of the element will be integral over A_l f_2 N_i of the element, dA, plus integral over $gamma_l$ intersection with $gamma_N$, $T_2N_i^l$ ds. This is going to give us the load vector for the element. Note very carefully that this is an integral over the area. This f_1 N_il dA is an integral over the area of the element. Area of the element is this. So for that we will have to use the integration points defined over the area of the element. While this quantity and this quantity are defined, these are integrals obtained over the edge of the element. Edge is a line. So this is a one-dimensional integration. So in order to do this integration, we have to transform this edge, from -1 to +1 edge and then do the integral. We should be able to do the transformation very easily. Take this edge to this edge, get the integration points because this is where I know the integration points are and do the integration by summation.

This has to be done as a separate loop in each element over the three edges of the element and we have to carry the information about, which edge of the element lies on the domain boundary and which does not. For an edge which does not lie on the domain boundary, like here, these internal edges, I do not do anything. For an edge which lies on a domain boundary with zero displacements or zero traction conditions, I do not do anything because those integrals on the right hand side become zero. Only for the case where the traction conditions are given, the traction boundary conditions are non-zero. In those cases, I have to do these integrals on the boundary and put them in the load term. Here we are doing things concurrently. The Neumann boundary condition is taken care of simultaneously with the load calculations. In the one-D case, we very easily handled the two end points separately. Here we do not because we do not have that luxury. We do it right away.

(Refer Slide Time: 33:06)



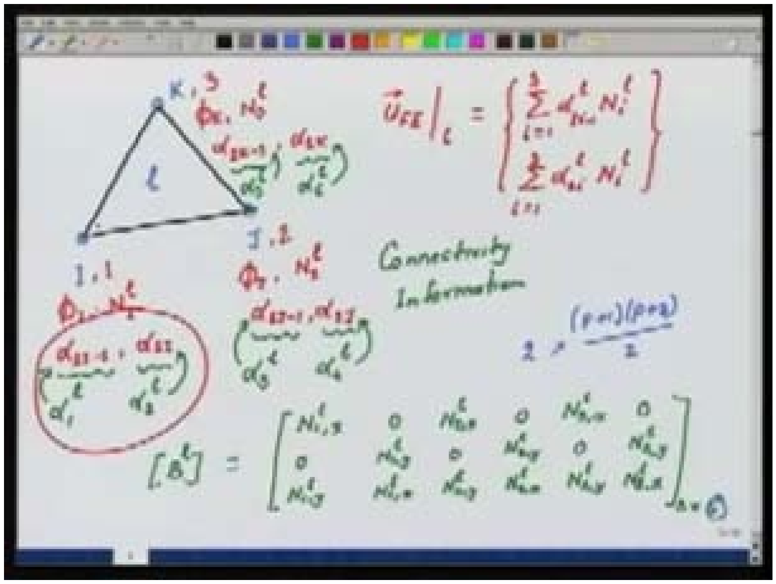
Again if I go back here (Refer Slide Time 19:06), if I look at these elements, we see for element 1, I do nothing on any of these three edges, because on these three edges, either zero traction condition is given or no boundary conditions or these are internal edges.

If I go to element 2, here the displacement is zero on the third edge of the element 2, on this edge of the element 2. So as far as this element is concerned, again I do nothing, as far as the integrals on the boundary are concerned. In element 3, I come to the second edge. Let this be edge 1 for the element 3. Here I have to do these integrals on the boundary, while on the other two edges, I do nothing.

Similarly, for element 7, I have to do the integral on this edge, do nothing here and for all the other elements, I do nothing. Finally, I am going to go and enforce the displacement boundary conditions explicitly just like we did earlier. That is we have to force, in this problem, $alpha_1$ $alpha_2$ is equal to 0 because the vector u is 0 on this edge.

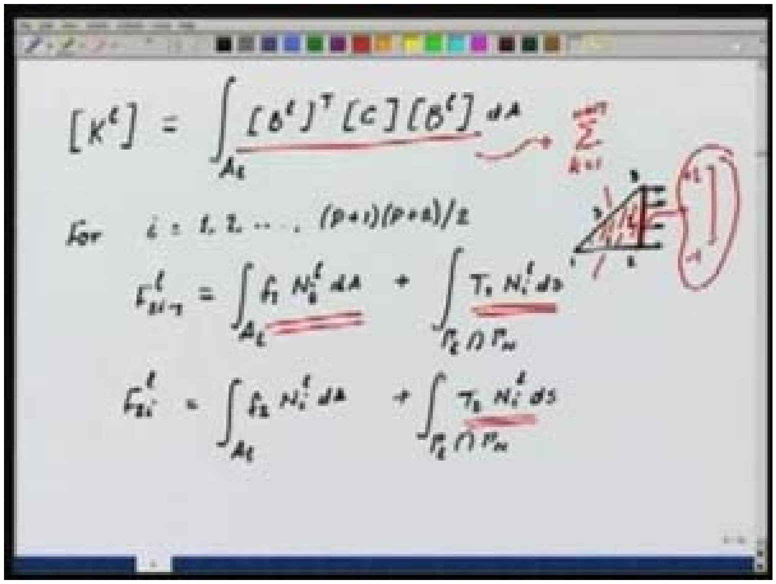
Similarly, I have to force $alpha_7$ $alpha_8$ is equal to 0 and $alpha_{13}$ $alpha_{14}$ is equal to 0. So this way, I can construct the element, load vector entries, and the element stiffness entries.

(Refer Slide Time: 34:40)



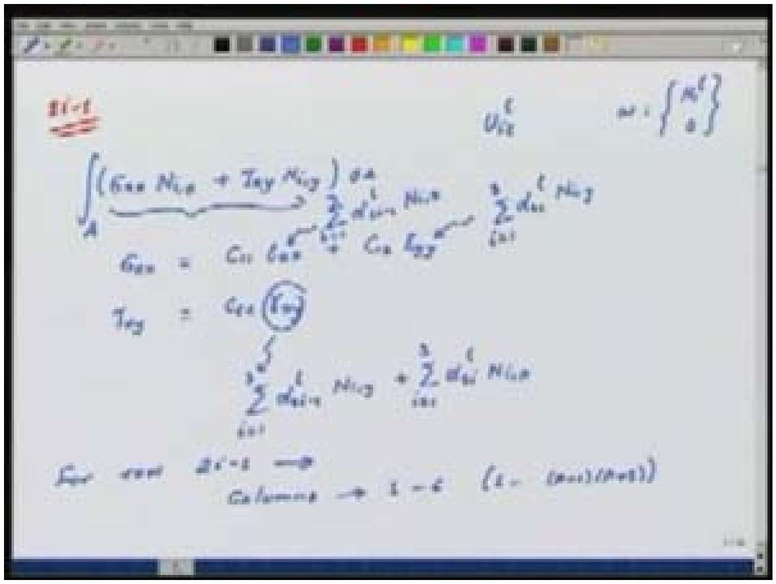
Once we have done this, then we go back to this connectivity information (Refer Slide Time 19:49) and simply assemble the entries of the element stiffness matrix and the load vector in the corresponding entries in the global stiffness matrix and the global load vectors. Here we will be doing things two at a time because corresponding to each phi$_{i}$ there are two unknown coefficients alpha$_{2i \text{ minus } 1}$ and alpha$_{2i}$. This is how one would go ahead and do the element calculations, assemble and setup the problem to be solved.

(Refer Slide Time: 35:28)



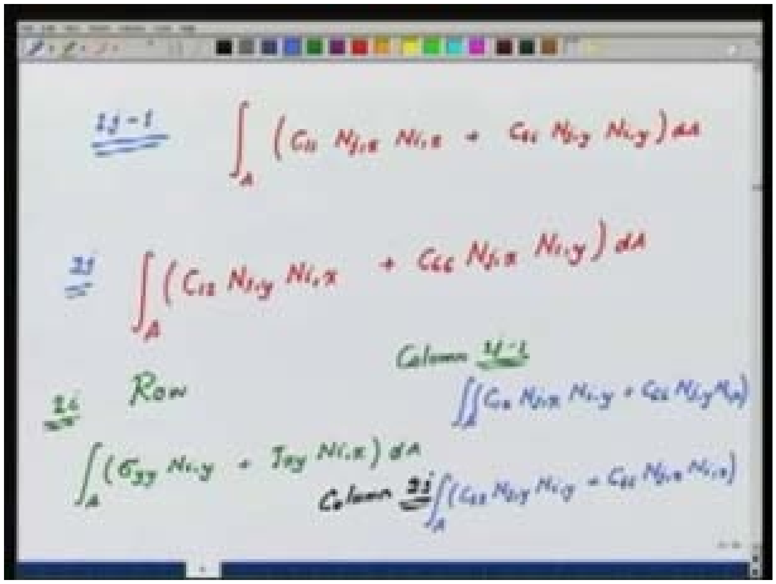
This is nice. Cannot we do this in a more explicit way? This B transpose C B approach is something that is commonly available in all the books and this is what people follow. But here something which I do not like personally is that here we are dealing with matrices. We have to store these matrices for every integration point because this integral will be written in terms of summation over integration points. At each integration point, I have to go and compute the values of the shape functions and the derivatives. Construct this matrix B to the power l and do B to the power l transpose C B at each integration point and then put it in. Cannot I do this explicitly? That will reduce the cost of computation, because, if we can do this job explicitly, it is quite easy.

(Refer Slide Time: 36:35)



So if I go back to our element equation, if I look at the 2i minus 1 equation, I will have integral over the area sigma$_{xx}$ N, I am dropping the l, $N_{i, x}$ plus tau$_{xy}$ $N_{i,y}$ dA. This is going to be the row corresponding to alpha$_{2i-1}$ in the element or corresponding to N_i in the element. Sigma$_{xx}$ is actually by what we have done, C_{11} E_{xx} plus C_{12} $E_{yy.}$ Tau$_{xy}$ is equal to C_{66} gamma$_{xy}$. I know that u_{FE} in the element is given by the representation, we have done already. This is going to be equal to sigma i is equal 1 to the number of unknowns in the element. Here we have taken 3, alpha$_{2iminus1}$ in the element, $N_{i, x}$. Similarly, this one will be sigma i is equal to 1 to 3 alpha$_{2i}$ in the element, $N_{i, y}$ and similarly, this one will be sigma i is equal to 1 to 3 alpha$_{2i-1}$ in the element $N_{i, y}$ plus sigma i is equal to 1 to 3 alpha$_{2i}$ in the element $N_{i, x}$. What am I trying to get? Corresponding to choice w is equal to N_i of the element 0, I can explicitly get the stiffness entries in terms of the derivatives of the N_is and the N_js. How will I do it? Corresponding to 2i minus1, for row 2i-1, take columns, column will go from 1 to 6 in this case or in the general case, 1 to P plus 1 into P plus 2.

(Refer Slide Time: 40:16)

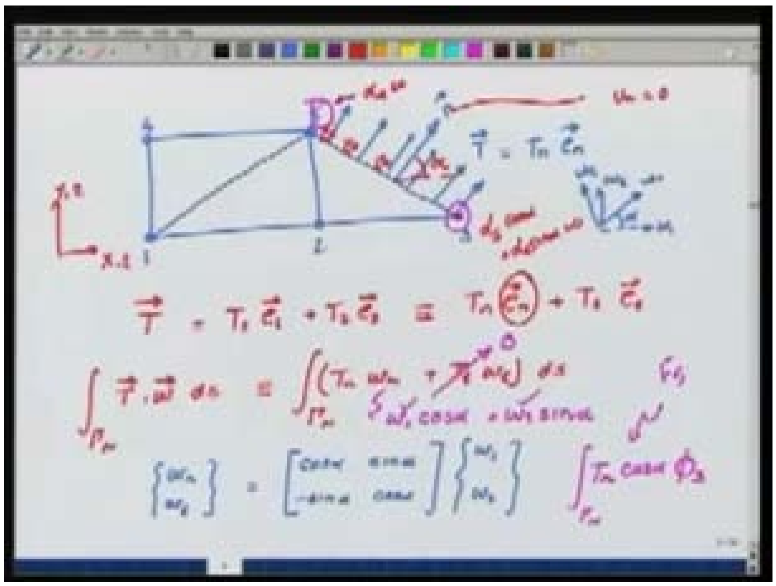


Let us now look at the column 2j-1. So for the row 2i-1, the column 2j-1 will have contributions due to, we see $alpha_{2i-1}$ will be this one and it will be this one. I will get this as C_{11} $N_{j, x}$ into $N_{i, x}$ plus C_{66} $N_{j, y}$ into, if I go back, I will have $N_{i, y}$, into $N_{i, y}$. Integrate this quantity over the area. This will give the column 2j-1. Similarly, remember for the row 2i-1, so column 2j will be the remaining part. It will be integral over the area of C_{12} $N_{j, y}$ $N_{i, x}$ plus C_{66} $N_{j, x}$ into $N_{i, y}$.

Where did this come from? I got this one from this part and this part. So very easily I can get these entries explicitly. I put it in the loop for the integration, loop of the integration points and compute these quantities.

Similarly, for the row 2i for the element, this will correspond to integral over the area $sigma_{xx}$ part will be 0, $sigma_{yy}$ $N_{i, y}$ plus tau_{xy} $N_{i, x}$ dA. This part, I can again write in terms of the 2j-1th column and the 2jth column. So I do column 2j-1. This will have integral over the area of C, if I go back to the material matrix for $sigma_{yy,}$ it is C_{12} into $E_{xx,}$ so C_{12} $N_{j, x}$ $N_{i, y}$ plus C_{66} $N_{j, y}$ $N_{i, x}$ integrated over the area. Similarly, column 2j, this would mean I will do integral over the area of C_{22} $N_{j, y}$ $N_{i, y}$ plus C_{66} $N_{j, x}$ $N_{i, x}$. So it is very easy to write these expressions and in someway this is explicit operation of doing this B transpose DB, has been written in terms of the expanded expressions that we would have obtained. One can check it that this is exactly what we will get out of the B transpose DB and we can write it in a loop over the integration points and get the job done. This will give me the element stiffness matrix. Similarly, we can handle the load vector entries and do the assembly to get the global system. Then in the global system, I explicitly impose these boundary conditions for alpha$_1$ alpha$_2$ alpha$_7$ alpha$_8$ and so on and solve the system to get the solution. There are one or two important issues that we should also look at.

(Refer Slide Time: 45:27)



Let us take for example a boundary, which is like this and let us say (again, I am doing it with a very simple mesh) that this is node 1, node 2, node 3, node 4, and node 5. I am solving it with piecewise linear approximation. We could do the same thing with the higher order ones. Let us say traction conditions are given like this here on this boundary. T is equal to T_n into the normal in the n direction. What is the normal? This is the normal in this n direction, the unit vector in the normal direction. If I am given the end load like this, how do I handle it? In this case, it is again very easy. We know that we had written our traction vector as T is equal to T_1 $e_{1,}$ where this is the x and y direction or this is the 1 and 2 direction plus T_2 e_2. This is also equivalent to writing as T_n e_n plus T_t $e_{t.}$ And how is T_n given? This is angle alpha. So e_n will be obtained quite easily. The components of e_n will be cos alpha$_i$ and sine alpha$_{j.}$ I can write it like this. So the work done when we do the work on gamma$_N$ T dotted with w ds could also be written as integral over gamma$_N$ T_n w_n plus T_t w_t ds.

Now the question is, our approximation or the finite element solution is defined in terms of the Cartesian components, the xy components, so how do I convert it to the n and the t components? It is very easy. w_n w_t is equal to, (here I essentially want components in this coordinate system when I have components in this coordinate system. If I look at w_n and w_t, w_n will be w_1 cos alpha plus w_2 sin alpha) cos alpha sin alpha. w_t will be equal to minus w1 sine alpha plus w_2 cos alpha So this is w_1, this is w_2. So it is minus w_1 sin alpha, w_2 cos alpha. This is in terms of w_1 and w_2.

I am talking about the function defined on this phase, which is this inclined phase. Now it is very easy that I put the boundary condition because here for example, the T is 0. Let us knock this off. w_n will now be given as w_1 cos alpha plus w_2 sine alpha. We have defined our w in terms of the Cartesian coordinates, that is, in terms of the components w_1 and $w_{2.}$ What will happen to the nodal equations? I will write T_n into w into cos alpha integrated against the phi corresponding to this one will give the first load term. Let us say corresponding to node 3, what will I get as the load contribution corresponding to phi_3. It will be integral over this edge T_n into cos alpha $phi_{3.}$ This will go into equation F_5. It will go to the x component of the load vector.

Similarly, the other one will have T_n sin alpha phi_3 which will go to F_6. If I go to the next one here, I will have T_n cos alpha phi_5 integral will go to F_9 corresponding to 2i-1 here and T_n sin alpha phi_5 will go to F_{10}. This way we can handle inclined boundary. What about an inclined displacement condition on an inclined edge? For example, if I have this edge and instead of this, I have displacement scales, let us say rollers, where the normal displacement is 0. That condition, that constraint has to be imposed. This means, in this case u_n is 0. u_n is given as a combination of u_1 and u_2 and u_1 cos alpha plus u_2 sine alpha has to be 0 on this edge. So we will do everything that we do in the standard way. Assemble the global stiffness matrix from the load vector and then go and impose this condition to eliminate one of the unknown coefficients. I can write here, from this edge, u_n is equal to alpha 5 alpha 5 phi 5 into cosine alpha plus alpha 9 phi 5 alpha phi 5 3 cosine alpha plus alpha 9 phi 5 cosine alpha plus alpha 6 phi 3 sine alpha plus alpha 10 phi 5 sin alpha is equal to 0. From there, I can write these alphas, that is, $alpha_5$ cosine alpha plus $alpha_6$ sine alpha is equal to 0 and from here, I will get $alpha_9$ cosine alpha, cos alpha plus $alpha_{10}$ sin alpha is equal to 0. This constraint has to be imposed on the stiffness matrix. I can eliminate one of them in terms of the other and then the equations also get properly modified and I will solve that system and get the solution to the problem.

With this, I would like to conclude the part on the element calculation and the construction of the global stiffness and the load vectors for the planar elasticity problem. Everything follows the same line of attack that we had started in the first lecture. In the next lecture, we will talk about how to post process the stresses that we get out of the finite element solution, because we know that those stresses, by construction, especially for lower P orders are essentially discontinuous, while the state of stress in the actual case is continuous. So I will have to do some post-processing just like we did in the one-dimensional problem in order to obtain a seemingly better or hopefully a better state of stress out of the finite element data and then we will go and look at some curve geometries. How do we handle curve domains? Here, we have not handled curve domains. In that case, what kind of an approximation for the geometry has to be done and how does it affect the scheme of things that we have?