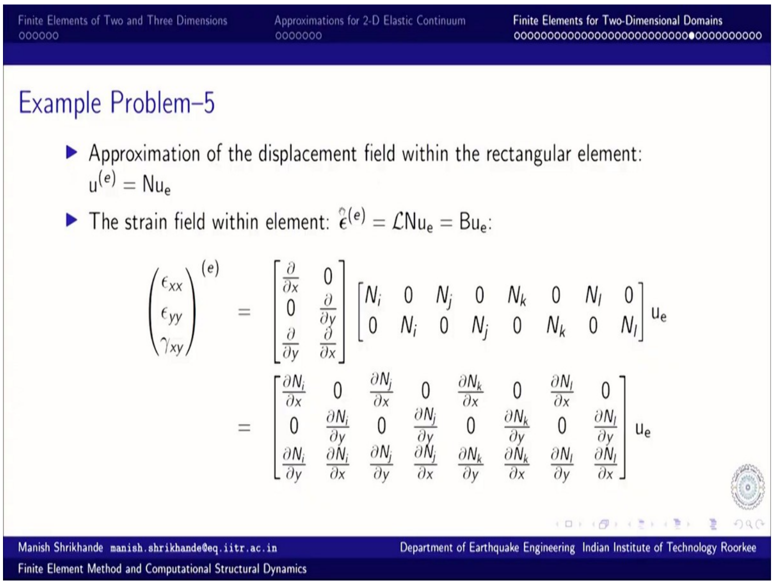
Finite Element Method and Computational Structural Dynamics
Prof. Manish Shrikhande
Department of Earthquake Engineering
Indian Institute of Technology, Roorkee

Lecture - 30
Finite Elements of C^o Continuityin 2-D and 3-D- XII

Hello friends. So, we were discussing about solution of a bracket problem, plane stress problem, steel bracket problem using Finite Elements. And we choose to discretize the domain by using 4 elements of 3 elements of 4 node rectangle and 1 element of 3 node triangle.

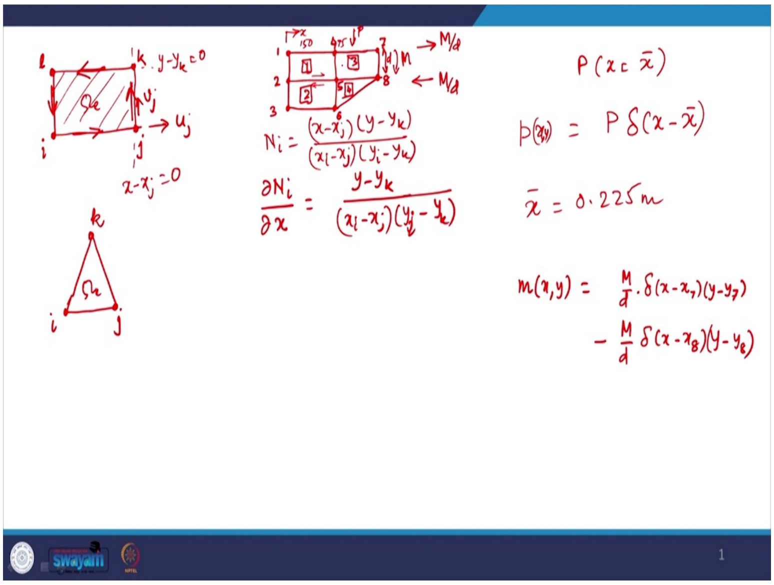
And we saw how the mesh was discretized, and the nodes, and what is the mapping between the local node numbering and global node numbering within the element, and how the displacements are interpolated once the coordinates are defined the of the nodes. The interpolation functions can be derived. And once the interpolation functions are available, they can be related to the derivatives.

(Refer Slide Time: 01:19)



So, the strain field and those derivatives the strain derivatives that these are essentially the derivatives of partial derivatives of interpolation functions. Now, these partial derivatives are completely defined when we specify the coordinates of the nodes. So, for example, Ni is defined by as x - xj multiplied by y - yk divided by xi - xj by yi - yk.

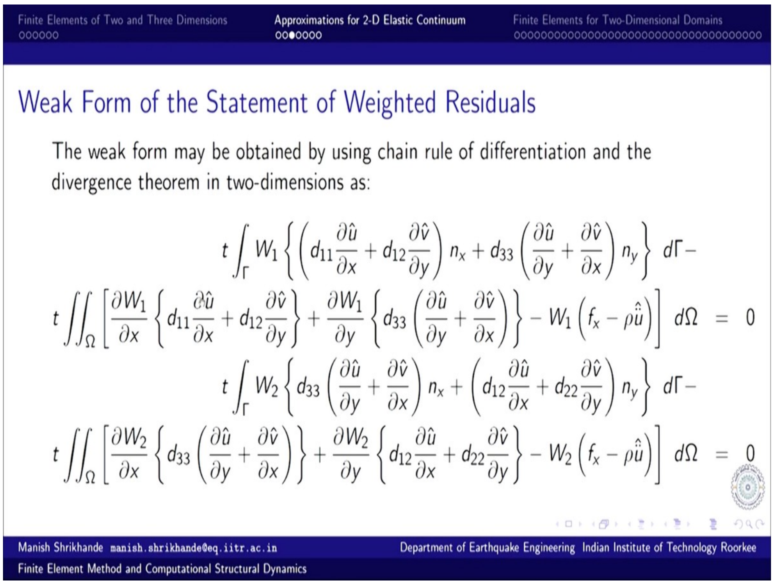
(Refer Slide Time: 02:04)



Now, the first derivative is of course, a function of y and the denominator. So, just to tell you, so $\frac{\partial N_i}{\partial x} = \frac{y - y_k}{(x_i - x_j)(y_i - y_k)}$. So, it is linear in y and similarly all derivative functions. So, N_i is essentially (x - xj) (y - yk) divided by numerator normalized at the node (xi - xj), (yi - yk). So, derivative is simply this remaining expression.

This is completely defined the moment we have the coordinates of all these nodes in the mesh. So, once we map these local node class identifiers i, j, k, and l, with the physical actual node in the global mesh, then these derivatives can be evaluated readily by using these coordinates.

(Refer Slide Time: 03:26)

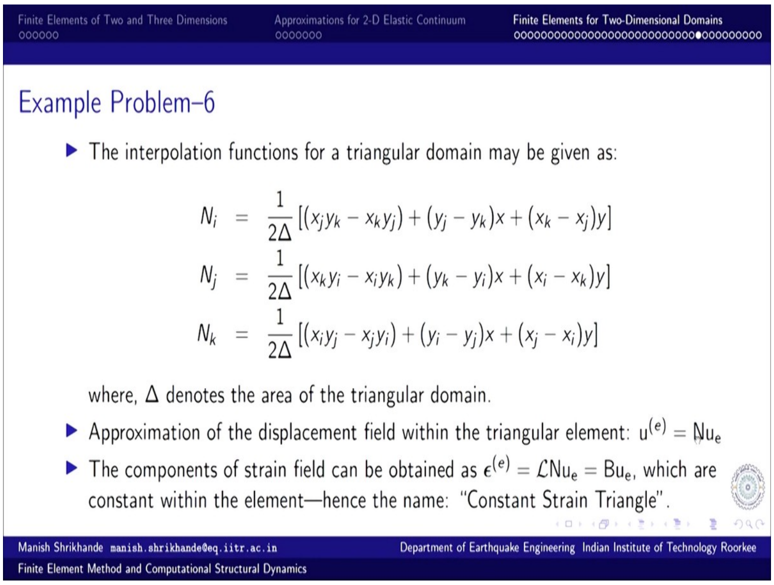


And that is how the terms, respective terms, derivative terms in the domain integral these are evaluated. So, $\frac{\partial u}{\partial x}$ will be defined by similar shape function derivatives, and once, because those are already defined in terms of nodal coordinates for each node, for each element the. And this is of course, the waiting function that will keep on changing W can be either N, Ni, Nj, Nk, Nl in succession.

So, again the derivative is defined in terms of the coordinate functions, coordinate variables, nodal coordinates. And this expression then can be evaluated with respect to x and y. So, domain integral and the this entire expression will be known as the stiffness matrix of the problem.

So, this is the approximation within the element of any rectangular element, or for that matter even for triangular element, the only difference is the number of variables nodal variables and the size of this matrix N. Other than that, there is no difference between the this representation of displacement field. Of course, this expression of interpolation function will be different for in the case of rectangle and for from the a triangular element.

(Refer Slide Time: 05:05)



So, for a triangular domain the interpolation functions can again be given in terms of nodal coordinates, as we had seen during this during our discussion of triangular family. So, Ni the interpolation function for ith node that we have, this is the ith node location in the local node numbering for element.

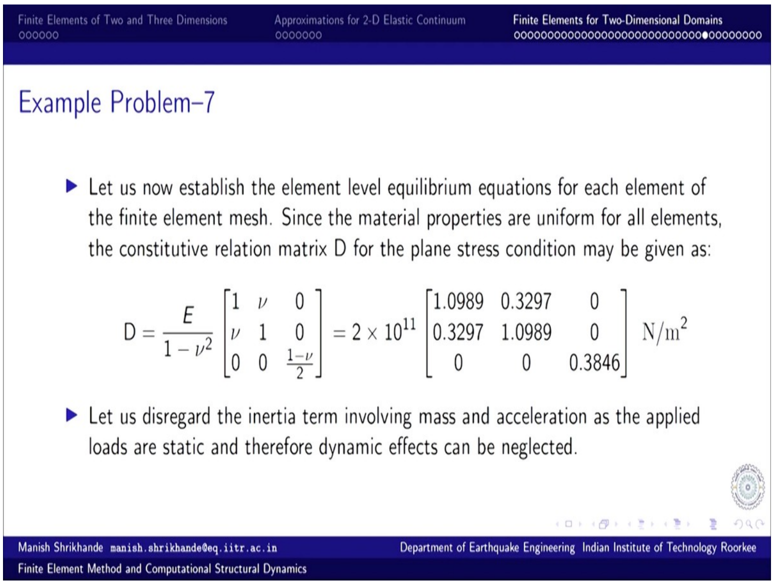
So, this is defined in terms of coordinates of node j and node k, as (x_j y_k) - (x_k y_j), +, (y_j – y_k) x. So, this is the coefficient of x and x_k - x_j multiplied by y. So, that is the coefficient of y and the entire thing normalized by twice the area of triangle. So, this is the twice delta is the determinant of the coordinates 1 x_i y_i, 1 x_j y_j, 1 x_k y_k. So, that determinant of 3 by 3 matrixis the; is equal to twice the area of the triangle enclosed by these 3 nodes.

So, similarly we have by cyclic permutation, we can had derive the interpolation for node j and node k, and you can see wherever we have j for the ith node interpolation that is replaced byk for j's node interpolation and so on. So, cyclic symmetryis holds in this case.

So, the approximation of the displacement field within triangular element is again similarly given by shape function matrix multiplied by the displacement, nodal displacement. The interpolation, the strain field is again same differential operator, operating on the interpolation function. And because this is all of them are first degree polynomials.

So, the derivatives are constants. So, this strain field is constant over the entire element. So, ε_{xx} is constant over the entire element, ε_{yy} is constant over the entire element, and similarly γ_{xy} is also constant over the entire element. And hence the name constant strain triangle or CST that is often used for in place of 3 node triangle element.

(Refer Slide Time: 07:39)



So, in order to establish the element level equilibrium equations, we need the constitutive matrix D for plane stress conditions, what are referred to as coefficients d11, d12, d33, etcetera. So, these are the coefficients of those constitutive matrix.

So, these are coefficients are of course, going to be different for plane stress conditions and for plane strain conditions. But for the once we idealize, once we consider, once we reassure ourselves that the problem that is being dealt with is consistent with the assumptions of plane stress conditions, then the constitutive matrix for plane stress condition which is given by this is can be derived in terms of the element properties that is Young's modulus and Poisson's ratio. And this is the constitutive matrix for plane stress conditions.

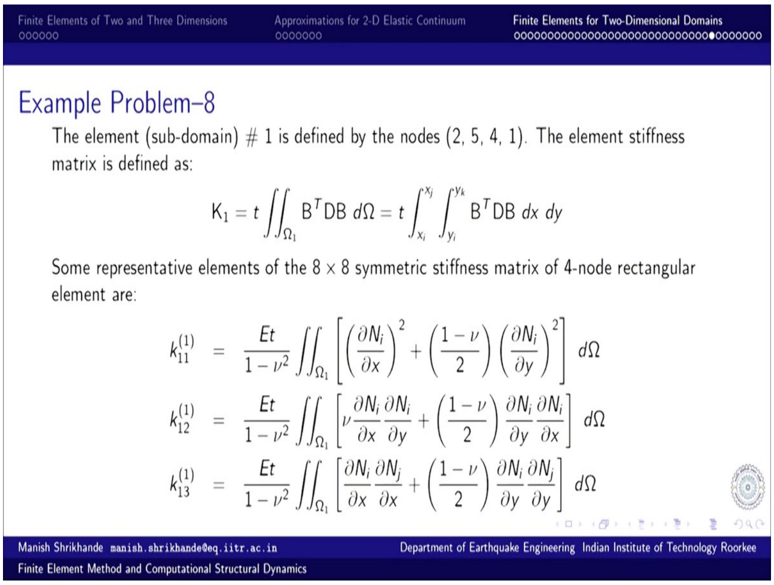
We will also in this particular case because the loading conditions, they are constant, they are not changing with time. So, there are inertia terms are not significant and we will drop these acceleration terms, right. So, integral of W times $\rho\ddot{u}$. So, this is the acceleration term. So, these are the inertia terms that define the govern the dynamics of this system, if the loading, if the system is subjected to rapidly changing loads or time varying loads.

Since, in this particular case we are not looking at time varying loads, so this calculation will not be required, otherwise this can be computed in the same way as other domain, other domain integral. The only thing that is being implicit, that will be implicitly assumed is the where, while in this part in this particular expression, we are looking at approximation of the displacement.

In this particular term, requires consideration of the variation of acceleration, how are we going, how are we approximating variation of acceleration within the element domain, right. So, we will revert to this return to this particular point after discussion of this static problem.

So, we ignore the dynamic effects. So, since the loads are static. So, now let us look at the formulation of element level equation.

(Refer Slide Time: 10:35)



We take element number 1 defined by 2, 5, 4, 1; 2, 5, 4, 1 as the element of our discussion. Process is exactlyidentical for all the elements. So, we will only refer to this in this discussion. Step by step process for all the elements and in going through the entire every single step, you can see in the solved example in the book that we have my book on finite element and computational structural dynamics.

So, I will not repeat the entire, I will not go through the entire series of calculations for all the elements in this discussion for paucity of times, because of paucity of time. So, the element stiffness matrix can be defined, I mean the all the equations that we have here, if I collect all these terms arrange it in matrix form, I will end up with equation of this type.

So, B transpose that is again derivative of the interpolation function. If you recall, strain displacement matrix involves derivative of the interpolation function and since we need to multiply, our weighting function is del v shape function by derivative of the weighting function multiplies with all of these terms. So, that is related to the strain displacement matrix again for Galerkin formulation and that is what we use.

So, essentiallyit boils down to expression of this type evaluation of expression of this type B transpose DB. So, B transpose is the strain displacement matrixin the case of borrowing the terminology from solid mechanics, D is the constitutive relation matrix, and B is the again strain displacement matrix, and this is to be integrated over the element domain.

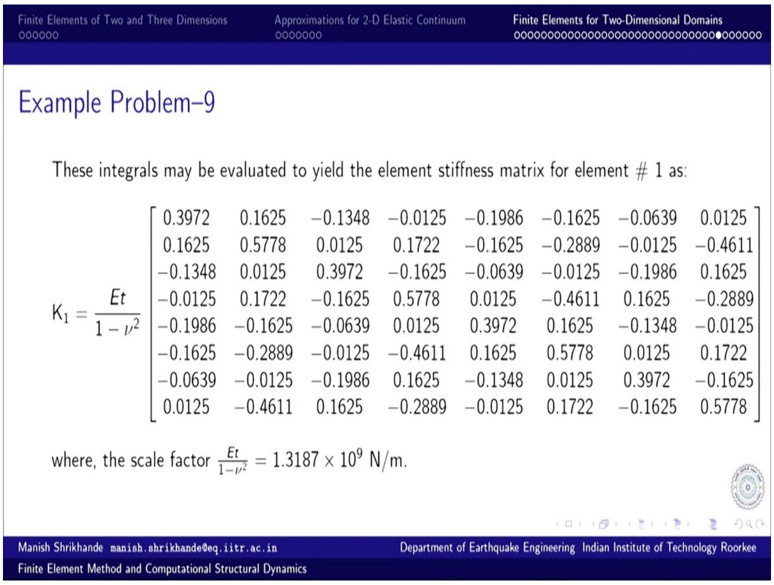
So, element domain is this entire element domain which is bounded by range x ranging from xi to xj and y ranging from yj to yk, right. So, these are the domains, and this is what omega 1 ranges from xi to xj, yj to yk, or yi to yk, does not matter. So, yi and yj they are identical. So, it really does not matter whether I use yi or yj. It is the same coordinate so yi to yk.

And within this integral, within this domain this these are to be evaluated. In this particular case, the element is the directions are oriented along the are parallel, the element edges are parallel to the coordinate direction. So, these integrals can be evaluated very easily or straight forward manner and eventually the 8 by 8 symmetric matrix terms can be evaluated individually as this. So, these are some of the representative evaluations that come out.

So, this is of course, E over 1 - ν^2 is from the constitutive matrix D, and rest of the terms they are coming from the product of B transpose DB individual elements that they come. And this is the superscript, 1 refers to element number 1, and subscripts they refer to the dimension. So, there are total 8 by 8 matrix, so 64 elements. So, the k 11 refers to element in first row and first column for the element stiffness matrix.

And similarly, I mean all these elements element by element evaluation can be done because all these interpolation function derivatives, they are known once the coordinates of these nodes are known and then they can be evaluated. These are one-dimensional, I mean linear functions of x and y, and these can be evaluated in, and the integral, definite integral can be evaluated.

(Refer Slide Time: 15:29)



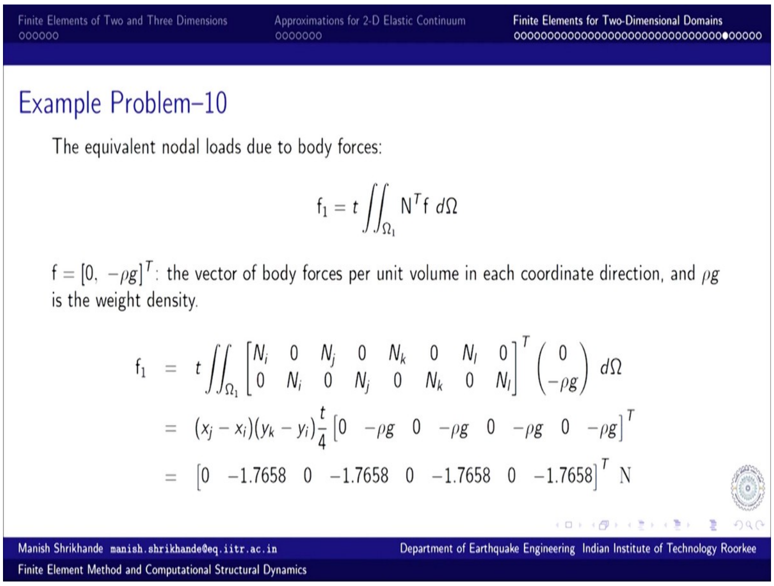
And this is the final integral that stiffness matrix that we have for element number 1. And this scaling factor that we have is based on this Young's modulus and the thickness t of the element and Poisson's ratio nu square. So, this is equal to 1.31 x 10^9 N/m, multiplied by this coefficient matrix 8 by 8 coefficient matrix. So, that defines the stiffness matrix for element 1.

Once we are done with this, then we move to the domain integral, the load vector. So, we are done with this. Now, we need to look at what is the body force component. So, what are the nodal equivalent of these? So, if you look at it essentially this looks like a, this is essentially the work done work equivalent.

So, what is the work; so, that the work done by these forces in moving through the displacement should be same as the work done by the nodal forces, what are we putting and the nodes, and that the work done would be same as work done by the nodal forces in moving through the nodal displacements with primary variable of that node.

So, that is what is evaluated it simply n f(x), x component of the body force and evaluated over the in domain integral domain interval.

(Refer Slide Time: 17:18)



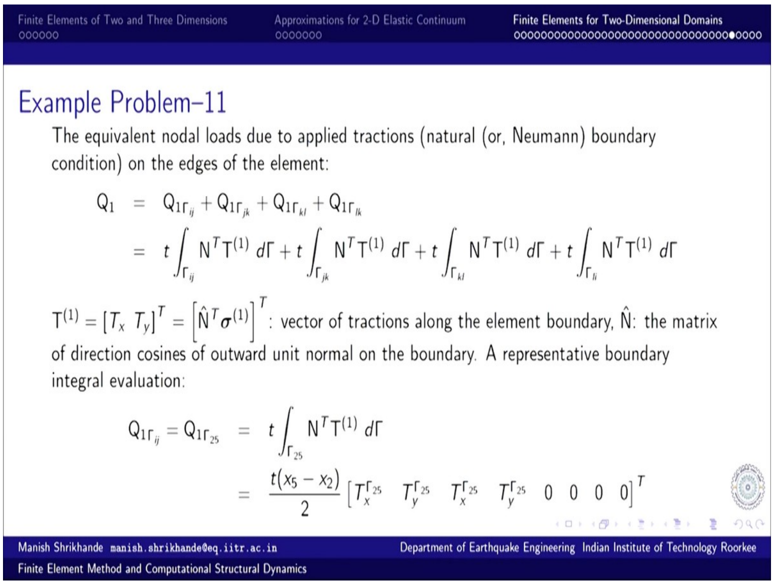
So, this is what the nodal forces will look like, evaluation of nodal forces. So, waiting function multiplication is represented by N^T f. So, f is the vector of nodal forces. In this case, there is no body force in the along the x direction, but there is a body force component along y direction that is self-weight, and that is referred to here as rho times g per unit volume.

So, rho is the density and - sign because weight acts vertically downwards and the positive sign is assumed upward as the positive y direction. So, this is the body force component along x and y direction, and when we multiply this with the weighting weight function matrix or the interpolation matrix to represent the weight function multiplication, domain integral.

And then, evaluate it along the domain xi to xj and yi to yk, we end up with this expressions, that you can check and straightforward manner. And once we substitute the values for all of these coordinates and for element number 1, and thickness and values of rho and acceleration due to gravity g, we get this vector for the element forces as this in terms of I mean Newtons, so many Newtons.

So, essentially equally distributed at all the nodes. So, 1.76 Newton downward at all the 4 nodes. So, that is the gravity load self-weight. So, that takes care of the boundary domain integrals in this case. So, this particular integral, both of these in both of these equations they are taken care of, accept this inertia term which we are dropping because acceleration is deem to be insignificant or negligible.

(Refer Slide Time: 19:51)



What remains now is this boundaryintegral. How to evaluate these boundaryintegrals? And these boundaryintegrals are again evaluated by segregating the boundaries, looking at each boundary at one time. So, this is gamma ij, the boundaryintegral is evaluated in counter clockwise direction, and in this particular case this boundaryij is common with the boundary for kl for element two. So, the those boundary terms would get cancelled.

Similarly, if this boundary term along this edge will get cancelled with the boundary term, similar boundary term coming from element number 3, and 1, 2, this boundaryit is of course, the essential boundary specified. So, Neumann boundary condition does not apply here. So, there will be reaction. So, though those reactions are unknown. And this is of course traction free.

So, boundary 4, 1, which corresponds to kl which coincides with the problem boundary the original boundary of the original domain. So, we only need to evaluate the boundaryin along this edge 4, 1, right. All other boundaries conditions are not really required to be evaluated, but nevertheless we will explain the process, how it is done.

So, together the rectangular element, there are any rectangular element is defined by 4 boundaries. So, γ_{ij} , γ_{jk} , γ_{kl} , and γ_{lk} , and again these are the order I mean the node numbering is the order in which we are moving. So, counter clock wise, i to j, j to k, k to l and from l to i, ok, fine. So, this one should be li not lk. So, that is a typo of here. Pardon for that error.

And this subscript one refers to element number 1. So, for element number 1 the boundary along the edge ij, for element number 1 boundary along jk edge, for element number 1 boundary along kl edge, and so on. So, that would be given byintegral. Again, thickness is of course, constant. So, integral along this edge weighting function multiplied by the traction, tractions for node element number 1. Along this edge and similarly for all these 4 terms, right.

And this traction, there are two components, I mean its two-dimensional problem. So, there can be tractions components along x direction and components along y direction. So, Tx or Tx and Ty and they are given by direction cosine, product of the direction cosine with the normal stress and tangential stress along the bound, along the bound element boundary.

So, this is what we will get, vector of tractions along the element boundary, N hat, the matrix of direction cosines of the outward unit normal. In this, for this particular geometryit is simply 1 and 0, and a representative boundary evaluation is what looks like this. So, $\hat{N} T_1$, so this is what Tx in along this edge Ty along edge 25 and again Tx along 25, Ty along 25. So, both of them divided equally between the two nodes 2 and 5, and then multiplied by what is the length and thickness of course remains.

And that completes the evaluation of the one boundary, one segment of the boundary, and similarly we can go for next segment, and third segment, fourth segment, and that is how we do that. And during the assembly, as we saw in the case of one-dimensional element these for the edge of 25 there would be a corresponding matching contribution coming from the element number 2 which would be from element 5 to 2. For element 1, we counter clockwise direction was from 2 to 5.

For element number 2, the correspond this edge would be evaluated from 5 to 2. And accordingly the direction cosines would of course, be outward normal, so they would be in opposite directions, and the corresponding terms would be of opposite signs and they would cancel out during the process of assembly, right. At the individual element level of course they do not, they appear, but during the process of assembly they would of course cancel out.

And therefore, generally we generally do not bother about calculation going through this calculation, unless this boundary coincides with the problem boundary. For example, this boundary of 4, 1 corresponds with the or boundary of 4, 1 corresponds with the actual problem boundary. So, we will evaluate it at this point. So, that brings us to the problem.

In this case, we have element number 3. We have this point load at the midpoint of this edge P and then there is a moment M. So, how do we model these? How do we account for these? So, P point load can be easily modeled as a Dirac delta function that s a discontinuity.

So, point load P at xis equal to x bar can be represented as function of P x as P delta x - x bar, and then this function P x can be treated as a continuous function of x and substituted in the integral. Integrant will be evaluated as the by using the property of Dirac delta function, and that would result in point evaluation sampling at x bar, and appropriately the calculations will be done.

So, all that we need to do is substitute for T x, the traction as instead of continuous function, this variation of Dirac delta function at x bar whatever the coordinates of x bar are. So, in this particular case it would be 225 because this is where the xis ranging from and this is at 75 + 150. So, this is 150 and this is 75, so together its 225. So, it would be xis equal to 0.225 meter. So, x bar is equal to 0.225 meter, for the problem that we are dealing with.

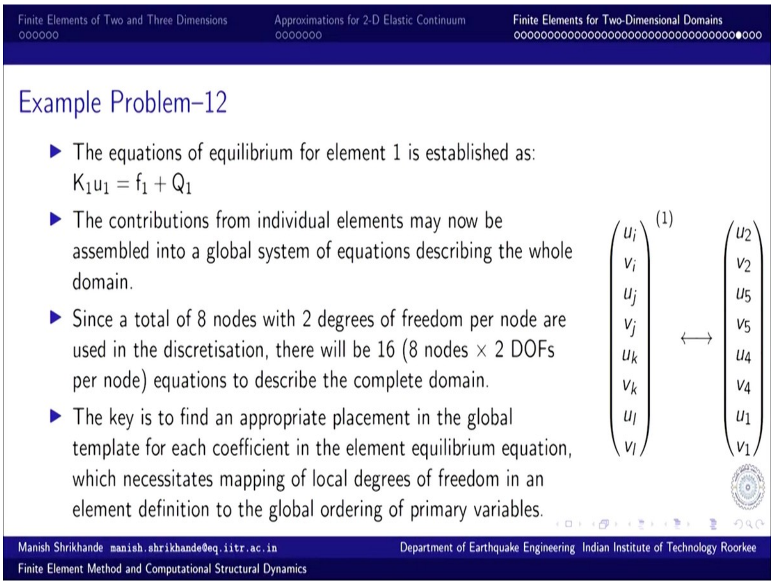
And that leaves us with another issue, how do we model the moment because there are no rotational degrees of freedom here, so how do we incorporate all the forces that we have, the allowance is only for the rectilinear forces, force along x direction, force along y direction, how do we model the moment, applied moment in this case, in this problem. So, the way to model moment is that this moment can be represented can be replaced by appropriate couple and this couple is equal to M by d, where d is this distance, right.

So, this constant couple separated over a distance d that defines the moment. So, while analyzing for moment we need to convert it into an equivalent couple that would be of pair of concentrated load, separated by a distance, d. So, in this case d is the depth of this element, and these would then be treated same in the same way as this Dirac delta function for this traction in y direction.

So, in this case, it would be traction along x directions of opposite signs. So, we will have delta x - x 7, multiplied by y - y 7, - delta x - x 8, y - y 8. So, that is the variation and using this variation it can be evaluated.

So, once we have this traction along x along this particular edge 7, 8, then this expression can be evaluated in exactly the same fashion as this. So, instead of traction, we will have these Dirac delta functions and then the integrals one-dimensional integrals can be evaluated.

(Refer Slide Time: 31:00)



So, once we have this evaluation, now similar equations the problem element number 1, now for the element number 1, the equilibrium equations look like stiffness matrix for element 1 multiplied by the nodal coordinates of for element nodal, variables for primary variables of our element number 1 is equal to the element force vector for node of element number 1, and the secondary variables reactions for element number 1. And these are then placed into appropriate locations of the global stiffness, global equation, equation for the entire system.

So, contributions from individual elements may then be assembled into global system. So, we are done with first element. And these element level equations are plugged into the placeholders for the global system of equations. Since, we have a total of 8 nodes with two degrees of freedom. So, there will be 16 equations in this particular problem, right. So, there are 8 number of nodes. So, total 8 number of nodes, each node has 2 degrees of freedom uj and vj. So, 2 degrees of freedom at each node.

So, there are total of 16 variables that we are looking at, so we need to have provision for 16 by 16 stiffness matrix, and 16 number of variable, 16 number of element level element forces, nodal equivalent forces, nodal equivalent forces and the secondary variable tractions.

Now, how does the assembly take place? The keyis to find appropriate placement and that is how the entire assembly process is, one different elements contribute the contribution of different elements goes in the along the degrees of freedom that they share or the nodes that they share with the adjacent elements. And one, the wayit is done, it is automatically done in the program.

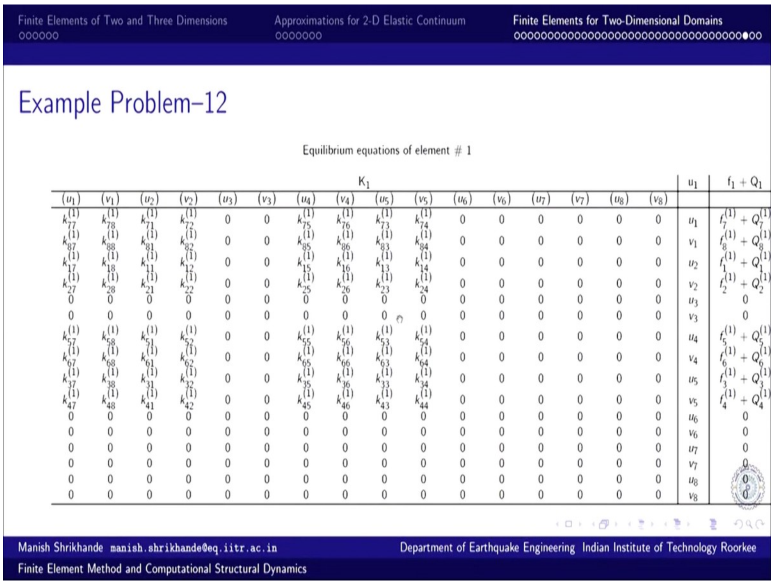
Again using very powerful data structuring techniques. If you are interested, you may look at the text book very excellent and easy to read text book on finite element programming by Hinton and Owen which beautifully explains this process of assembly, how it is done automaticallyin the in a computer program and how to prepare a data structure suitable data structure to facilitate that.

So, essentially what it involves is mapping of element level degrees of freedom we know there are two degrees two displacements at each node. So, u i, v i, u j, v j, u k, v k, u l, v l, we know that this i j, k l, are numbered in counter clockwise direction. In the global mesh, node i may refer to some particular node according to the mesh design. So, in this particular example, i^{th} node corresponds to for i^{th} node of element 1 corresponds to node 2, j^{th} node corresponds to node 5, k^{th} node corresponds to node 4, and l^{th} node corresponds to node 1.

So, appropriately what happens here is whatever is the coefficient or terms corresponding to u i, they would go to corresponding terms of node 2 or x displacement of node 2. So, in the global system of equations that corresponds to element 3 because node 1 will have u and v, and node 2 will have u and v. So, ui would go to third variable in the global system of variables and so on. And in this case ul, so that is the 4th node, 7th variable, 7 and 8th variable, in this element level variable, right. So, 1, 2, 3, 4, 5, 6, 7, 8. So, that is the 7 and 8th variable.

In the element level, they would correspond to the first and second in the global variable because the 4th node in this particular local node numbering is actually the first node in the global node numbering. And that is how the actual assembly goes.

(Refer Slide Time: 36:08)



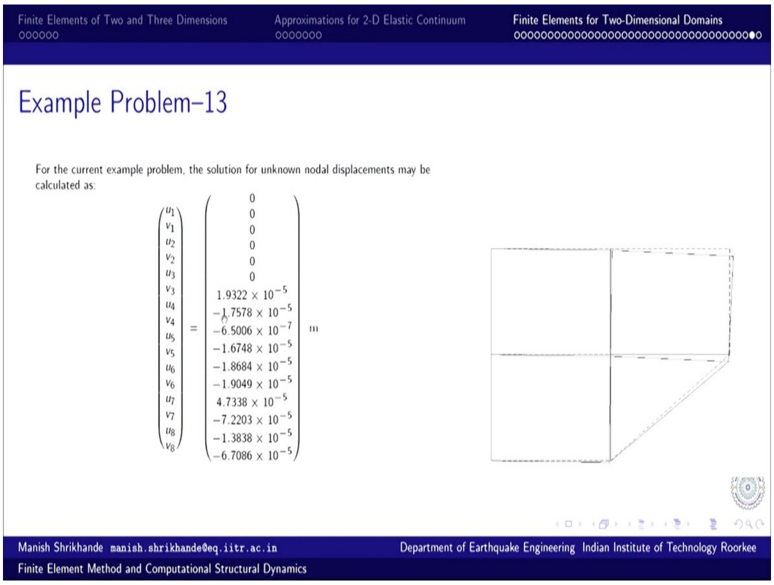
So, in this global system you can see that there are 16 columns and the all the elements are actually placed accordingly. So, this is the stiffness matrix. So, element stiffness, matrix stiff coefficient of stiffness matrix of element 1, they occupy respective positions corresponding to the mapping of local node variables with the global node global variables, and similarly the load vector, the tractions and the body force. So, they also figure, take the appropriate locations.

So, on this, so this is for element number 1. Contributions from element number 2, would be superposed on this, added to that. So, some elements get added up, some elements get subtracted, according to whatever their signs are, respective signs are. So, it is always algebraic addition.

And then, once we are done with element number 2, we are we will go to element number 3, and add super impose element number 3 equations on this and then finally, element number 4 equations on this. And once we have those all the 4 element equations in place, then we impose the boundary conditions that are available for u1, v1, u2, v2, u3, v3, that these displacement degrees of freedom should vanish.

And once we impose that boundary condition then the rest of the system of equations can be solved for unknowns, unknown variables.

(Refer Slide Time: 37:51)

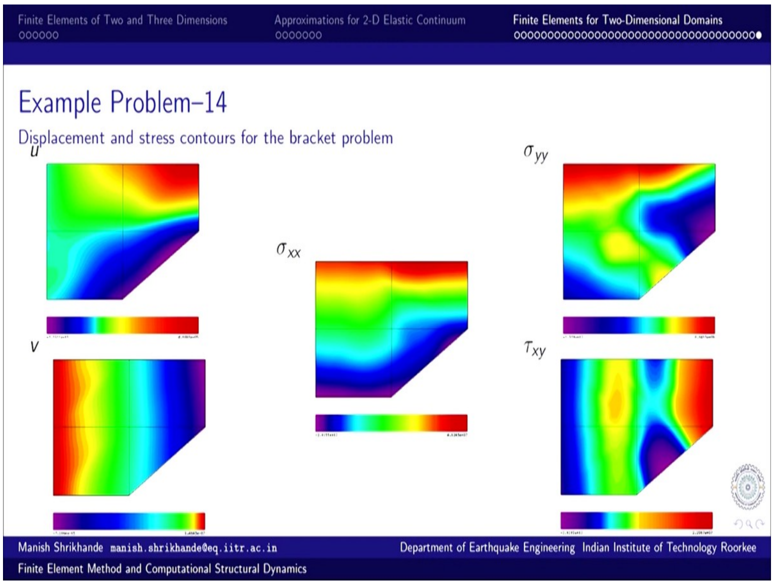


And once we do that these are the solutions. We will come to the solution of simultaneous equations how that is done. You might already be familiar with the simultaneous equation. For example, the elimination of variable and so on. And also, Gaussian elimination and so on. But we will discuss that in some lecture down the line.

So, this is what the solution of primary variables looks like. And if I plot, I mean once I do this then I can go back to interpolation model then I have the complete variation of displacement all through the domain with respect to x and y. The displacement is of course, continuous across the element boundaries. So, I can have the deflected shape of the bracket which is what this looks like.

This is often referred to as wire frame diagram of the deflected shape. And once I have, once I extract these and go back to the element level equations, I can derive the secondary variables the stresses and so.

(Refer Slide Time: 39:02)



This is the displacement u and v, all through throughout the as a function of x and y. This is σ_{xx} . You can see very beautifully the constant stress, I mean the tension at the top, and compression at the bottom, and very beautifully my color-coded contours can be seen. And this is σ_{yy} , and this is τ_{xy} shear stress. And these are just the representative of how the typical results of a finite element stress would look like, a finite element result look like.

Of course, these are what we call as smooth contours, in raw contours as we compute do not look so continuous and beautiful. They would be there would be jagged lines. Because the continuity of stresses is not guaranteed, continuity of displacement is of course, guaranteed along across the element boundaries, but not of these stresses because the derivatives are going to be discontinuous.

And should, therefore, the stresses should also be discontinuous over the across the boundary. And that level of discontinuity is what provides us an estimate of the error in the solution or adequacy of or whether we need to refine the mesh or we need to look at further refinement in the finite element results. But that is a another topic, error analysis. We will see if time permits.

So, we stop here. In the next lecture, we begin with a new topic called Mapped Elements or Distorted Elements.

Thank you.