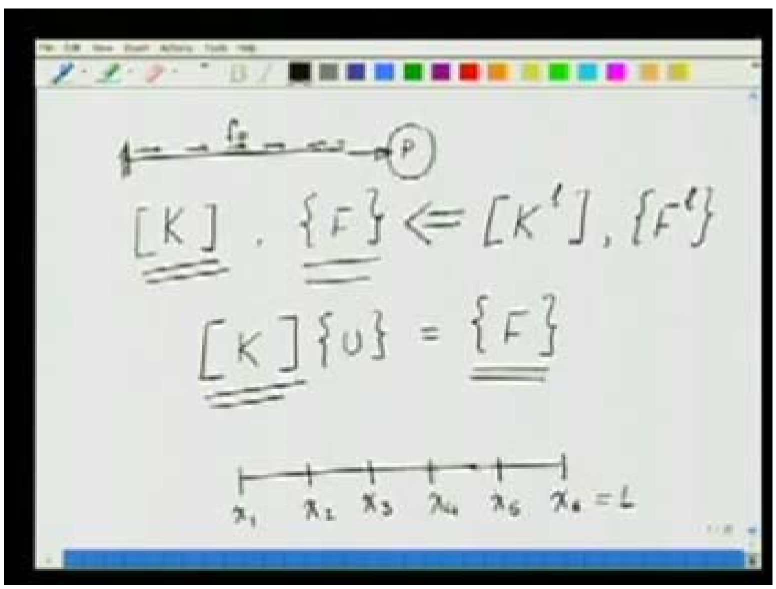
**Finite Element Method**

**Prof. C. S. Upadhyay**
**Department of Mechanical Engineering**
**Indian Institute of Technology, Kanpur**

**Module – 2 Lecture – 4**

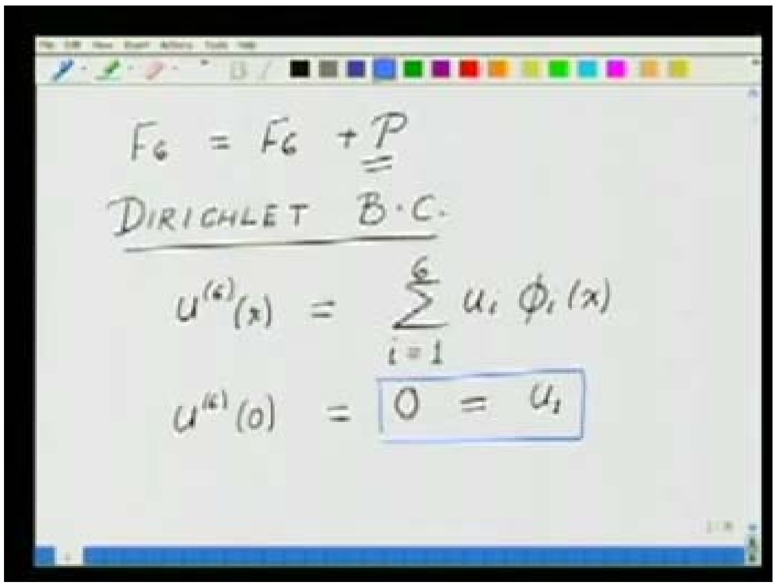
In the previous lecture, we had looked at the finite element method and we had done a simple implementation of that method using the hat-shaped functions or the linear functions to a problem of a bar.

(Refer Slide Time: 00:33)



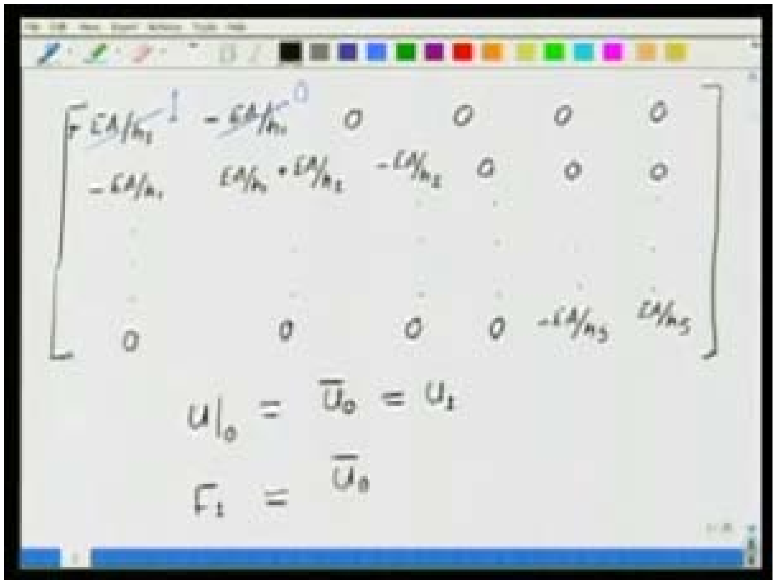

If I can draw that problem again, this is the bar with an end load P and subjected to a uniformly distributed load $f_0$. So, in that case, we had built up the element equations. We introduced the idea of element calculations and then we had discussed how to take these element equations and put them in the global matrices; that is, the global stiffness matrix K and the global load vector F. The element calculations which we had written as K of l, element l and the matrix, and the load vector from the element F due to the element l - how these equations will be put in these global equations in terms of K and F? We had said that this will finally after assembly lead to an equation of this form: K U equal to F. What we had done in the last class was we had developed this matrix K, we had developed this vector F, but we had not taken into account the effect of the boundary conditions. How do we account for the boundary conditions? We had taken care of this end force P here by saying that if we consider the domain is broken into 6 elements as we had done into 5 elements with nodes $x_1$, $x_2$, $x_3$, $x_4$, $x_5$, $x_6$. $x_6$ is equal to L and $x_1$ is equal to 0.

(Refer Slide Time: 02:53)




We had said that in the entry $F_6$, we updated it equal to $F_6$ plus P. That is in the final entry of the vector F, we added the value of this end load P which is applied and why we did it was discussed in the last class. So this part was taken care of. We had left at was a little bit more complicated part of an application of boundary conditions, which is how do we impose the Dirichlet or the essential boundary conditions? This is where we had trailed off in the last lecture. What do we have for our problem? We know that for our problem, u six is the approximation that we have made. This is equal to sum of i equal to 1 to 6 $u_i$ $phi_i$ x and we know that u six at 0 for this problem has to be equal to 0. At x equal to 0 which is the node $x_1$ only $phi_1$ is equal to 1. All other phi(s) are 0. So this is equal to $u_1$ (Refer Slide Time: 04:18). This condition has to be explicitly imposed in the solution. How do we explicitly impose it? Let us go to the system equations that we have. If you remember, what was our global stiffness matrix? Let us start with the system equations that we had written.

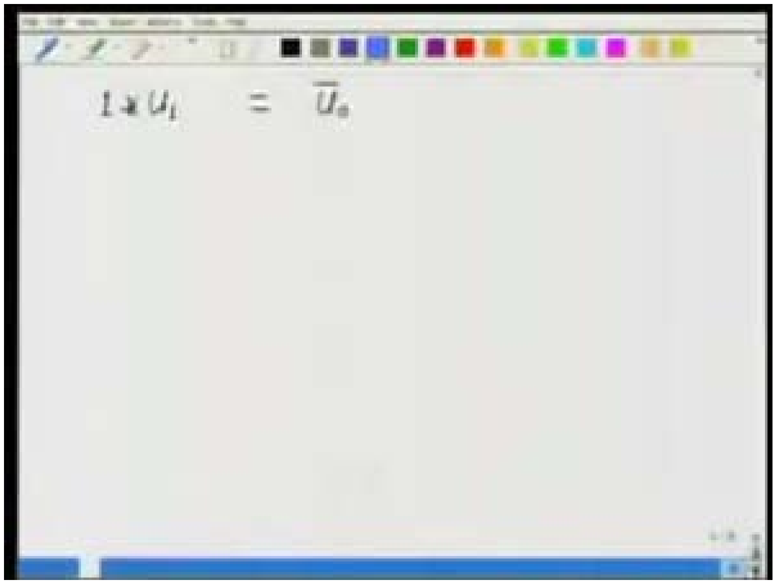
(Refer Slide Time: 04:53)




What did we have earlier? I will write it in a little bit general way. First equation was plus EA by $h_1$, minus EA by $h_1$ and 0, 0, 0, 0. Second one was minus EA by $h_1$, here I will get, because of the assembly, EA by $h_1$ plus EA by $h_2$. Third one will be minus EA by $h_2$, 0, 0, 0. This way, I will continue till the sixth equation which is 0, 0, 0, 0 minus EA by $h_5$ and here I will have plus EA by $h_5$. This is the global matrix K. This into the u was equal to the F.

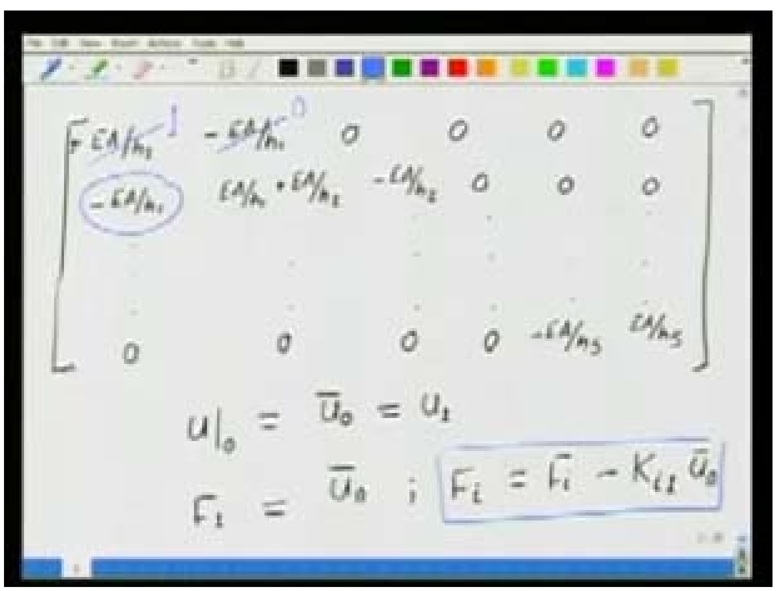
Here we see that nowhere have we really taken care of the value of $u_{1;}$ that is, we have not imposed the fact that the value of $u_1$ has to be equal to specified value. Let us now be a little bit further general. We will say that u at 0 is given the value $u_0$ bar. From what we have done earlier, this will be equal to $u_1$. How do we impose this in our solution? The idea is very simple. We will take the first equation and what I will do is I will replace this with 1 (Refer Slide Time: 07:15 min). The first entry, that is the diagonal entry, corresponding to the first row, that I am going to make as 1. The second entry I am going set it 0; that is, all other entries in this row, the first row corresponds to $u_1$. The diagonal entry of the first row, I am going to make it 1. All other entries, I am going to make it equal to 0. At the same time, what am I going to do to the $F_1$? I am going to put $F_1$ is equal to $u_0$ bar. Just look at the first equation.

(Refer Slide Time: 08:02)



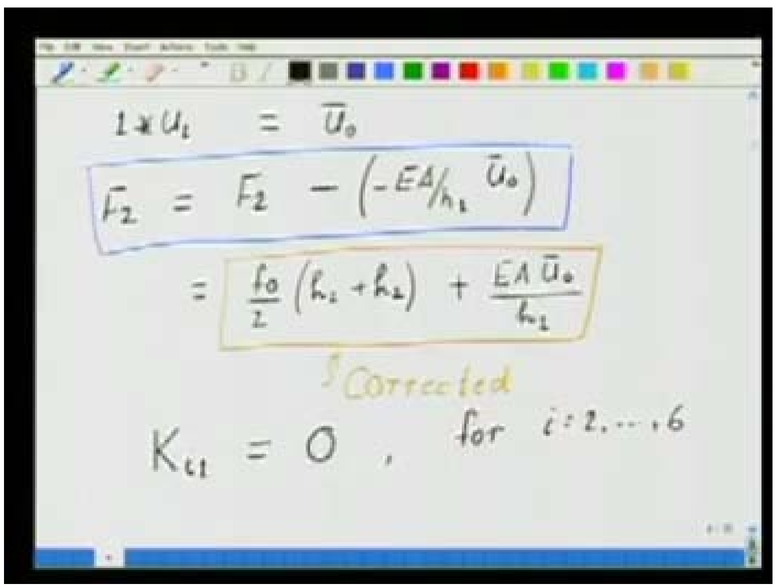

In the first equation, what do we end up getting? The first equation becomes 1 into $u_1$ is equal to $u_0$ bar. So this by modifying the first row of the stiffness matrix, I have enforced the fact that $u_1$ has to come out to be equal to $u_0$ bar. The question is that when we enforce this, the other equation has to change also.

(Refer Slide Time: 08:31)



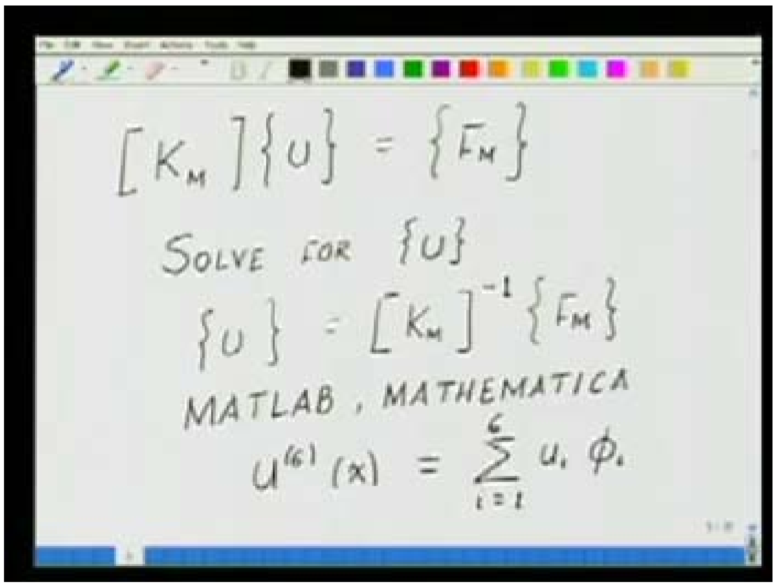
If I go back to the previous equation, in the second equation, this is the part (Refer Slide Time: 08:39) corresponding to $u_1$. This is the entry in the second equation corresponding to $u_1$. We know that $u_1$ is equal to $u_0$ bar. How do we remove this from this side and put it into the known part? Because $u_1$ is equal to $u_0$ bar which is known. The idea is very simple. What do we do next? For all other equations, we will put, that is for i equal to 2 and more, we will put $F_i$ is equal to original $F_i$ minus K; which entry are we looking for? The entry corresponding to $u_1$ in that particular equation. So the row will be I, column will be 1 into $u_0$ bar. So this is how we are going to modify our load vectors. You see what happens in our case is that for all other entries below, for the second equation, this has to be taken care of because this entry is not equal to 0, but from the third to the sixth equations, $u_1$ does not figure in the equations. So trivially, nothing has to be done, because, it is 0 which is being added or subtracted from the right hand side.

(Refer Slide Time: 10:21)




With that in mind, what do we have in the second equation? Modified $F_2$ will be equal to, by this token, what we had earlier as $F_2$ minus of, we had $K_{i1}$ that is the first column of the second row; it will be minus of EA by $h_1$ into $u_0$ bar. This will become the modified right hand side for the second equation. Let us rewrite it; what did we have as our $F_2$? $F_2$ was $f_0$ by 2 into $h_1$ plus $h_2$; so to that, I am going to add EA $u_0$ bar divided by $h_1$. This becomes the so-called corrected or the modified right hand side for our problem. Once we had transferred this information to the right hand side, then we are going to set $K_{i1}$ is equal to 0. That is we are going to blank out all the entries in the first column, for all the rows. This is for i equal to 2 to, for our problem, 6. Certainly, we are not going to do this to the first row because then it makes no sense. From the first row, we want $u_1$ is equal to $u_0$ bar and from the second row onwards, we want to take care of the known values of $u_0$ by suitably modifying our load vector.

(Refer Slide Time: 12:46)

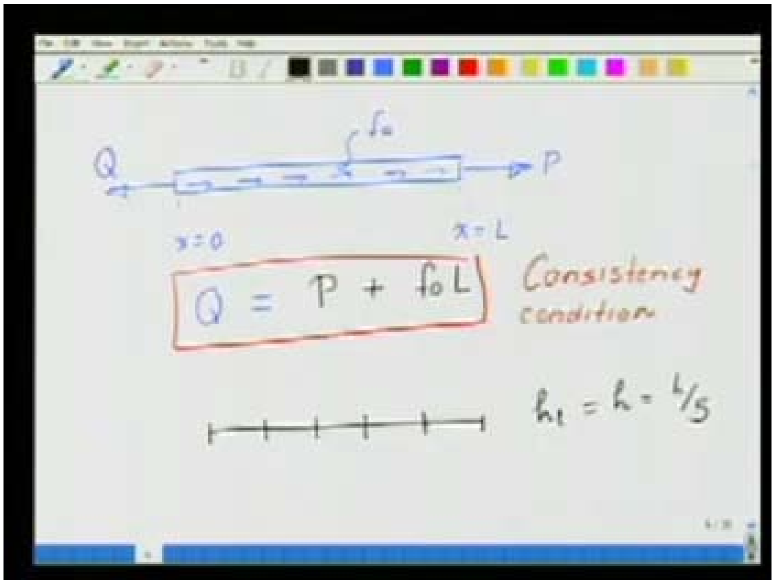



If I now use this and write the system, I will end up getting this system; I will call it $K_{modified}$ into displacement vector U is equal to the $F_{modified}$. Once I have these two things, then I can solve for U. How do I solve for U? I can solve for U by taking the inverse of K. U will be equal to $K_M$ inverse into $F_M$. Here, we are not going to talk too much about what kind of solvers. What I would request the student should do is to go use some commercially available code. For example, we can use Matlab or Mathematica to solve this problem. That is we can feed to these programs, the matrix K, the vector F and ask it to return back to the vector u. Once you know the components of u, then you know your solutions u six because this will be equal to sigma $u_i$ $phi_i$ x. This is how we will construct the finite element solution to a problem.

Now the question arises - why did we not ignore the first equation right away? Because, the first equation was meaningless from our point of view as we completely changed it. That is the first row because $u_1$ was known to us. We have to remember that when we are developing a method, it has to be a general-purpose approach. This was a particular example we took for which we had said $u_1$ is a known value.

For example, for our problem $u_1$ was actually equal to 0. So really these corrections that we did as far as the load vector entries are concerned, were also trivially equal to 0, the corrections terms, but we anyway did it to show that in case $u_1$ is the non zero number corresponding to a boundary value problem of interest; that is, I fix one end to a given deflection; that is also possible. In that case, what is it that we have to do? We see that the general-purpose approach gave us the right way of doing things out of which the special cases will also come out as solutions. So remember that whenever we are developing anything, we have to have the generality of the approach in mind. So with that in mind, we have done all these things.

(Refer Slide Time: 16:02)

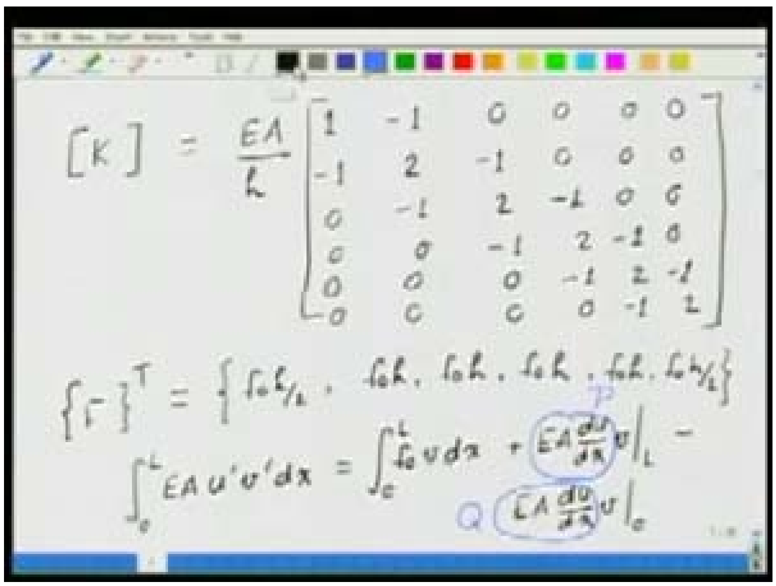



Another reason why we should assemble the first equation also is seen from the next example. Let us take a simple problem which is like this (Refer Slide Time: 16:02). This is again a simple modification of our earlier problem. This is a bar with distributed load $f_0$ and end load of size P at the point x is equal to L. At the point x is equal to 0, I apply a load Q. This is the static problem. We know that for a static problem, static equilibrium for the whole component has to be satisfied. Out of that, we know that P and Q cannot be arbitrary. That is, if I give $f_0$ and P, I know what Q should be for the system to be in static equilibrium. So looking at that what is our

Q? Q is equal to P plus $f_0L$, because in this case, $f_0$ is constant intensity distributed load. This we will call as the so-called consistency condition. That is, the load has to be consistent to give static equilibrium.

For this problem, what do we do? Remember that for this problem, we have already formed the stiffness matrix for the mesh of 5 elements. Let us keep that mesh of 5 elements, such that again here the mesh size $h_1$ is equal to h which is L by 5. For this 5-element mesh, we already know the stiffness matrix; that is the unmodified stiffness matrix; so, that, we will inherit from what we done earlier.

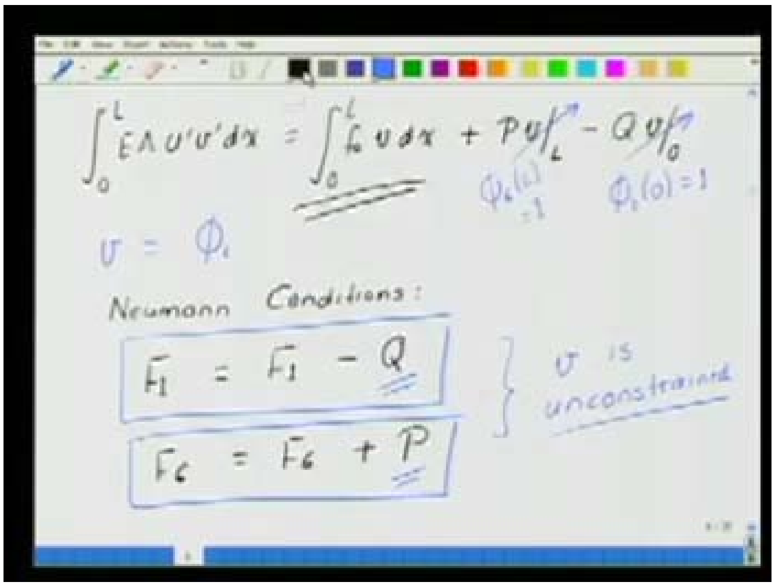
(Refer Slide Time: 18:18)



The K stays as what we had obtained for the previous problem before we applied the boundary conditions. This will be equal to, if I remember it correctly, it is EA by h into 1, -1, 0, 0, 0, 0; -1, 2, -1, 0, 0, 0; 0, -1, 2, -1, 0, 0; 0, 0, -1, 2, -1; 0; 0, 0, 0,-1, 2, -1 and finally we get 0, 0, 0, 0, -1, 2. This is the matrix K. What is F? F as we had said the unmodified F, that is before applying the boundary condition, so I will write the transpose of F, so that things are easily written. So 1, 2, 3, 4. Now comes the next part - that is we have to take care of, if we remember how we went about this thing - after we got the unmodified stiffness matrix and the load vector, then, we took care of the Neumann boundary condition first.

In this problem, both ends have Neumann boundary conditions. So if I look at the weak form - you should remember the weak form also - it is integral of 0 to L EA u prime v prime dx is equal to integral 0 to L $f_0$v dx plus EA du divided by dx into v evaluated at L minus EA du divided by dx into v evaluated at 0. The question is what is EA du divided by dx at the two ends? At the two ends, at x is equal to L, EA du divided by dx is P. At the end, x is equal to 0, this is equal to Q.

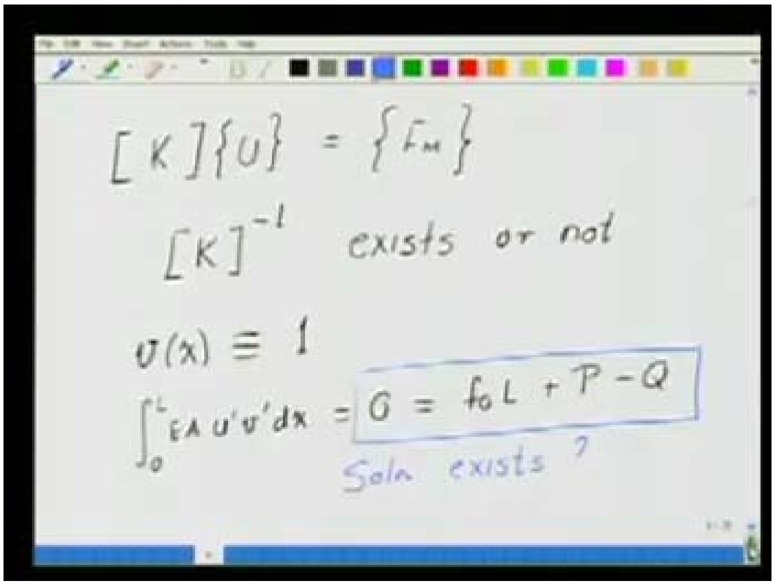
(Refer Slide Time: 21:16)




Let us go and look at it in greater detail. From 0 to L, we have EA u prime v prime dx. This is equal to 0 to L $f_0$v dx plus P into v evaluated at L minus Q into v evaluated at 0. I would like to ask this question - which of the v, because we have chosen v to be functions phi$_i$, which of v's will be 1 at point x is equal to 0 and 0 at the point x is equal to L? I would like to know which v will be non-zero here. By now, we should know that this will be equal to phi$_1$ evaluated at the point 0 which is equal to 1 (Refer Slide Time: 23:10 min). v at L, which of the phi(s) is non-zero at the point x is equal to L? This is phi$_6$ at L this is equal to 1. From this, what do we see? If I look at the equations, put v equal to phi$_i$ then this P has to contribute to the load vector term corresponding to phi$_6$ and Q has to contribute to the load vector corresponding to phi$_1$. So what do we do? Here we apply the Neumann conditions. The known $F_1$ that we have computed using this part to that we are going to add or subtract because Q is given like that. This will be now the modified $F_1$ (Refer Slide Time: 23:17 min). Just like what we did earlier, $F_6$ will be equal to the computed assembled $F_6$ plus P.

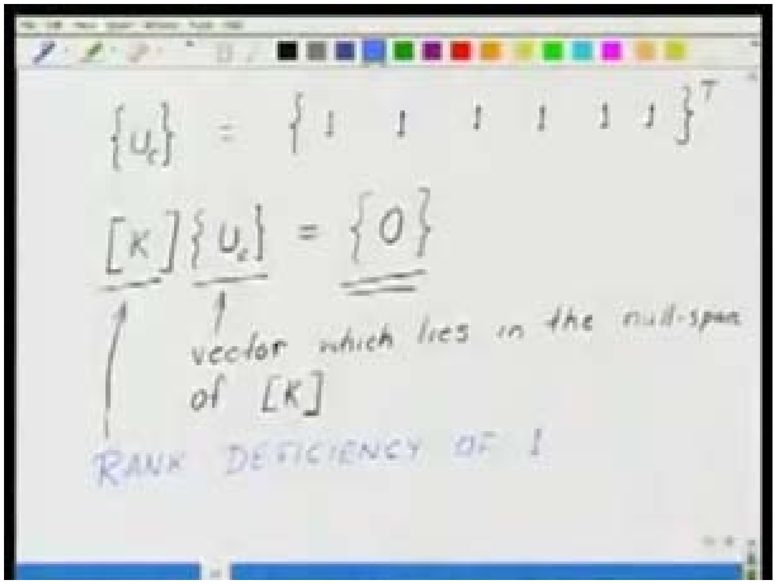
Application of the Neumann boundary conditions for this one-dimensional problem is very easy. If we go to the corresponding rows, just add a number, because Q is a number, P is a number. Once we have this, what do we know about v, for this problem? In this problem, v is unconstrained. What does that mean? Let us see what this will relate to.

(Refer Slide Time: 24:13)



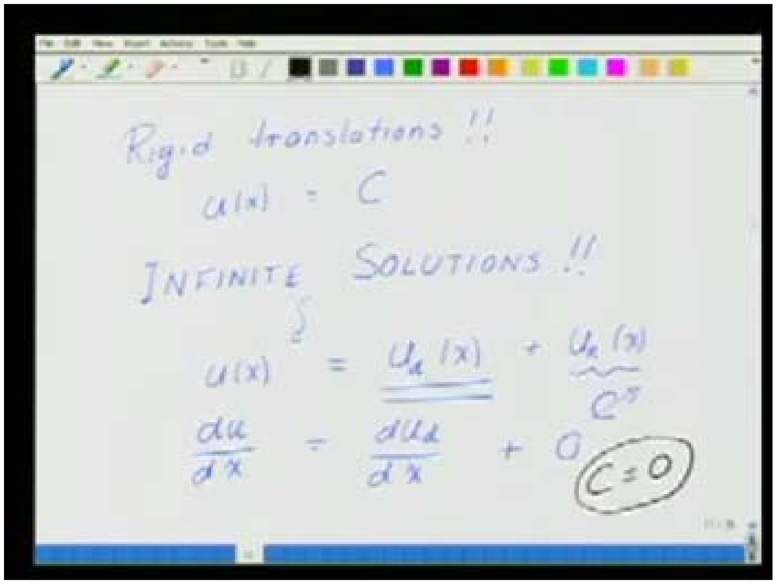

Remember that there is no geometric constraint to be imposed. So we will take the original K that we got out of assembly, keep it and your load vector will be modified - one which we obtained by previous step. The question is if I want to solve this problem, can I solve? That is I have to find K inverse. Before we find that, we have to know whether K inverse exists or not. Remember that v is an unconstrained test function. If it is an unconstrained test function, then I can very well use v(x) equal to 1, that is valid. Why is it valid? Because, it satisfies all the conditions or the constraints that we have put on v - that is the derivative of v exists, the energy due to v is finite, it is 0 actually. So we can use this. v(x) is equal to 1 everywhere. If we put it back in our weak form, what do we get? Integral 0 to L EA u prime v prime dx becomes equal to 0. This on the right hand side will be equal to integral of $f_0$v, so it will be $f_0$L plus P minus Q. So what do we see? We see that we had earlier stated the consistency requirement for static equilibrium and that has come out of this also. This is what we want our loads to do. Loads have to satisfy these constraints, in order to find the solution to the problem. That is for K inverse, we will see whether it is invertible or not, but at least here the issue is whether the solution can be found or not.

(Refer Slide Time: 26:34)




The next question is, what can we say about K? If we see one very curious thing that if I take the unmodified K and I choose U to be equal to the vector 1, 1, 1, 1, 1, 1 transpose, then what is K U? I will call it U choice. K $U_c$ will be equal to 0. It will be equal to the 0 vector, and then what can we say about the $U_c$? $U_c$ is essentially the vector which lies in the null space of K. That is for certain there is 1 non-trivial vector, we see for which K $U_c$ is equal to 0 which means that certainly K has a rank deficiency of 1. The question is - are there any other vectors which are not the same as $U_c$ and which are nontrivial, for which K U is equal to 0? The answer is no. There are no other vectors which are different from $U_c$; that is we can multiply with the constant that does not change the vector as such. So except $U_c$, there is no other vector for which K $U_c$ is equal to 0. So K has a rank deficiency of 1. This is fine from mathematics.

(Refer Slide Time: 28:54)



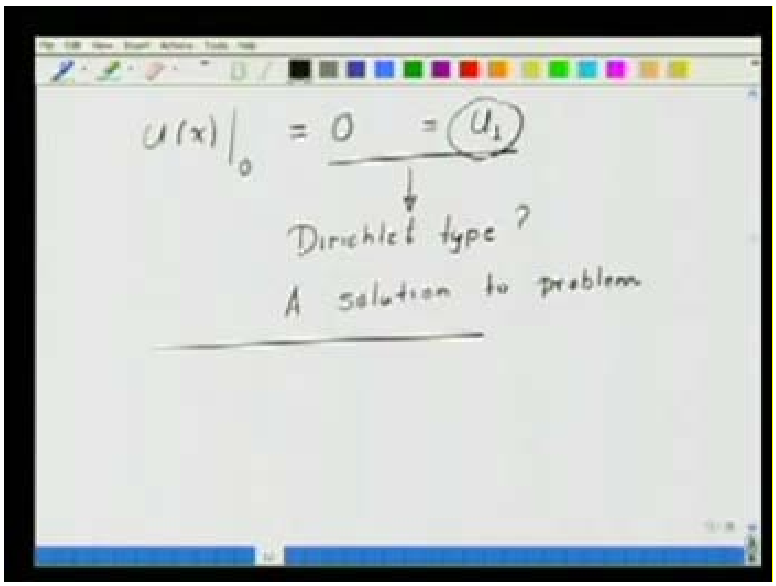

From physics what does this mean? It means that for rigid translations, which means that if u(x) is equal to some constant that is the whole bar is moving as an entity, in that case, the stress and the strain at any point in the bar are 0; because, it is a rigid body motion, there is no relative deformation of one point on the bar with respect to the other point on the bar. If there is no relative deformation, there is no strain. If there is no strain, there is no stress. The rigid translations correspond to 0 strain energy for the system, which is what we got. So the 0 strain energy modes are the rigid modes and for this bar, this is the only mode which is there and this shows in the rank deficiency of our system. There is no other rigid motion which is permitted for this kind of a bar. So the rank deficiency of the stiffness matrix that we got is 1.

The rank deficiency is 1 and we saw that our system equations are consistent. So solution will exist, but the question is - here we will have infinite solutions, because, so many solutions are available, what do we mean by that? Infinite solutions which will be of the form u(x) is equal to u deformation of x plus $u_{rigid}$ of x, that is plus C. That is the second part corresponds to the rigid translation and it could be anything - a constant C; and this part (Refer Slide Time: 30:45) is the one which leads to non-zero stresses and strains in the bar.

Really what we are interested out of this problem is in knowing the state of stress and strain. Stresses and strains will require derivatives. If I look at this du divided by dx, it will be equal to

$du_d$ divided by dx plus 0. That is the rigid part does not contribute to the derivative. If the rigid part does not contribute to the derivative, that is, its presence does not change the strain information, then, I can fix this C to be anything and get the same derivative or the strain. So we will put C equal to 0, we can put it. C is equal to 0 means that $u_{rigid}$ is 0. If $u_{rigid}$ is 0, we can say that how do we enforce this in our problem?

(Refer Slide Time: 31:49)

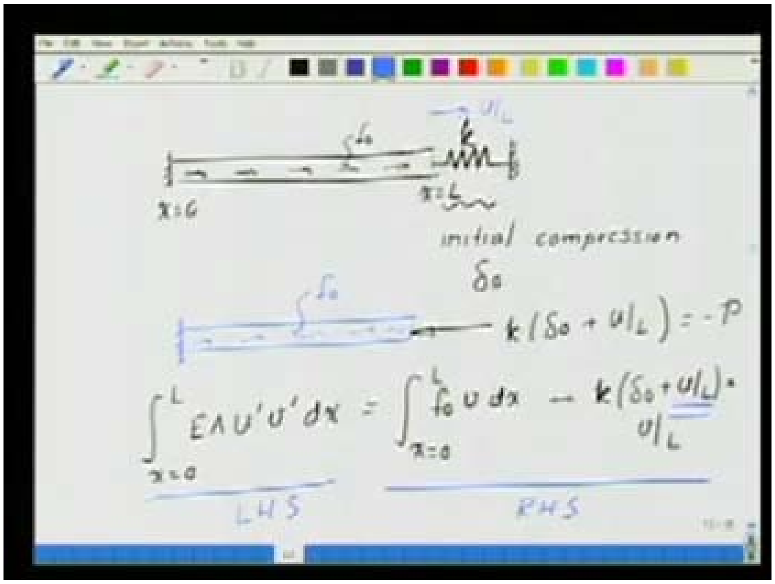



We can say fine, I am going to say that u(x) at point 0 is equal to 0. I can enforce this also to fix the rigid motion which is equal to $u_1$. Once I do this, then I will follow the same route as we followed in applying the Dirichlet condition to get a solution to the problem. We are fixing u(x) at the point 0 equal to 0. It is not the same as putting C is equal to 0, such that we get a solution to the problem and how do we impose this in our code? It is by modifying the equation corresponding to $u_{1.}$ That is, $K_{11}$ we are going to make it equal to 1; $K_{1i}$ for all i is greater than 1, we are going to make equal to 0 and similarly, $K_{1i}$ is equal to 0 for all i(s). Under this condition, we are going to get a solution to the problem.

We see that there was a very important issue that we have tackled here – that is a solution to the problem may not exist. If I apply the loads wrongly or if I apply the boundary conditions wrongly, I will get garbage solution.

Let us look at another problem which has a little different boundary condition.

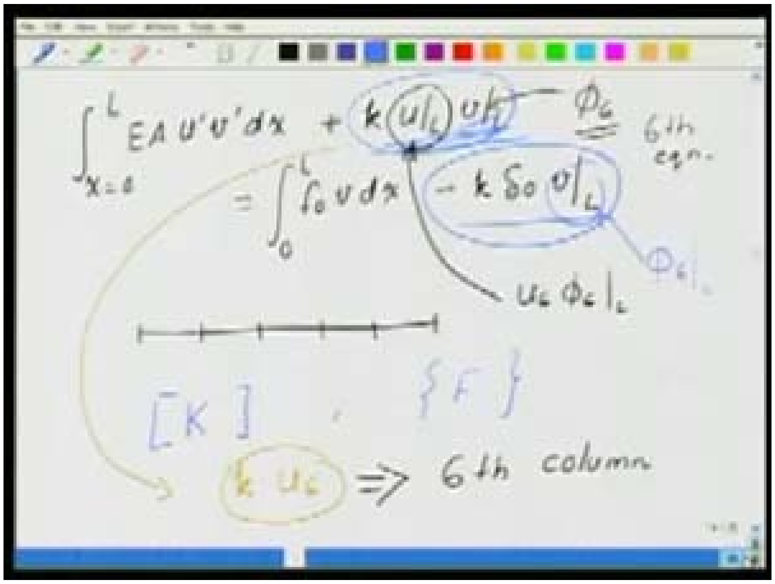
(Refer Slide Time: 33:45)



Let us go to the next problem. Since we are playing with problems, let us do this one. Here is the bar. Again, let us keep it to $f_0$, but here what I do is at this end I fix a spring, with spring constant K and I give an initial compression to the spring $\text{delta}_0$. This is the point x is equal to 0. This is the point x is equal to L (Refer Slide Time: 34:25). If I give this initial compression $\text{delta}_0$ then due to the action of this force, this bar is going to elongate, that is the end x is equal to L will move by an amount u at L due to the action of the forces. So u of L is unknown. For this u of L that you get at this end, what is the total force that the spring is applying on the bar? The spring applies compressive force of size k into $\text{delta}_0$ plus u at L, because, the total compression of the spring now becomes $\text{delta}_0$ plus the displacement of the end L which is u at L.

In our formulations that we have done, let us write the weak form for this problem. You should always write the weak form to the particular problem that you are trying to solve to understand what is going on. Weak form will be integral x is equal to 0 to L, $f_0$v dx plus P, that is the applied external force at the end x is equal to L into the v at the end x is equal to L. What is this force equal to for us? It is actually, if we remember, P was tensile earlier, now it is compressive; so it is minus P. This plus becomes actually a minus. I will write minus k $\text{delta}_0$ plus u at L multiplied by v at L. This is the weak form we get. We see that something very curious about this weak form. This unknown u at L, we had said that in the weak form, we are going to collect unknowns on the left hand side and knowns on the right hand side, but in the right hand side, even an unknown u at L is sitting. So what do we do? We shift it to the left hand side.

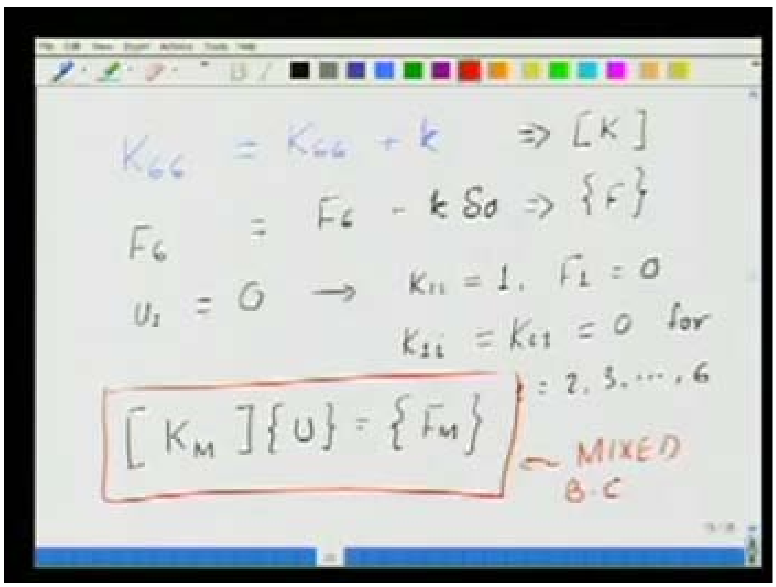
(Refer Slide Time: 37:12)




When we shift it to left hand side, weak forms becomes x is 0 to L, EA u prime v prime dx plus k into u at L into v at L is equal to integral 0 to L $f_0$ v dx minus k $\text{delta}_0$ into v at L. With this what happens? Let us go to our finite element formulation; let us again take the 5-element mesh that we had drawn. For the 5-element mesh, tell me what is it that we have to do to respect to the applications of these conditions? First thing that we do is we ignore these two terms (Refer Slide Time: 38:10). We do the standard element calculations to find the element matrices KL and FL; assemble them into the global matrix K and the global vector F. This condition k uL vL will be something that we have applied to the stiffness side. Tell me in the stiffness, for which P, that is for which row this part is going to cause the change? v at L, which v is non-zero at L? This is true for only $\text{phi}_6$; so v at L is equal to $\text{phi}_6$ that is for the sixth equation.

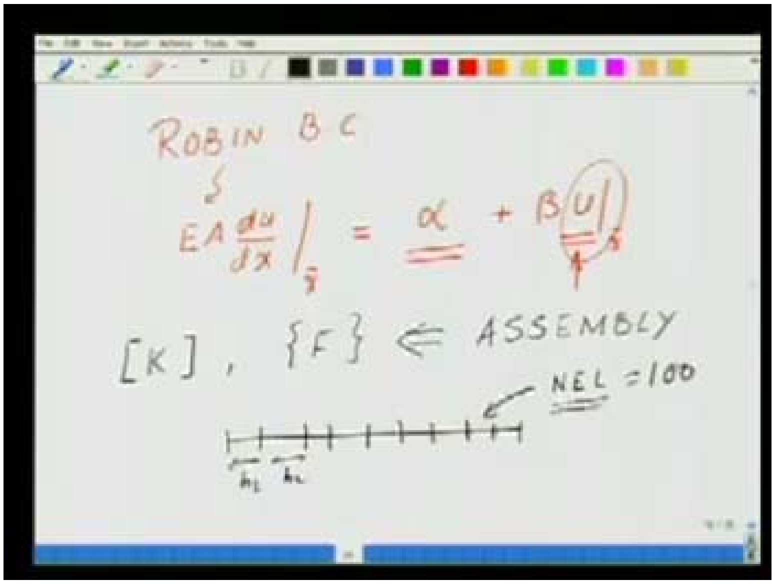
We will have to modify the stiffness term. This is the row which is fixed. The columns come out of u at L that we have. u at L, if you remember, what is the representation for the u? It is sigma $u_i$ $\text{phi}_i$. So which of the $\text{phi}_i$ is non-zero at x is equal to L? It is again $\text{phi}_6$. This will be essentially u at L will be equal to $u_6$ $\text{phi}_6$ at L. What does this lead to? The whole expression, if I want to write it somewhere, this will become $ku_6$. This tells me very clearly, which column this has to go in. This means, this entry goes in to the sixth column. In the stiffness matrix, that we have obtained out of our assembly we go and add k, because, $u_6$ stays in the displacement vector. I will add k in the sixth column of the sixth equation. So that is the first thing we have to do. Second thing, let us see as far as the load, now this term looks very much what we have handled as far as the load is concerned. So this is very easy, because, v at L is only non-zero for $phi_6$. This minus k $delta_0$ goes to the sixth load vector. Let us do that.

(Refer Slide Time: 41:14)




What else do we have to apply? We put $K_{66}$ is equal to $K_{66}$ plus small k. $F_6$ is equal to $F_6$ minus k $delta_0$. So these give us our modified stiffness matrix K and the modified load vector F, but our job is not yet done, because we have to also put that $u_1$ is equal to 0. For that we will again make $K_{11}$ is equal to 1, $F_1$ is equal to 0, $K_{1i}$ is equal to $K_{i1}$ is equal to 0, for i is equal to 2, 3 up to 6. Once we do this, apply these conditions, then I will get $K_{modified}$ which is what we have to solve (Refer Slide Time: 42:28 min). We see that here, because of the spring at the end x is equal to L, even the stiffness matrix needed a modification. This kind of boundary condition which is in terms of a spring applied at one of the ends is called a Mixed B C or a Robin Boundary condition which is that the end load, that is EA du divided by dx, at the point x bar is given in terms of some constant plus beta times U at x bar.

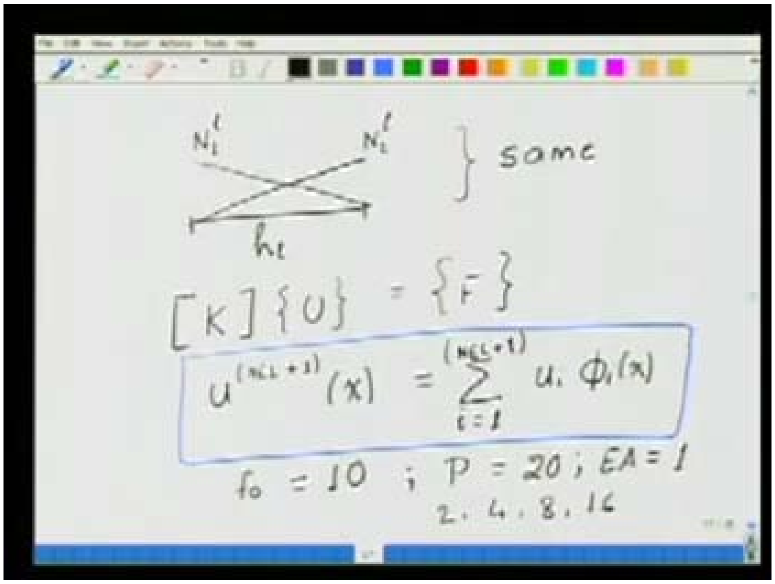
(Refer Slide Time: 43:02)




That is, the boundary condition is given in terms of some known and a component which is unknown, because U here is unknown; it is mixed in the sense that it has both known and unknowns, sitting in the boundary conditions. Because unknown U is sitting there, this has to be taken to the left hand side and it leads to a change in our stiffness matrix. These were some of the problems that we can handle, but remember that our basic assembled K and F that we obtained out of assembly, they remained unchanged, till we came to the next part, where we first applied the force boundary conditions whichever end it is given and then we applied the displacement boundary conditions whichever end it is given displacement or the even mixed one. This is the procedure here we are going to follow in any finite element computation. You see the beauty of it is that out of our element calculations, the stiffness matrix and the load vector came out automatically.

The question is if I want to change the number of elements in the mesh, no problem. If I want to change from 6 to let us say some number, let us say the total number of elements is NEL. These could be non-uniform. So non-uniform means that the mesh size is not the same for each element. Then it does it pose any problem? From what we have done, the stiffness matrix for any number of elements, let us say NEL is equal to 100, can be easily formed using these stencil that we have given and what is the stencil?

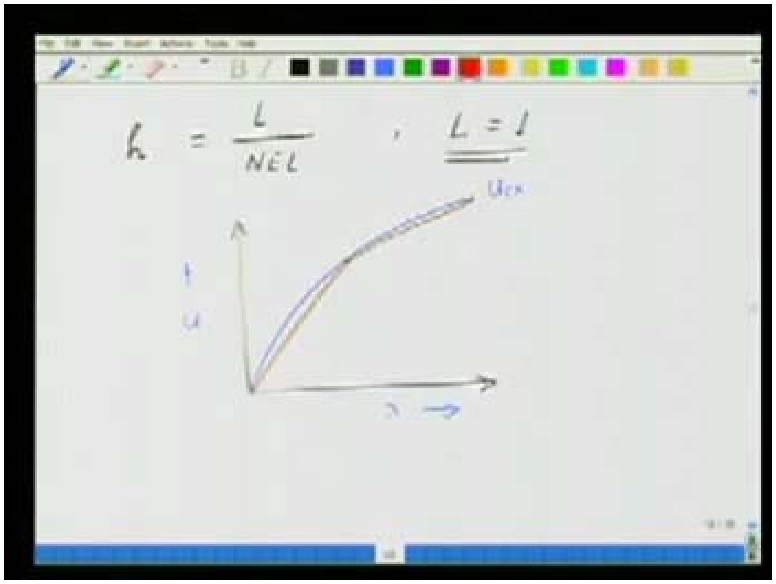
(Refer Slide Time: 45:37)



It is that the equation will have out of the element contributions due to $N_1$ of the element l and $N_2$ of the element l. So the equations for the element remain unchanged, irrespective of whether we are using 100 elements or 5 elements. Only thing that changes is that h that is the mesh size, what we have done and the material parameters that you may wish to change. So the element matrix representation remains same. Then it is the matter of assembly which we can do. Using this approach that we have outlined for this problem, very easily we can form the stiffness matrix and the load vector for a mesh of any number of elements or with any number of elements with either uniform meshing or non-uniform meshing. Once we do that, we can form the global matrixes K U and F solve; after we assemble, then we apply the force boundary conditions; after that modify the matrices to take care of the displacement boundary conditions and solve the problem. The thing is that bigger the size of the system, certainly, it cannot be solved by hand. So we have to go to a computer to invert the matrix K, but in principle, we can easily form this matrix K and solve it. Then obviously, we have to remember that u here will be equal to, let us say, number of elements plus 1 is equal to sigma i is equal to 1 to (NELplus 1) $u_i$ $phi_i$ x. So this u finite element can be easily obtained (Refer Slide Time: 47:50 min).

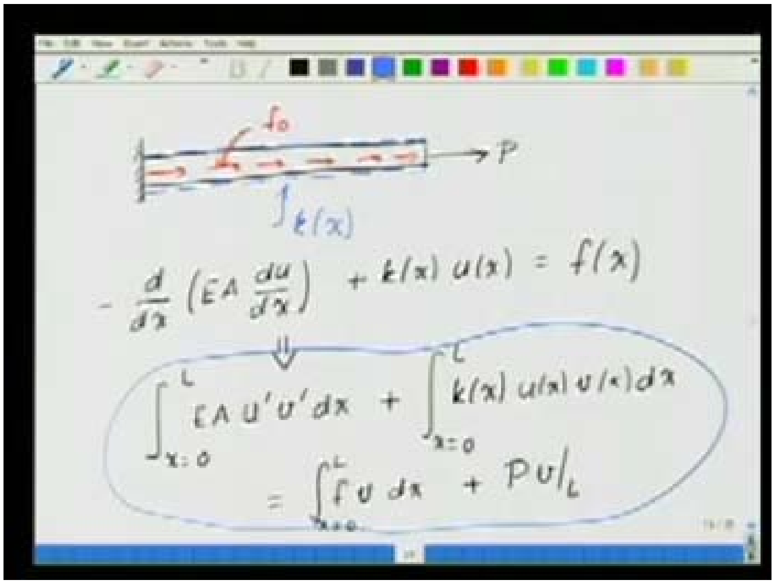
What I would like is as an exercise, you can take $f_0$ is equal to 10, P is equal to 20, EA is equal to 1 and solve this problem using 2, 4, 8 or 16 elements and so on; elements of uniform mesh size.

(Refer Slide Time: 48:28)



That is, in this case, uniform mesh size means h will be equal to L by NEL. Let us take L is equal to 1. One can easily attempt the solution and see that out of this, that if this is x (Refer Slide Time: 48:48), this is u, let us say, this is $u_{excat}$ then this will be the 2 element solution, then this will be the 4 term solution, then the 8 term solution. I will make it with blue again, we can see that. Essentially, what will happen is the solution will keep on coming closer and closer to the exact solution. You see that as the number elements are increased, our approximation does very well. This is true for any of the boundary conditions you take. The question is that do we always have this type of differential equation to handle? The answer is no.

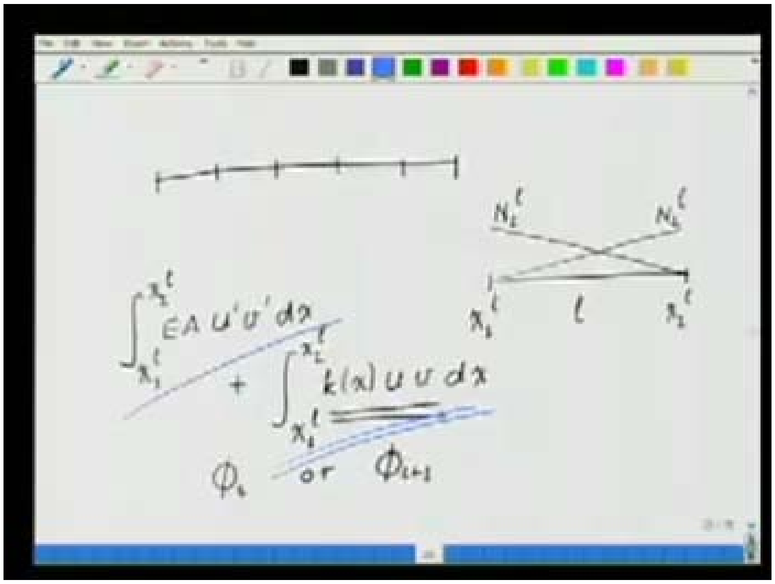
(Refer Slide Time: 49:55)




Let us make this differential equation a little bit more complicated. Still we have a straight bar with an end load P, but here we will have a distributed spring support. So it is an actual support which leads to a distributed spring constant k(x). As if the bar is resting on an rubber padding and this rubber padding is applying an uniform resistance with the force proportional to the displacement and the proportionality constant is k(x), because it is changing with the position.

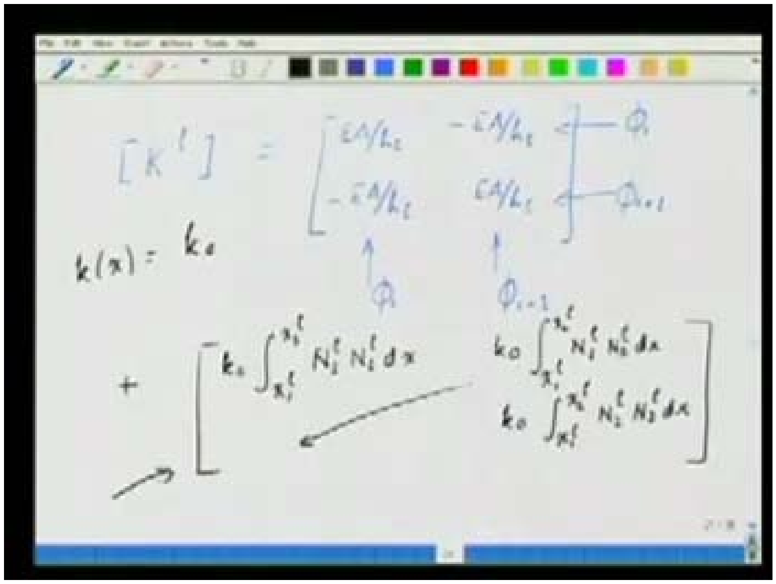
In this case, I will simply write the differential equation again. It will be d divided by dx plus k(x) into u(x) equal to f(x). This is the differential equation. The boundary conditions, we have already stated them. The weak form for this problem will be integral x is equal to 0 to L, EA again by doing integration by parts, u prime v prime dx plus integral x is equal to 0 to L k(x) into u(x) into v(x) dx. This is equal to integral fv x is equal to 0 to L dx plus P into v at L. This is going to be the weak form of the problem at hand. I say that let us do the finite element approximation corresponding to this problem.

(Refer Slide Time: 52:09)



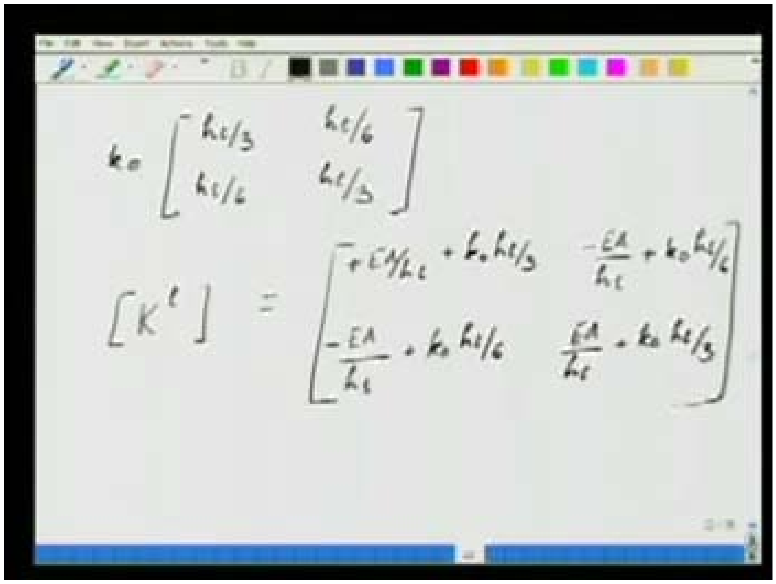
Let us go back to our 5-element mesh. For this mesh, what is going to change as far as the element calculation is concerned? If I take the generic element out, element l, $x_1$ of l, $x_2$ of l, this is $N_1$ of l, $N_2$ of l. What is going to change? What we are going to change is that, look at the element level, so $x_1$ of l through $x_2$ of l, you are going to evaluate this term plus now we are going to evaluate this contribution also. The load vector side remains the same. So this part (Refer Slide Time: 53:14) now is an addition. So what do we do? We replace this wherever u is there we will do u is equal to $phi_i$ or $phi_{i \text{ plus } 1}$. Similarly, v is equal to $phi_i$ or $phi_{i \text{ plus } 1}$. So what will happen is that to our standard element stiffness matrix that we had calculated earlier, we have to add this part corresponding to v equal to $phi_i$ or $phi_{i \text{ plus } 1}$. Similarly, u is equal to $phi_i$ or $phi_{i \text{ plus } 1}$.

(Refer Slide Time: 53:55)



So all we are going to do is change so the K element will be equal to, what do we have if we do this integration? Earlier, we had EA by $h_l$, minus EA by $h_l$, minus EA by $h_l$, EA by $h_l$ and we had said this corresponds to $phi_i$ (Refer Slide Time: 54:22 min), this corresponds to $phi_{i \text{ plus } 1}$. Similarly, this row corresponds to $phi_i$, this row corresponds to $phi_{i \text{ plus } 1}$ (Refer Slide Time: 54:35 min). To this, we are going to add plus, what are we going to add? Let us take k(x) is equal to some fixed value $k_0$. We are going to add $k_0$, if I integrate out, into integral $x_1$ of l to $x_2$ of l, $N_1$ of l into $N_1$ of l dx $k_0$ integral $x_1$ of l to $x_2$ of l, $N_1$ of l $N_2$ of l dx and this will repeat here. Here we will add $k_0$ into integral $x_1$ of l to $x_2$ of l, $N_1$ of l $N_2$ of l dx and this will repeat here $k_0$ into integral $x_1$ of l to $x_2$ of l, $N_1$ of l $N_2$ of l dx. This additional part, if we work it out, will be equal to $k_0$.

(Refer Slide Time: 55:47)




I have k to be constant, but it need not be. So the additional part becomes $k_0$ into $h_l$ by 3, $h_l$ by 6, $h_l$ by 6 and $h_l$ by 3. This has to be added to the element stiffness matrix that we got out of the first part and that will give the total element stiffness matrix K of l which will now become equal to plus EA by $h_l$ plus $k_0h_l$ by 3, minus EA by $h_l$ plus $k_0h_l$ by 6, minus EA by $h_l$ plus $k_0h_l$ by 6, EA by $h_l$ plus $k_0h_l$ by 3. This is all the modification that has to be done to the stiffness matrix. Curiously, in this case, the problem with forces at the two ends will have no problem as far as the solution is concerned, because in this case, the global stiffness matrix is invertible, because there is no rigid mode. If we had a rigid displacement, it would lead to deformation of the springs, so the energy of the system is not going to be 0.

With this, let us now close our exercise that we did with respect to a very simple finite element formulation using hat functions. We saw how we construct the element calculations, how we apply the rules of assembly to form the global stiffness matrix and the global load vectors, how we apply the Dirichlet boundary conditions and first the Neumann boundary conditions, then the Dirichlet boundary conditions. Whether we have to check for consistency or not, that has to be problem dependent and we further extended the problem to include various types of boundary conditions and also finally, the case where there is a distributed spring attached to the member.

In the next class, in the next lecture onwards, we are going to refine whatever we have done further. We will address the issue of how do we go from this setting to a setting which will be more amenable to a computer implementation and also how do we address the issue of improving the accuracy of the solution, not only through mesh refinement, but also by increasing the order of approximation.

Thank you.