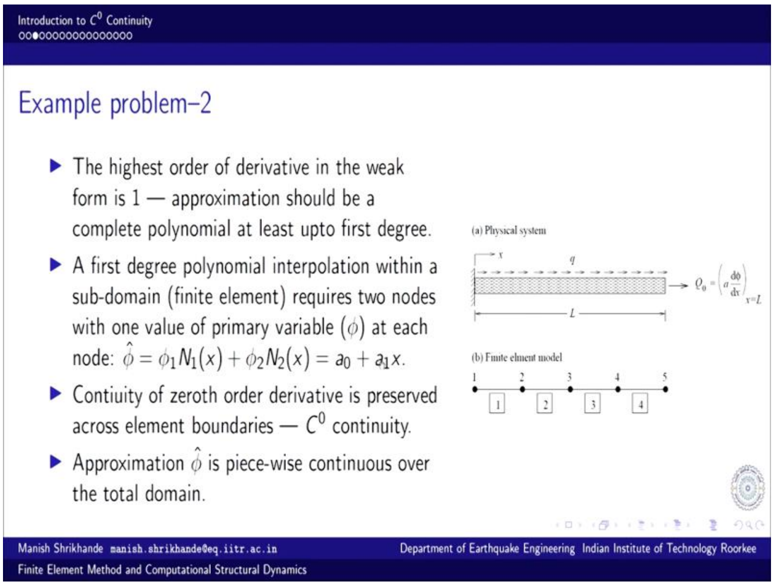
**Finite Element Method and Computational Structural Dynamics**
**Prof. Manish Shrikhande**
**Department of Earthquake Engineering**
**Indian Institute of Technology, Roorkee**

**Lecture - 15**
**Finite Elements of $C^O$ Continuity in 1-D-III**

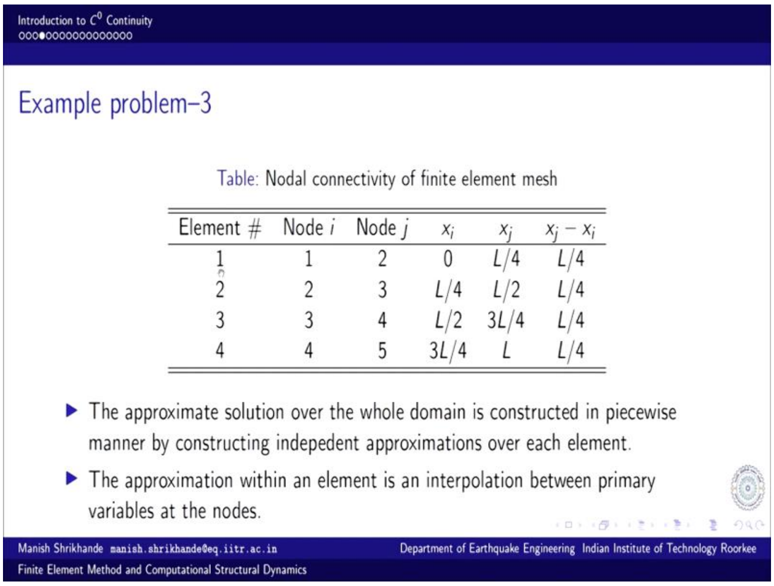
Hello friends. So, we were discussing the problem of axial vibration, axial deformation of this rod.

(Refer Slide Time: 00:39)

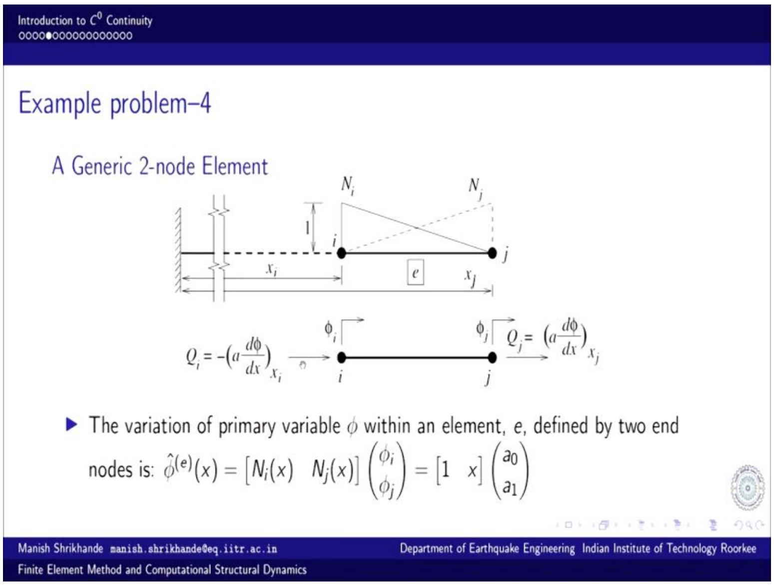



And we are discretized this domain 0 to L into a set of 4 finite elements, 2 noded element. Each element comprising of 2 nodes and equi spaced. We considered to keep simple matter simple because the geometry is uniform distribution is uniform. So, there is no such need for any different discretization. So, we divided the whole domain into equal parts, 4 equal sub domains of L/4 lengths each were quarter of length.

(Refer Slide Time: 01:17)




(Refer Slide Time: 01:24)

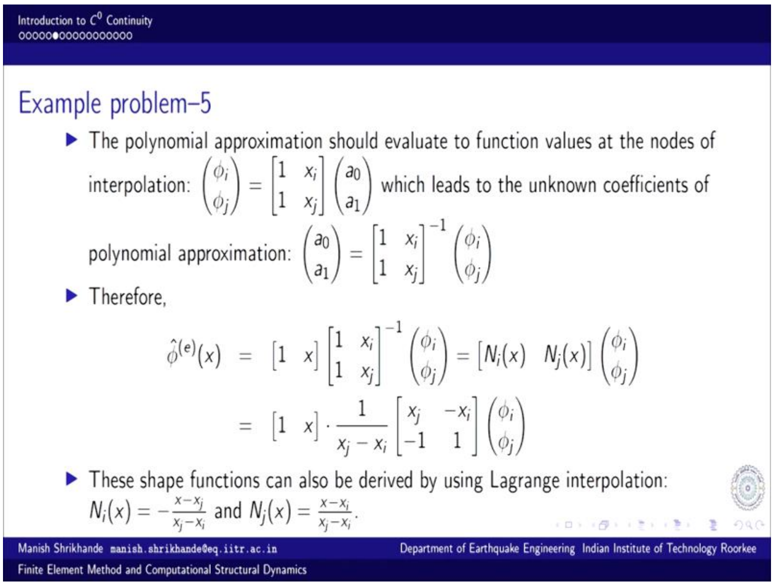



And we saw that after we develop the weak form of the domain residual. We found that it was the first order the highest order term was only the first degree and therefore, only $C^0$ continuity only the I mean the primary variable is the basic unknown of the problem the unknown function of the problem itself and the continuity of primary variable only is desired and therefore, the zeroth order derivative is required to be continuous, so this is called $C^0$ continuity.

And the approximation can be built in terms of interpolation of this primary variable. So, $\phi_1 N_1$, $\phi_2 N_2$ as a 2 node interpolation and; obviously, this is straight line two points two values can define, one first degree polynomial. So, this is what first degree polynomial expression it will come out to be, and this allows us to derive the interpolation function. What the form of interpolation functions these can be. Either from deriving these unknown coefficients $a_0$ $a_1$ based on the interpolation criteria that the function $\hat{\Phi}$ should yield $\phi_1$, when evaluated at $x_1$ and it should yield $\phi_2$ when evaluated at $x_2$. Alternatively, we can also develop these interpolation functions $C^0$ continuity by using Lagrange's interpolation formula. And this is what a generic 2 node element interpolation looks like.

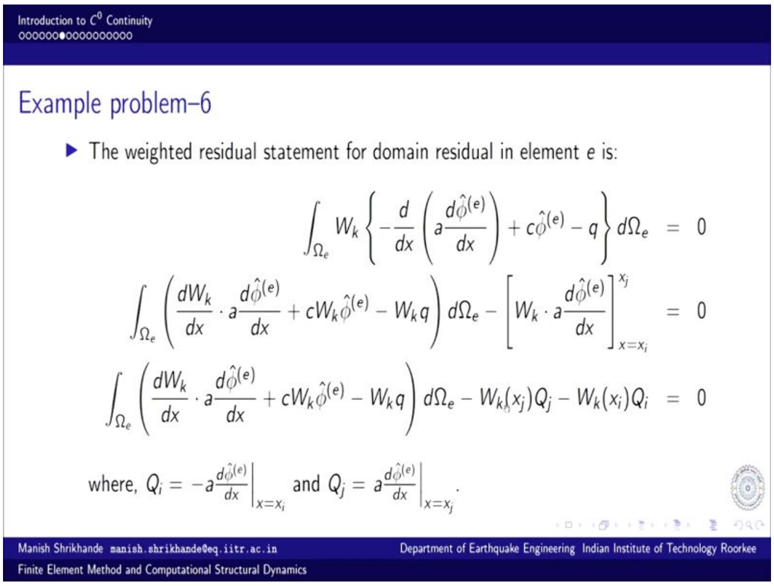
So, we consider $i^{th}$ node as the left side node and $j^{th}$ node as the right-hand side node of any 2 node element e. 2 node $C^0$ continuity element. And obviously, the interpolation function for corresponding to node i takes unit value at node i and it vanishes linearly to 0 at node j. Similarly, interpolation function for node j increases from 0 at node i to unity at node j linearly. And the $\phi_i$ and $\phi_j$ they are the primary variables at these two nodes and this is the basic expression of interpolation. So, $\phi_i N_i + \phi_j N_j$ is equal to the variation of the primary variable within the element e.

(Refer Slide Time: 04:07)




And, this is how by imposing the interpolation condition we can determine $a_0$ $a_1$ coefficients and the interpolation functions can be derived and they are can be derived by imposing those conditions and they are identical to what we can also write straight away by using Lagrange interpolation formula.

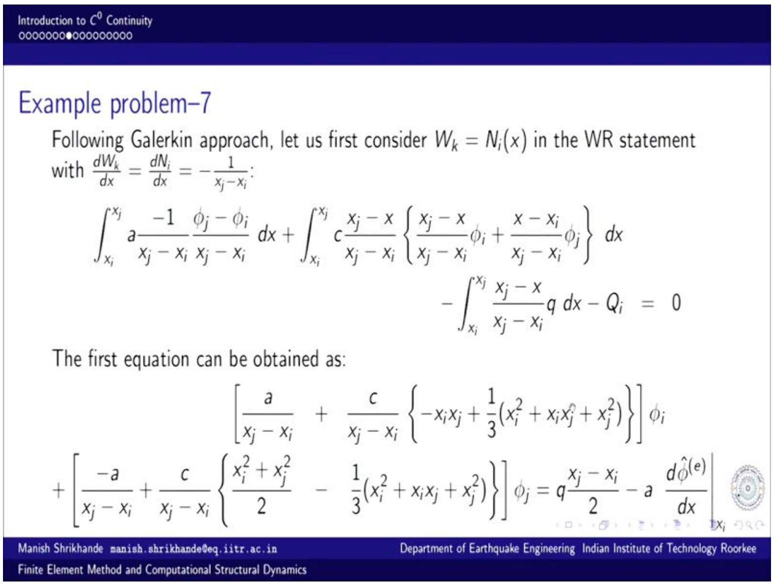
(Refer Slide Time: 04:52)




So, as I said developing the weak form within the sub domain, domain residual and we can find that the highest order of term here is first order derivative and the boundary term has this weighting function term without any derivative. So, replacing this weighting function by the unknown of the problem, so that becomes the primary variable of the problem and this is of course, the secondary variable of the problem and we define secondary variables as $Q_i$ and $Q_j$ just for notation purposes.

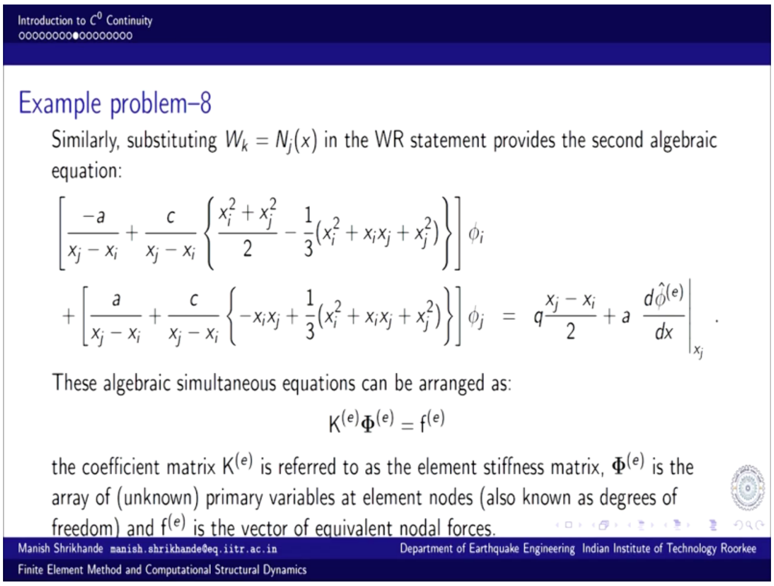
And, we take care of these signs I mean this is important this is how i mean if $Q_i$ is positive indicated in this direction. So, $Q_i$ is actually minus of a times d ϕ / dx and Q j is equal to a times d ϕ / dx and that is what is indicated here. That comes out from by substituting the evaluating these conditions these weighting functions one after the other substituting N i and N j for these weighting function. So, one of the terms will remain. So, $W_k$ as $N_i$ will vanish at $x_j$ and $W_k$ at $N_j$ will vanish at $x_i$. So, only one of the two boundary terms remains at one time at a time.

(Refer Slide Time: 06:29)




So, this is what by following Galerkin approach using weighting function as the interpolation functions. So, first using $N_i$ we obtain the first equation after evaluating the integral. We obtain algebraic equation in terms of $\phi_i$ and $\phi_j$ and this is the boundary term corresponding to $N_i$ ,W is equal to $N_i$ weighting function is equal to Ni.

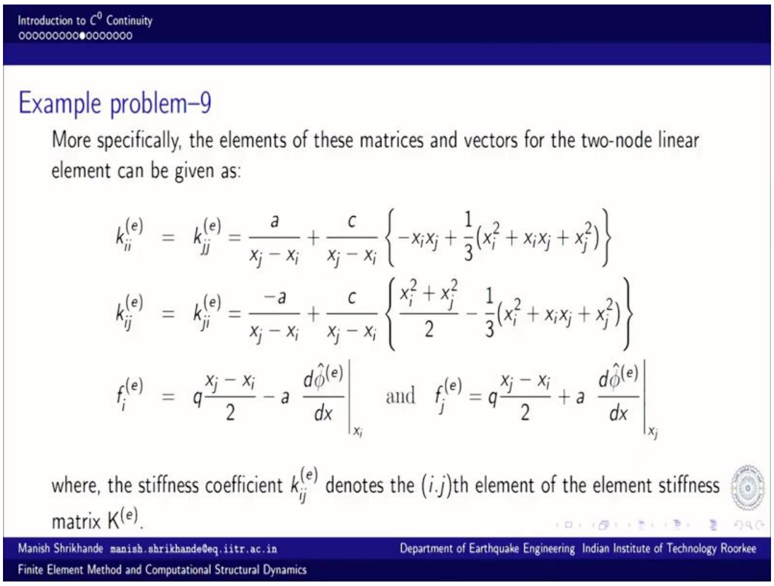
(Refer Slide Time: 06:55)




And this is by substituting second equation for second equation substituting weighting function as N j and again evaluating the same weighted residual statement integrals, we end up with this algebraic equation in terms of unknowns $\phi_i$ and $\phi_j$, and again one weighting function one boundary term remains here corresponding to node j. So, these two equations can be arranged. I mean both these equations this and the previous one, they are equations in terms of ϕi and ϕj as unknowns. So, two simultaneous equations and they can be arranged in the matrix form.

So, we have this 2 by 2 system, K is referred to as the element stiffness matrix in structural mechanics domain, other I mean it is general the name has got stuck. Even if we just refer to it as stiffness matrix, although it is a coefficient matrix for simultaneous equations in other domains of application. ϕ is the vector of unknowns or degree of freedom also known as degrees of freedom and f is the vector of equivalent nodal forces. So, if you look at it, it comprises of this term q is a distributed traction, distributed along the length, so that is the intensity of distribution. Now, this is essentially equivalent translates this distributed force, distributed over the length domain into equivalent force which is applied at the nodes. So that the work done by these nodal equivalent forces will be same as the work done by these distributed forces in moving through the displacements. So, that is vector of equivalent work equivalent nodal forces.

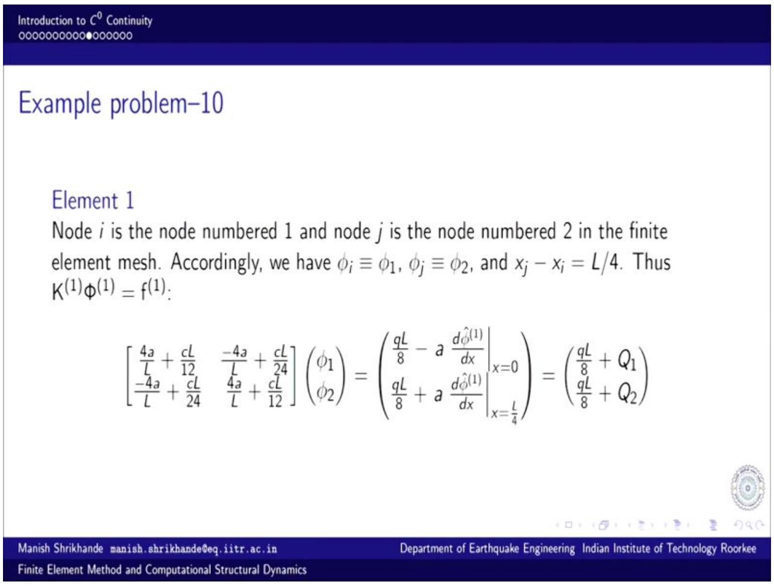
(Refer Slide Time: 09:01)




So, generally if I look at the terms. So, $k_{ii}$ diagonal terms, they are identical if I look at it. If we check these equations off diagonal terms are also identical. So, it is a symmetric matrix and the loading term corresponding to $i^{th}$ node. We have this simple equivalent, the total intensity q multiplied by length of the element and then equally distributed at two parts. So, this is what it becomes for the distribute from coming from the traction distributed traction and this is from the boundary term. So, secondary variables evaluated at respective boundary terms. So, $k_{ij}^{e}$ represents the i $j^{th}$ element, $i^{th}$ row and $j^{th}$ column of the element stiffness matrix $k^{e}$.

Now, this is how it goes and we can move from one element to another element that we derived this for a generic element e. Now we can look at it from specific element from element number 1, let us say element number 1 node i is 1 and node j is 2 coordinate of node i is 0 coordinate of node j is L / 4 and based on that we can derive the equations.

(Refer Slide Time: 10:32)

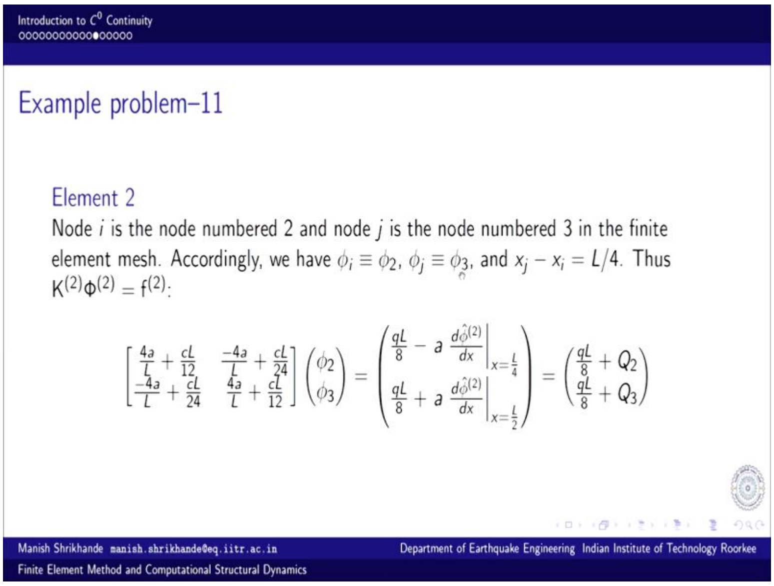



And this is what the equations K ϕ is equal to f for the element number 1. So, K for element 1, ϕ for element 1 is equal to f for element 1. So, these are element level equilibrium equations that have been derived from the evaluation of weighted residual statement weak form.

So, node i in element 1 is the node number 1. So that is the left hand side node. Node j is the node number 2 in the finite element mesh that we used for discretization of this domain from 0 to L.

So, accordingly $\phi_i$ is same as is identical with $\phi_1$, that is the variable primary variable corresponding to node 1 in the mesh and $\phi_j$ for element number 1 is identical with the primary variable at node 2 the displacement at node 2. So that is phi 2 and the length of the element is the difference between the 2 coordinate so $x_j - x_i$. So that is equal to L /4. So, this gives us the basic equation K ϕ K 1 for the element 1 multiplied by the degrees of freedom for element 1 is equal to the forces at in element number 1. Similarly, we can this is what we evaluate and we keep it aside. Now we move to the next element. So, increment e counter from 1 we go to 2 and we get similar set of equations.

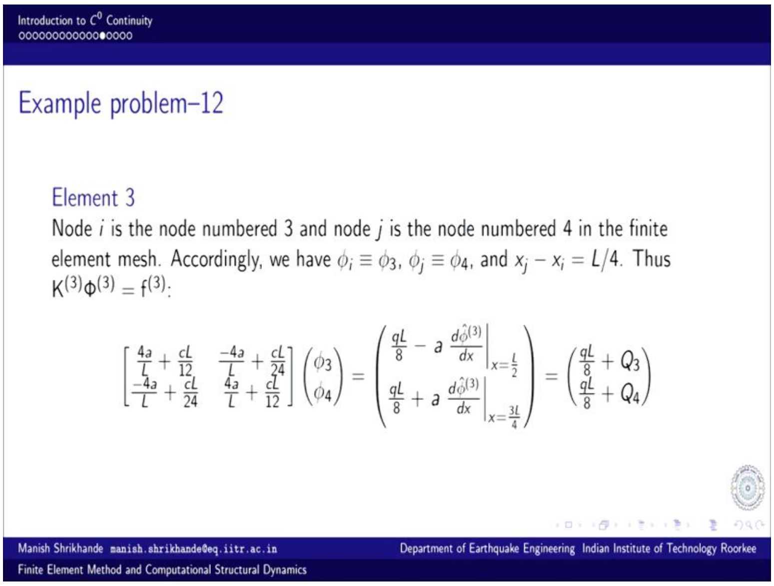
(Refer Slide Time: 12:11)




Now, all that has changed is the interpretation of node i. So node i for element number 2, the left hand side node in element number element number 2 is node 2 in the mesh node 2 in the mesh. So, this is what. So node 2 in the mesh is this. So, this is the finite element mesh. So, for element number 1 node i is node number 1, node j is node number 2, for element number 2 node i is node number 2 and node j is node number 3 so on.

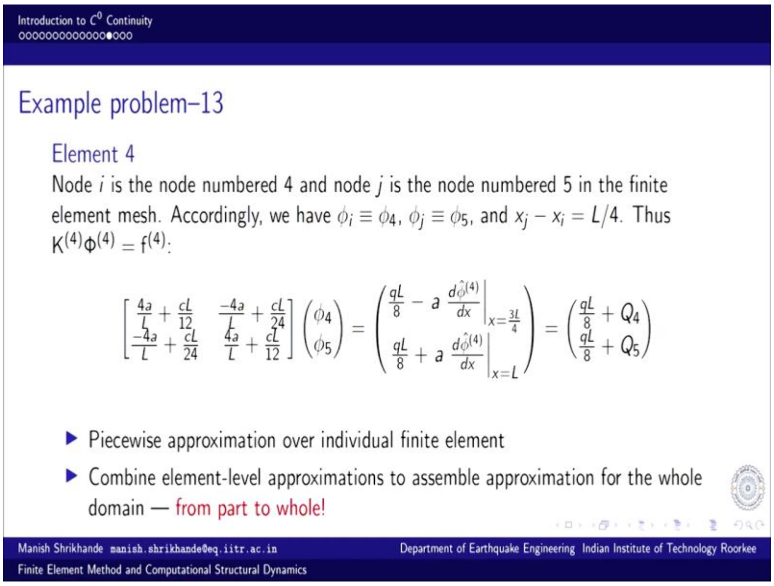
So, $\phi_i$ is same as the primary variable at node 2 and $\phi_j$ is same as primary variable at node 3, and length of the element is same $x_j - x_i$ is same L / 4. We used uniform mesh and the equilibrium equation for element number 2 can be developed arranged in similar fashion by evaluating the integral and we end up with this 2 / 2 system of equations 2 unknowns. So, in this case the unknowns are $\phi_2$ and $\phi_3$.

(Refer Slide Time: 13:38)




Then we again move the increment the element counter move to element number 3. Now for element number 3 the node i, the left hand side node is node 3 and node j the right hand side node is node number 4 and so on. So, the primary variables are also $\phi_3$ and $\phi_4$, length of the element is L/ 4, again we end up with similar set of equilibrium equations.

(Refer Slide Time: 14:00)



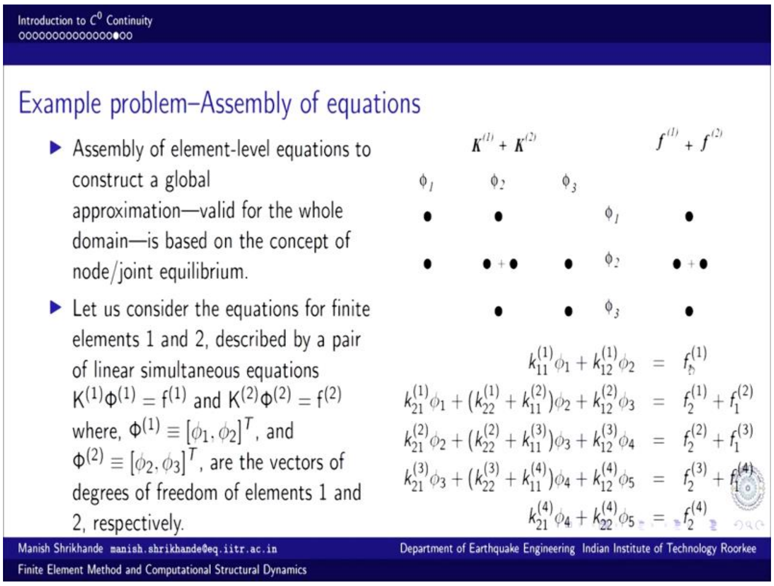

And then finally, element number 4 similarly, defined by nodes 4 and 5 and the primary variables are defined by correspond to the variable at node 4 and at node 5. So, $\phi_4$ and $\phi_5$ and that is what the element level equations are.

So, these are the qL / 8 comes from so L / 4 / 2. So, qL /4 half of that. So, that's qL / 8 that comes from the distributed tractions and this is from the secondary variable the boundary term associated from associated with the weak form of the domain residual.

Now, all these equations for element number 1 to element number 4, if you recall have been derived based on piecewise approximation over individual finite element. So, we considered individual finite element one at a time. So, element number 1 we developed a linear interpolation between $\phi_1$ and $\phi_2$ and develop the approximation linear approximation for $\phi$ over element 1 and then evaluated the weighted residual statement weak form integrals evaluated over the sub domain and then the whatever algebraic equation was presented itself. We assemble that as the element level equation and so on for all the four elements and eventually all four elements give us these element level equilibrium equations which have been derived based on the piecewise approximation over individual finite element or sub domain.

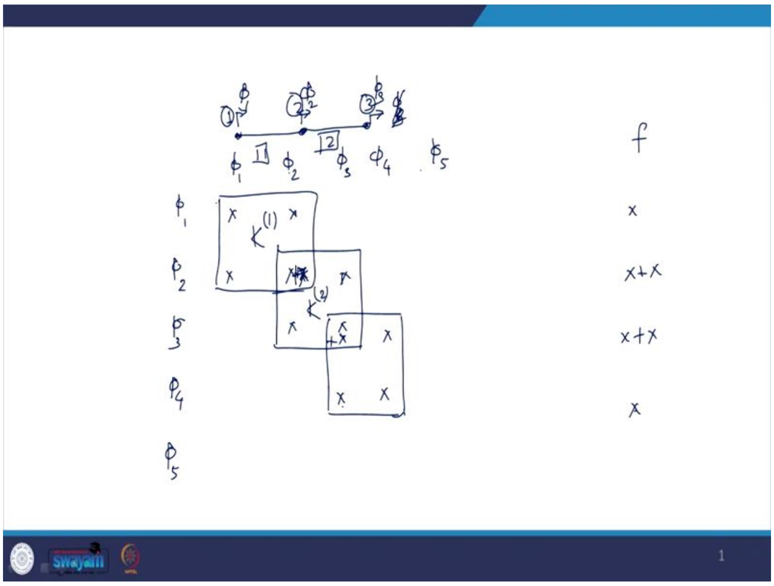
Now, we need to combine them somehow so that we can construct the approximation for the whole domain. Not just from 0 to L/ 4 or from L /4 to L /2 or from L /2 to 3L/ 4 or from 3L /4 to L. So, individual elements they were valid only for that sub domain length of individual element. So, that does not help us much, unless we can find a suitable way to combine the individual contributions to make up for the whole which is no approximation at all. So, we combine element level approximations by assembly, by the process of assembly. And by assembling these approximations we construct the approximation for the whole domain and that is what we call I mean it is a familiar term for civil engineers at least from part to whole. From approximation for small parts and then we patch it up to make the whole.

(Refer Slide Time: 17:21)




How do we do this? That is called assembly of equations. So, we consider assembly of element level equations to construct a global approximation which is valid for the whole domain and this is based on the concept of node or joint equilibrium. Equilibrium comes naturally to civil engineers or mechanics-based formulation. Of course, similar constructs are applicable in other domains. So, we consider as the demonstration of the process, we consider the assembly of elements equations from element number 1 and element number 2. Now, element number 1 and element number 2 if you look at it.

(Refer Slide Time: 18:08)



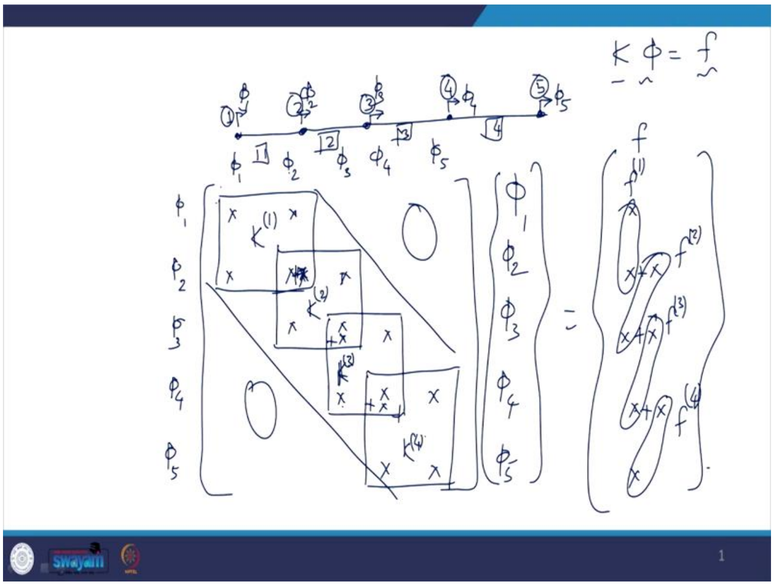
So, this is element 1 and it has node 1 defined by node 1 and node 2 and similarly element 2 is defined by node 2 and node 3. So, $\phi$1 is defined by element number 1 is defined by the interpolation between $\phi$1 and $\phi$2 and interpolation between phi 2 and phi 3, that defines the variation within element number 2. So, if I collect these, let us say, if I collect let us say I mark these rows corresponding to different variables $\phi$1, $\phi$ 2, $\phi$3, $\phi$4, $\phi$5.

So, there are total 5 number of nodes in the mesh and I define similar terms $\phi$1 column arrangements as well. So, for element number 1 this is what element number 1 is and the corresponding term would be in the force vector would be $\phi$1 and $\phi$2. So, this comes from element number 1. Element number 2, because element number 1 is defined by $\phi$ 1 and $\phi$ 2 and that those elements will go this is what K 1 element number 1 stiffness matrix of element number 1 is, so 2 by 2 and these are the 2 vectors 2 elements of the first element load vectors.

For element number 2 which is defined by variables $\phi$2 and $\phi$3, so element number 2 this I would say this is $K_1$ and then I have $K_2$ right. So, this is $\phi_2$ and $\phi_3$, they come from element number 2. Now, $\phi$ node 2 is common to both of them, so the contributions get added up at correspond node at the degree of freedom common to both the elements.

Similarly, for element number 3 which is defined by nodes 3 and 4, so the variables are common variable is 3 and $\phi_3$, so there would be, so this term comes from this one comes from element number 2 and these two terms come from element number 3.

(Refer Slide Time: 22:17)



And similarly, for the element number 4 which is defined by nodes 4 and 5 I would have let me just complete the mesh. So, this is element number 3 this is element number 4 and this is node 5; this is node 4 ϕ 4ϕ 5 so node 5 element number 4 defined by these 4 terms.

So, ϕ 4 ϕ 5 and again corresponding to node 5 and there would be 1 element here and 1 element here. So, this one comes from f 1, this one comes from f 2, this one comes from f 3 and this one comes from f 4. The element superscript indicate the element numbers.

And this one is K 4 and this one is K 3 the stiffness matrix for. So, stiffness matrix for element 1 load vector for element 1 stiffness matrix for element 2, node 2 is common the degree of freedom at node 2 ϕ2 is a common degree of freedom, so the terms get added up here. And similarly, for the force vector the 2nd degree of freedom is common. So, the terms corresponding to 2nd degree of freedom they get added up and similarly for element number 3 is K 3 phi 3 is common f 3 is common, so these two terms get added up and from 4th element degree of freedom 4 phi 4 is a common degree of freedom. So, this term gets added up with the element from with the contribution from the 3rd element and so on.

So, together this becomes global stiffness matrix K and this is multiplied by the unknown vectors, vector of unknowns. So, together this becomes K ϕ is equal to f. This is the global system of equations, this is what we call as global system of equations. And so that is the process for just for just to fit the graphic into this space I used only two elements. And this is the pictorial representation of what we did.

So this 4 elements come from the first element stiffness matrix, so K 1 and these 4 elements come from the stiffness matrix of second element, so that is K 2 and ϕ 2 is a common degree of freedom. So, these two contributions get added up. Similarly, for the force vector we have this contribution at common node getting added up. And if I look at the assembly of all equations all 4 elements so these are the terms. Now, these have very interesting structure these equations, if you look at it the influence I mean it is a banded structure, just look at this system it is a banded structure this is entirely 0.

The if degrees of freedom they have very localized effect they can influence only adjacent elements. Rest of it I mean approximation over element 1 has nothing in common with approximation over element 4 or for that matter element number 3 right. It has common with the element number 2, the 1 degree of freedom is common and of course, so because of that coupling the equations are coupled. But the approximation the development of the approximation as such is limited to the immediate neighborhood. The influence is limited to the zone of influence is limited to the immediate neighborhood of the element. The degrees of freedom which they share with the adjacent elements.

So, that way the structure of the equation becomes very banded in nature. So, it is a sparse matrix most of the elements are 0 and the entire thing can be compacted in very small number small band.

So, it is a banded matrix structure and moreover Galerkin approach weak form we already discussed that it lends itself to a symmetric structure for the coefficient matrix and therefore, this equation this coefficient matrix is symmetric, so we can actually work with only the upper half and that leads to a very efficient compact storage schemes for numerical solution, but and that is how the commercial finite element solvers are usually arranged.

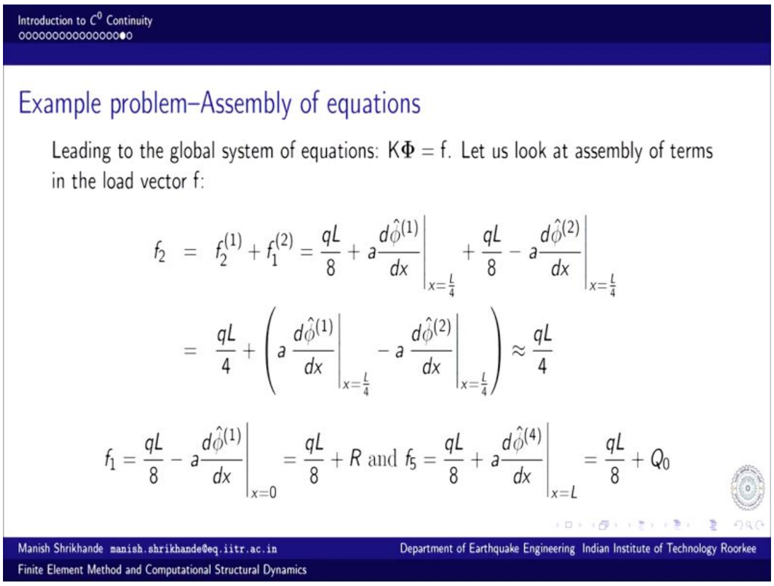
So, you can see that the influence is very limited. So, ϕ 1 figures only in the first 2 equations, ϕ2 figures only in the first 3 equations, ϕ3 figures only in the next 3 equations and so on ϕ1 ϕ5 only figures in the last 2 equations.

So, this is how and here I used subscript 1 and 2 as in place of i and j. So, this is another common notation that we adopt, i and j are good for let us say for developing the formulation, but use of i and j is not really convenient when it comes to coding right.

We do need some hard numbers for coding. So, that is what we use as local numbering system and global numbering system. So, i and j instead of calling i and j referring to nodes as $i^{th}$ node and $j^{th}$ node as left hand side node and right hand side node, we just use term as node number 1, local coordinate, local node numbering, node 1 is same as $i^{th}$ node and local node j is same as $j^{th}$ node and so on. And this has how it works out.

And then we have a mapping defined between local node numbers for element given element with the global ordering of the variable, so node number so that will map the degrees of freedom correctly and then the assemblies can be carried out conveniently.

(Refer Slide Time: 29:26)




Now, this assembly is an important thing it needs to examine the root right hand side vector can be examined little closely these 2 terms. So, let us look at this term f $_2$. So, the contributions coming from two terms.

So, this is what it is qL / 8 plus this boundary term qL / 8 plus this boundary term. Now in one side the node 2 is receiving contributions from element number 1 and element number 2 and for element number 1 node 2 is the right hand side node, for element number 2 node 2 is the left hand side node and the sign of the boundary terms they are of opposite signs.

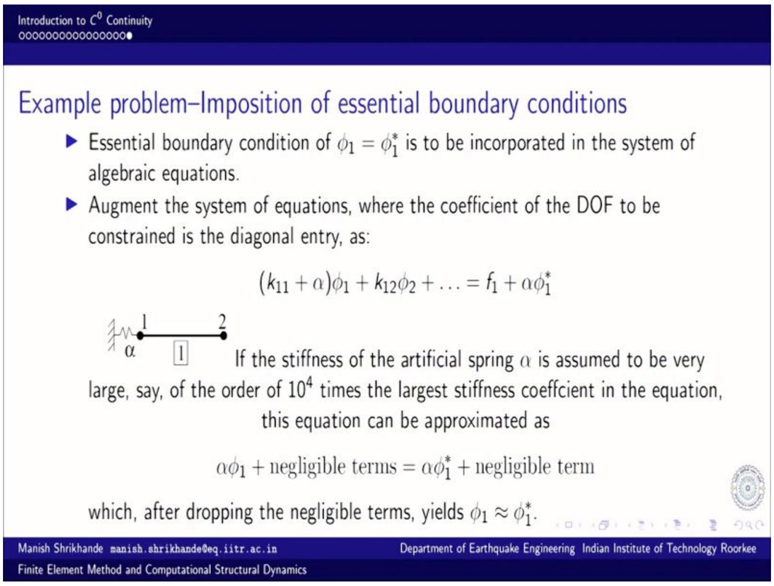
In whether with regard to I mean it is of one sign if it is a right hand side node and if it is a left hand side node it has a different sign, but the node is same. So, when we assemble the contribution so; obviously, this qL / 8, qL / 8 they add up and we get qL/ 4 and these are added up. So, these are of opposite signs, so obviously, they should cancel each other. So, that is how we get the we consider these opposite sign and they cancel each other out and the resulting vector is the node equivalent of the distributed traction.

Now, if you look at closely this is at least in theory it should cancel out, but if you look at it closely this is the derivative of the approximation in element 1 evaluated at node 2, and this is the derivative of approximation in element 2 evaluated at node 2.

Now these approximations are independent of each other. So, they will in general not cancel exactly in this case. In theory they should, that is what the equilibrium implies, but in practice in actual approximation finite element approximation they do not cancel exactly, and in fact, this difference is often taken as a measure of mesh refinement or how good is the approximation, what is the quality of approximation, so what is the error in satisfying the equilibrium.

And then similarly, first element we have this qL / 8 and this is the reaction force that will be evaluated during the force, because at the first node it is a fixed support, so it will develop reaction in response to deformation and that is the unknown reaction R and similarly at the free end there is a axial load acting and that is what is substituted for this and this is the natural boundary condition that is applied. So, this stress resultant is to be equal to is to be imposed as the applied boundary condition. And with this we have the complete assembly of the system.

(Refer Slide Time: 32:40)




And, we can then proceed with the imposition of boundary condition. Now, natural boundary condition we already applied. The essential boundary condition is that is we need to apply phi at x is equal to 0, it has to take some prescribed value. So that value has to be specified. So, let us say that value is phi 1 star. That has to be incorporated in the system of algebraic equations.

So, that is done I mean there are several ways of doing that. The easiest thing to do is by using what we call as stiff spring approach. I impose I consider this I consider there is a very stiff spring of stiffness alpha between the support and the node 1 and then I impose a force of alpha times whatever is the prescribed value and then augment the force vector accordingly. So, that is what and then similarly, I add α to the diagonal term corresponding to ϕ1.

So, the corresponding equation row of equation first row of equation becomes like this. So, $k_{11}$ + α multiplied by ϕ 1 + other terms in the row and then the right hand side becomes $f_1$ + α times ϕ 1 star. This is the value that needs to be specified.

Now, α is considered to be a very large number. Is usually taken as a very large number, that is of the order of 10 raised to the 5, 10 raised to the power 4, 10 raised to the power 5 times the largest stiffness coefficient k elements in these equations.

So, that once we have this then it actually this row, actually looks like $\alpha$ times $\phi_1$ + some negligible terms in comparison to this right hand side becomes $\alpha$ times $\phi_1^*$ plus some negligible terms. And, if we solve for this it will yield $\phi_1$ is approximately equal to $\phi_1^*$ and that is what the imposition of boundary condition is. So, that satisfies the boundary condition and that completes the solution of this problem.

So, we stop here. It has and in next lecture we will develop the other kind of finite element formulation which will involve continuity of the primary variable as well as the first derivative of the primary variable that is called C 1 continuity. So, we will discuss that in our next lecture.

Thank you.