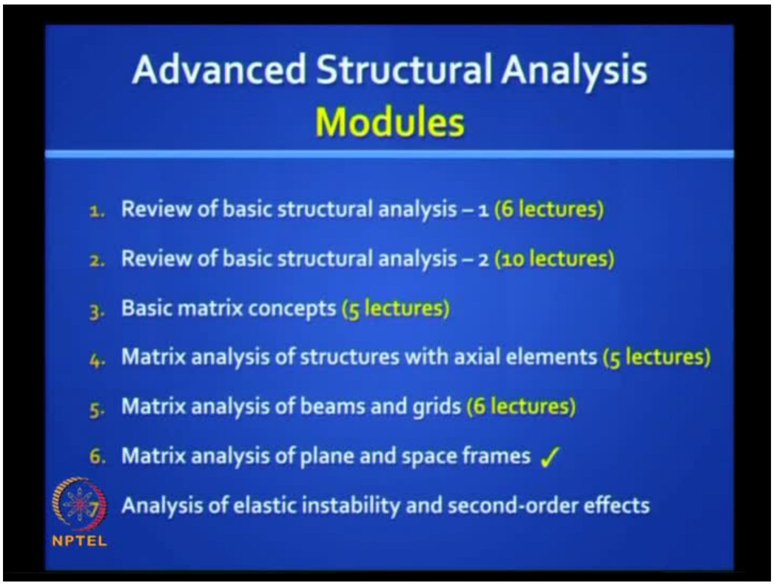
Advanced Structural Analysis
Prof. Devdas Menon
Department of Civil Engineering
Indian Institute of Technology, Madras

Module No. # 6.1
Lecture No. # 33
Matrix Analysis of Plane and Space Frames

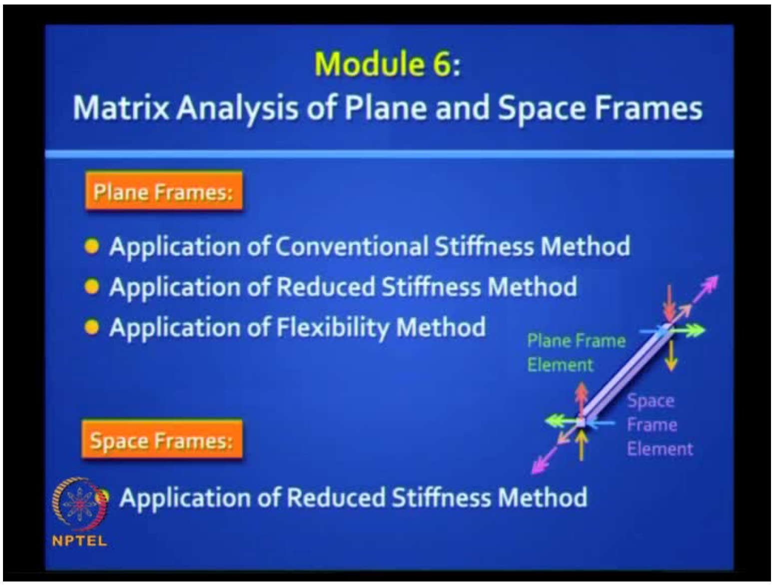
Good morning. With this lecture number 33, we are starting a new module -- module 6, which deals with the matrix analysis of plane and space frames.

(Refer Slide Time: 00:24)



If you recall, we have already covered 5 modules and you are now in a good position to quickly understand more complex structures like plane and space frames. The methodology is the same. And we have one last module left, which we will take up. It is a second order analysis and the study of the elastic instability of beams and frames.

(Refer Slide Time: 00:55)



The space frame element is the most generic, most complex element that you can get in matrix analysis. It is a very powerful element. As you can see, you have six degrees of freedom at each end of the elements, so we have 12 degrees of freedom, which takes care of bending about two orthogonal planes, vertical plane and horizontal plane. It takes care of torsion, which also takes care of axial force.

If there is a variation in bending moment at the two ends, you have a shear force coming into play. So the shear force can be in both vertical and horizontal planes. All other elements that we have studied are actually special cases of this one element. If you take the stiffness matrix for a space frame element -- 12 by 12, if you delete rows and columns, respectively, you can downgraded to a plane frame element to a grid element to a beam element to a space truss element, to a plane truss element, and to 1 degree 1 D axial element.

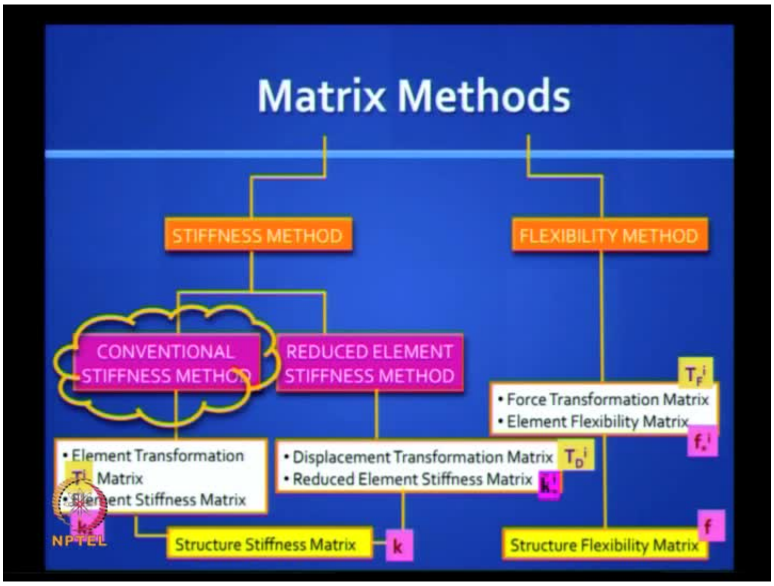
There, all special case of this element are as usual. We will learn to apply the conventional stiffness method, the reduced stiffness method and the flexibility method for plane frames. For space frames, we will just demonstrate the application of reduced stiffness method because it takes lot of space and time to do a big frame.

So right now, we will be looking at the application of conventional stiffness method to plane frames, using the plane frame element. As you can see, there are six degrees of freedom, three at each end. You have an axial degree of freedom, which you get from the truss element, the axial element, and you have the beam element with bending about the vertical axis, as shown here, and shear force.

It is quite easy. We just have to put together the stiffness matrix that you derived for the axial element along with the beam element. And we conveniently assume that there is no interaction between the two and you get the stiffness matrix for the plane frame element. This is covered in the chapter. It is chapter number 6 in the book on Advanced Structural Analysis, and we begin with the conventional stiffness method.

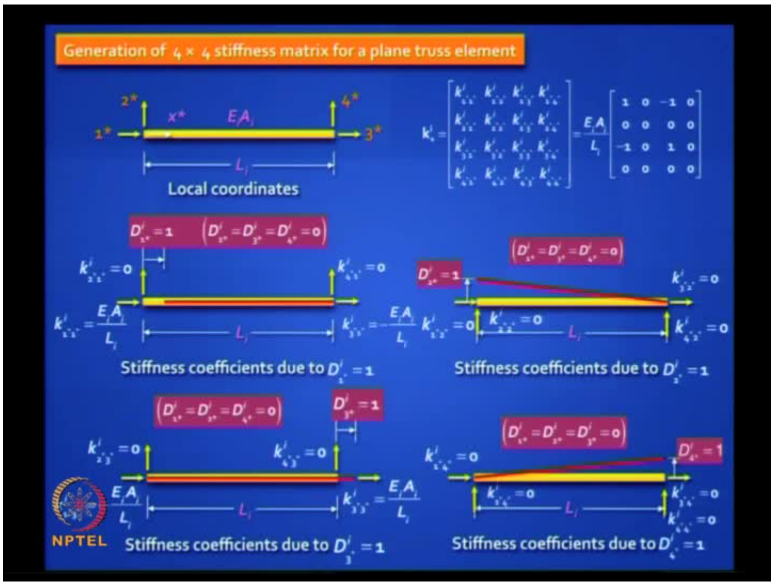
I have tried to show the same map in all the lectures that we have done, to show that we basically following very systematic way of solving problems. The conventional stiffness method is the method that is used in software programs commercially, which is used extensively in actual Structural Analysis.

(Refer Slide Time: 04:33)



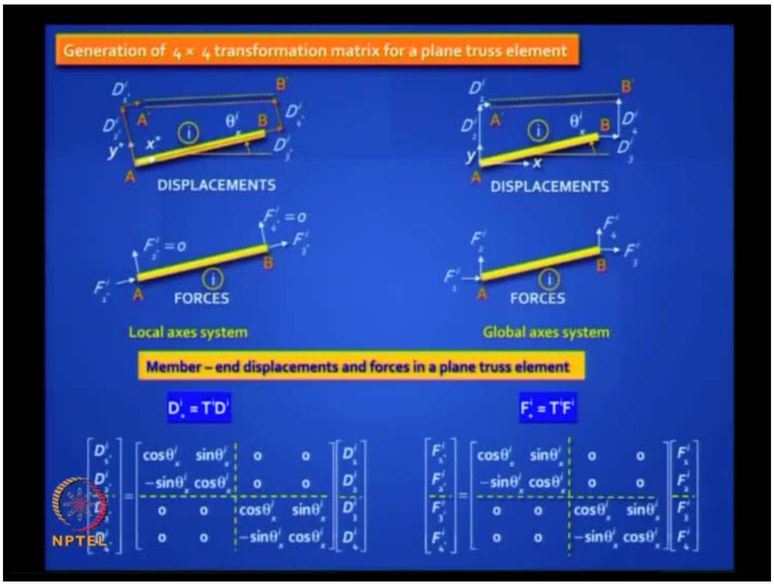
Reduced element stiffness method is very good if you want to program yourself. And some of your senior students have developed beautiful programs using MATLAB, which actually can do space frame analysis. I like to demonstrate this at some point, using matrix methods. Reduced element stiffness method can also be used and there is also the flexibility method, which on a case to case basis, you could use because it is an appropriate to use the flexibility method when the degree of static indeterminacy is very small in relation to the degree of kinematic indeterminacy.

(Refer Slide Time: 04:52)



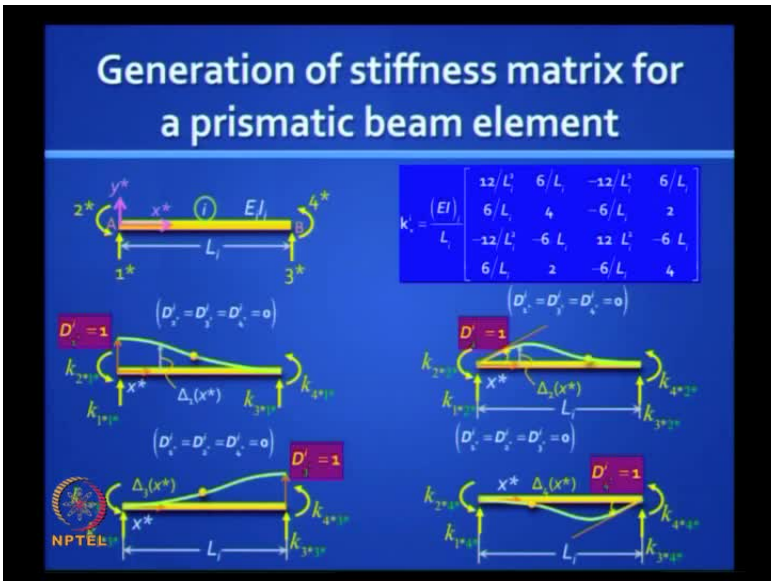
So I am just playing back to use some slides that we have already seen. You remember, we began with a plane truss element, which had a 4 by 4 stiffness matrix -- 4 degrees of freedom. It is very easy to remember the stiffness matrix. Two rows and columns will be in 0s and the others are simply EA by L and minus EA by L.

(Refer Slide Time: 05:45)



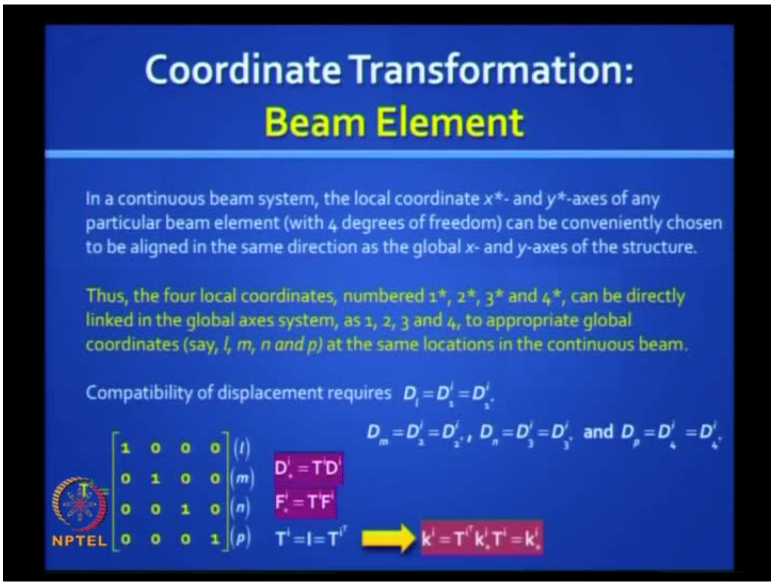
This is familiar to you. In this truss element, you will find that the 0 rows and columns correspond to the shear force. If you use it in a plane frame element, you have ability to take shear, so they are no longer going to be 0s. That is a big difference. Then, we looked at the plane truss transformation matrix. You remember, we used this theta sin theta transformation. We need to invoke the same transformation when we do a plane frame element.

(Refer Slide Time: 06:01)



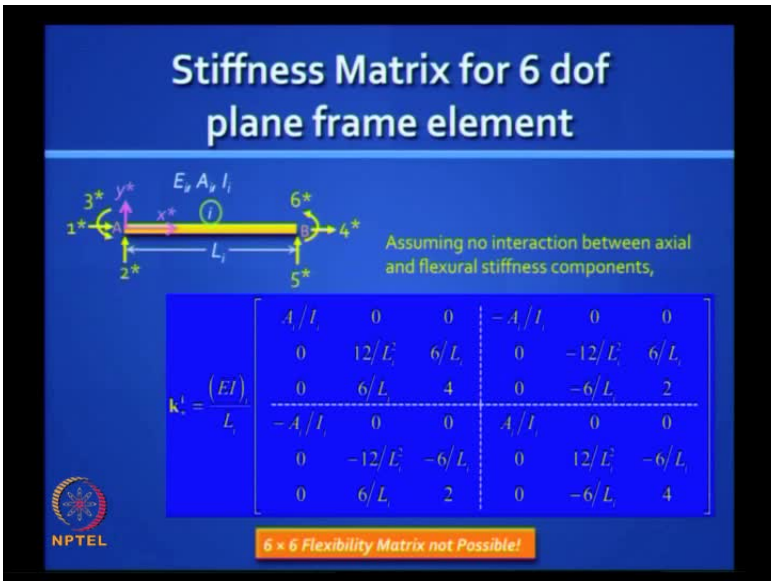
We next looked at the beam element. In the beam element, you had bending moment shear force at two ends, you had four degrees of freedom, and we used this element not only for the beams, we also used it for the grids. We are familiar with this. And we learned to generate the stiffness matrix using different alternative approaches, including the conventional displacement approach, where you assumed a displacement function using geometry, and then you worked with that and generated the stiffness matrix.

(Refer Slide Time: 06:40)



We also looked at the transformation matrix for a beam element. We found that the T i matrix is an identity matrix because conveniently the local axes can be made to align with the global axes. So, x is x star, which you cannot do in a frame. In a frame, the element can be orientated in any direction.

(Refer Slide Time: 07:06)



Now, with that back ground, it is quite easy actually to put things together and straight away derive the stiffness matrix for a plane frame element. Will you try it out? Write down the 6 by 6 stiffness matrix for a plane frame element with the coordinates as shown here. We have number these coordinates -- 1 star, 2 star, 3 star -- to align with the x y and z axis. 1 star refers to an axial degree of freedom; 2 star is a translation, a deflection which is normal to the longitudinal axis, the corresponding force is a shear force; 3 star is the rotation in the x y plane, which is with respect to the z axis. We follow the same directions and numbering sequence for the end node. So, you have 1 2 3 at the start node, 4 5 6 at the end node. Just write down the stiffness matrix assuming that there is no interaction between the axial degree of freedom and the flexural stiffness. No interaction between axial stiffness and flexural stiffness. If you do that, you will find that you just have to add one additional row or other two additional rows and columns, corresponding to 1 star and 3 star to your traditional beam element stiffness matrix.

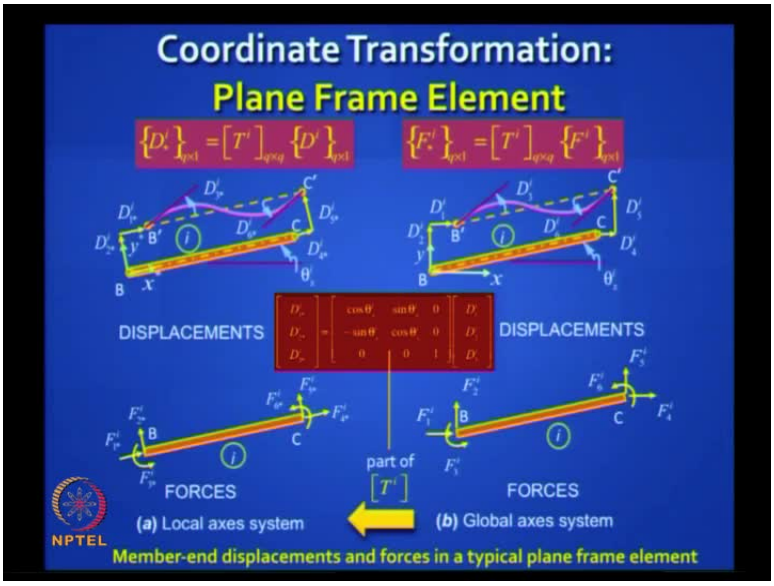
You remember we did something similar for the grid element. In the grid element, we added GJ by L, here you add EA by L. There is also one notable difference -- in the grid element, which we discussed yesterday, 1 star and 2 star referred to the flexural degrees of freedom and 3 star was a torsion degree of freedom. Here 1 star corresponds to the axial degree of freedom, so it comes on top. You can instead of writing EA by L, if you take out EI by L outside the brackets, then you would say A by I. Incidentally A by I is also 1 by r square, where r is the radius of gyration.

It is the only additional terms, you have, EA by L, plus and minus which is easy to add. It is not at all difficult once you are familiar with whatever we have done till now. Is it clear? So, we found that the plane frame element is essentially combination of the truss element and the beam element. Do you think there is some possible interaction between the axial and flexural stiffness components? Are they really independent, as assumed here?

We will study this in the next module -- module 7, where you realize that there is an interaction between the axial degree and the flexural degree. In fact, if the axial force is compression, it can be quite significant. If the compressive force is high, you have a phenomenon called buckling that is possible. If the axial force is close to the critical buckling load, the flexural stiffness degrades to what? To 0. That is why when buckling takes place, there is no flexural stiffness left in the beam. Which intern, suggest that axial compression can actually reduce your flexural stiffness, and conversely, axial tension can enhance your flexural stiffness. But these are second order considerations. They do not come in the realm of first order structure analysis. In this module, we are doing first order structural analysis, we do not look at what are called p delta f_x. We conveniently assume that these two stiffness components are uncoupled, so that there is no interaction between them.

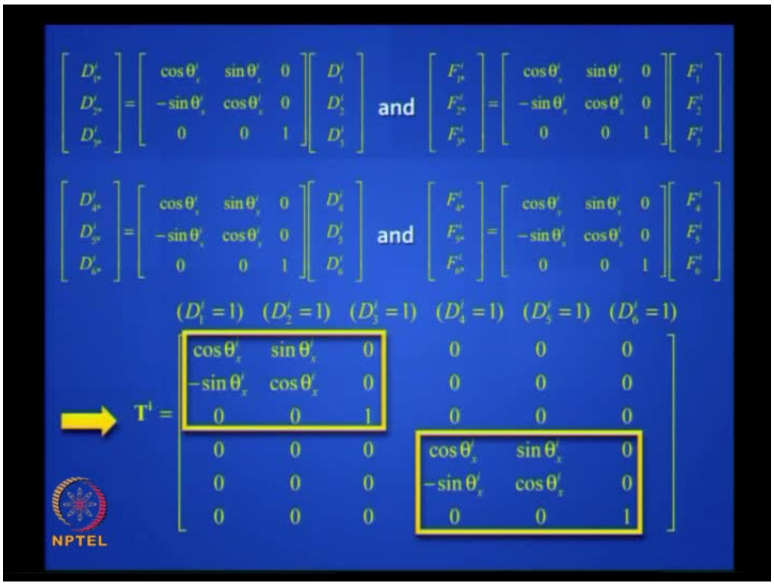
This is reasonably true if you are dealing with well proportion members, which are not slender and your axial forces are not very high -- not close to your critical buckling load. The other interesting thing to notice the rank of this matrix is not full. What is the rank of this matrix? 3, because the element itself is physically unstable. To hold it in place, you need to arrest 3 degrees of freedom and we will do, when we do the reduced stiffness method. That is why this matrix is singular, it cannot be inverted. There is no flexibility matrix possible by inverting this 6 by 6 matrix. You have to reduce it to a 3 by 3 matrix, which we will do in the reduced elements stiffness method and the inverse of that is the flexibility matrix.

(Refer Slide Time: 12:37)



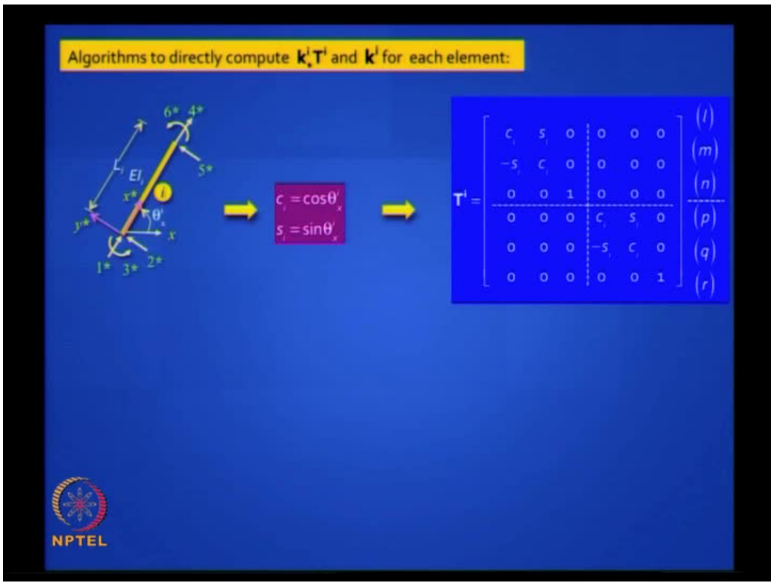
As far as a coordinate transformation is concerned, I am showing a picture which I have showed in module 3, where we tried to show the relationship between the degrees of freedom expressed along the global axis, with respect to those expressed along the local axis and that transformation is T i. If you take one of those ends, it forms a familiar transformation, the rotation matrix, where you have cos theta minus sin theta cos theta sin theta, and you have one corresponding to the third degree of freedom because there is no need to do any transformation with respect to the z axis, because we are talking of rotation about this z axis.

(Refer Slide Time: 13:39)



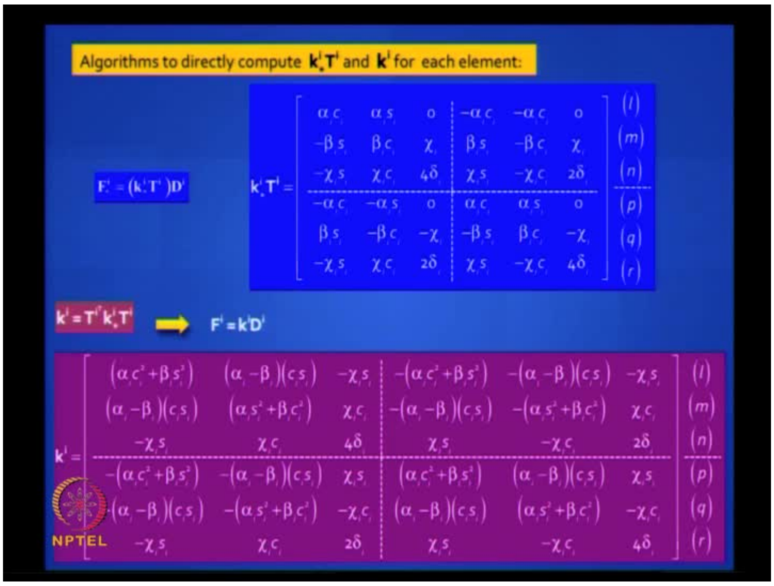
I hope you are familiar with this. Knowing this, you can easily write down your transformation matrices at the 2 ends. Put it together and you have this symmetric T i matrix, where each of those blocks in the main diagonals are familiar to you. So we have to use this, it is familiar to you. So T i matrix is no problem and the inverse of this matrix is the transpose because it is an orthogonal matrix.

(Refer Slide Time: 14:32)



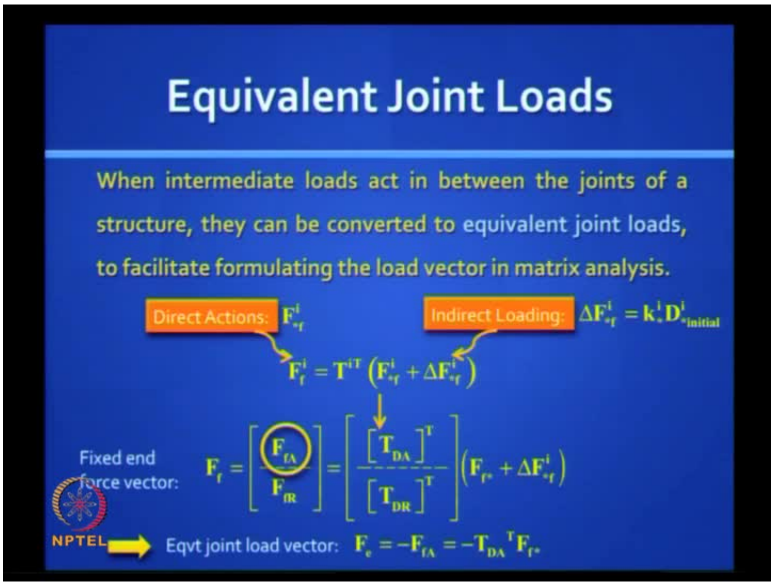
You also need to indicate the global coordinates in parenthesis, as we did earlier. This is familiar to you. This is a path we will take to do transformations. You will find that if your programming, it might be beneficial to do it in a systematic manner. For example, for each element if you pull out the direction cosines, in terms of theta i, and you call them c i and s i and if you know the length of the element, you can generate some properties straight away. This is a typical transformation matrix for any element in your plane frame. Once you input the properties, once you write the coordinates, it can generate this automatically for all the elements. It is important to note that you must also put the linking coordinates, which I have shown here -- l m n p q r and it is very easy to assemble the stiffness matrix.

(Refer Slide Time: 16:00)



From the properties, you can write an algorithm like this, in terms of alpha beta zeta and delta. And if you want program, it will do the stiffness matrix generation for at the element level effortlessly. You can go one step further and do this product multiplication. You can feed in these coefficients, these elements of your k i T i matrix because this is necessary for you to get the internal forces. You can go one step further and pre-multiply this matrix with T i transpose and you can feed in this, in terms of the geometric and material constants that we had expressively. If you are interested in programming, you can directly feed in these values but you can also allow each multiplication to be done separately. The choice is yours.

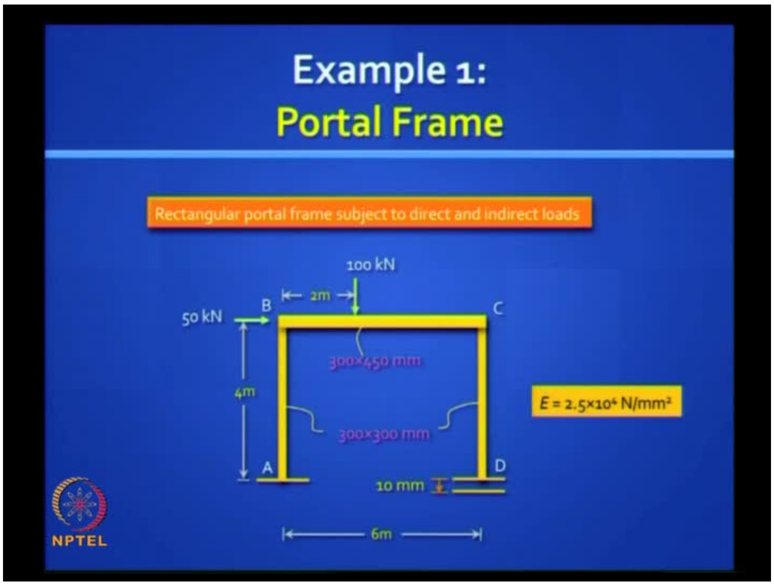
(Refer Slide Time: 16:34)



We also looked at equivalent joint loads. Now, when we did trusses, the equivalent joint loads came not from nodal loads and not from distributed loads but from lack of fit and temperature effects. When we did the 1-D axial element, you also had the possibility of intermediate loads. When we did the beam element, we did not have temperature loads but we had intermediate loads. We also had support settlements, which you also had in the truss.

In the frame, you can have anything. You can have a mixture of everything. You can even have temperature effects. So we will take a look at the large variety of problems that you can get. You have to follow the same procedure, you have to find the fixed end force vectors, you have to include any additional fixed end force vectors that you get from initial displacements. You have to do this transformation to slots your fixed end forces, along the global coordinates and you have to work out the equivalent joint load vector. Procedure is same, map is same, territory is familiar.

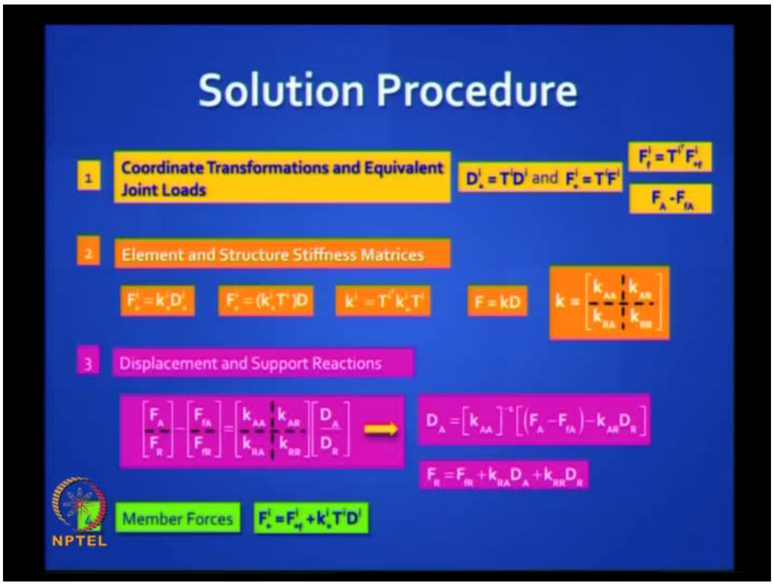
(Refer Slide Time: 17:50)



We will do two problems. I hope we have time to do two problems or at least one problem exhaustively in this session. Take a look at this plane frame. It has got all the familiar complications -- you have an intermediate load there, 100 kilo Newton, you have a nodal load there lateral load of 50 kilo Newton. We will also throw in a support settlement of 10 millimeter. The E value is given, I value is not directly given. It is given in terms of the cross section -- 300 by 300. The 2 columns are square and the beam has a depth of 450 mm and a width of 300 mm.

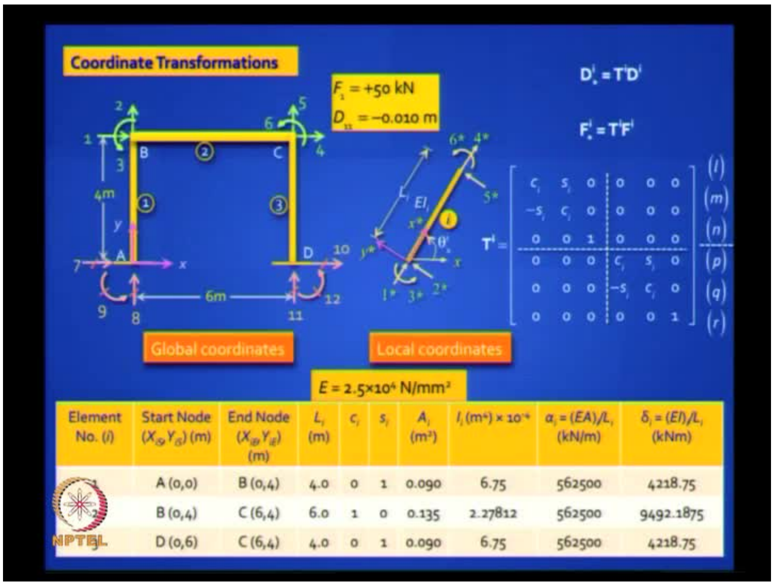
It is a typical singles bay portal frame. Let us learn how to analyze this. We should know what the displacements are, at least at the joints. Have maximum displacements. But more important, we must know, are the support reactions. We must know the bending moment diagram, shear force diagram, the axial force diagram, which is all easy once you have the free bodies. So how do we proceed?

(Refer Slide Time: 19:06)



This procedure is very familiar to us. We are following exactly the same steps. First, the coordinate transformations fixed end force vectors equivalent joint loads, next the element in structure stiffness matrices, next we write down the equilibrium equations and in this case, the support settlements are there. So you have to bring in that into solving those two equations. You solved the first equation, you find the unknown displacements at the active coordinates to plug in those values in the second equation. You get the support reactions, and finally what is a last step member forces. There you are. So, it is very familiar. The member forces are nothing but your equivalent slope deflection equations. You have the fixed end forces and you have the additional end forces that you get from the joint displacements. So I hope, now you are comfortable dealing with any problem because the procedure is well laid out. You just have to do your transformations properly.

(Refer Slide Time: 20:12)



You will find that a conventional stiffness method is actually very straight forward. Not much thinking to do. It is a reduced stiffness method, which can be tricky because you are taking shortcuts in reducing your degree of kinematic indeterminacy. Personally, I think that is a real challenge if you want to do things manually and write your own programs. Flexibility method, similarly, is very challenged. But conventional stiffness method, except when you have local complication like internal hinges of very straight forward systematic, you cannot go wrong. Let us demonstrate this.

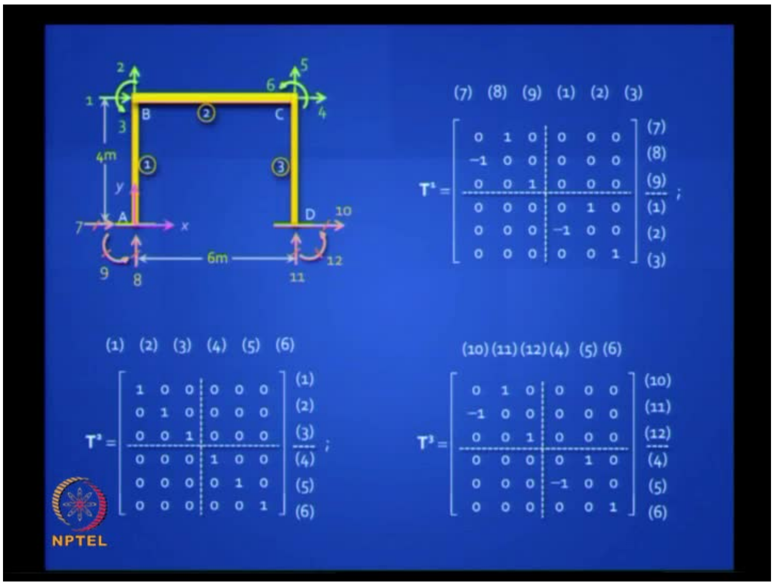
First, we have to identify the coordinates the global coordinates. As usual, we will start numbering from the active degrees of freedom. Since the ends A and D are fixed, only B and C have the active degrees of freedom, since we choose the origin at A with x pointing towards the right and y pointing towards the upper region, we follow the same sin convention -- 1 2 3 at joint B and 4 5 6 at joint C. Those green colored arrows are active degrees of freedom. We also have restrained degrees of freedom and they are 7 8 9 10 11 12, following the same sequence. And once we know those, we know the support reaction is 1.

What is the input data given to you? Do you have any nodal forces? Yes. You will find that F 1 is plus 50 and D 11 is minus 0.010 meters, rest are all 0s. You also need local coordinates, so instead of writing three separate figures, we can write one common figure for all. This is a standard picture, which will show in all plane frame elements.

In this particular problem that angle theta with respect to the global x axis is either 0 degrees for element 2 or 90 degrees for elements 1 and 3. But in general, it looks like this -- 6 degrees of freedom. This is the standard T i matrix, the transformation matrix, where you shift from global axis to local axis. It is easy to generate this if you make a table, which is what I suggest you do, whenever you deal with any frame. You are shown a single story single bay frame, you can have a 100 story 20 bay frame, it makes no difference.

You have to systematically follow this procedure. You have to identify your elements, you have to identify the start nodes and the end nodes. You have to write the coordinates of those start and end nodes, which is what we did when we did the plane truss. The plane frame is similar. You need not explicitly calculate those theta values, you do not need the theta values, you need the directions cosines cos theta and sin theta, which is very easy to generate from the coordinates. Even the length comes from the coordinate. We have done this before so and go ahead. You can also, in this table, put in your EI values and your EA values EA by L and so on. All that you need to generate your T i matrix and your stiffness matrix. So, the moment you give the input, you can also, if you writing a program, ask it to generate a visual picture of the structure, so that you can verify at one glance whether you missed out some element is miss located, which you can do when you do your programming. In fact, all software programs do that. They also have the facility where you do not do all the steady business, we actually sketch a picture and it will generate this stable on its own. So, those are all tricks that you can carry out. We are not studying those tricks here. We are just studying how this black box works, what is the algorithm inside it, and can we do a minimal amount programming, and we able to generate make the computer do structural analysis for us. It is a systematic method -- it follows certain laws and it is easy to understand at this stage.

(Refer Slide Time: 25:17)

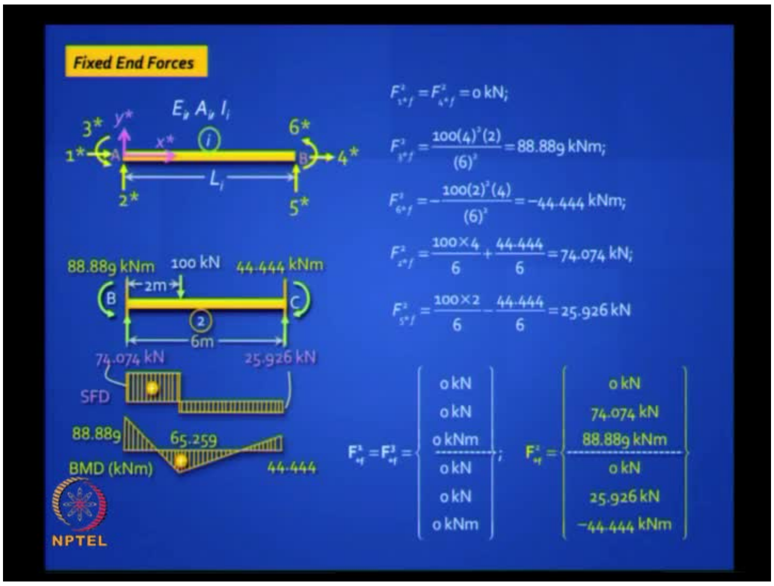


It is very easy with that table. You can generate the Ti matrices for the three elements – T 1, T 2, and T 3, where theta is 0, cos theta is 1; where theta is 90 degrees, cos theta is 0. So these are very straight forward. Very easy to generate, if you look at the element 2 – T 2. It is like a conventional beam element and that is why it is an identity matrix. It is like a beam element

The columns, all frame elements, the linking coordinates are very clear. For the second element, it is clean 1 2 3 4 5 6 because they are all active degrees of freedom. For the first element, it begins with 7 8 9 and then 1 2 3. And for the third element, it is 10 11 12. We have to start the start node, it has is at D, not at C in this particular case or you can choose your own sign convention -- 10 11 12 and 4 5 6.

Remember when you do a large frame, it is better to follow those suggestions, we gave in the third module of reducing your band width by numbering it along the shorter direction. But we are not looking into those accepts at this stage because we are dealing with very small frames. It does not really matter when you are dealing with a small frame, you can do this easily and you can program it.

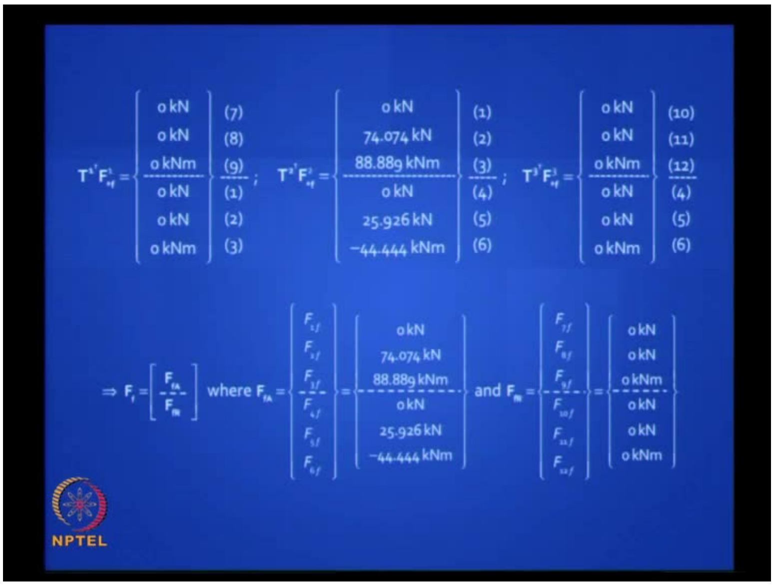
(Refer Slide Time: 26:33)



Next job is to find the fixed end forces. What are the fixed end forces? It is easy to generate. It is a beam element you have. Of the 3 elements, only element 2 will have fixed end forces because element 1 and 2 have no lateral loads on them. You have a concentrated load located 2 meters from the left end. You know the formula -- w a b square by l squared -- put the signs correctly, and work out your vertical reactions. You do not need to draw the shear force and bending moment diagram but if you wish, you do that.

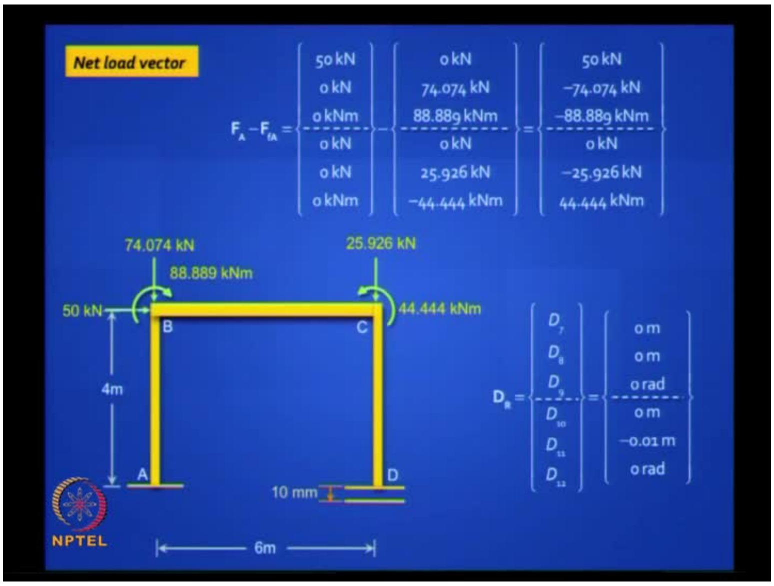
You can pull out the fixed end force vector for that second element. And for the first and third elements it is going to be null vectors. Nothing new. This is what we have been doing for the beam and the truss and the axial element. So you got this? What is the next step? You have to switch from local to global. How do you do that?

(Refer Slide Time: 28:06)



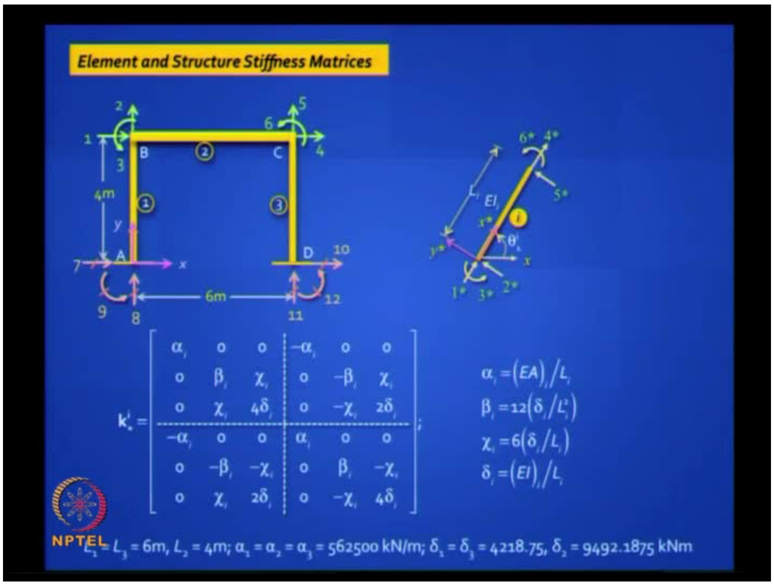
You just have to pre-multiply with the transpose of the corresponding -- T i matrix -- and you put alongside the linking coordinates. You know the linking coordinates, so you first do that product. And after you have done that, you have to do the slotting. That means, whatever you get in coordinate one from the 3 products, you add up algebraically. After, get the answers, you can just go back and check and see if it make sense because visually you can inspect and see whether it make sense. And you will find that in this case because only the top beam, the horizontal element has loads, nothing is happening at your F_{fR} level. There is no fixed end force going to your restrain coordinates.

(Refer Slide Time: 28:46)

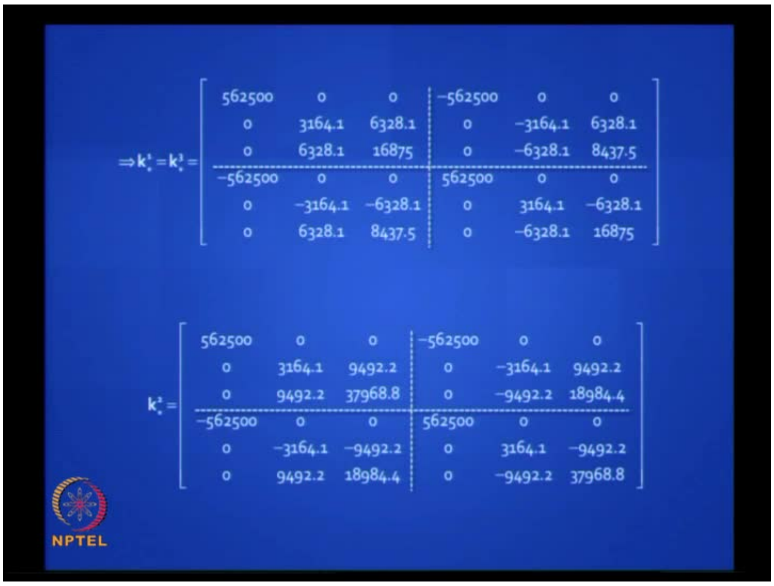


You can also draw a sketch after you find the net load vector. Net load vectors F_A minus F_{fA}. F_A has only one nodal load. F 1 equal to plus 50 kilo Newton. You do this product and draw a sketch. You have actually converted your original problem to this problem and the equivalents is -- we all have the same D_A vector, the active displacement vector. You will also notice that, in addition to the forces, you have support settlements, which come from the D_R vector. You have D 11 having a value of minus 0.01 meters.

(Refer Slide Time: 29:31)

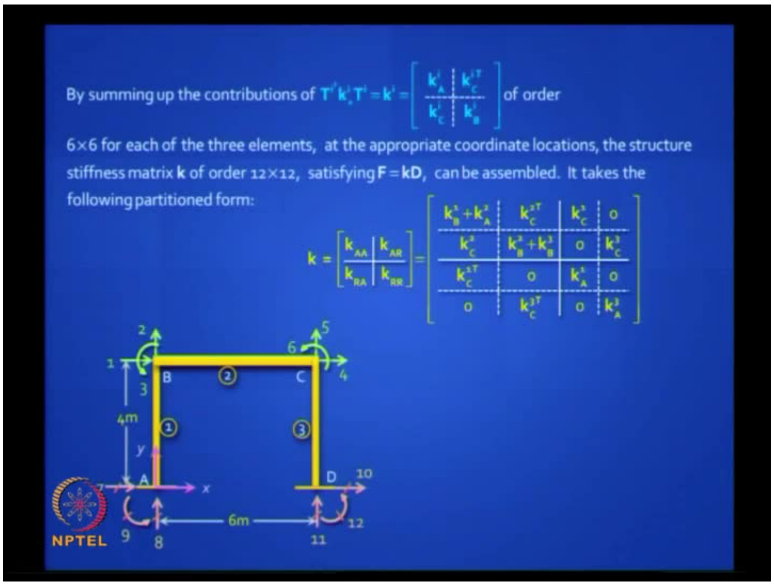


(Refer Slide Time: 29:49)



Next, you generate the element and structure stiffness matrices. Actually, you can do this earlier because the computer does not wait for the loads. It straightaway does all these. These are properties of the structure, you can write this algorithm. We have already plugged in those values in the table so in a (( )), it will generate all this. There is nothing much in it.

(Refer Slide Time: 30:31)



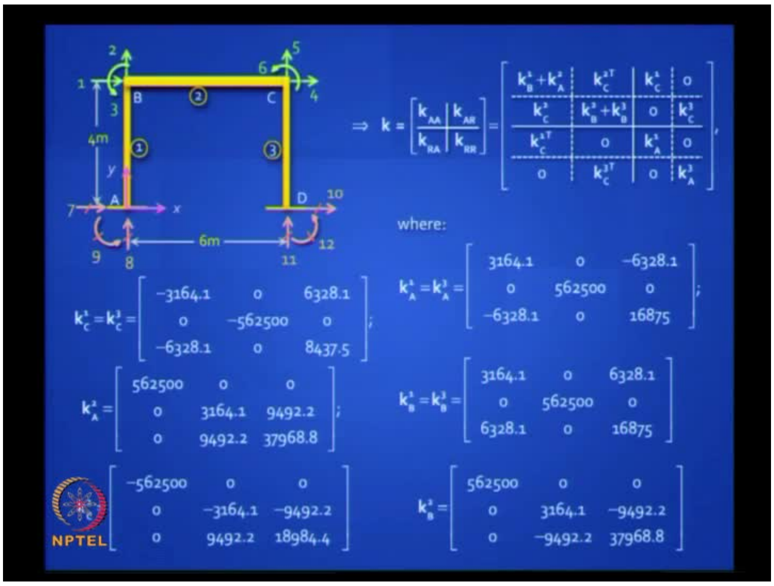
Once you have done this, your next job is a little tricky. What you need to do now? You have to assemble all these matrices by first converting them from local coordinates to the global axis, and then slot it. That takes a little thinking. By summing up the contributions of T i transpose, k i star t i, you get k i. k i is the same stiffness matrix, realigned along the global x y and z axis. It will typically have the coordinate format as shown there. You can say, it is k_A, k_B, k_C, and k C transpose. On the other side it is going to be symmetric.

So I can generate for each element these three values -- k_A, k_B, k_C, and I put a superscript i to identify which element is where. Now, I should do my slotting very carefully. The slotting comes from understanding the linking coordinates. For each of them, I have a 6 by 6 matrix, and at the appropriate coordinate locations the structure matrices matrix k of order 12 by 12 satisfying F equal to k_D can be assembled. It takes this form. Now, look carefully at that form. The coordinates 1 2 3 will be affected by which elements? By 1 and 2.

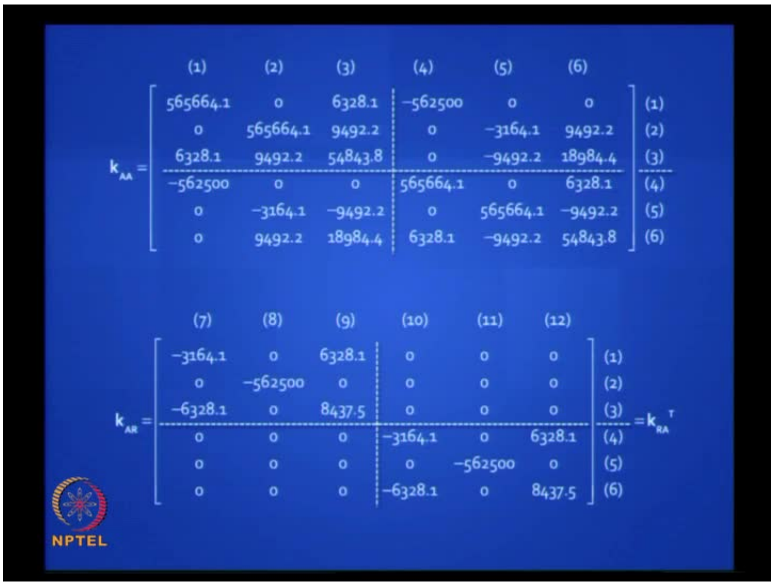
It is going to be affected by the tail end of 1 and the start end of 2. That is how you write k_B of 1 plus k_A of 2. That is a clever way of doing it. Now, you can program it to do this automatically. But if you are doing it by inspection, you have to do it carefully.

You need to assemble this with some care. And would you like to do an assignment -- one problem of this type, so that you get a feel for it. Your last assignment you will do this. It is going to take time if I am going to explain this by myself but I would like you to generate it make sense of this, and generate that matrix. It should be symmetric. So, it is easy to generate one side of it.

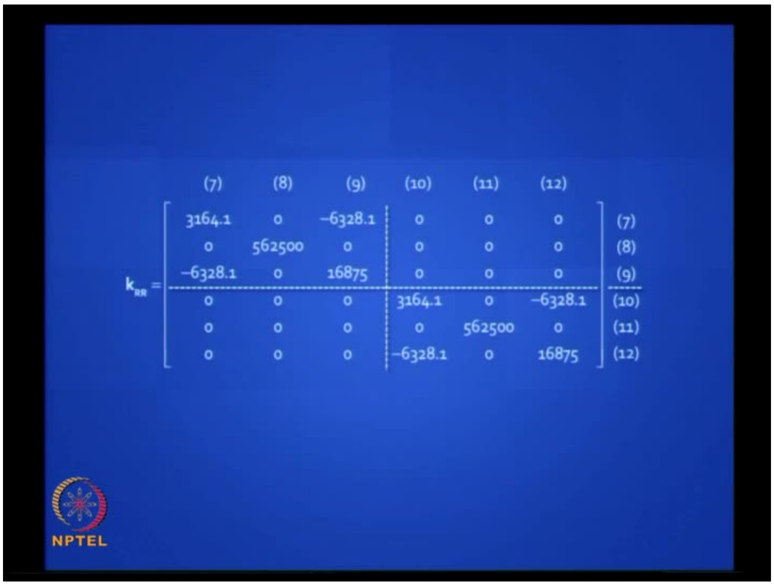
(Refer Slide Time: 32:57)



(Refer Slide Time: 33:00)

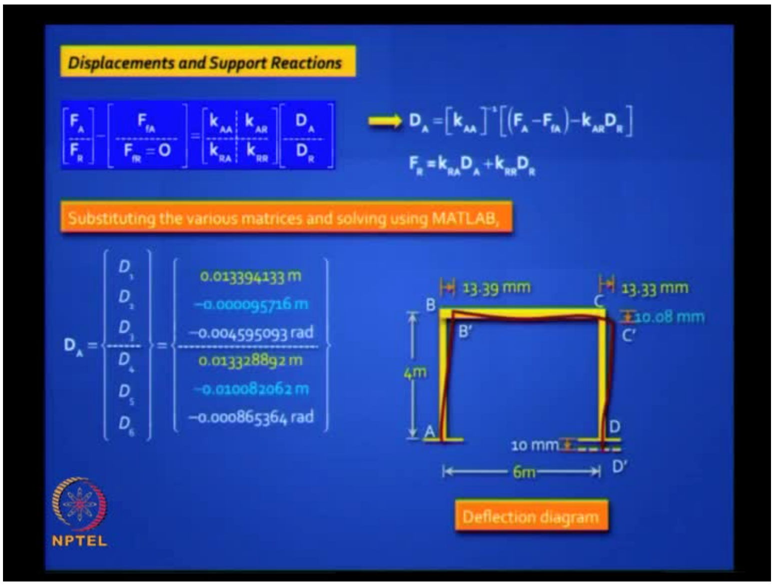


(Refer Slide Time: 33:08)



You can generate this from all those three matrices. So this is k_A 1 k_B 1 k_C 1 k_A 2 k_B 2 k_C 2 k_A 3 k_B 3 k_C 3, which once you assembled, you put all together and you get the full matrix -- you get k_{AA} k_{AR} k_{RA} transpose and k_{RR}. So, you got the full structure stiffness matrix but it takes a while. You do not attempt doing this manually. It should be done through MATLAB. May be to make your life simpler in your assignment I will give you -- a 2 bar 2 member, I think. So, you have to add only for 2 members. Let us see how you do that.

(Refer Slide Time: 33:48)

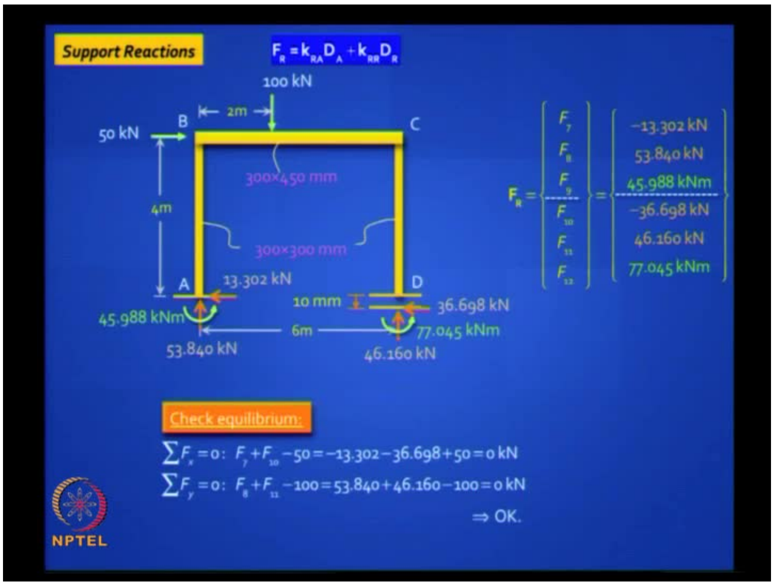


Make step is very clear. You have got your structure stiffness matrix, you have got the transformations, you got the net load vector, you have this familiar equilibrium equation and you plug in those values. And you can find the active displacements by solving that equation -- you k_{AA} inverse. The computer can do it for you. You have a MATLAB inversion program, which can handle it your matrix is well condition. So you are pretty well assured of the results that you get.

It is nice at this stage to look at those numbers. Do they make sense? Is that the kind of values that we get? Do not get scare looking at them. Try to draw the deflected shape. It will look like that. And we will find that everything make sense. D_1 tells you that joint B is going to move to the right by 13.39 mm. That is okay. 13 mm, is fine for a big structure like this.

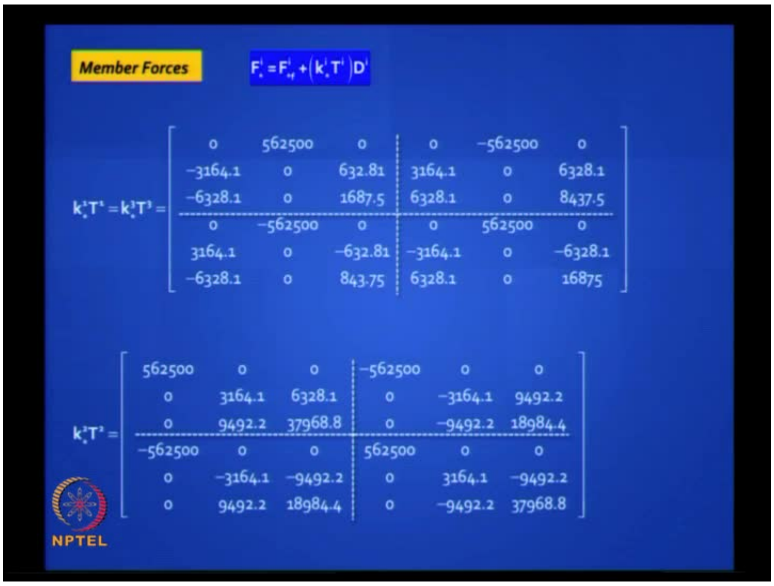
The right end will move 13.33. The little differences is because of the axial deformation in that member and bit of rotation. Joint D does not go down at all because at far too many 0s there in that but you will find that C goes down and that corresponds to D_5 goes down by 10.08 mm, which make sense because your support is going down by 10 mm at D. So, this is a kind of feel you should get once you get the output. Do not just get the numbers and see I have got just give me full marks for what I have done. See what you are doing. Understand the physics behind the problem. It should sway the way you expect to sway. So, that is how the displacement vector should be interpreted in practice.

(Refer Slide Time: 35:47)

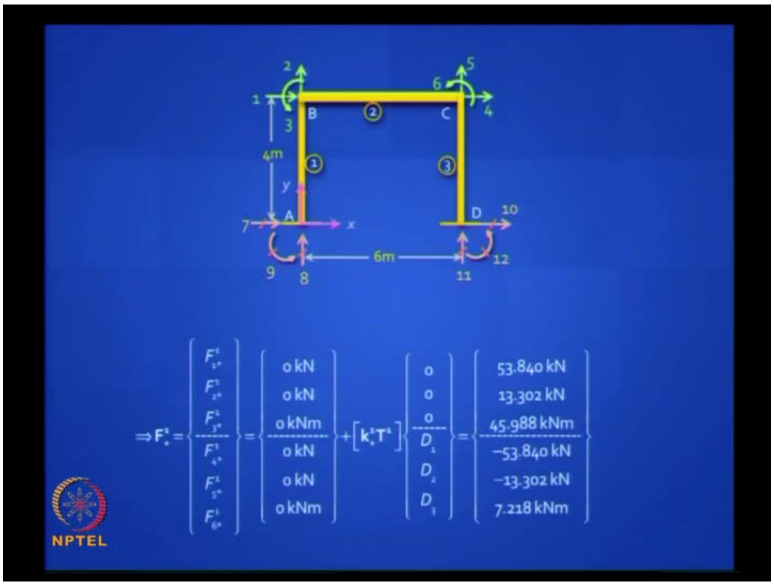


Then, your next step is to get the support reactions, which you can get by solving that equation. You plug in those values and see whether it make sense, see especially whether you are satisfying equilibrium. That check you can do. You need to do many checks including the moment equilibrium check, but at the very least, you can add up the forces in the vertical and horizontal direction. It should all add up to 0 then you say, at least my solution satisfies equilibrium that is for sure. Hopefully, it also satisfies compatibility. It will because the stiffness method begins with compatibility and equilibrium is the final solution.

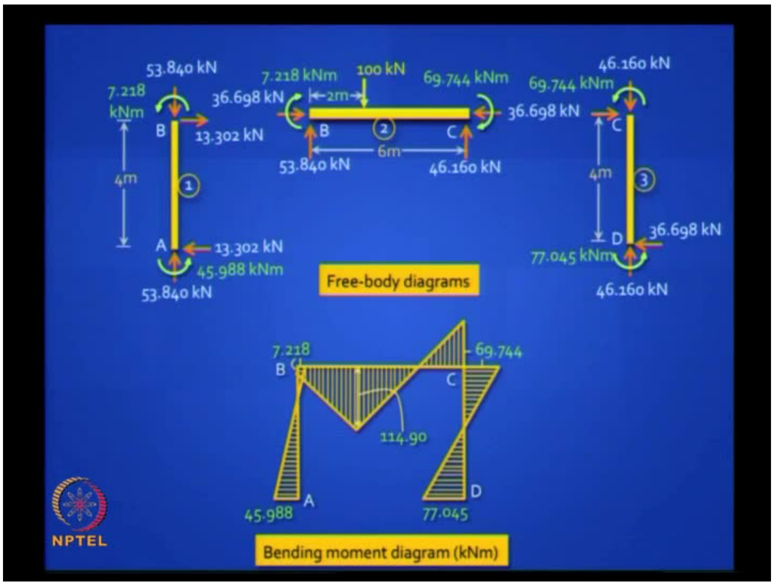
(Refer Slide Time: 36:28)



(Refer Slide Time: 36:49)

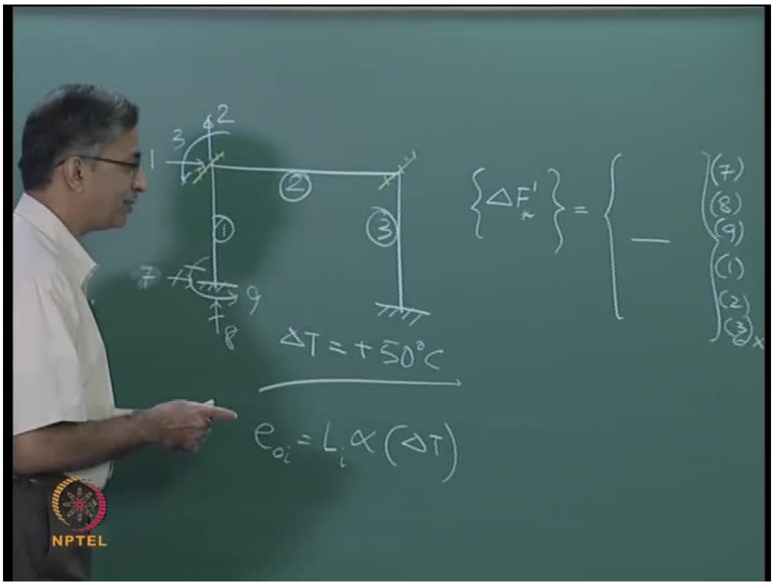


(Refer Slide Time: 37:03)



Then your next step is to find the member forces, which you can do you. Already have these computed and stored in your memory the k i T i. You plug in those values, you get the member forces for all the three elements. Then you draw free bodies. When you draw the free bodies, you will find you are now in a position to draw the bending moment diagram, and if you wish the shear force diagram, etc. As easy as that. So, we have done 1-plane frame element, solving by this method. Looks like, we have finished early, so it is a good time for you to raise some questions on whatever we done till now. Do you have any questions? We would able to go fast only because we have travelled quite a distance to reach this stage. We have done the truss element, we have done trusses, we have done beams, we have done grids and what now? We are doing frames. Let me ask you a question. Take that same plane frame. Let us subjected to temperature loading -- let us just heat it up. How would you solve that problem?

(Refer Slide Time: 38:23)



Let us say the temperature is increased by. You can have seasonal variation of temperature of this order. How would you deal with this problem?

(( ))

Fixed end moments. How do you get fixed end forces? There are no forces given.

Minus (( ))

Sorry

Point b star

Let her answer.

We find the changing length and force we deduct.

What kind of force will you get? You find the change in length. You have elements 1 2 3, the change in length will be used as a notation e naught – e naught i will be L i alpha delta T. You got this, now what? This is the free elongation, if it is allow to freely elongate.

(( ))

So you should say that

(( ))

I take the primary structure. In the primary structure, all these ends are restrained artificially. In the primary structure, you have a temperature raise. You end up with axial forces in all those members which you can calculate. Then what? That is what you did in a truss remember.

(( ))

Added to what? there is no other loading.

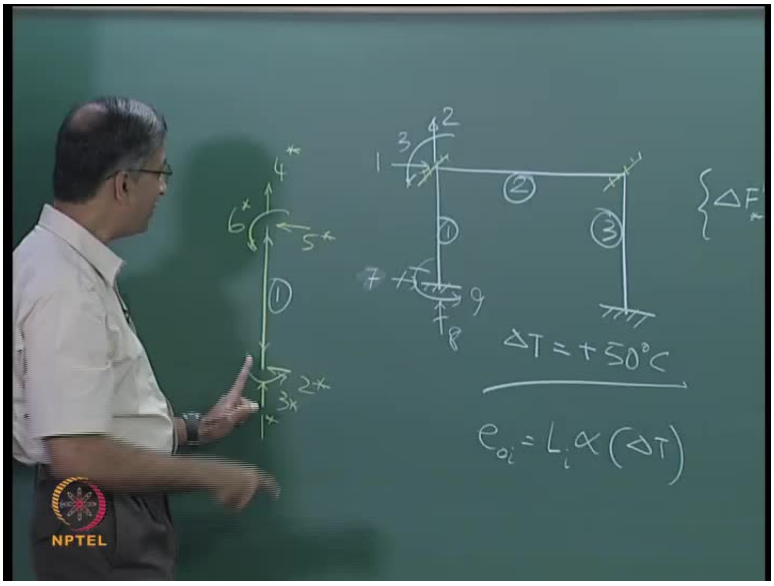
Sir that is the vector (( )) because of the elongation there is an actual component that is the first order.

So, we are saying, you have a delta F vector at the element level. Let us do it for element 1. What will it look like? For element 1, what will be the size of this element?

6 by 6.

It will be 6 by 1. What do you write for the left end? This had if you remember, 7 8 9 and 1 2 and 3.

(Refer Slide Time: 42:11)



The linking coordinates are 7 8 9 and 1 2 3, and the element itself had 1 star 2 star 3 star 4 star 5 star 6 star. How do you fill up this? What is the first element? We are looking at an element like this with this as 1 star, this as 2 star, this as 3 star, 4 star, 5 star, 6 star. This is the element 1. This value that you get, what do you do with that? What should I write?

EA by L.

Is it plus or minus?

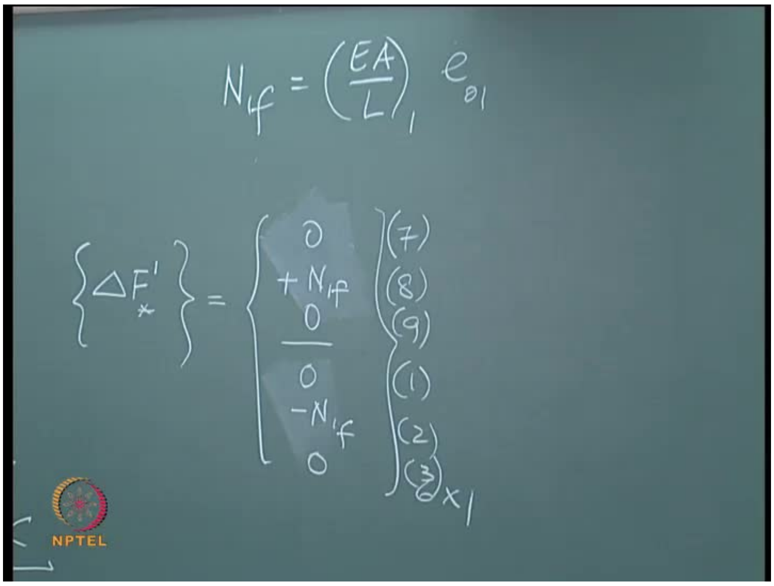
Minus.

If you heat this up, it is going into compression. So, what are the end forces that you get? This will be positive or negative?

Positive.

It is going to be positive. It is going to compression means you press it down like that, so this will be positive and this will be negative.

(Refer Slide Time: 43:32)

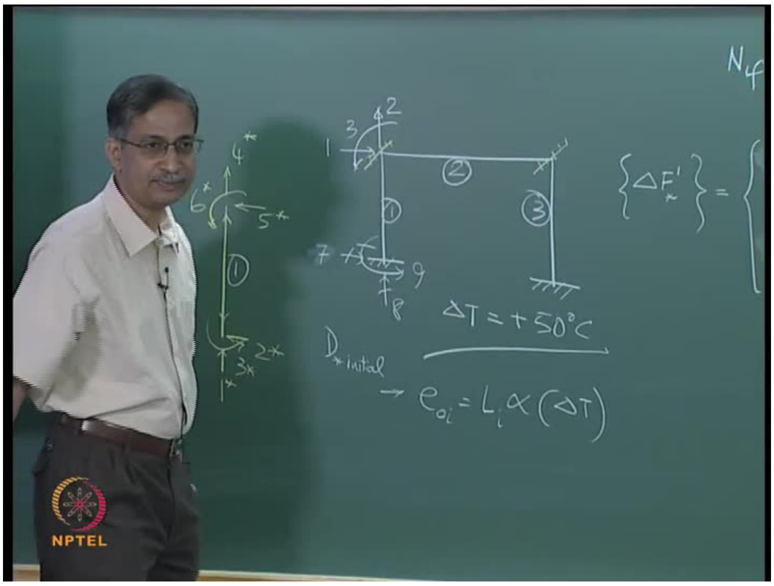


What would this quantitative? This will be positive and this will be negative. Let me say, plus N 1f, minus N 1f, where N 1f will be

(( ))

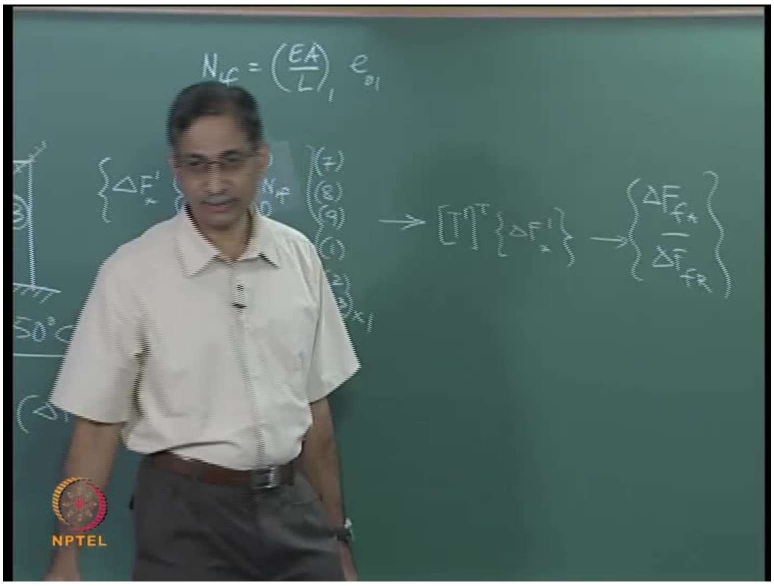
EA by L for the first element primes e naught 1. So you will do that. What do you get here? 0. Here? 0. Here? 0. Here? 0.

(Refer Slide Time: 44:36)

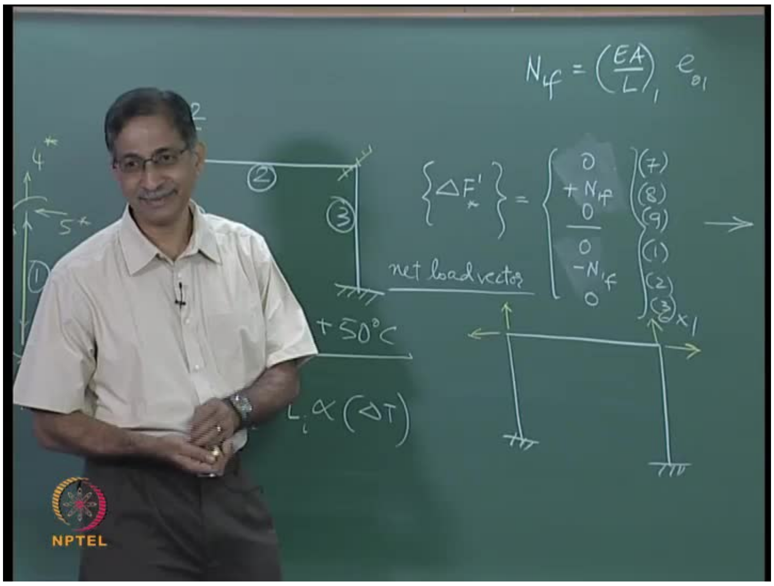


Likewise, you can generate this vector for all the members. In fact, we used a word called delta star initial for this. You have done that then what you do?

(Refer Slide Time: 44:54)



(Refer Slide Time: 45:34)



You now need to convert this into the global coordinates by pre multiplying it with T_1 transpose. And what do you get? If you add up for all the elements, you will end up with delta F_{fA} delta F_{fR}. So, you have got nodal forces, you have got equivalent joint forces, and you find that you actually going to analyze a structure with what kind of forces? You have to apply it in the negative direction. What is your net load vector going to look like?

What is it going to look like? Show me here. Do you have force acting down? Yes. We will be acting down or up?

Down.

No, you had compression acting here, you have to reverse it because it kind of had to hold it down. You have to let go. So will it act down or up?

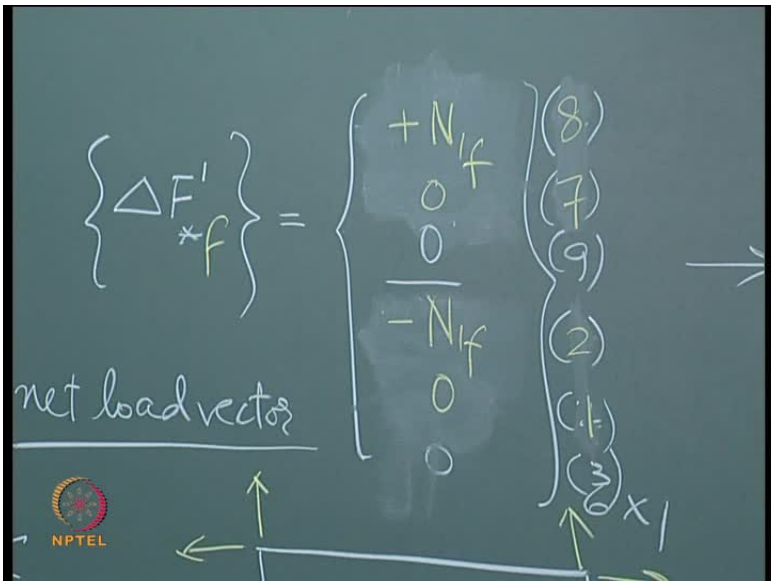
Up.

It is going act up, so you have a force acting up here, acting up here, what about here? This wanted to expand, you held it back. Finally, a plus. You had some support reaction, so you actually analyzing a structure like this and will this cause bending?

Yes.

For sure it will because if this moves laterally, this does not move. You got a chord rotation, so you will have a curvature. You will have a bending moment diagram. It is a very interesting problem. One which I wish you will solve on your own. So, the subject is beautiful, if you can link here left brain with your right brain. Understand the physics of the problem and you have got a powerful tool to handle any kind of loading on any skeletal structure. Thank you.

(Refer Slide Time: 47:55)



(( ))

I am glad you raised this point. You are right; we have to correctly put the linking coordinates. As far as the element is concerned, 1 and 4 are always axial degrees of freedom. So you are right. This should find a place here and this here, because this corresponds to 1 star, this corresponds to 4 star, and this incidentally is F star f. So, the mistake we made was in putting the linking coordinates. 1 star actually matches with 8 star, not 7 star; and 2 star matches with 7 star. That is a correction we need to do. This will be 8, this will be 7, this will be 2, and this will be 1. You strictly follow the fixed end forces at the element level and place the linking coordinates which come with your T_i matrix; be careful. Thank you.