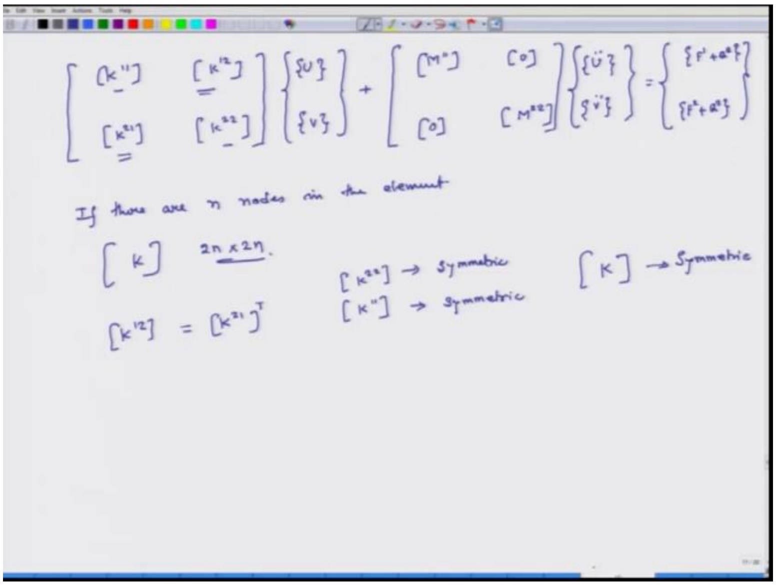
Basics of Finite Element Analysis – Part II
Prof. Nachiketa Tiwari
Department of Mechanical Engineering
Indian Institute of Technology, Kanpur

Lecture – 48
Plane Elasticity Problems: Closure

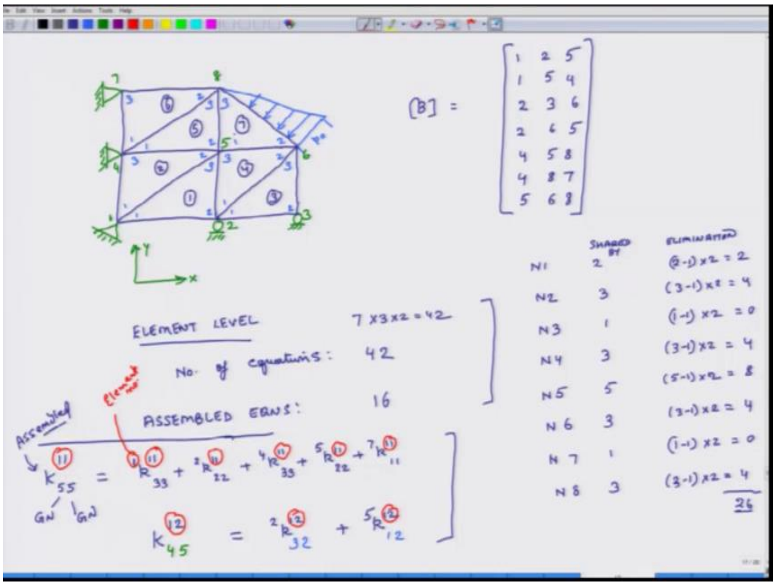
Hello, welcome to Basics of Finite Element Analysis Part II, today is the last day of this course and what we plan to do in this lecture is close the discussion on plane elasticity problem and then after that, if we have time we will make some general remarks and some important remarks in context of how to develop a (Refer Time: 00:36). So, that is what to we plan to do today.

(Refer Slide Time: 00:43)



What we will do today is; so yesterday we have developed the relation at the element level for the entire plane elasticity problem. What we plane do today is, do a simple example and see how do we go around doing the assembly.

(Refer Slide Time: 01:01)



So, consider a problem which looks something like this, so this is the domain and I have broken it up into several elements. So, total number of elements I have here this is 1, 2, 3, 4, 5 and I also have one element here 7, 6, so total number of seven elements are there and then my global degrees of freedom r 1, 2, 3, 4, 5, 6, 7 and 8 and then, what are my boundary conditions. So, I will actually just rub this, so this is global degree of freedom 2, so the boundary condition is that it is rigidly fixed here, it is also rigidly fixed here. So, this is node 4 it is also rigidly fixed here; here it is fixed in the y direction, so this is y; x and this is my y direction and at this thing also it is fixed in the y direction at node 3 and then on this surface, I may having some traction in the normal direction.

So, let us say this number is p 0 and my local degrees of freedom r 1, 2, 3; 1, 2, 3; 1 so you will note that in every case I am putting my local degrees of freedom in counter clockwise direction, so we have to be careful about that 1, 2, 3 as long as it is counter clockwise I am fine, I can start one from anywhere, but I have to do this be counter clock wise 1, 2, 3; 1, 2, 3; let us write the connectivity matrix for this, so it will be 1, 2, 3. So, the connectivity matrix will have how many rows? 7 because there are 7 elements, it will have 7 elements, 7 rows and how many columns 3 because the maximum number of nodes in any element is 3, so there will be 3 columns.

So, the first row corresponds to element number 1 and what are the global nodes 1, 2 and 5, second row corresponds to second element; the global nodes are 1, 5 and 4 and again we have to go in the right order 1, 5 and 4. Third element corresponds to third row, so the numbers are 2, 3, 6; fourth element 2, 6, 5; fifth element 4, 5, 8; sixth element is 4, 8, 7 and seventh element 5, 6, 8, so this matrix will help us do the assembly. Second thing total number of nodes, at element level how many nodes we have 3 and total number so element level.

Each element has 3 nodes and total number of elements is 7, so number of equations each elements will give us how many equations for plane elasticity problem; it will give a 6 equations, each elements has 3 nodes and all are triangles so n each has 3 nodes. So, 6 equations for each element and they are 7 elements, so total of 42 right, 7; 6 to 42, 7 times, 3 times 2 is equal to 42. Global level once I have assembled; assembled number of equations; once I have done the assembly, how many nodes I am left with, how many nodes do I have? Once I have done the assembly, we have 8 and each node has 2 degrees of freedom. So, assembled number of equations is 16, so where do all these other equations go? They get observed by continuity relations right for instance node 1, so this is global nodes 1; global node 1 is shared by two elements. So, this eliminates, so this is shared; shared by, so elimination. How many equations get eliminated because of that 2 minus 1; times 2 is equal to 2.

Node 2 is shared by how many elements? 3 elements, so how many equations gets, 3 minus 1 times 2 is equal to 4. Node 3; how many elements it is sharing; are shared by it; it is shared by 1 element, so 1 minus 1 times 2 is equal to 0. Node 4; node 4 is what? Node 4 is here how many elements sharing it 3. So, how many equations get eliminated 3 minus 1 times 2 equals 4. Node 5, node 5 is being shared by 5 elements 1, 2, 3, 4, 5, so it is 5 minus 1 times 2, why am I multiplying it by 2? Because they are equations right, so that is why, 5 minus 1 into 2 is 8.

Node 6; node 6 is how many 3, is it node 6; equation element 3, 4 and 7 right, so it is 3 minus 1 times 2 equals 4. Node 7; node 7 how much, 1 element uses it, so 1 minus 1; times 2 is equal to 0 and node 8, 3, so 3 minus 1, times 2 equals 4 add them up together 2, 4, 6, 10, 18, 22, 26; 26, what is the difference? 42 minus 26 equals 16, so we get 16 assembled equation because all these things are getting eliminated. Which degrees of freedom are getting eliminated, that you can see either from the picture or from this connectivity matrix, so this is important to understand?

So the elimination process of variables is same, there is no difference. Now we will very quickly see; how do we assemble the elements. So, when we eliminate and hence we also have to add up, when we are doing assemble we also have to add up several equations, so we will do one or two examples and get comfortable with it. So, let us find out, so we know that the k matrix has four sub matrices k 1 1, k 1 2, k 2 1, k 2 2, so let us find out the global element k 5; 5, for k 1 1 sub matrix you know, this is at assembly level. At assemble level, how do we calculate this; it will be an addition of several element level stiffness's.

So, let us look at, so this is node 5 this is global node, this is also global node and when once we done assembly we only worry about global nodes. So node 5 is shared by how many elements, it is shared by 1, 2, 5, 7, 4, so how many components it will have, it will have 5 components. So, it will be equal to, so I am using lower case for element level it is stiffness matrix. So, it will get 1 component from element number 1, so let us call this, now this side, I am putting element number then it will get another component from element number 2, then it will get another component from element number 4.

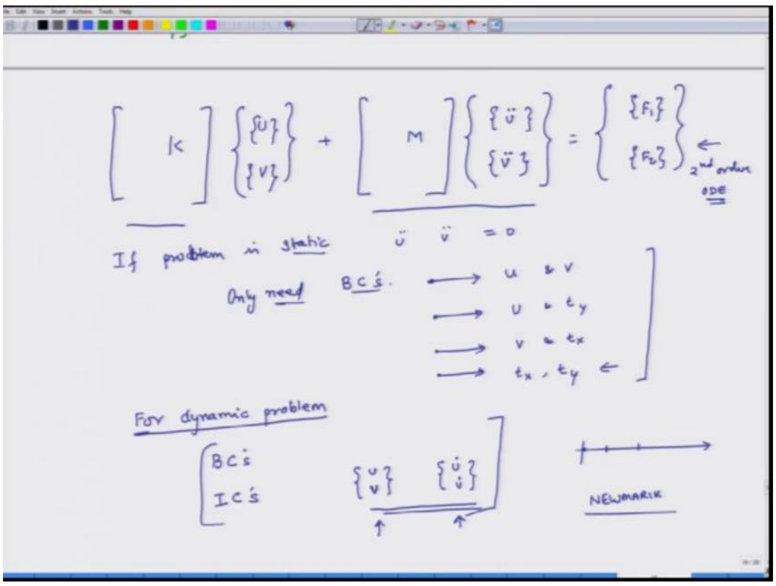
Then it will get another component from element number 5, then it will get another element component from 7 and what are the subscripts. So, this 1 1 corresponds to what; it corresponds to this sub matrix, this corresponds to element number and then it also should have a subscript, all these terms should have a subscript. So, for the first element, what does global node correspond to; global node 5, it corresponds to 3, so it is 3; 3 understood.

For the second element, global node 5 corresponds to which element, so it is 2 2. For the fourth element, global node 5 corresponds to node 3; 3 3. For the fifth element global node 5 corresponds to; so it is 2 2, for the seventh element global node 5 corresponds to 1, so it is 1 1 understood. So, you have to patiently do this and we will be able to assembly in this way, we will do one more example or I do not know, I think we have explained this sufficient number of times.

So, let us try to find out global matrix and let us say 1, 2 and the global nodes are 4, 5. So, this 4 and 5 it means that there is a line, that boundary is being shared by how many elements; it is being shared by 2 elements, so there will be two components right. The first component will come from which element; element number 2, the second component will come from element number 5, what about these superscript 1, 2; do they change or if there remain same, they remain same and what about the subscripts, 4 global node in element 2 means 3 and 5 global node means node 2 in element 2. Similarly for element 5, global node 4 means local node 1 and for element 5 global node 5 means local node 2. So, again the process of assembly does not change fundamentally, the only thing addition, the only additional thing is that we have this extra superscript and we have to track it otherwise everything else remains same.

So, we can assemble it for single degree of freedom, as long as we maintain the superscript 1 1 1 2 1 3, same process works for assembly. So, I think this completes the discussion of this two dimensional finite, this plane elasticity problem and this also brings us to the closure of this course. I think what I have captured in context of plane elasticity, you can use it, you can develop element level equations, you can develop; you can assemble them, you know how to apply boundary conditions and then we have to solve these, there is one small thing I would like to mention that once we have done the assembly I get a k matrix then I get a u vector.

(Refer Slide Time: 18:22)



So, this is the global assembly, the assembly of all the equations then I get a big m matrix which you have four sub matrices and then I have a; so when I am trying to solve this, if problem is static which means u double prime, v u double dot and v double dot are 0, then I do not have to worry about these terms. In that case, I only need what, I only need boundary conditions, I do not worry about what are the initial conditions in the system and there could be four sets of boundary conditions.

The first set of boundary condition is that on the entire boundary, I know u and I know v which means what do I know which type of variables? Primary variables I know and I am specifying essential boundary conditions. The other condition could be that I know u at some places and I know t x at some other places, sorry t y. Then I have a make situation the other case could be I know v at some other places then I know t x at some places. So, it could be either one set of, one of these sets.

At a particular nodes you cannot know both either know you know or you know traction and the fourth condition could be that you know t x and t y. In this case we are going to specify the secondary variables and in force natural boundary conditions. So, these could be four different situations, so we have to implement these conditions in the force vector accordingly and then once you implement this, then you have to solve the equation.

For dynamic problem, you not only have to know the b c s, but you also have to know the initial conditions and what are the initial conditions, that at time t is equal to 0, what is the value of u and v in the whole system, not at just one point and also what are u dot and v dot. So, once you know u and v and u dot and v dot boundary conditions you already apply right then what you do? You integrate it in time; see we have already integrated able in x and y that is how we got this stiffness matrix, but in time we start or solution from t is equal to 0 and then we go to t is equal to 1, t is equal to 2, so we march forwarding time and how do we march forward, we have developed a methodology for that, we had discussed it in the first finite element method using new mark integration method right.

So, you use the time integration methods and for each integration of time, for each step of time you find out the solution. So, they were couple of methods, but here because it is a second order differential equation in time, so this is if there was not time component then this would be a set of algebraic equations, but now this is a second order o d e, it is an ordinary differential equation in time. So, that is why I need initial condition because when I integrate to integrate u twice, I will get two constant and those two constant will be satisfied by these two; we have to determine them, we have to these two in a sets of initial conditions.

So, we use an integration method and we keep on marching in time and that is again numerical integration approach and that is how we solve this plane elasticity problem and if you want to learn more details about please visit our first finite element course, all the details are there. So, this brings us to the closure of this course and what we have accomplished in this course are several things.

In the first week we did a recap of whatever we did in f e a one part one of the course then we moved on to understanding, how errors are computed and estimated and we developed several measures of errors then we moved on to understand how numerical integration happens using Gaussian Quadrature, and Newton course method and that we developed all the details of it in context of only problems and then after we were had understood it and then we moved on to solving 1 d, two dimensional problem using a single variable and in that context we spent a lot of time trying to formulate the problem, develop its weak form, develop finite element equations, learnt how to do the assembly learnt how to apply the boundary conditions and also do the post processing.

And then in last two weeks, we moved to some new sets of information's and we covered how do we go around doing numerical integration in two dimensional systems and in two dimensional systems and in two dimensional systems, you have Quadrature points, in two dimensions if you go to three dimensional system the same method apprise and you basically project those points in the z direction also and you again do the same numerical integration scheme. So, the movement from or the transaction from two d to 3 d pretty straight forward, if you know about 2 d, you can do 3 d relatively easily.

And then finally we ended up solving or formulating the plane elasticity problem, which involves more than one variable and it is in two dimensions. So, I hope let this course has been fruitful and rewarding to you and it was a pleasure interacting with you all, if you any questions please send us email and also prepare for your final examinations well.

Thank you very much and best wishes bye.