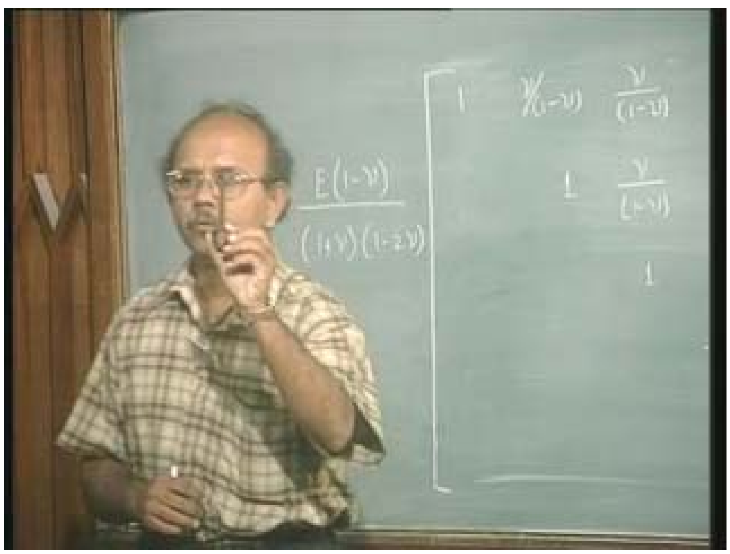
Introduction to Finite Element Method

Dr. R. Krishnakumar
Department of Mechanical Engineering
Indian Institute of Technology, Madras

Lecture - 12

In the last class, we were looking at some examples and we derived the stress strain relationship and so on. Now, let us proceed on further discussions with further discussions on this stress strain relationship, especially for the axi-symmetric element, before we go and look at further examples. To summarize what we saw last, in the last class, we said that there is a general behavior which I would call as a three dimensional behavior for which I have put down a 3D element.

(Refer Slide Time: 1:37)



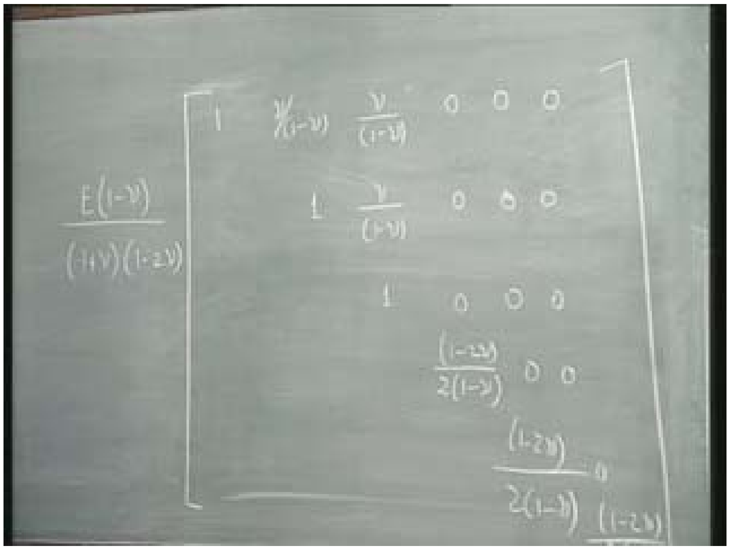
For example we saw that for connecting rod, this kind of connecting rod, we have to use a 3D element and then we looked at certain other examples. For example we looked at the rotor, LP rotor and said that in order to study LP rotor we have to put down what is called as an axi-symmetric element. We also saw gear and we said that for the gear, depending upon its thickness we can put either a plane stress or plane strain or may be 3D and so on or in other words the philosophy is that you have to look at the type of behavior; you have to classify the type of behavior and then from that classification we can arrive at what is the type of element that has to be used. That is an important part of finite element model. There are two questions that may come to your mind. Number 1 is that why do you want me to do this kind of two dimensional approximation? Why should I have an axi-symmetric element, a plane strain element, plane stress element?

I am going to classify, I mean many more behaviors in this class, but before we go further you may ask me a question. Why not I have only one element called 3D element, because anything in this world is 3D? Why do you want 3? Why do you want to have all this kind of elements? That is a question which can come to your mind. There are two things to this issue. One is that, it is much easier to generate a mesh or other words, define a mesh and very accurately and very, say, easily with respect to a 2D geometry. 3D mesh generations are still difficult, you can be stuck most of the cases; 100% of the cases. Why most? 100% of the cases you will be stuck with elements called tetrahedron elements which has or which does not have certain good properties. We are going to see all that later in this course. So, good elements, we can generate nice elements to study using the 2D case and the results, when your assumptions are right, will not be any different or will be very, very, very slightly removed from the results that you would get with the 3D element. That is number 1.

Number 2, when you go to 3D, the problem size also increases tremendously; the problem size is not going to be small. So, the problem When the problem size increases, your computer space increases, time to solve increases; all sorts of complications. In fact, I should add one more thing to this: post processing or looking at the result becomes very difficult when you go to 3D rather than 2D. You may not understand it that easily and I mean, the visualization become slightly difficult, but whereas in 2D all these things may not be there. At the same time a word of caution: you should not rush to 2D or plane strain, plane stress and axi-symmetric instead of solid all the time. So, overall that is the first thing.

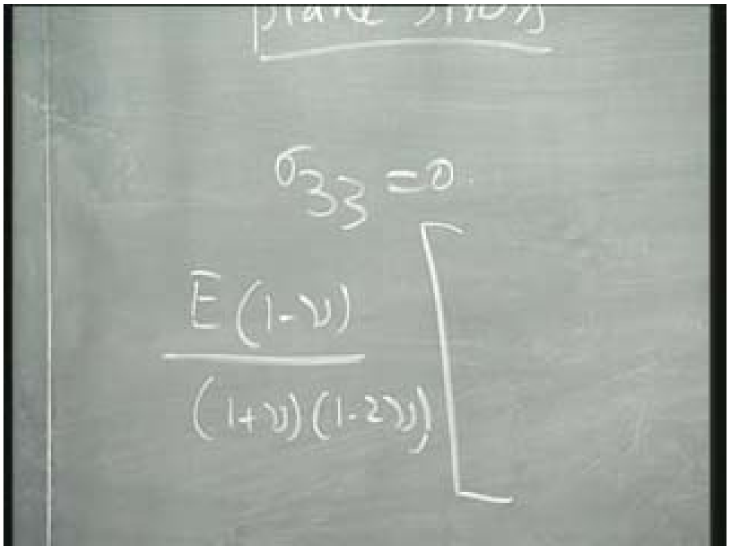
The second one is that there are certain behaviors like bending. These behavior, bending behavior can be caught in a much better fashion by certain specialized elements like a beam element; bar does not. Please note bar is not for bending. It is only for an uni axial loading. You can catch the effect of bending very clearly or very nicely in a beam element, plate element and as well as the shell element. These things capture it very nicely. In fact there may be In fact you would introduce errors when you use solid element or you have to use lot more solid elements in order to capture same effect as what you see in a beam element. So, this kind of problems or in other words the behavior need not all the time be a three dimensional behavior. That also we will expand as we go along like what are the problems that you would face if you go to a solid element in order to capture a bending behavior.

(Refer Slide Time: 6:20)



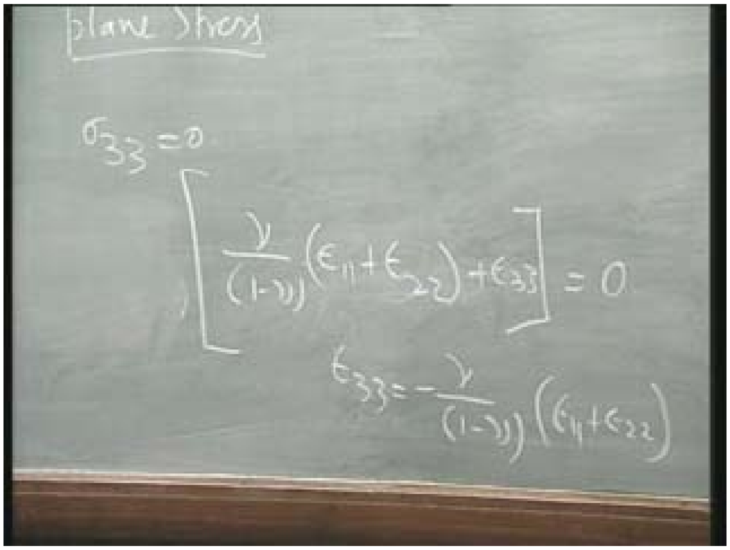
But, before we proceed with further examples on these two or these three types of behaviors, let us have a look at this matrix and quickly derive a matrix for plane stress. I want to derive a matrix for plane stress. Usually plane strain is very straight forward because this epsilon$_{33}$, what see this is associated with epsilon$_{11}$, this with epsilon$_{22}$, 33; what is this? Gamma$_{12}$ and so on. Usually it is very simple to reduce this for plane strain because epsilon$_{33}$ is equal to zero. On the other hand, it is not a very straight forward task to reduce this to plane stress and I want to emphasize that point and that is why I want to derive this. What is the condition for plane stress?

(Refer Slide Time: 7:18)



Sigma$_{33}$ is equal to zero. In other words what is the relationship between epsilon$_{33}$ and epsilon$_{11}$ and epsilon$_{22}$? We can very easily write it, from this particular matrix. For example you can write that E divided by 1 minus nu into 1 plus nu into 1 minus 2 nu into, that is the first one; 1 plus nu into 1 minus 2 nu into, how do I write this?

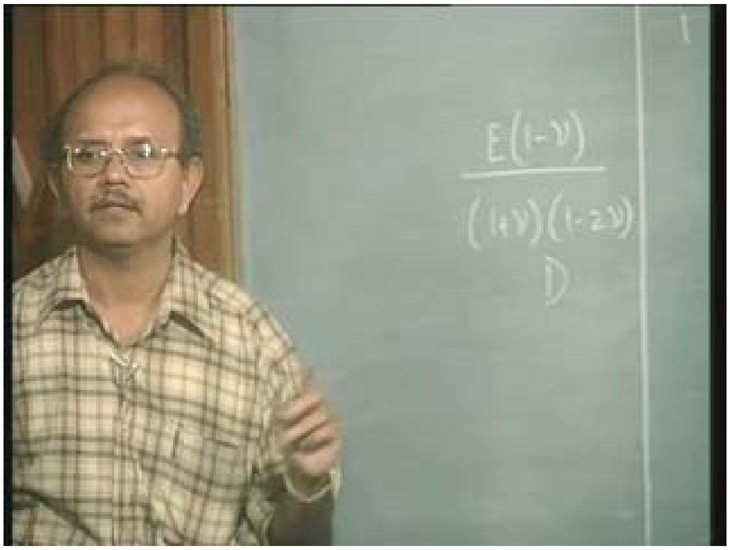
(Refer Slide Time: 8:08)



What are the terms that would come into picture into nu divided by 1 minus nu into epsilon$_{11}$ plus epsilon$_{22}$ plus epsilon$_{33}$ is equal to zero. Is that clear? Just that term alone, which means that I can say that, that is equal to zero; which means that

$epsilon_{33}$ is equal to minus nu into 1 minus nu into $epsilon_{11}$ plus $epsilon_{22}$. Is that clear? Now, what is my next step? I have to substitute for $epsilon_{33}$ whatever I have got here nu into minus nu into 1 minus nu into $epsilon_{11}$ plus $epsilon_{22}$. Where do I substitute this? For example, I can substitute it in $sigma_{11}$ equation. What do I mean by that?

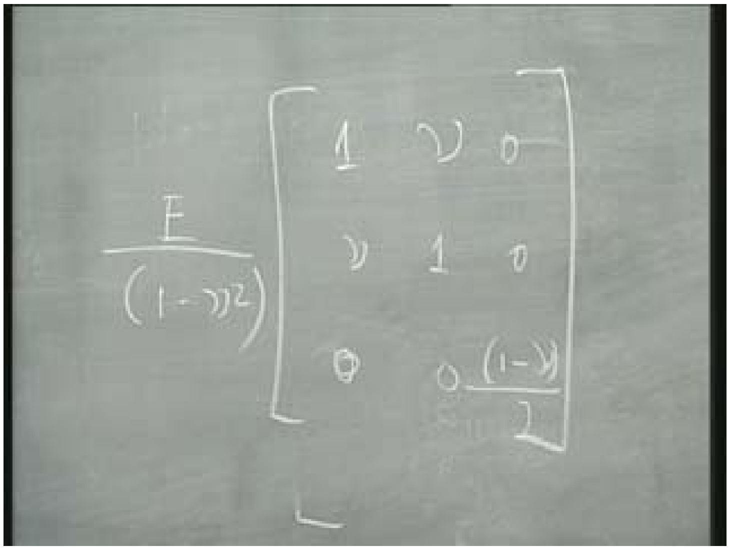
(Refer Slide Time: 9:44)



I mean to say that $sigma_{11}$ is equal to this term. Let me call this as a D term just for convenience, so that I need not write it again. This D into $epsilon_{11}$ plus nu by 1 minus nu into $epsilon_{22}$ plus nu into 1 minus nu into $epsilon_{33}$ that will go; I mean in that place this particular term will go or nu into nu divided by 1 minus nu into minus of this term. Is it clear? Then, I will give you a minute; let us see whether you are able to write down. I will leave this; let us see whether you are able to write down the D matrix correctly or C matrix.

By the way how many rows and columns will be there? How many rows and columns will be there for plane stress? Three; $epsilon_{11}$, $epsilon_{22}$, $gamma_{12}$. $Epsilon_{33}$ will not enter and will go separately because $sigma_{33}$ is equal to zero. So, you should be very clear about that as well that for both plane stress and plane strain we are going to have, we are going to have only 3 by 3 matrix and that matrix, that is simple algebra. That I want you to write and that matrix can be written as, has anyone completed it?

(Refer Slide Time: 11:43)

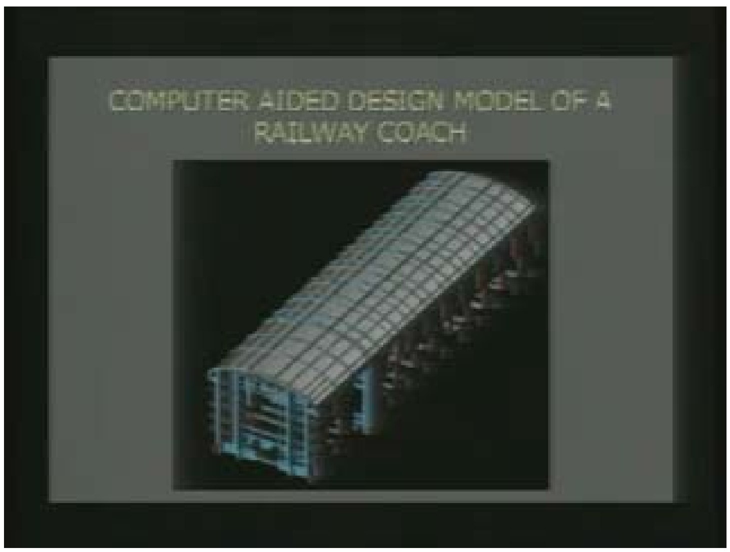


E divided by 1 minus nu squared into 1 nu 0, nu 1 0 and what will happen to this? 0 0; what will happen to $gamma_{12}$? What will happen to this term, whether plane stress or plane strain? What I mean, In other words what will happen means, what will be the entry here? What should come at that place? Please note that at other places 11 22 and all that are affected because $sigma_{11}$ will affect 22 and 33. 1 minus, look at this; is it 1 minus 2 nu, 2 nu? What will come there? You know, forget about this. Will the shear, 1 minus nu divided by, no, this will just be G, by 2. That is it, good; because the shear will not get affected, please note that carefully.

The Poisson's effect is felt only in 11 22 and 33. So, it will not be felt in shear term and hence E divided by 1 minus nu squared into 1 minus nu divided by 2. Is that clear? What will happen to the axi-symmetric? How many terms or what will be the matrix size? I do not think I wrote it last class. So, what would be the matrix size, this size in the case of axi-symmetric element? What are the terms that are present? What is it? See, this size is very important because in a later class when I write stiffness matrix, you should know what should be the size of this matrix? In order to understand that, at this point of time it should be clear what are the things that go into the calculation for stress strain relationship? What are the terms? How do I get this number? What are the terms that are present as far as strain is concerned in an axi-symmetric situation? We have studied that in the last class.

We said that epsilon_{rr}, $\text{epsilon}_{\text{theta theta}}$, then epsilon_{zz} and then gamma_{zr}; these are the terms that will be present. That is what we studied. Is that clear? Though we will be looking at geometry in the rz plane, we said $\text{epsilon}_{\text{theta theta}}$ will be present. That is because of the fact that there is going to be an expansion and so on. So, that will be a 4 by 4 matrix, but fortunately that is quite straight forward because this kind of, what we had in plane stress that complexity will not arise here and so it will be a straight cut off 4 by 4 matrix. We will stop here for a moment and get back to a slide and see whether anything different can be looked at.

(Refer Slide Time: 15:44)

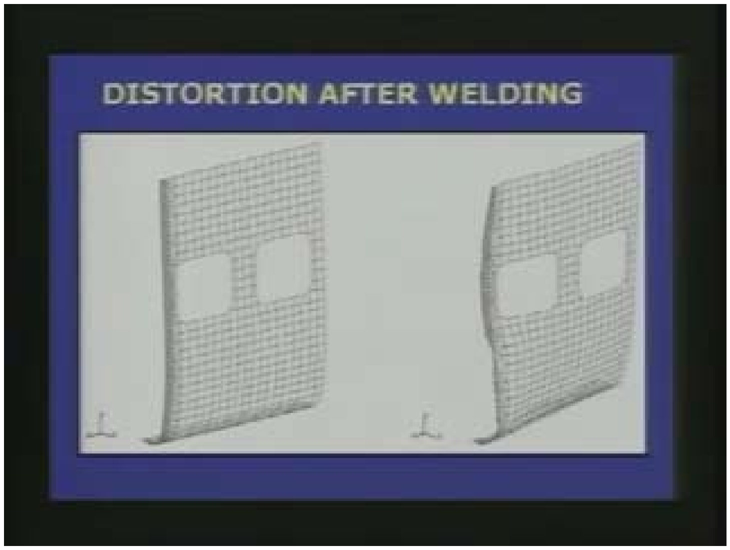


This as I had told you before is the computer aided design model of a railway coach. Now, I want to model this coach using finite element analysis. What does this coach contain? This coach contains what are called as an undercarriage. You see those blue things, lines running right across in the ground; in the floor where you are going to stand, that part and where your seats are going to be put, there are a number of beams that run. There is an undercarriage which is nothing but the floor portion, the whole load being taken by these strips of steel, with a channel section or I section steel, which run right across and the next thing that you are going to see in that picture is the side wall.

(Refer Slide Time: 16:43)

Probably this picture gives you a better view of it. You can see those lines, you can see those lines there; they are actually the beams. Let us get back to the previous picture and see that again very closely. Those lines, they are nothing but the lines that you see here on the roof. In other words, there are some beams that run in the roof as well. We will come to the roof in a minute, but let us look at another component of this particular picture.

(Refer Slide Time: 17:17)



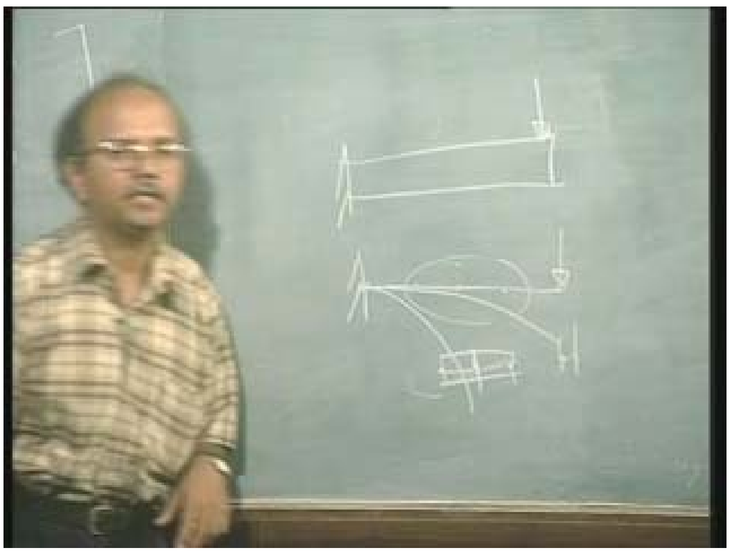
This shows the side wall. Our interest is also in the side wall. This shows a piece of the side wall, this shows a close up view as we assemble it. But the previous slides show the complete picture of the side wall as well. This as well as the previous one gives the complete picture of the side wall. The side wall sheets are 2 mm, 2 mm thickness. The question here is, how do I model or what are the elements that I use to model this kind of coach?

Let us look at only the undercarriage. Just have a look at that undercarriage there. What is the type of behavior that you see? What is the type of behavior you see there? Can you guess? Those blue things; what are the types of behavior? What do you think, how it would behave? Bending, very good; so, the behavior is one of bending. Hence I use what is called as a beam element. We will discuss the beam element in a minute. But before that let us look at the same thing; side wall. You see that red portions there, that is the side wall. It is 2 mm. What is the type of element that you will use? What is the type of element that you will use?

Plate; so, I have heard terms plate, shell and so on. Let us now discuss this beam, plate and shell and complete our vocabulary to certain extent on the type of the elements that are seen. What is a beam element or how does it or what are the sorry degrees of freedom of a beam element. Let us have a look at it. We will not derive now the equations for a beam element. We will reserve it for a future class, but let us now look at just the beam element and the corresponding degrees of freedom that are required from a physical point of view. Whatever I am saying now can be very well proved by mathematics as well which I will do in a later class. But right now, let us only look at the beam and see what the type of degree of freedom is, that you will require? Remember, what is the type of degree or freedom you required for solid element? 3; u v and w or u_1 u_2 u_3 and for the others, per node; we are talking about per node, 2 degrees of freedom.

We are already familiar with nodes. Though we have not yet discussed stiffnesses and all that, we know what a node is. We have to divide the whole body into number of nodes; that much we know.

(Refer Slide Time: 20:37)

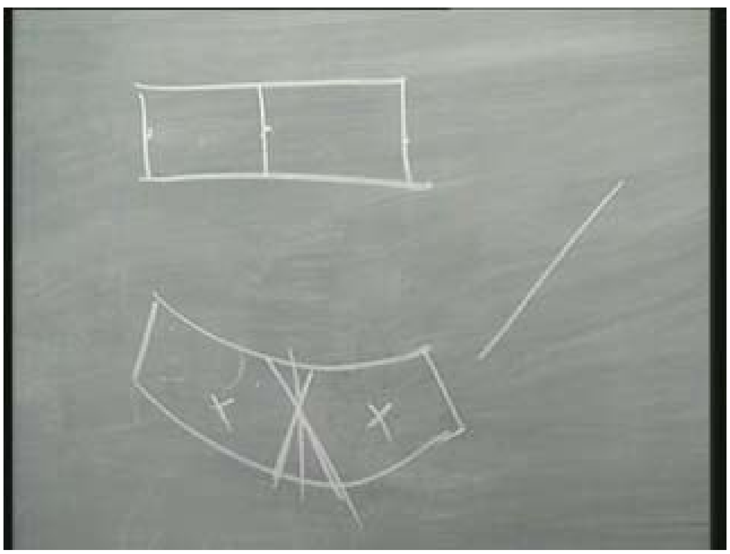


Now, let us look at, say, just a beam. Let us say that some load is applied. Now, I can possibly divide it into number of elements and then apply a load as well. The first question is, Let us take this element out. These two elements are like this. This element here, I have just shown it here. My first question is how many degrees of freedom are present in each of the nodes that are here? 2; what are the degrees of freedom? x and y, x and theta. We will first clarify the first one. We are now looking at only bending behavior and what we are doing are all small displacement problems. In other words, when there is a displacement like this, under the application of the load that you see there, effect of this movement here is extremely small; that is extremely small. So, this is a small displacement problem whereas, suppose this beam is made up of rubber and you apply lot of load; may be that this beam would take the shape something like that. It would just fall down; your load may fall down. We are not looking right now at this kind of problems. We are looking at small displacement and small strain problems.

You already know what large strain is. You remember that slope of displacement is far less than 1; that we had already seen. You look at this thing. The displacement is w, very good. Some of you answered that the second degree of freedom is theta. Why did you answer or how you know that it is theta? How do you say that it is theta? Yeah; one end of it does not rotate, so you said this. That is a very good answer, because you said that in one end when you apply boundary condition, in your earlier classes you would have said that the slope there is equal to zero; cantilever beam slope is equal to zero.

In order to specify a boundary condition, you obviously need a degree of freedom and hence this beam, a two dimensional beam, let me call this as two dimensional beam, should have theta as well as a displacement. Is that clear? This can also be looked at from a compatibility point of view. Let me draw a line here, imaginary line at that place.

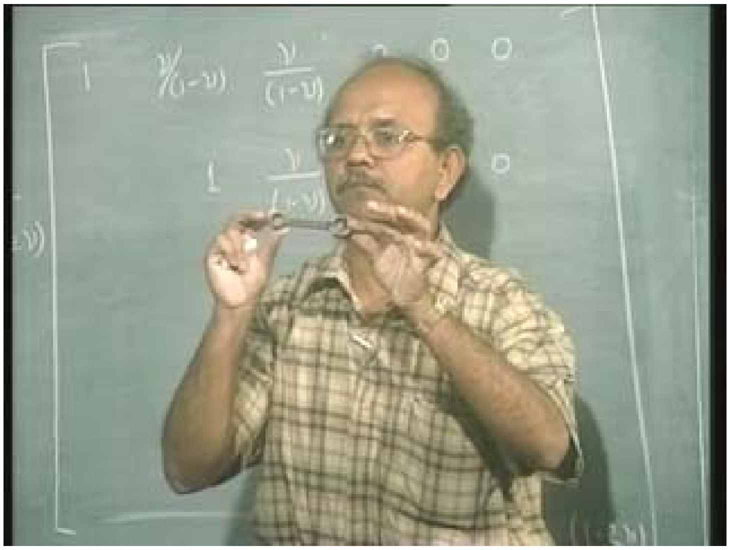
(Refer Slide Time: 24:00)



Let me draw that more clearly here; let me draw a line here. Let us say that this part of the beam bends like this. If theta is not included as a degree of freedom at the center node, what would happen? This line would look like this for me. This would look like that for me. Why, because this corresponds to this element and this corresponds to this element. So, both of them I have not specified as a degree of freedom. This is an important concept that you have to notice that the degree of freedom is shared at that node by these two elements. In order that that line In reality what will happen to that line? It will remain the same. in order that the in other words um In order that the situation is compatible, that line has to appear to me as one line and hence I have to specify theta as a degree of freedom, because in that case theta will be calculated for this element and theta will be calculated, I mean, for this node and that theta will be the same for this element and this element, because that node is shared by these two elements. Hence from a compatibility point of view itself, it is very in a from a very physical argument, it is very clear that theta should be a degree of freedom. But the theory, the mathematical theory is quite deep.

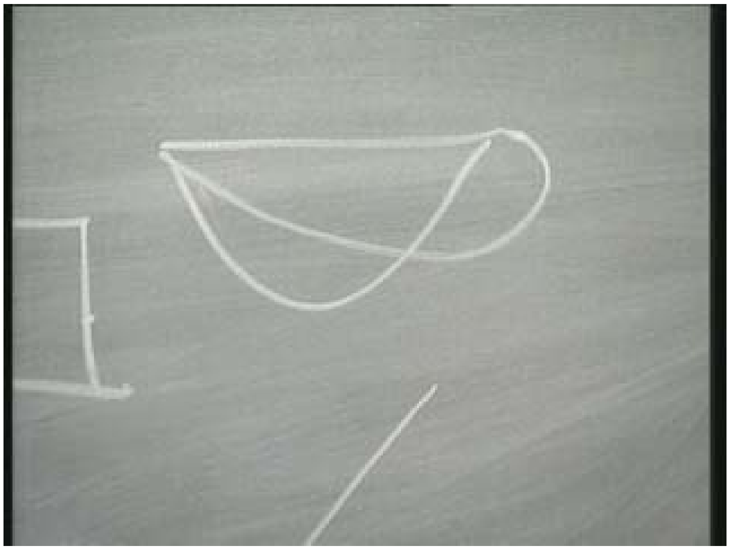
Right now, we are yet not ready to discuss this. We will postpone that to a future class, but right now, let us understand that this has 2 degrees of freedom. But the situation was not this simple in the figure that you saw because the beam was more three dimensional. It was not in a plane, it was running in the x y and z direction. So, what is that I should do? I should extend this picture or the degrees of freedom to 3D. How do I extend it? What does it mean? It means that my beam may be in space like this.

(Refer Slide Time: 26:47)



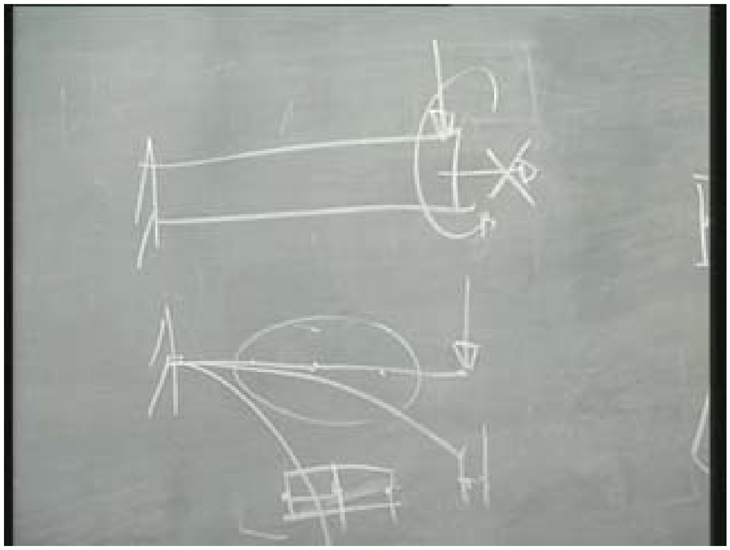
Let us say that this is the beam. Let us say that this is the beam. Beam may be such that it can bend like this or it can bend like this or in other words that can be theta x and theta y. It can bend like this, sorry theta y and theta z. It can bend like this, in this fashion or it can bend in this fashion; in both ways it can bend.

(Refer Slide Time: 27:14)



So, like this take the beam or in this fashion, it can bend either side. In other words, there can be a, I add, how many degrees of freedom I add? 2 more, because there will be a displacement as well as rotation in directions perpendicular to what I had initially for a two dimensional case. I want to introduce one more thing. Why, because the beams that you saw may be such that it may also take some tensile loads; it may take also some tensile loads.

(Refer Slide Time: 28:02)

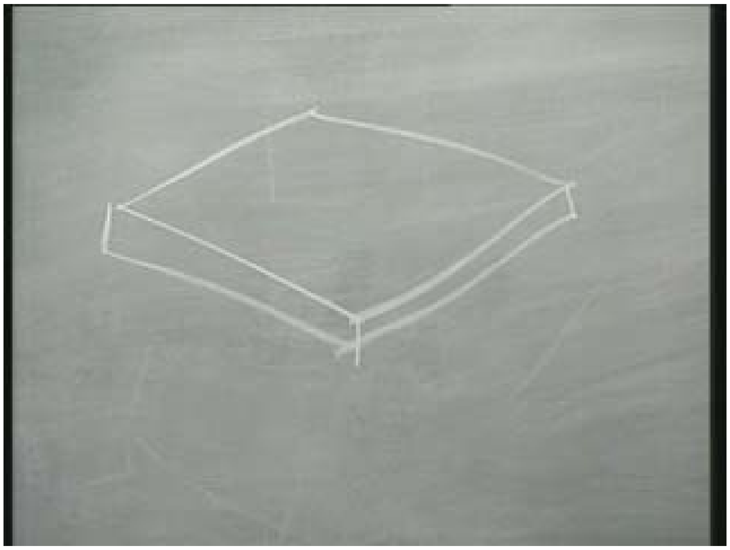


If it starts taking tensile load, then this beam theory what we have studied so far cannot take it, because beam means it is bending, we know and this fellow cannot be simulated by a beam behavior. So, what should I do now? What should I do now? What do you think I can do now? We have studied what is to be done at that place? Super position, very good; that means that if I can put the bar property along with the beam, introduce one more degree of freedom and for that degree of freedom put the corresponding stiffnesses, then I can simulate both the beam as well as the bar type of behavior. Only one is remaining. Still I have not completed the case. What is it? Twist; so, there can be torsion and I leave it as an exercise for you to work out how to derive a stiffness matrix for twisting.

Can someone give a clue? What is that you will use? What is that you will use? What is now stiffness in a twist? Correct; someone answered beautifully or torque, you apply a torque to it. What is the stiffness in a torque? How do you specify torsional stiffness? e by theta; so, in this case, in the beam case or in the sorry, the bar case you add ae by l as a term to indicate the stiffness. For a torsion case, for the same beam what is that you will have instead of ae by l? Fantastic; that is this gj by l; so, instead of ae by l, you will have gj by l. Now, you have got the point as what stiffness is. If I introduce all these things, first I started with the 2D bending, then three dimensional, then introduced torsion, introduced bar effect and so I will get per node, how many degrees of freedom? 6 degrees of freedom; so, this beam has 6 degrees of freedom. Is that clear?

Now, let us extend this beam in the third dimension; let us extend this beam in the third dimension. Let us see what I get out of it?

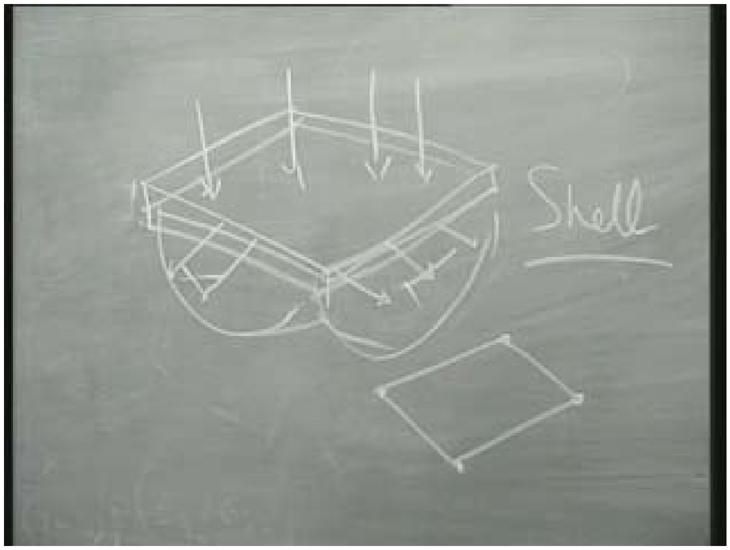
(Refer Slide Time: 31:03)



That means that let me extend that. See, there are lot of assumptions yet, you know; we have not yet come to all that in a beam. The assumptions that you had when you did your beam theory in the earlier classes still hold good. What are the assumptions? Romberg; yeah, isotropy is general. Yes; that means the planes remain normal; planes remain plane and they remain normal to neutral axis. What does it mean? There is no shear, shear of x are not there. But there are places where this kind of assumptions are not valid. Do you remember when? When the beam becomes quite thick, this kind of assumptions are not valid, then we have to move to some other type of formulations.

I will not right now deal with that type because physically I am trying to introduce this concept of beam elements. I am not going to deal with that right now. We will deal with it later, but right now, I just want to summarize saying that this beam element has 6 degrees of freedom. Let us now extend this plate in, I mean sorry, beam in the other direction and get to a plate. Now, see what happens?

(Refer Slide Time: 33:34)



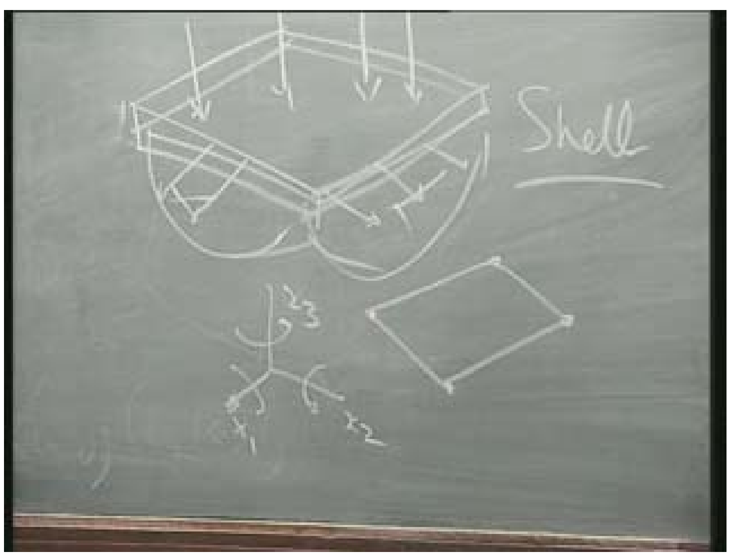
The plate is subjected to out of plane loads and under this loading condition the plate is going to bend, it is going to bend something like this and so on. It is going to bend. Because of bending, again we have degrees of freedom present. But before that let us just quickly recapitulate how we can model this beam or in other words what is the type of element I will use to model the beam.

I will just introduce node at the centroids and then get an element, say, something like this. Though the beam is 3D, though we are going to look at three dimensional effect, the element looks like a 2D element. Actually it is not 2D, because we are going to say that or we are going to see that can even geometrically span a three dimensional space, though the behavior or the looks of it looks like a plane element. Is that clear? These are the nodes now, that is of interest to us. What are the degrees of freedom for these nodes? What are the degrees of freedom for these nodes? Is it 2, 4, 6? 4, because there is displacement in the, how are the displacements?

Is it 4 and how are the displacements? Look at the load now. I mean we will have a look at this load. There is no in-plane, there is no in-plane loading. What is in-plane loading? I have not applied loads like this; that I have not. It is just a plate. It only takes bending. It just bends. That means there is a w displacement, but it can bend on either direction; so, theta in one direction, theta in the other direction. But, nowadays people do not use a plate element separately. Like for example, bar element can be combined with beam. Look at that beam, which we did. There is no need for a separate bar element, because that beam had the effect of a bar as well. Similarly, though technically speaking this is a plate, the in-plane behavior or the in-plane forces and corresponding stresses can also be taken into account together with this bending behavior and the result is what is called as shell, so, the shell looks exactly similar to plate, but there are lot more theory to shell.

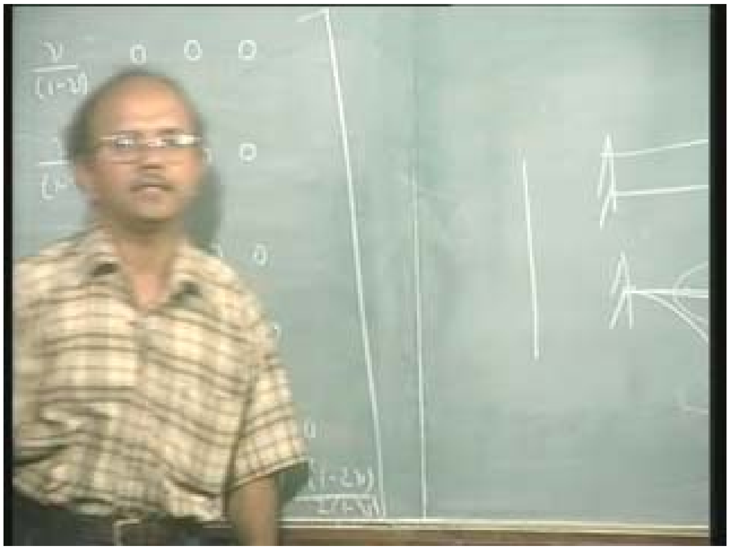
Nevertheless, at this moment in order to model, it is important to understand that when there is bending, out of plane bending and when there are loads that have to be taken in-plane what are called as membrane effects, then I would look at it as a shell. I have a problem at this stage. What is the problem? In a beam case it was easy to convince you that there are 6 degrees of freedom, but here how many degrees of freedom do you see?

(Refer Slide Time: 37:12)



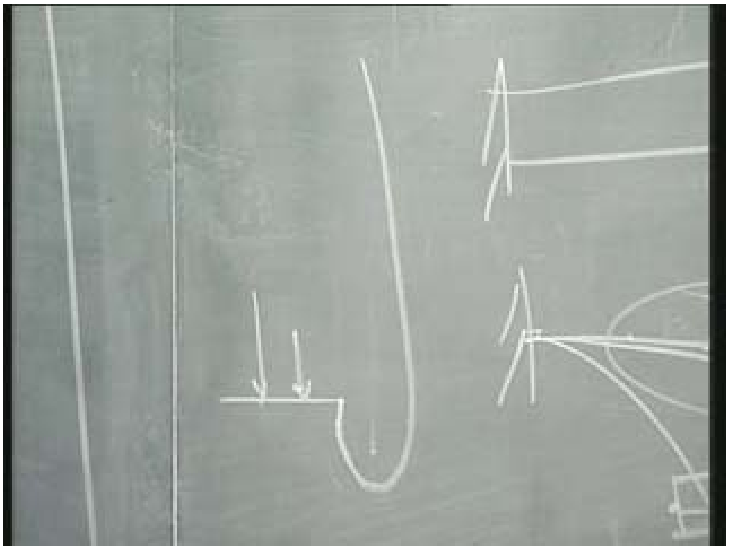
Not 3; 3 plus there is going to be displacement in, say, this is x_1, x_2 and x_3; so, 5. Is that clear? But the other, that means that there can be rotation in this and rotation in this and so on but the rotation in this particular plane is a sort of a problem to understand, right now at this stage. The only thing right now you can say that the stiffness in that direction will be what, small or high or very high? That degree of freedom is called as drilling degree of freedom, because it is something like perpendicular to the plate or the shell and ……. degree of freedom. But, unfortunately we may not deal with only planes, when we deal with this kind of shells.

(Refer Slide Time: 38:35)



In fact the same shell that you saw as an example in one of the slides is that, the side wall sheet is not a plane like this; it is not that. When you look at it from the side, it is not a straight line like that. It has quite a complex shape.

(Refer Slide Time: 38:50)



In fact the shape is something like that. This is a cross section. So, this sheet is very thin and it has a cross section which is something like that. When I now model this sheet with a number of shell elements, then at these joining places, you can easily imagine that if I do not have a fifth degree I am going to get into trouble and so on. We will expand that; if we have time we will talk about shells later in this course, but right now, it is important to understand that, just to keep in mind that, the fifth degree of freedom, it is a question mark. What am I going to do with the fifth degree of freedom?

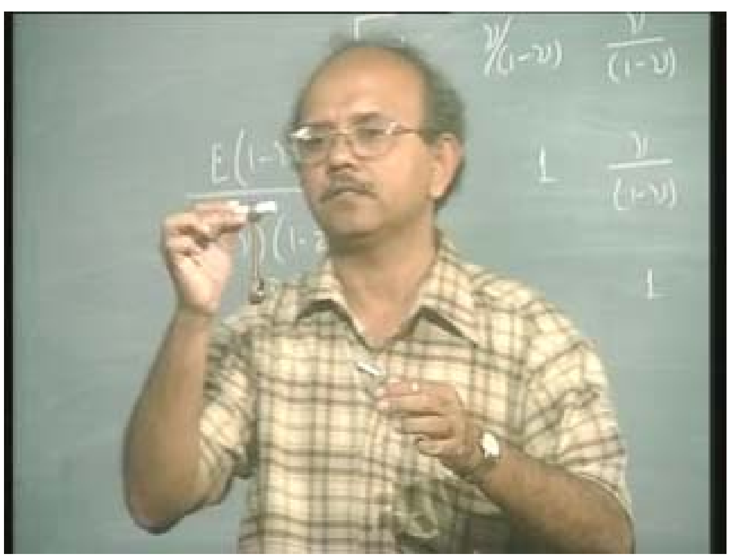
There has been lot of work that has been done; some assumptions, some simplifications, all these things have been done as far as that degree of freedom is concerned. We will not worry about that right now, but my whole aim at this point of time is to tell you what a shell is and where it can be used. In this case you cannot use a plate, because there can be in-plane stretching, because this sheet is not only going to be bent but also going to be stretched because this is welded to the under carriage and hence this behavior becomes complex and hence we have to use a shell element. But as I told you before, most of the commercial packages have plate shell element and hence we do not have much of a problem. One can argue at this point of time that shell is nothing but a plate plus, no; what is it? What? No, plate is this, what is this in-plane behavior, thin sheet in-plane behavior? No, bar is only to beam; plane stress, that is it.

At this point of time, one can argue that, look this behavior is very similar to plane stress plus plate. So, can I do the same thing as I did for a beam element where I had a bar effect and a beam effect? There are formulations like that where people just add without cross inking, plane stress and plate behavior without cross linking, without delineating or uncoupling degrees of freedom. There have been cases, which have worked well as well, but there are lot more issues in shell. Shell finite element analysis is one of the hotly debated, extensively researched, still being extensively researched area in finite element analysis. The last word is yet to be said. There are still issues which have to be sorted out especially in case of nonlinear and contact and other things and hence still it is a very highly debated area. Usually we cover the shell theory and shell elements only in advance classes. We do not usually introduce it at a lower level class or on the introductory class, because it is not very easy to understand. If time permits, later in the course 1 or 2 days I will just highlight the issues and the approaches towards the element formulation for shell. But that does not mean that you will not use shells, but as long as physically you understand what a shell is and how to use it and right now get convinced that 6 degrees of freedom, somehow people put it inside. It should not be an issue to use it in a practical case.

Hence that particular problem that you saw for the coach is modeled by means of, what are the two types of elements? Beam as well as shell element. At the end of these discussions, you know that the major elements are solid element, plane strain, plane stress, axi-symmetric, beam and shell. These are the major types of elements apart from a simple spring element, bar, I have included it into beam. If you want, you can exclude it and talk about a bar or a truss element; it may be good and so on. These are the major types of elements and you also know what the degrees of freedom are for each of the nodes that go and occupy or that go and press in this element. We are still talking about only what are called as lower order elements. We have not yet come to other niceties of it. We are only talking about how to classify or what type of element to use? Is that clear?

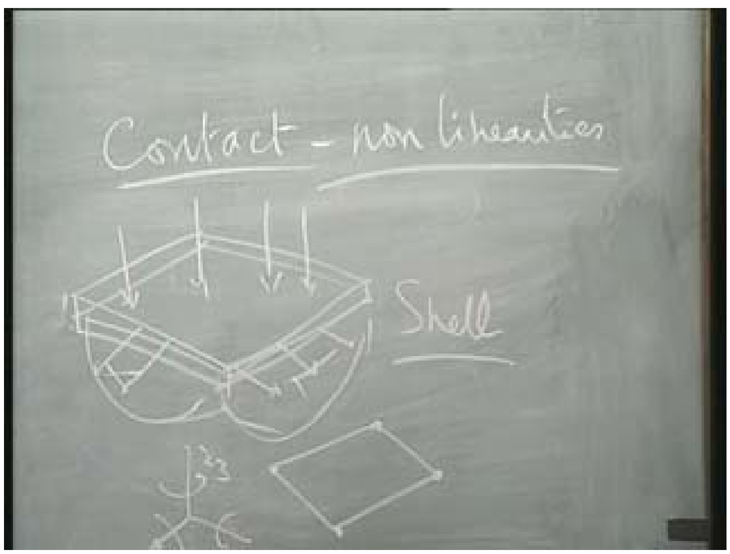
Apart from this there are quite a few important elements, especially for mechanical engineers. Now, let us look at this connecting rod.

(Refer Slide Time: 44:16)



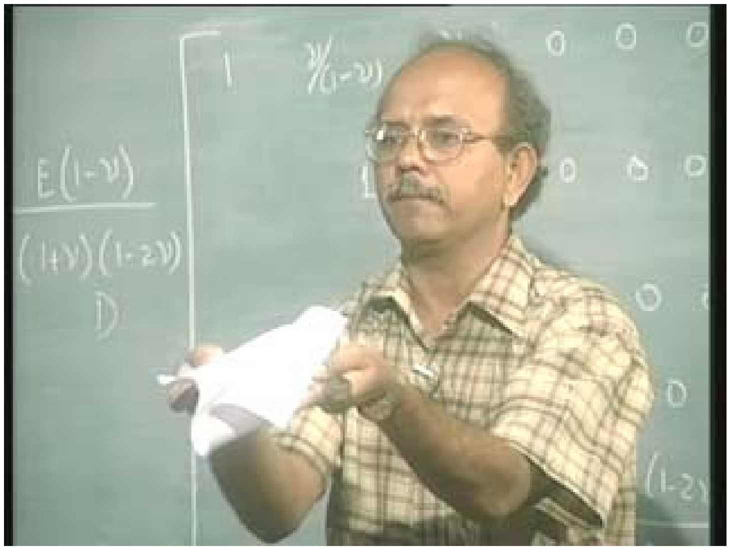
Connecting rod is very easy example, because people understand the good things to learn about. Now, inside this small end goes the, what is it? Gudgeon pin; Gudgeon pin goes inside this. It is in contact with the gudgeon pin and inside here is the crankshaft. When I model this connecting rod, I have certain things to settle about with respect to this pin. What are the issues here? The first question that I am asking is whether this connecting rod will be a loose like this or will it be tight? It will be loose. How am I going to model that part between the connecting rod and the pin, gudgeon pin? How I am going to model this? There are lots of things that requires to be modeled by elements which are different from what you have seen.

(Refer Slide Time: 45:33)



For example, this kind of situation has to be modeled by using what are called as contact elements, which are called as contact elements. These contact elements model the contact between two bodies. These two bodies can be such that one of the body can be a rigid body, it can be assumed to be a rigid body and another body can be assumed to be a deformable body or the contact element can be defined between two deformable bodies. This gudgeon pin can be assumed to be deformable as well and contact elements can also be defined within the same body. Suppose I take a sheet, I take a sheet. Now, I apply loads such that the sheet starts buckling. The same sheet can come into contact at some point.

(Refer Slide Time: 46:52)



Look at that, there is a contact. This is what is called as self contact. Contact can be between two deformable bodies, one rigid body and one deformable body or contact can be between different parts of the same body. Is that clear? They are special elements; these contact elements are special elements and more importantly they introduce what are called as nonlinearities. Once I introduce contact, then the problem is no more linear, the problem becomes nonlinear. Again we will postpone a discussion on this to a later class, but nevertheless your knowledge as far as now, the present moment is concerned, you should have or you should keep in mind that there are elements like contact elements which are also to be used, when the situation warrants, when there are contact between different bodies. Is that clear? Yeah, any question?

Yeah, but contact cannot exist between two rigid bodies because then we cannot model them using finite element analysis. So, only one of these three conditions we will use contact. In fact in mechanical engineering contact becomes extremely important as a modeling tool. We will see why that is after about 5 or 6 classes. Now, with this background, let us again get back to certain derivations from solid mechanics and ultimately land up in the derivation of a stiffness matrix. That we will do in the next two classes.