Retrofitting and Rehabilitation of Civil Infrastructure

Professor Sriman K Bhattacharyya
Department of Civil Engineering
Indian Institute of Technology Kharagpur

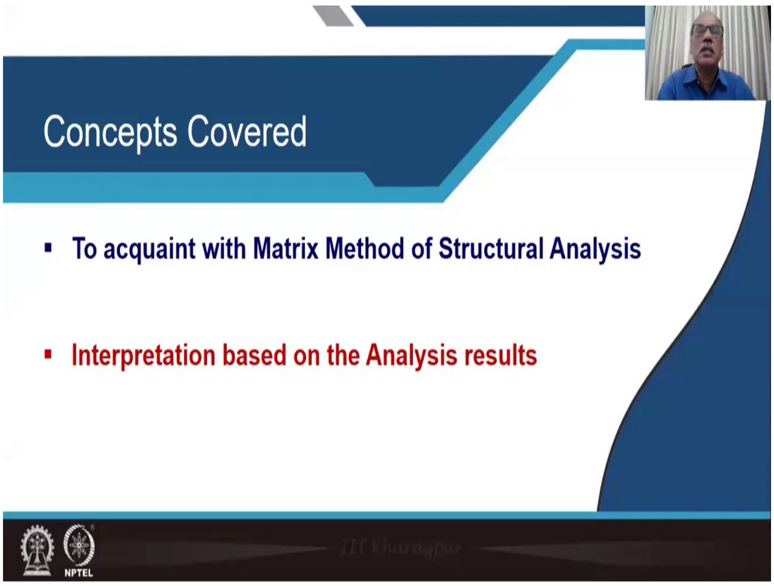
Lecture 47

Review of Analysis Methods

(Refer Slide Time: 0:43)

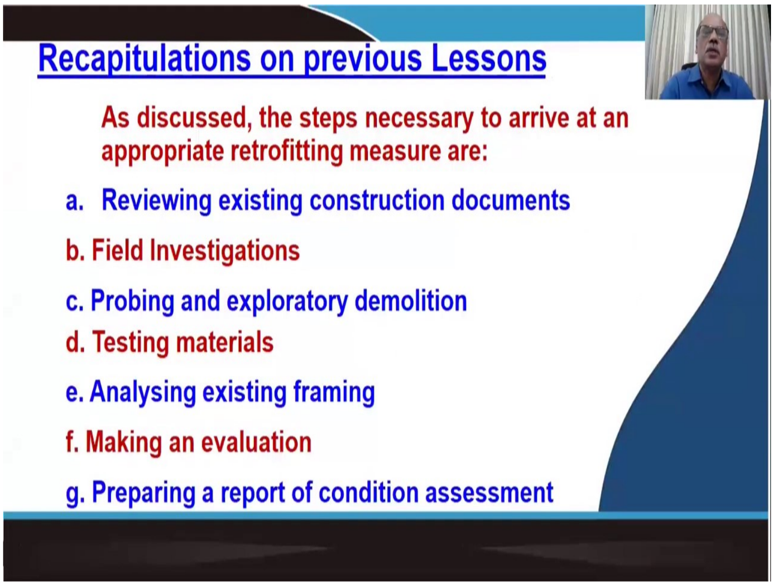
Welcome to the fourth lectures the module on overview. And in this particular lecture, I intend to discuss with you on the aspects of analysis methodologies that we have spoken about in our earlier discussions, where we have stated that the analysis of the structural system will be necessary after we acquire the data.

(Refer Slide Time: 1:03)



So, in this particular lecture, I expect that you will get acquainted with a particular method of structural analysis namely matrix method of structural analysis, we will be discussing some aspects of that. And once we carry out the analysis, and you get the results out of it, we should be able to interpret those results fruitfully, so that we can decide that what are the kinds of actions that are necessary to be taken so far as the retrofitting of the structural system is concerned.

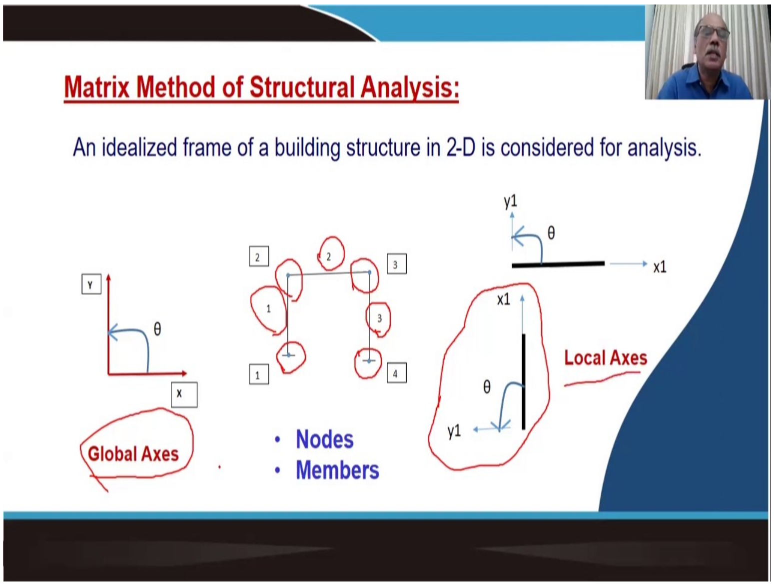
(Refer Slide Time: 1:36)



Just to give you and take you back to what we had discussed earlier, if you remember, we have spoken about these seven steps for arriving at the proper retrofitting measure that is required to be adopted in a structural system. And we have already discussed up to the four points we were in the last lecture, we discussed about the different kinds of materials that we normally use for retrofitting and the different test aspects that we normally carry out on different structural systems like masonry structures or concrete structures, steel structures. So, we have looked into those.

Now, once we acquire the data, once we get the characteristics of the material, we go for the analysis of the structural system, we try to idealize the structural system and then we try to analyze it, to understand that what are the effects of the loads on the individual members, and how they are stressed. So, that proper measures can be taken in terms of retrofitting of the system.

(Refer Slide Time: 2:53)



So, I would take you through certain basic steps that we normally adopt, in case of matrix method of structural analysis. First of all, any structural system is three dimensional in nature, what we try to do is to simplify our analysis methodology, we transform them in two-dimensional system, keeping certain aspects in mind, which I am sure you are aware of. So, I will just talk about certain basic steps that we need to adopt for matrix method of structural analysis in two-dimensional plane.

So, here we have shown is a simplified frame, which you call as a portal frame. And this portal frame is a single base, single story portal frame having three members, and let us say the bases are fixed at the support. So, this particular frame, first thing that we need to do is identify how many numbers of joints and how many numbers of members are there. Now, in this particular framework, as you can see, that we have three members member 1, member 2 is the horizontal beam member and member-3 is a vertical column.

So, two columns, one beam, these are the three members. And members are connected through the joints, there are four joints, joint-1, joint-2, here you have the joint-3 and you have joint-4 and normally we call these as nodes these joints. So, we have four nodes and three members in this particular frame. And at each node in a two-dimensional plane, you have three degrees of freedom, the movement in the x-direction, you see here, I have given the coordinate reference system. I will come back to tell you what is meant by global axis reference.

So, you have displacement in the x-direction, you have displacement in the y-direction and you have a rotation at the joint. So, at each node, you have three degrees of freedom thereby for this entire structural frame, since there are four nodes, so total degrees of freedom will be 12. And we will see subsequently what is the meaning of these 12 degrees of freedom that we take into account. So, we have four nodes and three members in this framework.

Now, two points, I would like to tell you, when we will be going for the formation of the matrices for each individual member, we will notice that, we need to first form the global stiffness matrix for the system. Now, for forming the total stiffness matrix for the structural framework, what I will do is that, the reason that we have divided into different members, we try to calculate the stiffness for each individual member.

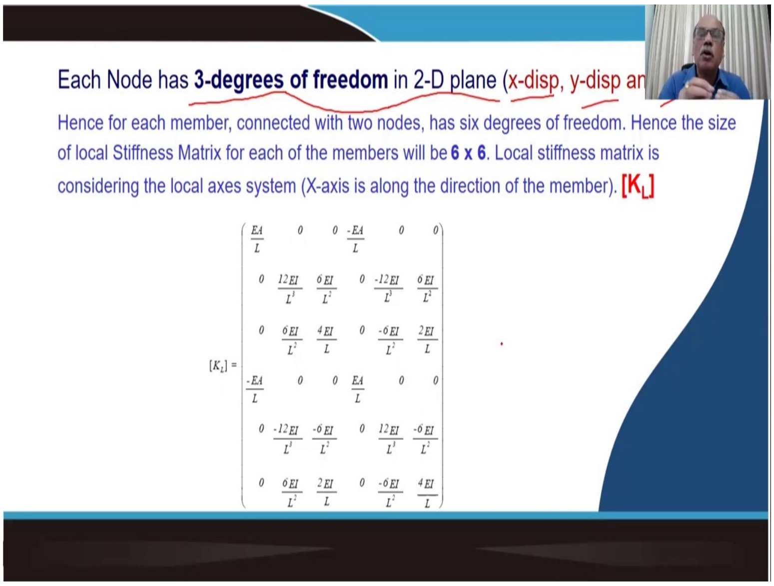
Now, let us say if I take member 1, member 1 is connected with node-1 and node-2. Now, this is a vertical number. So, normally what we do is, we define the local axis system, we take the x-axis along the length of the member, that is what I have shown over here is x1 and perpendicular that is the y-axis which is y1 and of course, rotation depends on z which is theta.

So, you see, this is the local axis reference for the vertical member 1. Likewise, if I take member 2, you see the local axis, x-axis system is along the member axis from node-2 to node-3 direction and this is x1 perpendicular to this is y1 and then again probably is theta. So, as you can appreciate from here that two members, one vertical, one horizontal have two different orientations of x and y, the local axis.

Now, in this, if I write down the matrix for each individual member, I cannot connect them algebraically. So, what do we need to do is that, we need to transform these with reference to a particular reference axis system, which we normally call as global axis. In the Global axis system what we do? We do that we have global x and global y and theta, so, we transform whatever we write in terms of these local axis system for each individual member to the Global Axis reference system.

So, then, once I do that, then I can add the contributions of this particular node from vertical member to the contributions of this horizontal member at this particular node which is common for both. So, that is why we need to look into the two-axis system one is local axis system and another is the Global Axis system.

(Refer Slide Time: 7:56)



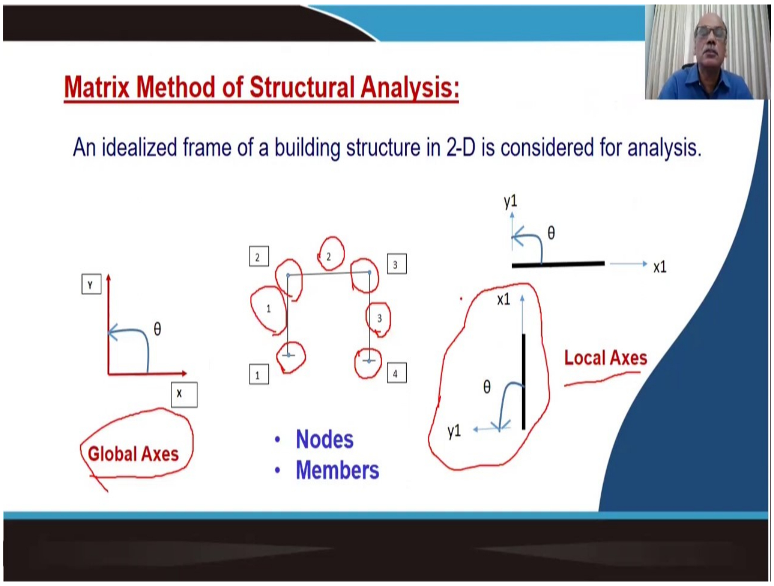
Now, having given this background, so, what we normally try to do is that, I have already told you that we have three degrees of freedom in two-dimensional plane which is x displacement, y displacement and rotation. Now, for a member, you have two nodes, the near node and the far node and at each node you have three degrees of freedom, so, you have total 6 degrees of freedom in a particular node.

So, you have x displacement, y displacement, theta, and likewise in other side also x displacement, y displacement, theta. Now, connected with that you have the corresponding stiffness parameters, this is called axial stiffness and then the flexural stiffness and the rotational stiffness.

And thereby, you get a, if you write forces equals to stiffness times displacement, you get the corresponding stiffness matrix which is of size 6 by 6 and here you please note the elements of the stiffness matrix is a function of the material characteristics which is E modulus of elasticity, cross sectional area A, length L, and moment of inertia I. So, it is basically geometric properties of basic member and the material properties, with this if we know these two aspects, we can form the stiffness matrix.

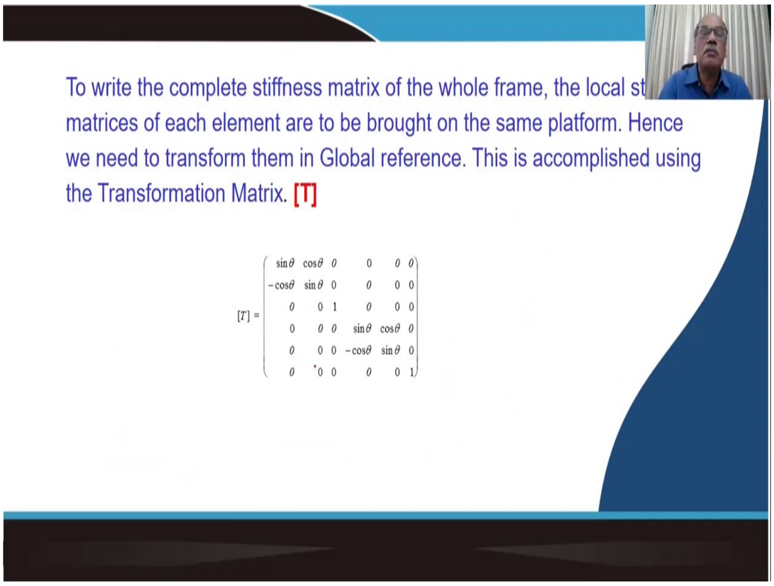
So, thereby stiffness matrix for a given member with given material is known to us. So, what do we do, for each of the member we found the local stiffness matrix and then as I said that we are forming the stiffness matrix in with reference to the local axis system?

(Refer Slide Time: 9:46)



Now, at a common node like the one I have shown over here that at this particular node member 1 and member 2 are getting connected, but they are in two different directions. And likewise, the local axis we have considered one x1 is this direction x1 in this particular direction. So, what we intend to do is that we transform this stiffness matrix in the global reference system with global capital X and capital Y and thereby, we can sum them algebraically.

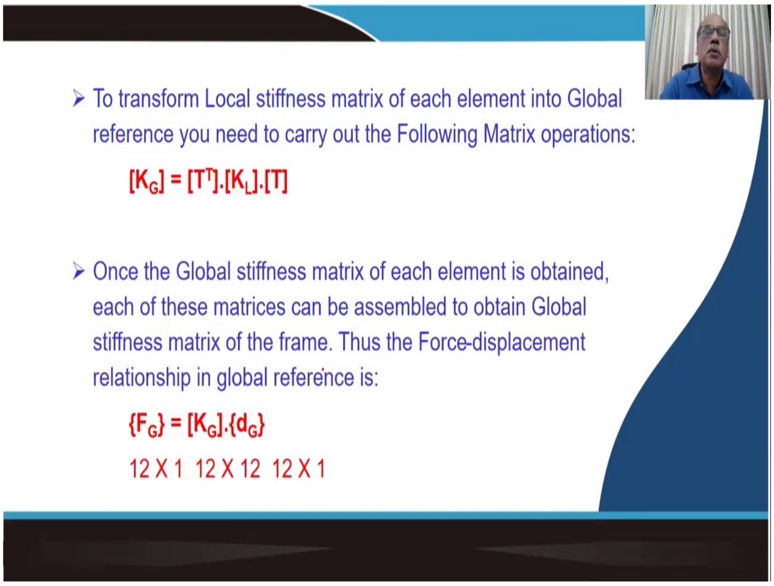
(Refer Slide Time: 10:17)



And this is what I have given over here, you can see this matrix again is of size 6 by 6 and is a function of the angle theta, which defines the angle of the member with reference to the global axis. So, once I know the angle theta, I can compute this transformation matrix T.

Now, this transformation matrix T has a property that if you take its transpose is equal to its inverse. And, you make use of this transform matrix to transform the local axis, the stiffness matrix in local axis system and transform them global with reference to the Global Axis system.

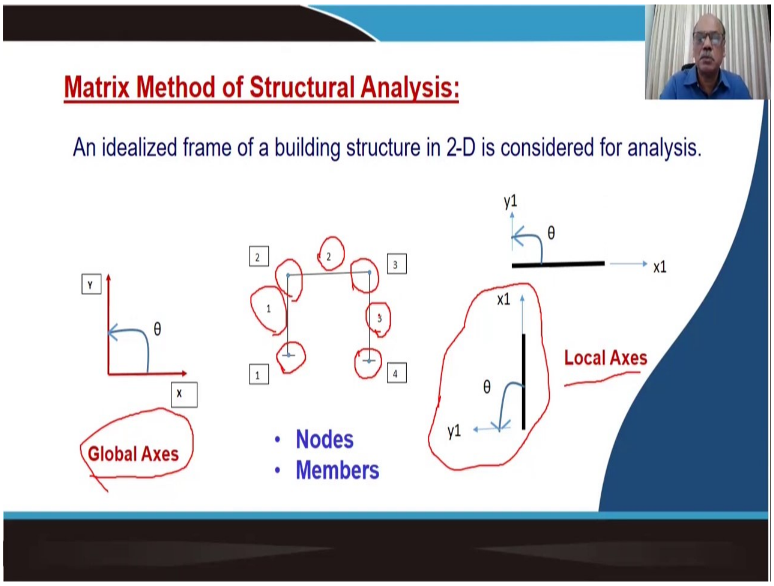
(Refer Slide Time: 11:01)



So, the local stiffness matrix K_L can be transformed to this global axis system K_G by multiplying with the transformation matrix, pre multiplying and post multiplying, in fact, this is the operation that can be done and those who are acquainted with this matrix method, I am sure that you know this derivation. So, in making use of these transformers in matrix, we can transform the global stiffness in the global reference to the stiffness I mean in local reference to the global and this is going to be helpful as I said, now, that for each member, if I do that, I can connect them algebraically.

So, once this global stiffness matrix of each element is obtained, Now, each of these matrices can be assembled to obtain a global stiffness matrix of the frame. Now, what is the meaning of the assembly process, assembly process, meaning of the assembly process is joining the matrices of each of the element in an appropriate manner, looking into the connecting nodes or keeping the connecting nodes in mind.

(Refer Slide Time: 12:18)



Again, you may go back to the framework over here. Now, let us say for member 1, I formed the local stiffness matrix which is of size 6 by 6, multiply these with the pre multiplying with T transpose and post multiply with T, then I get the global stiffness matrix for member 1. Now, since for the entire structure over here, you have 4 nodes and thereby 3 degrees of freedom at each node and total degrees of freedoms are 12 by 12.

So, that matrix either global stiffness matrix for the entire structure is a size 12 by 12. Now, out of that 12 by 12, when you say node 1, you have x1, y1, theta1 and node 2, you have x2, y2, theta2, at node 3, you have x3, y3, theta3, and at node 4 you have x4, y4, theta4. Now, though for member 1, it is connected with x 1, y 1, theta1 and x2, y2, theta2. So, you place that 6 by 6 stiffness global stiffness matrix into the entire global stiffness matrix of the structure depending on the position of this degrees of freedom

Likewise, for member 2, you form the global stiffness matrix, which is again going to be a 6 by 6 and placed them in the respective position keeping in mind that this common node which is x2, y2, theta2, can be placed in an appropriate manner, so that the elements that you have already positioned for member 1 will get superimposed.

So, likewise, for the other member also we can form the global stiffness matrix and we can position them in the overall global stiffness matrix of the entire structural system. And once we do that, by doing so, we will obtain the global stiffness matrix of the entire structure.

(Refer Slide Time: 14:15)

Now, while assembling the global stiffness matrix for the entire structure, likewise, you have 12 displacement components, as I said that at each node, you have x displacement, y displacement and theta. So, for 4 nodes you have 4 times three 12 displacement and this is what is represented by this vector d_G and correspondingly in each of those degrees of freedom, you have the corresponding force component, force in x direction, force in y direction, and corresponding to theta you have moment at that joint.

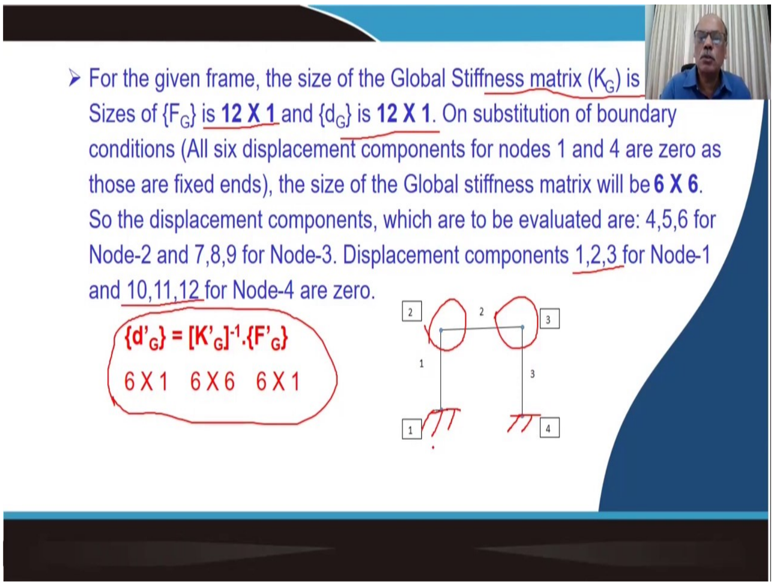
So, again you get three sets of course components for each joint and thereby you get a force vector which is of size 12 by 1. So, it Eventually, you have a 12 by 1 vector force forces for each node, you have displacement component which is 12 by 1 and the corresponding coefficient matrix which is a stiffness matrix, global stiffness matrix, which is of size 12 by 12.

Now, once I found this global stiffness matrix, mind that what is our objective? Our objective is to evaluate the displacement because we know the external forces that is acting on the member and we have for the stiffness matrix, for the entire structure, now, we need to compute the displacement. But the point is you can very easily say from this particular equation that if I invert K_G multiply with F_G then I can get d_G.

But what will happen is, if you try to do this you are not going to get the solution ready, why? The reason being that at this point of structure every time, the structure is unconstrained that means the frame as a whole can move and which you call as a rigid body motion. So, you are actually not going to get any solution, to get a meaningful solution, you have to put a constraint to the structural system and the constraints are that as we said that at the base level the frame members are fixed.

Now, if the frame is fixed at the base level, it means that the displacement components corresponding to those nodes, node 1 and 4 those displacements are 0. So, x displacement, y displacement, and rotation for node 1 and node 4 these are 0. So, out of 12 displacement components actually 6 displacement components are known to you.

(Refer Slide Time: 16:54)



So, what you can do is that the whole by matrix operation you can separate out the known and the unknown displacement components and since these are 0 here, so, readily in fact the corresponding rows and columns of the stiffness matrix can be eliminated. So, as I said for the given frame the size of the global stiffness matrix is 12 by 12, then size of F_G is 12 by 1 which is a force vector and the displacement vector is 12 by 1.

Now, once they substitute the boundary conditions then the corresponding to node 1 you have displacement components 1, 2, 3, and for node 4 you have 10, 11, 12. So, if I rearrange the rows and columns of the matrices in such a way that all 6 displacement components which are known, they are brought to one place and balanced 6 displacement components which are unknown there are brought at another place.

So, by doing this arrangement, I can eliminate the corresponding rows and columns of the stiffness matrix for which the displacements are 0 or they are known. So, thereby the resulting matrix that will be is equals to a transformed displacement vector or the displacement vector which is unknown and the corresponding force vector and multiply it by the reduced size of the stiffness matrix which is inverted.

So, and thereby you can compute the unknown displacement components by adopting the boundary condition to the structural system. So, what I mean in this is that the boundary conditions that we adopt for that this end and this node 1 and node 4 are fixed. So, by fixing these nodes you have already stipulated that the displacement components are 0.

But, sometimes it may so, happen that let us say this displacement gets separated by some amount, so that means that displacement component is not zero, but it is known now, if you get a known value, you can multiply the corresponding coefficient of the stiffness matrix and can you can take it to the right-hand side to the force vector. So, your force vector gets modified, so, effect of the displacement can be taken into account into the force vector and thereby you can get a solution of this.

So, we tried to bring in the whole stiffness matrix for the structure into a reduced form by applying suitable boundary conditions onto the system. And once we do that, then the reduced stiffness matrix that we get that can be inverted to get multiplied with the corresponding modified force vector and the corresponding unknown displacements will be known. So, by doing so, here the unknown displacement of node 2 and unknown displacement of node 3 are evaluated.

Once you know the unknown displacements of these, now displacement of all four nodes are known to you. And once you come to know about the displacement, I mean displacements at all four nodes then at least there you have an idea about the displacement component. But what is our objective?

Our objective is that we try to calculate what are the member forces and thereby, what will be the corresponding stresses in each of these members, because, you have carried out different kinds of tests and you have already tried to establish that what is the strength of the member and so, we need to know that the present strength that we have, whether they are matching with your analysis result, whether the members are showing the similar kinds of results or not.

Now, to calculate the member forces. So, what you have done now, from the unknown displacement you have got the global set of displacements for the entire structural framework. Now, you need to go back to member level again and for each of the member from the global displacements, you try to find out the local displacement. Let us say, after you have computed the displacement at node 2 and node 3 the six displacement components are known, and already because of node 1 and node 2 are fixed your displacement components are 0.

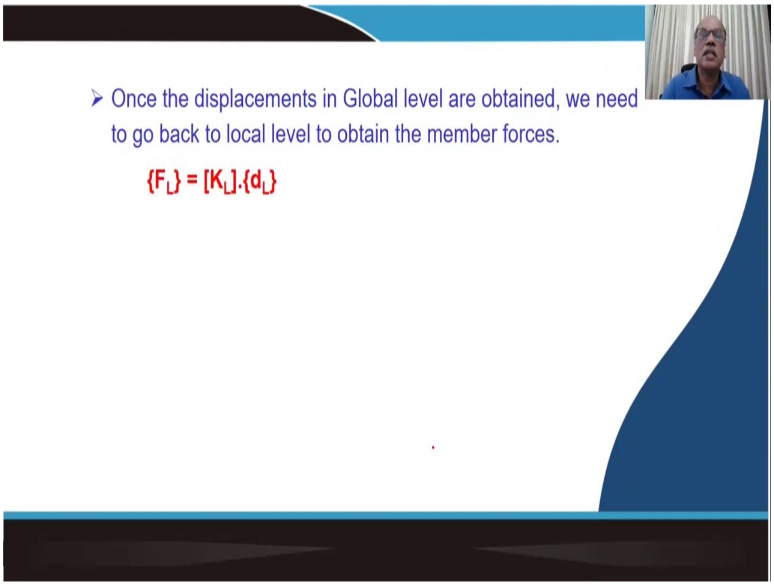
So, thereby all 12 displacement components are known which are on the global system. Now, I go back to member 1 and for member 1, I know what are the displacement of node 1 and node 2. So, I take those 6 by 1 displacement components and I tried to multiply those local displacements with the local stiffness. Now, if you multiply stiffness with the displacement, what are you going to get? You are going to get the corresponding force components.

So, from the known displacement values, you multiply to the corresponding local stiffness matrix and you compute the corresponding forces. So, once you do that, what you will get is at node 1 how much is the force along the member, how much is the force perpendicular to the member, this is a shear force and how much is the force in the direction of theta means, how much is the moment?

So, you get axial force, you get shear force, you get bending moment, at each end of the member. So, from this you will be able to plot now, your bending moment diagram, your shear force diagram, and you will know that how the transform or the rotational behavior of the entire frame work is case.

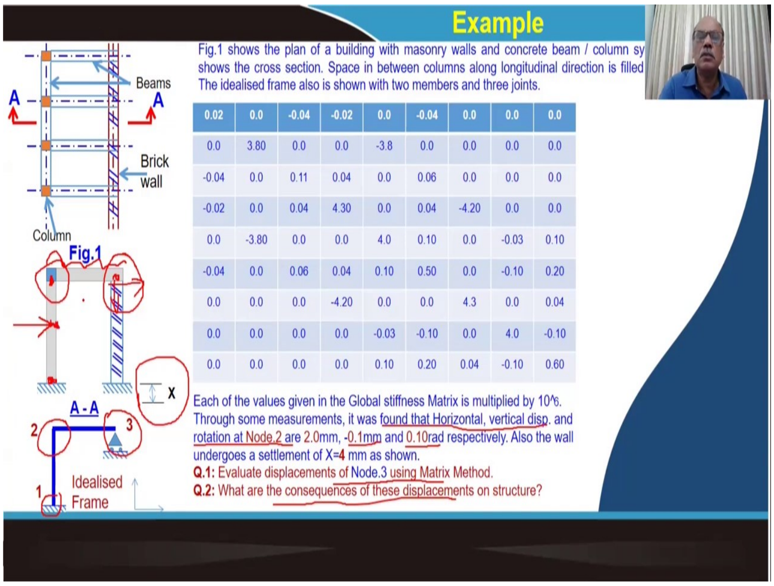
So, if you can do that, once you do that, then you will be in a position to compute the stresses in each of the members. And once you do that, you can compute the stresses in the member, you can find out that how much is the reserves strength and for the forces or the loads that the member is subjected to, this is the amount of actual stresses that is getting generated. And from the measurement, you have already established the strength and you can make a comparison between the two. So, that is why this analysis helps you to get an idea of what is happening.

(Refer Slide Time: 23:22)



And this is what I have explained over here, that once the displacement in global level adopted, we need to go back to local level to obtain the member forces, and this is what is our objective. We intend to compute the displacements at the nodes and the global level and then we go back to the local level and try to compute the member forces.

(Refer Slide Time: 23:49)



So, just to give you an idea, I have taken an example. Here you see, this is the plan of a building, where there is a brick wall over here, the brick masonry wall, and over onto top of this there is a beam, reinforced concrete beam and connected to a column. So, these are the column, the shaded parts are the columns, and again in along these directions there is a reinforced concrete beam which is connected to the column and in this direction, you have a beam member which is resting on the masonry wall.

Now, if I cut across a section over here, the cross section looks like this, that you have a column member, you have a beam member, and you have is resting over the masonry wall. And of course, here in the longitudinal direction you have beam also. Now, so, if I look into the idealized configuration of this particular framework is like this, you have a column member, member 1, your beam member, member 2.

So, here, these are two members and how many nodes you have? You have node 1 here, you have node 2 here, and you have node 3. So, thereby, as we have in a two-dimensional plane, you have three degrees of freedom in each node. So, you have 3 times 3, 9 displacement components for the entire framework.

Now in this, so, it is expected that the total size of the stiffness matrix for the entire structure is going to be 9 by 9. And the values of the stiffness matrix for this particular configuration has been given over here, the values are known. And what is needed is and also what is given here or that the displacement found that horizontal or vertical displacement and it was found by measurement that horizontal vertical displacement and rotation at node 2 are 2 millimeters minus 0.1 millimeter and 0.1 radian respectively. So, and also the wall undergoes a settlement of X is 4 millimeters.

So, these values are given, that means, three displacement components and node 2 are given and node 1 the displacement components are known because this is a fixed support, so, x, y and theta are 0. At node 3 the support is on the roller; we have idealized this as a roller support because it is resting on the masonry wall. So, at this wall that the interface between the beam and the wall, the beam can move in the x direction. But in the vertical direction as such it is not allowed to move assuming that the masonry wall is rigid.

So, here you will get x displacement at the rotation. So, here you have two displacement components which are unknown, rest of the displacement components are known. Now, and this node here incidentally it has been indicated that the support settles by certain amount. Now, if some support settles assuming this as a rigid the whole thing settles down. So, it comes down again by that much of amount which is 4 millimeters.

So, you have displacement given in the vertical direction also is known. So, you have known displacement components of this particular joint 1, and two displacement components namely displacement in the x direction and the rotation over here are unknown for node 3 and these are the two displacement components that we need to compute.

So, what we do is that, we have a 9 by 9 stiffness matrix and 9 by 1 displacement vector and out of 9 by 1 displacement vector 7 displacement components are known to me. So, balance 2 displacement components which are unknown those are to be evaluated. So, what I do, the stiffness matrix multiplied by the displacement vector, which is 9 by 1. So, 9 by 1 displacement vector out of that 7 displacement components are known.

So, I can partition the stiffness matrix in the form of 7 by 7, 7 by 2, 2 by 7 and 2 by 2, and then you will have a 2 by 2 matrix multiplied by 2 by 1 unknown displacement components is equals to the post vector which is again going to be 2 by 1. Now, by inverting the stiffness matrix this 2 by 2 stiffness matrix, we can compute the unknown displacement that will be happening at node 3. So, we can make use of the of the matrix method to analyze this kind of structural system.

Now, interestingly if you look into the question that has been given over here, it is stated that evaluate displacement of node 3 using matrix method, this is one. Second thing is that what are the consequences of this displacement on structure and that is interesting. Now, what is going to happen if this particular joint, the beam member moves in the x direction, that means rotates at this point. So, naturally this is going to have effect on these masonry wall.

So, the depending on the strength of the masonry wall and also there is a settlement in the wall. So, there is a possibility that wall might crack, there will be a possibility that the joint at the interface between the beam and the wall might crush. And also, because of the rotation on that, the corner also can rotate.

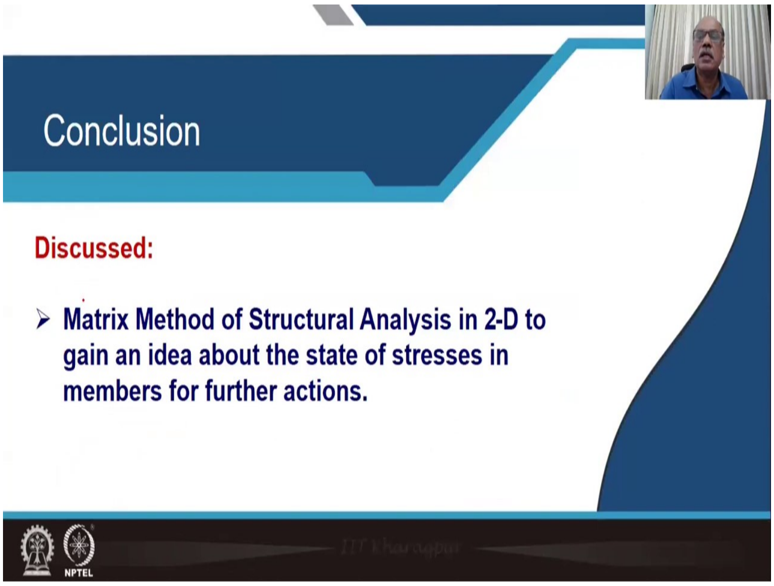
So, the masonry wall will undergo distress because of these kinds of axis that the frame is subjected to and because of this you can identify those kinds of distresses, and you can adopt appropriate retrofitting measure to repair that kind of thing. So, it is important to find out that what kind of distresses have happened and how to retrofit these by using appropriate methodology.

We will be discussing about the retrofitting methodologies when we go for the masonry structural system and looking into that you can adopt it appropriately. Now, just one point I would like to mention over here now, the kind of loading that we use for this kind of two-dimensional system, it could be distributed loading or it could be concentrated loading.

Now, whatever loading you have, let us say you have some kind of a distributed load in this, and let us say you have some kind of a concentrated load at this particular column member. Now, if you have this normally all these distributed loads, we transform them in terms of joint load, and we try to put in this particular joint.

Now, when you have a concentrated load in member, since, we are defining the loads as a joint load, so, what we can do is that, we can introduce a node at the concentrated load point and we can discretize the frame as like this, we can take a node here, we can take a node here, we can take a node here, we can take a node here, and accordingly we can say instead of two members is going to be three members, so, your degrees of freedom are going to increase, the metric size is going to increase, and accordingly you need to analyze.

(Refer Slide Time: 31:40)



So, this is in general, that we, we try to adopt the matrix method for the analysis of the system. So, basically what I wanted you to get exposed to, I am sure that all of you have done this matrix measure of structural analysis, just to give you some recapitulation on this, I thought to give you certain ideas, that how do we take these into account to analyze the whole system.

So, basically, we have discussed the matrix method of structural analysis in two dimensions, and to get an idea about the member forces that will be acting because of the loading, and thereby we will be able to compute the stresses in the member.

(Refer Slide Time: 32:18)

Thank you. So, with an overview of this analysis, and as I am going through that, I am giving you an overview of different aspects, then we will look into that how we compile the whole thing and we like to come out with the reports that what are the things that have happened, and how we can go for the analysis of the system, and we can go further retrofitting of the system as such. So, this is what we thought to explain to you about the analysis methodology. And thank you very much.