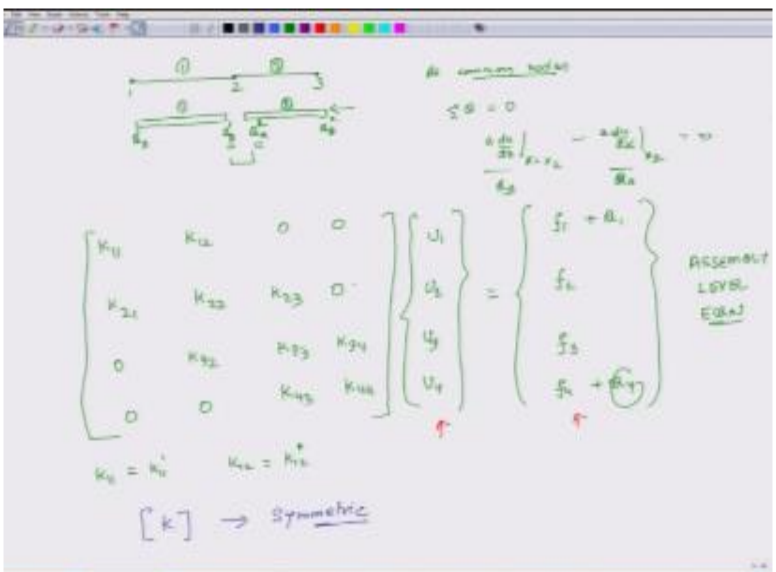
Indian Institute of Technology Kanpur
National Programme on Technology Enhanced Learning (NPTEL)
Course Title
Basics of Finite Element Analysis

Lecture – 27
Assembly of element equations, and implementation of Boundary conditions

by
Prof. Nachiketa Tiwari
Dept. of Mechanical Engineering
IIT Kanpur

Hello welcome to basics of finite element analysis, the fifth week, of this particular course and today is the third day in this week, in last class what we have learned was, how do we assemble all our element level equations and bring them to an assembly level. We will continue that discussion further and what we will learn today is, how do we impose boundary conditions on these equations so that is one thing we will learn, and second days we will do one more example, in context of how to develop assembly level equations, so we will.

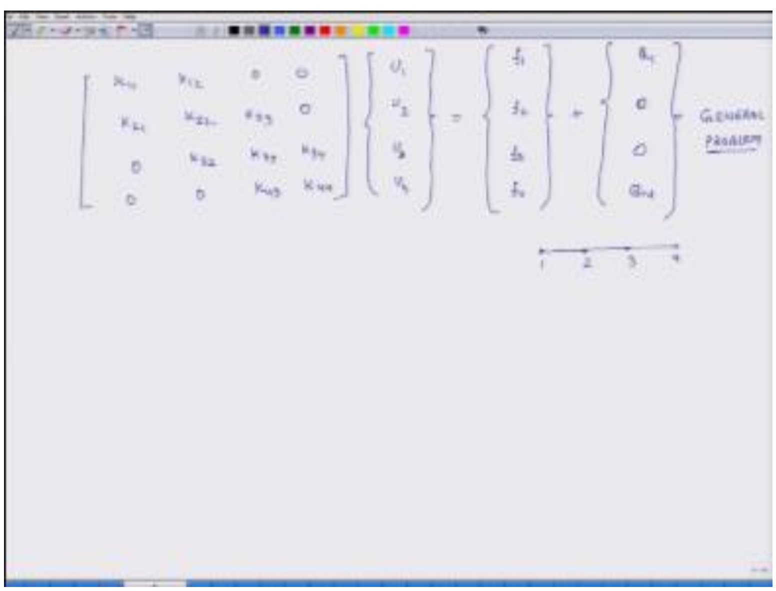
(Refer Slide Time: 00:58)



So these were our, in the last class what we had seen, that these were our assembly level equations, this was the first equation at assembly level, the combo of second and third element equations, was the second assembly question, then this was the third equation, and this was the fourth equation, and if we put all these four equations in the matrix form, then our global K matrix is this thing, and you should note that in this global K matrix, we do not have a super script term, because that whole super script referred to the element number. So all we have is K_{11}, K_{12} and so on and so forth.

And then, we have this global vector for unknown constants, and then global force vector, so that is there, at this stage we should again remind ourselves that at add the element level all the assembly level, at the element level, all the K matrices where symmetric, and when we use these matrices to construct the global K matrix, this global K matrix is also, so this is my global assembly matrix, this is also symmetric, this is also symmetric. The third point, I would like to make it make here is, that you see a lot of zeros in this global K matrix, and this is something you will see again and again, in a lot of assembly equations, and we will discuss the reason for this, and also what are the mathematical implications for such type of a matrix.

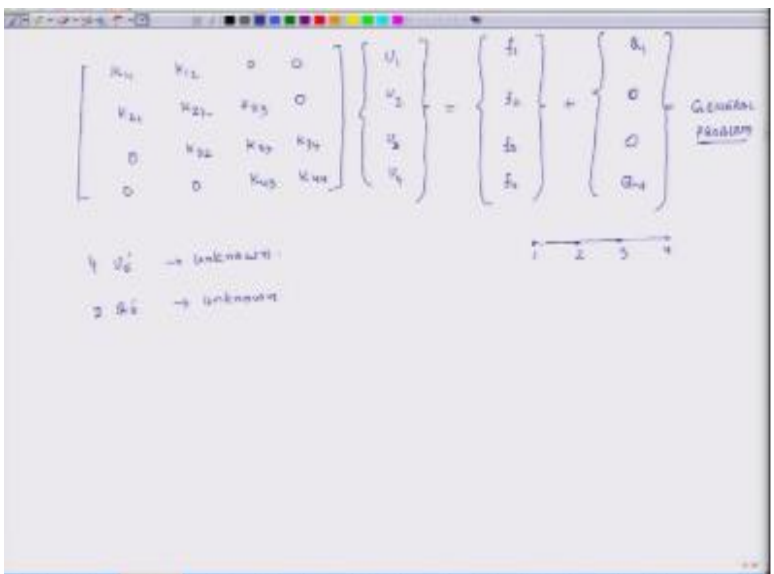
(Refer Slide Time: 02:54)



So, we will rewrite the global equation so K_{11}, K_{12} 0, 0, this term should be zero, multiplied by global degrees of freedom. Now, what we see is that in our original problem statement we had broken the domain into three elements right. Node 1, node 2 node 3, node 4, so there are four degrees of freedom, because each degrees each node is associated with one unknown which is u, so there are four degrees of freedom, and we have four equations, so this is the first thing.

So this is one, the second thing is, so this is the general problem, these equations are valid for all sorts of bar problems. Because here we have not given any boundary conditions, it is the boundary conditions which make the things unique okay. So these are, this is the, these are general equations.

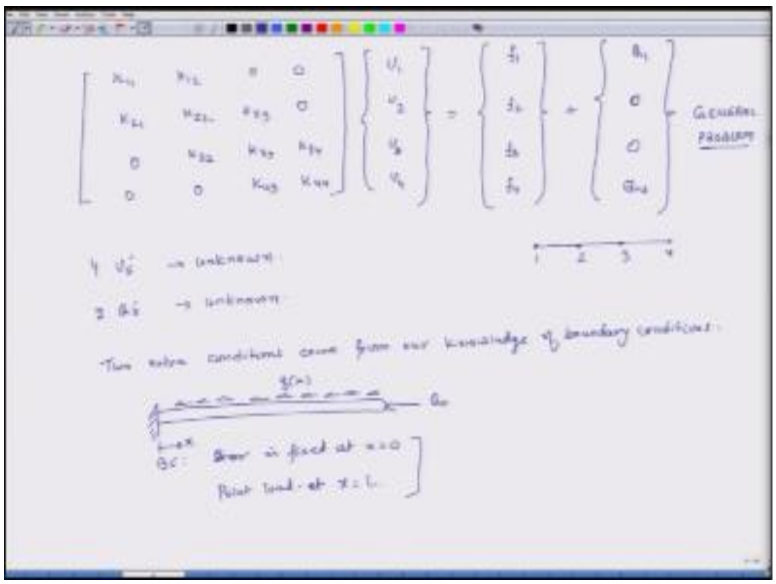
(Refer Slide Time: 05:13)



For any one dimensional bar under compression or attention, we can construct K matrix, we can figure out what is f and this, these are the Q's. In this general set of equations, how many unknowns we have, so we have four U's which are unknown, and then we also have two Q's, they are also not known. Q_1 and Q_4 everything else in this equation is calculated, so we know everything, we know all the f's, we know all the K's. But what we do not know U's and Q's okay, so there are six unknowns and four equations.

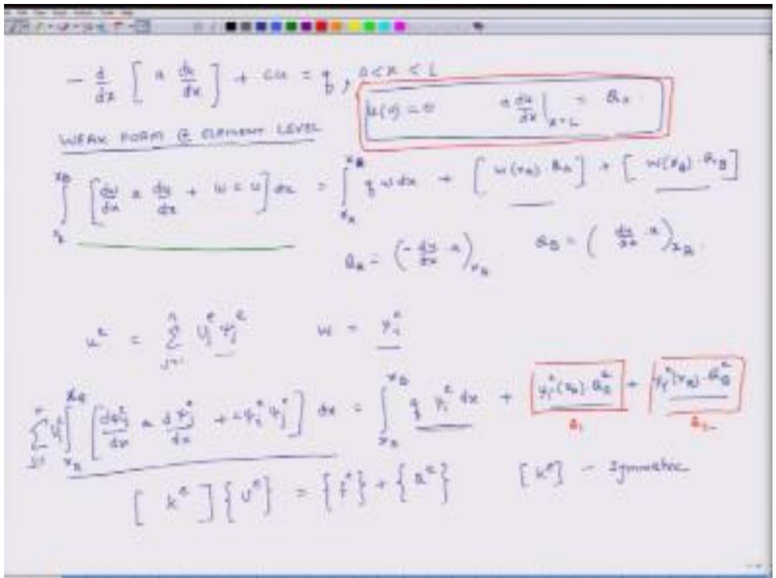
So we have to get, so this will always be the case, if there is if we have N nodes, we will always have N+2 unknowns and those so for a second order boundary value problem, in second order boundary value problem we will always have two extra unknowns, two extra unknowns okay. And we get the, so we need two extra conditions, we need two extra conditions, so that we can solve all these six unknowns.

(Refer Slide Time: 06:38)



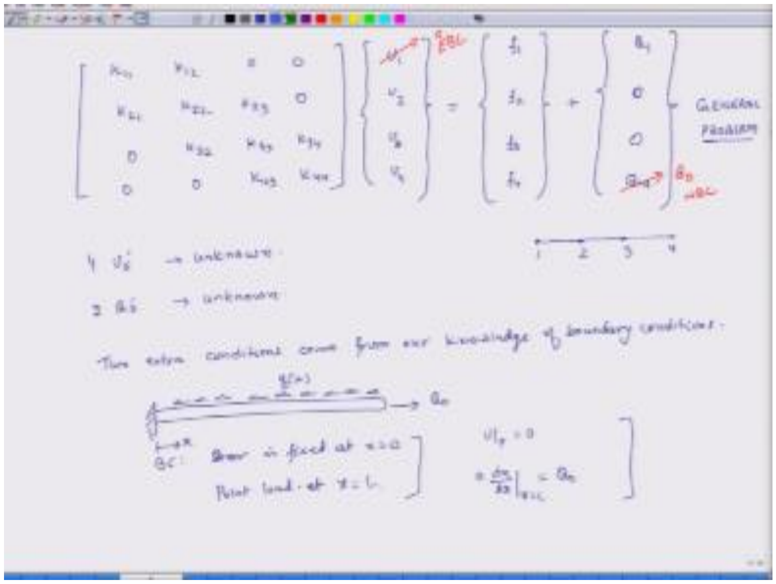
So those two extra conditions come from our knowledge of, our knowledge of boundary conditions. In this particular case, we had said that there is a bar right, and it is being exerted by some force, distributed force q(x) and there is an end force and this is Q_0 okay, this is the physical statement of the problem. The mathematical statement, so these are the boundary conditions so what are the boundary conditions, BC one that bar is fixed at x = 0, this is my x coordinate, so bar is fixed at x coordinates, is a x=0, and the second boundary condition is point load at x =L. So this is the physical meaning right, mathematically we had specified two boundary conditions.

(Refer Slide Time: 08:25)



Which were these, that so physically the bar is rigidly fixed mathematically it means that u is zero at x is equal to zero, at the other end, a which e times cross-sectional area, times strain so that is the force at x is equal to L is Q_0 okay.

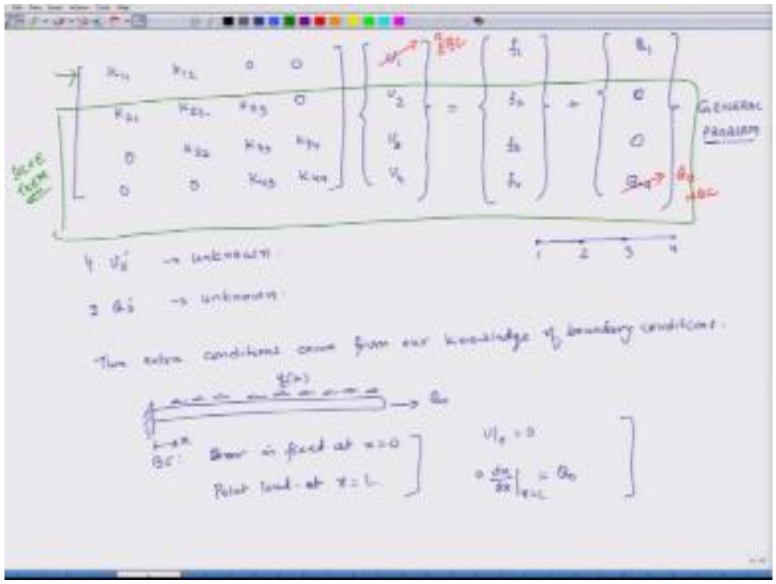
(Refer Slide Time: 08:54)



So mathematically these boundary conditions mean, that u at 0 is equal to 0, and a du over dx, at x is equal to L is equal to Q_0 okay, and actually I am going to pull it, Q_0, so this is what it means mathematically, which means that in this equation in this general assembly level equations I have to make u at 0 equal to 0, which means that this term is zero okay, so this is once we put this value to be 0 we have enforce the essential boundary condition okay. And then we put this value as Q_0, so this is my EBC I implemented my EBC, this is what I have implemented NBC, this is the primary variable, that is the secondary variable okay.

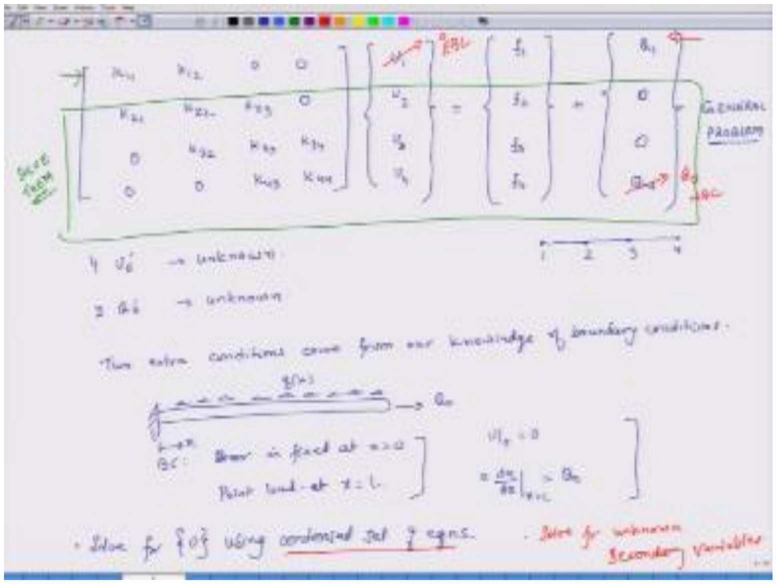
So, this is what I do when I specify boundary conditions they could be some other boundary condition and I again either change the value of essential boundary condition or the value of natural boundary condition accordingly. So, now once I have done this I have now four unknowns and four equations right, four unknowns and four equations. So the four unknowns are u2, u3 and u4 and Q1, and these are the four equations. So at this stage, what I do is once I have implemented all my boundary conditions, then I only consider those equations where I have not specified the essential boundary conditions. So I have.

(Refer Slide Time: 10:54)



Specified the essential boundary condition in this equation, so now at this stage I do not consider the first equation at all, and I only consider the second, third and fourth equations okay. So here, I have three unknowns and three equations, and I solve these guys okay, so I have an assembly level equation and once I impose the boundary conditions, there is a condensed set of equations for which I have not specified U’s okay, I only choose those equations not the other ones, where I have specified U’s, and then I solve for U’s okay.

(Refer Slide Time: 11:40)

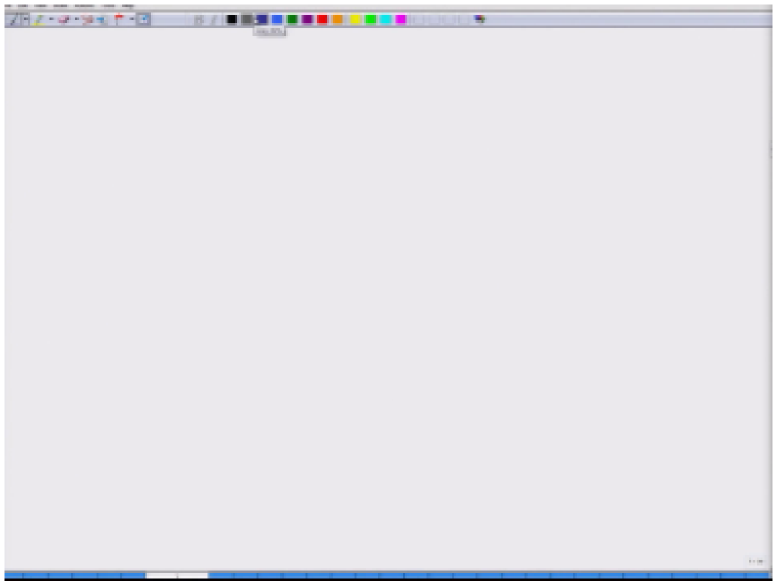


So my next step is once I have implemented the boundary condition is that solve for u using condensed set of equations okay. Once I have solve for U's using condensed set of equations, then once I have done that then, solve for unknown secondary variables okay. So first is that we solve for u, once I have calculated that u, then I plug that u_2, u_3, u_4 in my first equation, and I solve for this secondary variable which is also unknown okay. So first I always calculate primary variable, once I figure it out all primary variables, then I figure out my unknown secondary variables. So this is the overall finite element analysis process. And you see that this can be, if we understand this logic it can be implemented in computers.

At every stage you can break a line into twenty different elements, that can be done on a computer, you can develop an algorithm, you can integrate weight functions and their derivatives over the domain of l element, and that will help you generate the K matrix, So that is automatable, then you can you, you have a logic to assemble different equations which I have explained that can be automated, and then you have a logic using which we specify boundary conditions and develop a condense set of equations, that can be automated and finally we know how to solve linear equations using computers.

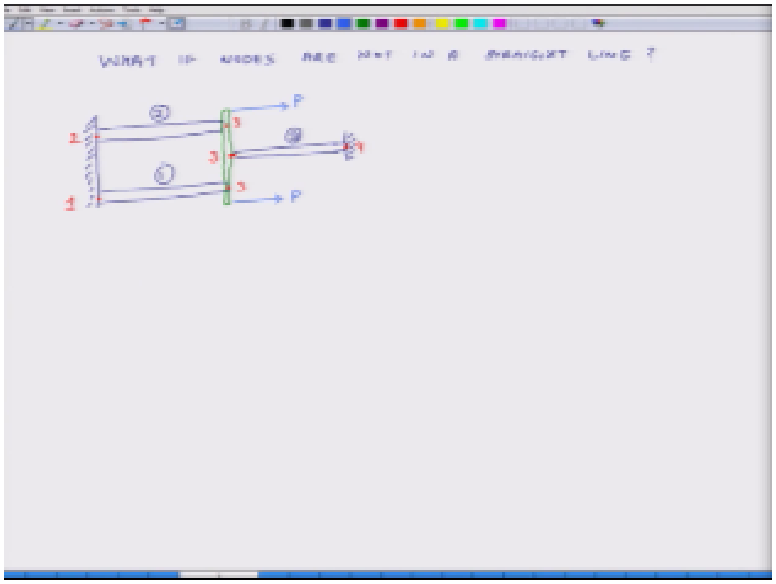
So then I know my primary variable, from primary variable I can calculate Q’s. So this entire process can be automated and that is what makes the finite element analysis very powerful.

(Refer Slide Time: 14:00)



Next what we are going to do in today's lecture is, we will consider one more example okay. Now in the example which we had discussed earlier it just happened that all the nodes where in one straight line, it where in they were in one straight line. But there is no reason to think that that should always be the case, we can have a one-dimensional problem and still nodes need not be in one single straight line. And if that happens then how do we address those problems and how do we make the logic which we have discussed till so far, even more general so that even if nodes are not in one straight line, we can figure out how to assemble element equations in a methodical logic programmable way, so.

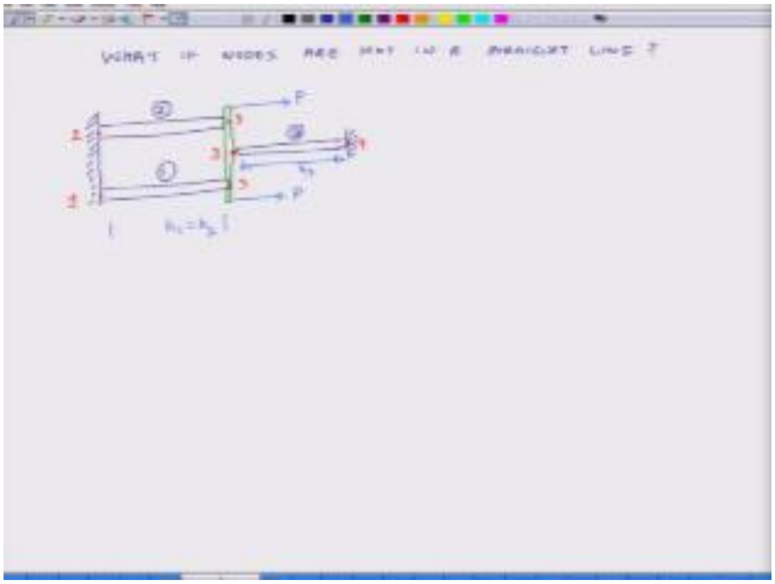
(Refer Slide Time: 14:59)



So the point is that what if nodes are not in a straight line, so we will do an example for that, so let us say I have a bar, I have two bars, two elastic bars and both these bars are connected with some rigid un-deformable plate right or some link. And the middle point of this is connected to a third bar, which is rigidly held at the other end, and for purposes of simplicity we will consider this entire system as a each bar we will consider that it is one element, just for purposes simplicity.

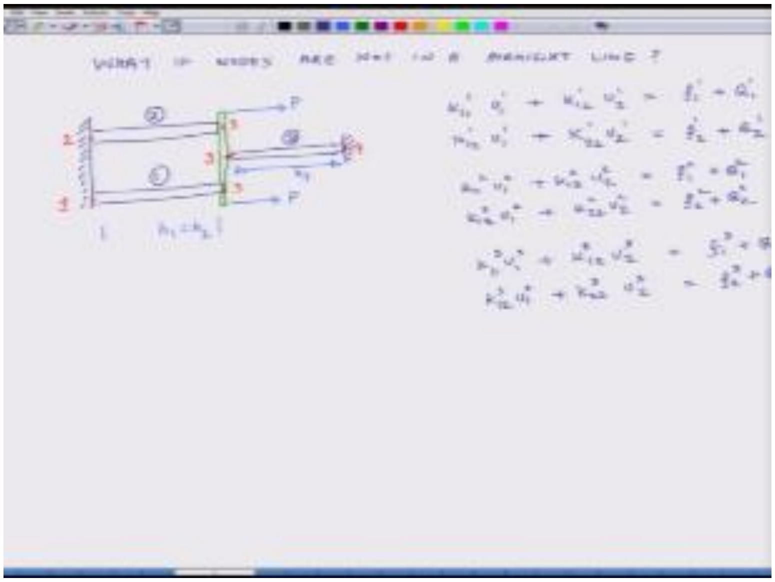
So this is my element number one, this is my number two, this is my number three and I am going to put node numbers, so this is node 1, this global node, this is global node 2, this is global node 3, okay which is same as this point three, which is same as this point three, and this is global node 4. And then this system what I am doing is, I am applying a force of P here, and another force of P here, so I am uniformly pulling these bars stretching and compressing these bars. So the problem is still one dimensional, because there is no gradient of u in y direction or z direction, it is only in x direction, further.

(Refer Slide Time: 17:16)



I say that this element length is h_1 and this is equal to h_2, and this element length is h_3. So we have three elements each element has two nodes so at element level how we will have we will have a total of six equations right, we will have total of six equations.

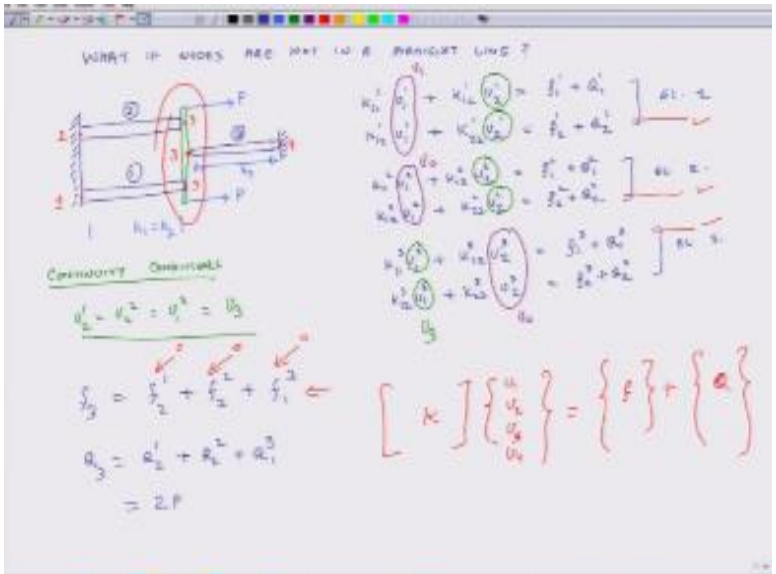
(Refer Slide Time: 17:46)



So we will write down those equations, so K_{11}^{1} times u_1+ K_{12} u_2 equals f_1+Q_1 and all these parameters are related to the first element, so I have a super script one, K_{12} u_1+K_{12}, so this is for element one, similarly I have K_{11} u_1 +K_{12} u_2 = f_1+Q_1, K_{12} u_1 + K_{22} u_2 = f_2+Q_2 and all these terms are for the second element, so you have a subscript of two. And for the third element K_1 so these are the six equation okay, now we have to assemble them, we have to assembled them and what we see here is that things are not in one straight line right, there are some elements which are connected in parallel, there are some elements which are in series.

But the laws of assembly are still the same, so what are the continuity conditions.

(Refer Slide Time: 20:21)

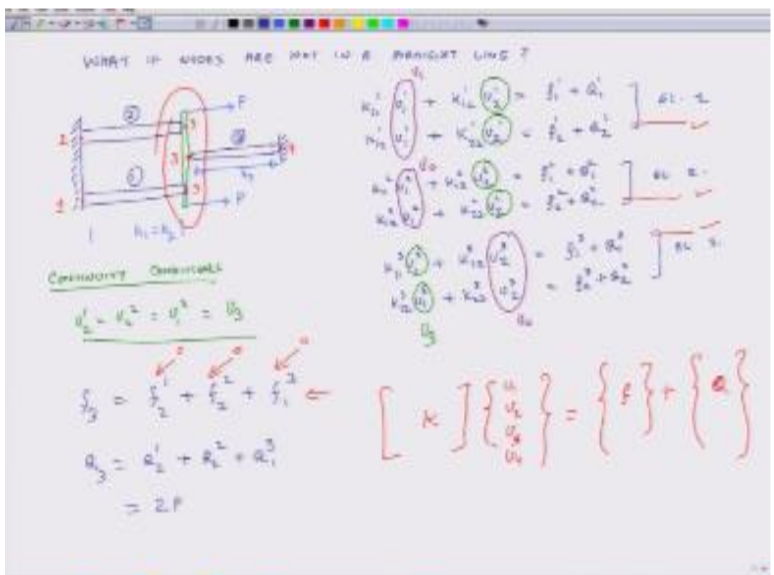


What are the continuity condition, the continuity conditions first is that u_1^2 is equal to this is the continuity condition. What does it mean, the displacement of first elements second node is same as displacement of second elements second node, is same as displacement of third elements first node that is the continuity condition yeah, okay. So what that means is, excuse me that I replace this variable, also this variable, and this variable by u_3 in the equations right. The second thing is, so this is the continuity condition, the second thing is, that they replace these by u_1 these by u_2, and these by u_4, because 1, 2, 3, 4 which are marked in red, there are my global degrees of freedom.

So that is there, the second condition is that the force at node 3 will have three components right, $f_3$is equal to $f_2^1+f_2^2+f_1^3$ agreed, and the same thing is true for Q's also, same thing is true for Q's also. So, so I have to add which equations, f_1^2 which is this equation, plus f_2^2 which is this equation, plus f_3^1 which is this equation, these are the three questions I will add up agreed. This is the force balance in this, at this interface agreed. So once I add these three equations up I get an overall K matrix times my global degrees of freedom u_1, u_2, u_3 and u_4 and that I get as some f matrix plus a Q vector.

It just happens then in this case okay, so this is the first case, in this case is there any externally distributed force in the picture there is no. So when we will calculate the values of these f's they will be zero, where is force coming from, it is a point load at node 3 right, so when I do this summation Q_3 is equal to $Q_2^1+Q_2^2+Q_1^3$ this will be equal to 2P, because I am actually applying a point load which is equal to 2P at that particular node. So this is one example where you apply a point at a node in that case, the sums of Q's does not become zero okay, then it becomes whatever the force you are actually applying, if you are not applying any external forces, then internal forces will cancel out they will balance each other and they will cancel out. But here I am applying an external force and that is actually going to drive.

(Refer Slide Time: 25:14)



The displacement in the system, so this is something I wanted to discuss, and this concludes our lecture for today, in the next lecture we will continue this discussion, and we will look at some other ways which are used to automate the assembly process. Thanks.

Acknowledgement
Ministry of Human Resource & Development

Prof. Satyaki Roy
Co-ordinator, NPTEL IIT Kanpur

NPTEL Team
Sanjay Pal
Ashish Singh
Badal Pradhan
Tapobrata Das
Ram Chandra
Dilip Tripathi
Manoj Shrivastava
Padam Shukla
Sanjay Mishra
Shubham Rawat
Shikha Gupta
K. K. Mishra
Aradhana Singh
Sweta
Ashutosh Gairola
Dilip Katiyar
Sharwan
Hari Ram
Bhadra Rao
Puneet Kumar Bajpai
Lalty Dutta
Ajay Kanaujia
Shivendra Kumar Tiwari

an IIT Kanpur Production