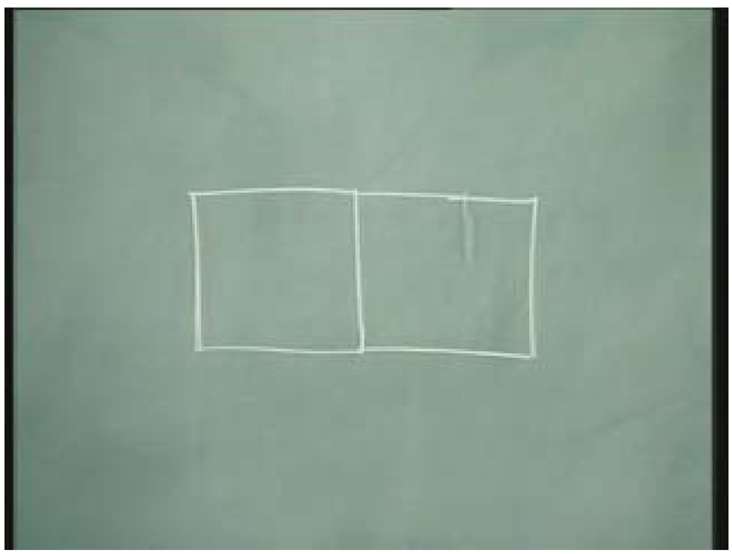
Introduction to Finite Element Method
Dr. R. Krishnakumar
Department of Mechanical Engineering
Indian Institute of Technology, Madras

Lecture - 28

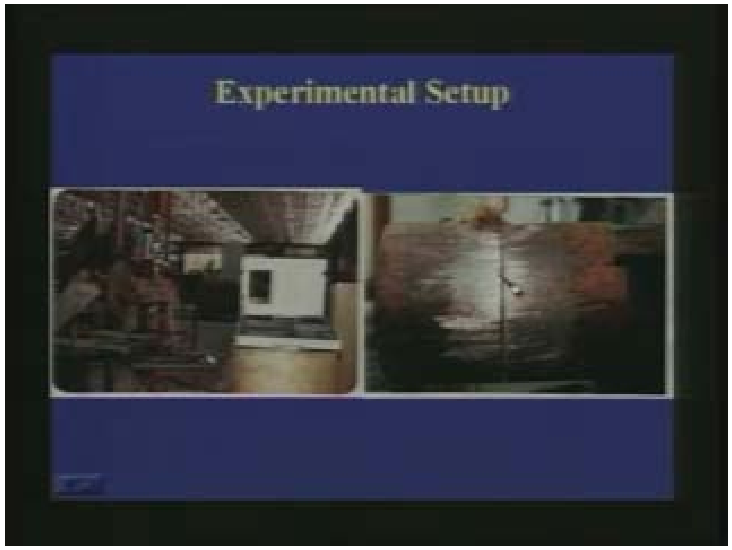
Yesterday we were discussing a very interesting case study on pipe welding. Remember that we said that this problem is one where we have a coupled analysis to be carried out. We have to have a thermal problem; we have to do an elasto-plastic problem. This is a very interesting problem, because we can learn quite a bit from boundary conditions. So, I thought it is high time that we discuss this as well.

(Refer Slide Time: 1:35)



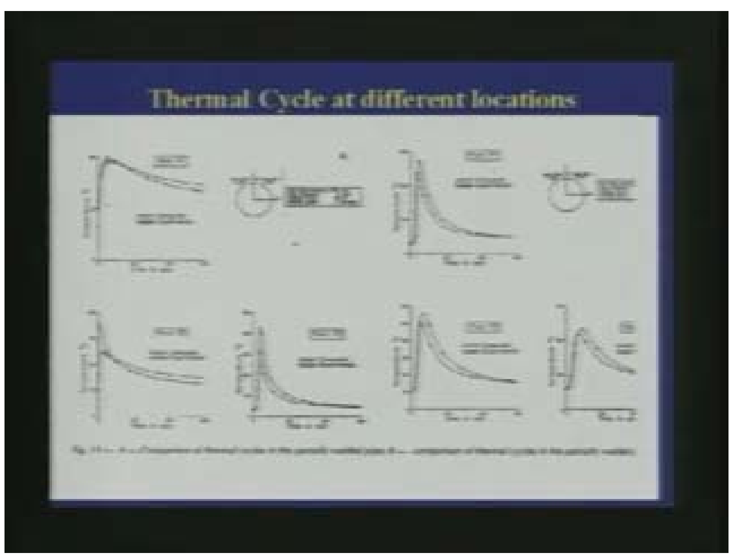
Instead of I telling you what can be the boundary condition, let us see how to develop a boundary condition for this problem. In order to do that, first let us look at the experimental set up. Look at the experimental set up now and you can pick up some very interesting things from that experimental set up.

(Refer Slide Time: 1:53)



See that the experimental set up gives you a close view of the welding that is being carried out. Especially on the right hand side, we see two pictures. In the right hand side, you see that the two pipes are joined. You also observe that there is a small gap. You know, you can see a small gap. As this cursor moves you can see that at that portion there is a small gap between the two things.

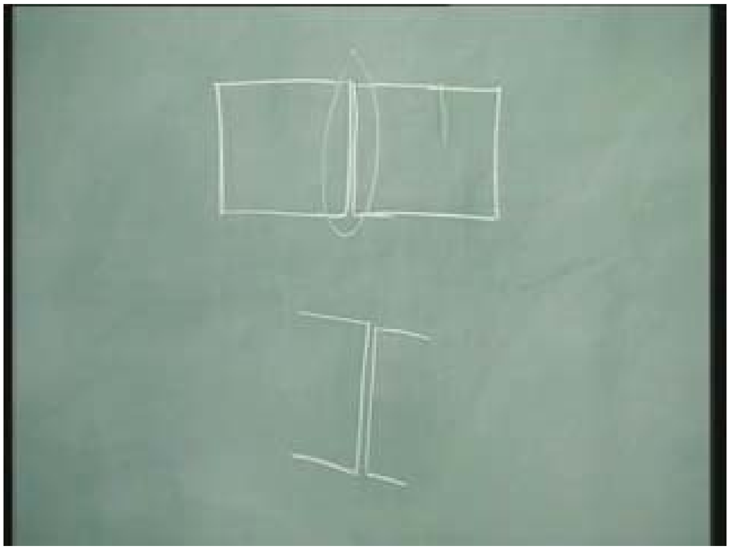
(Refer Slide Time: 2:22)



These are results, but let us not go into the results now. Let us now get back to our board and see how we are going to a model or put down boundary conditions for this.

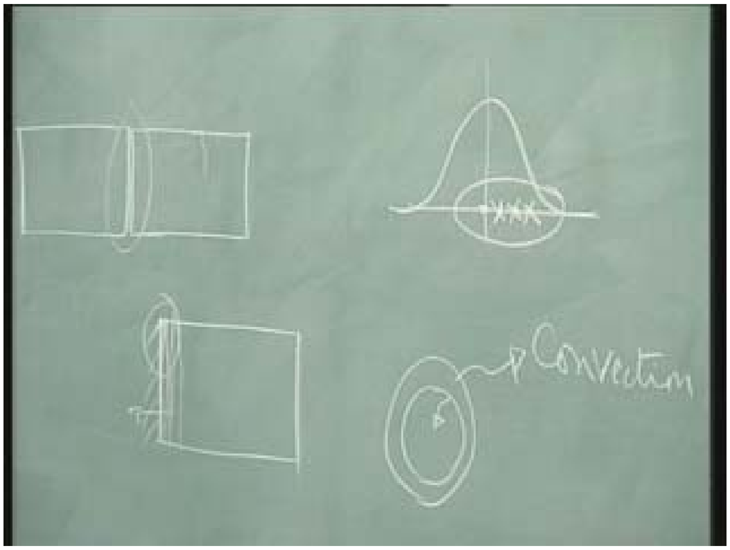
In other words, this is a very practical problem. What does it mean from even our drawing the sketch?

(Refer Slide Time: 2:40)



It means that actually I have two chaps who are as close as that. You can see that if I just magnify this that means that there is a gap between the two, there is a gap between the two. Now, I have to put boundary condition. We have discussed the thermal boundary condition. That did not look very difficult, that did not look very difficult for us. But, what about the other boundary condition for the mechanics problem? How am I going to put the mechanics boundary condition? All of you said that we can take symmetry, fine.

(Refer Slide Time: 3:35)

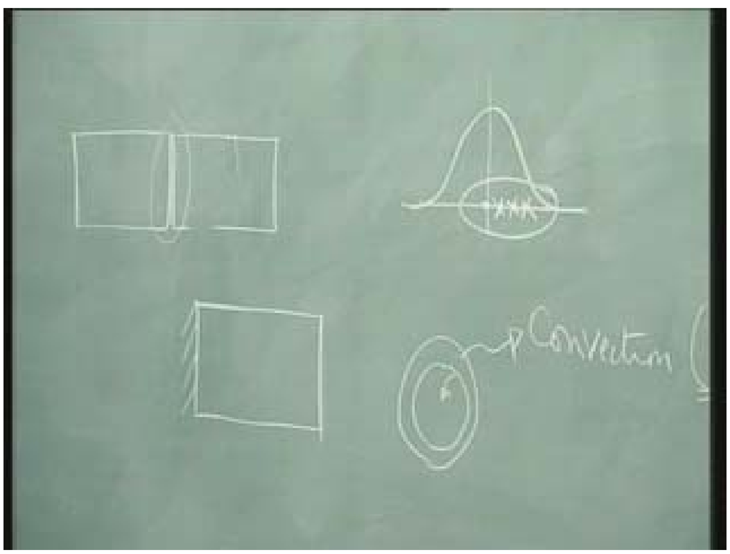


Actually this is a pipe. This view looks like this. This is a pipe like this. We said that we will use a shell element throughout, 4 noded shell element to model this. That is all fine and also the thermal boundary conditions were very clear; thermal boundary conditions were very clear, because we said that we can put that there is no heat transfer perpendicular to it. It is insulated. This side is insulated and that heat transfer takes place from this through convection and that since the pipe is large, you can also include convection from inside, though the h value would be different. Note that h value would be different from the top as well as from inside, which may be much lower than outside and you can also include radiation boundary condition and that the input is given. As the arc moves along this line, the input is given as q and that if I just magnify this place, then you can see that that is the centre line. That point is same as this point. Then you can see that the flux is or has a Gaussian distribution.

In other words, all the nodes, I have number of nodes here, all the nodes sitting here in this place which I say for example are here, are going to have the input, heat input. Look at that quite closely. Look at this, at these points. They are the nodes that are sitting very near in the place. As far the heat transfer boundary condition or thermal problem boundary conditions are concerned, it is not a big problem. But there is an issue when I put down the boundary condition for the mechanics problem. How am I going to put down the boundary conditions for mechanics problem, especially say, as I weld it? How am I going to put it? Is the question clear? Let us say that I am holding it. For example, I may not hold it, but in this case I might have held the pipe at certain position, hold it very firmly, because welding there may be some fixtures. Assume that there are fixtures; assume that there are fixtures here. So, at that fixture position, you may hold the nodes, but it depends, you may not even have a fixture. Because, large pipes when you weld it, you may not have a fixture to it.

Then how do I put the boundary condition especially at this place? What are the issues here? The issue is that there is a gap, there is a gap. Can I go and fix all the nodes? Let me redraw this position; let me redraw that place.

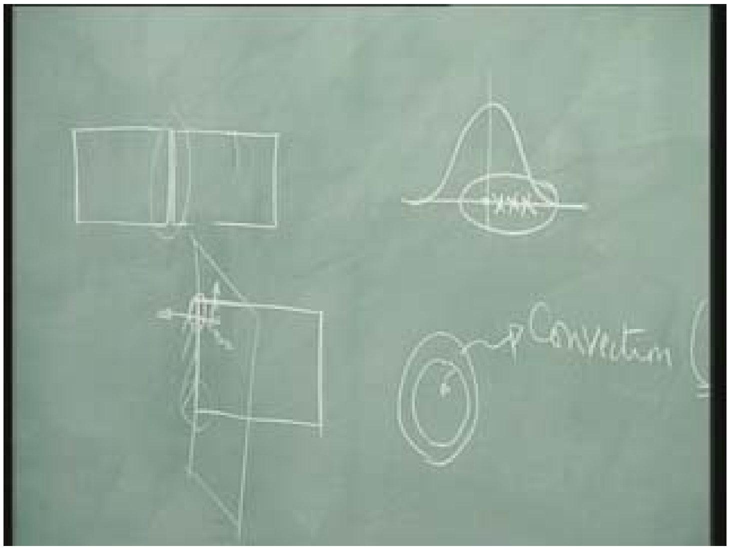
(Refer Slide Time: 7:15)



Can I go and fix all these nodes here? No? Very good. Why is that I cannot fix it? Yeah; you know from a mechanics point of view, I cannot fix it, because there is a gap and there may be a small expansion, which may actually be accommodated or if there is no a small gap is there because usually in the welding procedure they may leave a small gap; if they do not leave it and they are, you know, flush one against other. Let us start from there. Suppose they are flush, you know one against the other, they just touch each other, then how do I put the boundary condition? Boundary condition is now very simple. Then you put this and so on. You just fix all the nodes which are sitting in this plane, be done with it, because there is a symmetry boundary.

Infact, what boundary condition you would put? You would put a symmetry boundary condition. But, is it correct to put it like that? No, very good; so, how do you put the boundary conditions, because some accommodations will be there.

(Refer Slide Time: 8:40)

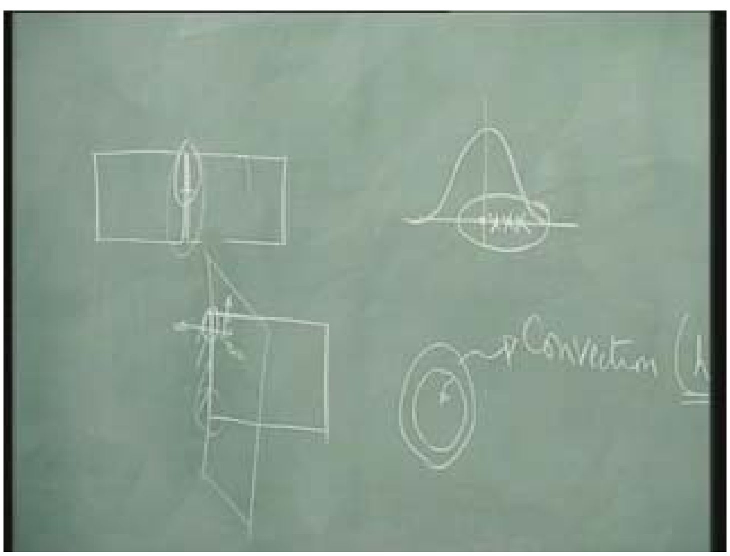


Suppose I started welding here, some accommodation will be here in this portion. I cannot say that as I start welding here, these portions which are at the bottom here they would be flush and will be restrained. I cannot say that. So, how do I handle it? Is the issue clear? Are you convinced now about the issue in this boundary conditions? Yeah? No? No? Now, let me get back. These are the niceties of boundary condition. That is why I took this problem to explain it. How do I put a boundary condition? Say, I am welding these two plates. How do I put a boundary condition at this interface? That is my first question. You said yesterday that we can take symmetry. I will agree with you.

Let us say that we take symmetry about this section. Is that clear? But, now I have raised a doubt. What is the doubt I have raised? I said that there is a gap. Can you put symmetry boundary condition here? That means that the displacements will not be there in this direction, symmetry boundary condition and in this symmetry plane no rotation will be allowed. When I put a plane here, that is a symmetry plane, then no rotations will be allowed along the symmetry plane and perpendicular to the symmetry plane, we do not allow displacement. This is symmetry boundary condition you wanted to put. But, are you correct in doing it? That is the question.

Why is that you are not correct? Because actually they are just, they may be flush or there may be a small gap. They may not touch each other and restrain actually their movement. The nodes movement may not be restrained. When do you say it is symmetry? When the nodes do not move at all along the boundary, then you say that there is restraint, but in this case can you say that there will be complete restraint? No, so, then how do you do it? Are the issues now clear? Yes. How do you put the boundary condition and how do you solve this problem? Actually what is happening? Just think about it, you know, just think what is happening to these nodes? Yeah, I will give you one more clue. What happens to these nodes with respect to time? Because I am marching now in time, what happens to these nodes with respect to time? Yeah, so, the clue here is that there is a change in the boundary condition with respect to time.

(Refer Slide Time: 12:04)



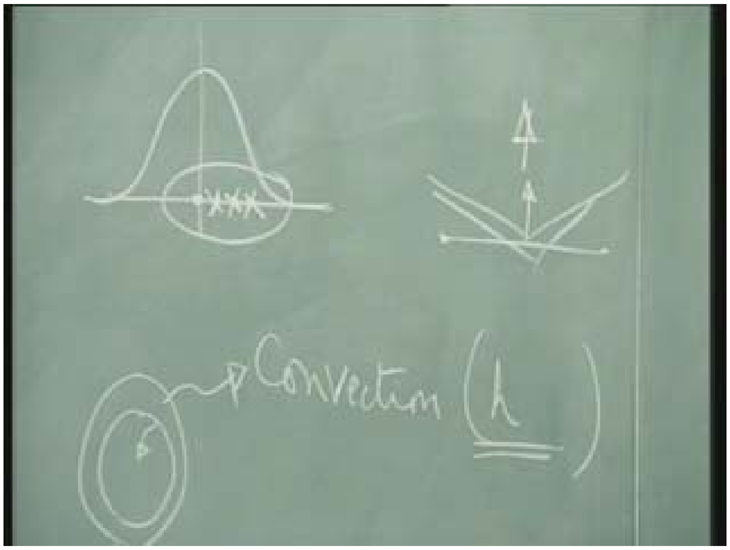
For example, suppose I go to this point. I say, I have welded up to this, anyway it is a three dimensional welding, does not matter. Suppose I just welded only up to this. Then up to that point I can say that there is going to be a symmetric boundary condition. There is no gap, both of them are welded; hopefully the fellow has done a good weld, so it is going to be one integrated piece. So, no expansion, thermal expansions and all those things will be allowed. At that place, I can still assume that there will be a symmetry boundary condition, but as I go down, as I go down, it may not be possible to hold this kind of boundary conditions in these places. What is that I can do as a first approximation? There are lot of issues here. It is not a straight forward problem. As a first approximation, I can say that I will change the boundary condition with respect to time. That is possible.

As I keep welding, as I keep welding, I can change the boundary condition, with one step to another step. I can say that in the first step, I will fix the boundary condition or I will release the boundary condition and later as a next step, I will fix those nodes. Is that clear? It is possible to change boundary conditions, but may not be possible to do it in every commercial software, but some of the softwares it is possible to change the boundary conditions; it is possible to change them. Is it clear? So, you fix them up. As you do the problem, you can fix up those nodes when they are being welded.

The first lesson is that you can change the boundary conditions. But, when you start the problem, at least some, one node at least should be fixed. You have to remove the rigid body motion. That has to be taken care of properly when you start it. May be even the first node, when you start it you may have to fix that node, when you start welding it. That node has to be fixed, very first node in the very first time step. Please note that many times it may be required to change the boundary condition with respect to the steps. As you move further, it is possible to change the boundary condition.

Some of you may think that why not I apply a contact problem or why not I make it as a contact problem? Put a contact element between the two, like what? If you had noticed one of our earlier examples, for example a dye or the punch and the material, the dye and the material they are in contact. So, you can put a contact algorithm between the two. Can I now put a contact algorithm between this body and this body, may be your question. Can I put a contact algorithm there? Very good; pardon. No; no no, they will be in contact later. The problem will be that contact elements, contact elements are such that when there is a tensile load due to some reason, there is a tensile load say for example and the gap increases then the contact element will not act. Only it will stop penetration or in other words penetration which I call delta, say for example, penetration one.

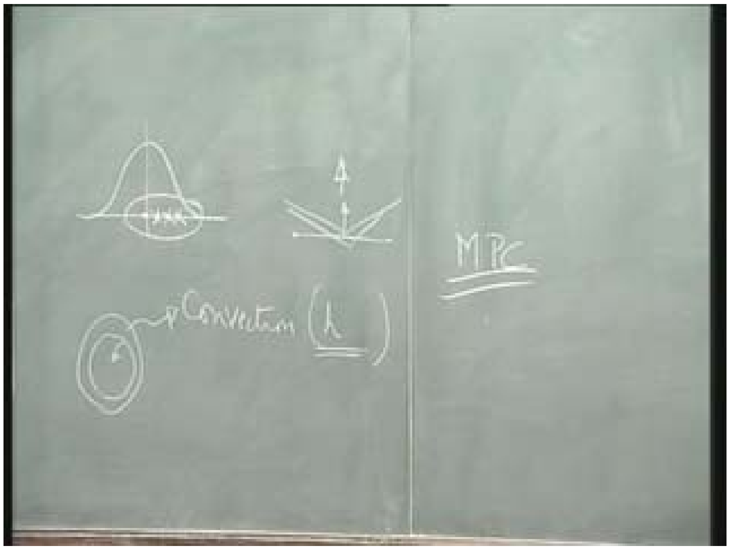
(Refer Slide Time: 16:21)



There are two nodes and one node trying to penetrate, that penetration is what is stopped. On the other hand, if a node is sitting here like this and there is a tensile load and the node starts moving out, contact algorithm will not do anything. A contact algorithm is only there to prevent the penetration of one node into the master node or master nodes or the master surfaces or the master line or whatever you can call it. Is that clear? So, you cannot use contact algorithm.

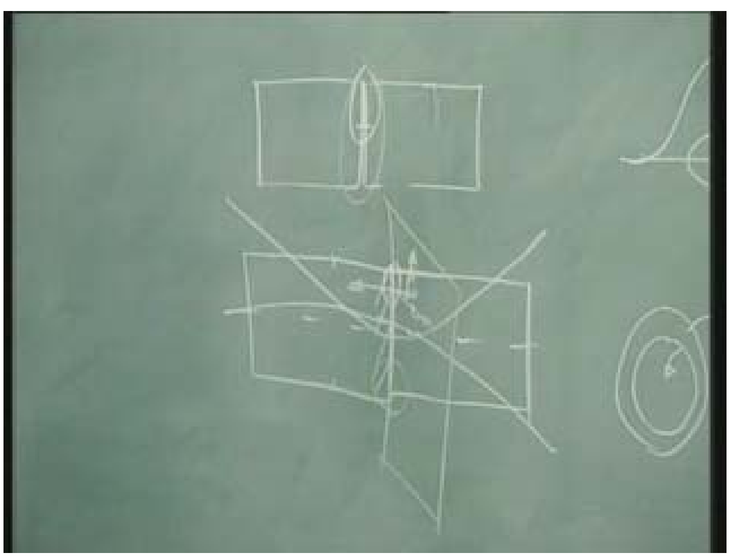
On the other hand, there are other techniques, in which case you cannot use symmetry. You can tie these two nodes on either side as you weld it. As you weld it, you can tie these two nodes and through what is called as multi point constraints.

(Refer Slide Time: 17:35)



When two nodes behave in the same fashion, you write down what is called as multipoint constraints or MPC’s. We will see about MPC in the next class; if there is time we will see in the next class. But, yeah, yeah; absolutely, I said I accept your argument for the time being. But, see what the distortion is. That is right. How does the distortion take place? How does or how does the distortion take place?

(Refer Slide Time: 18:11)



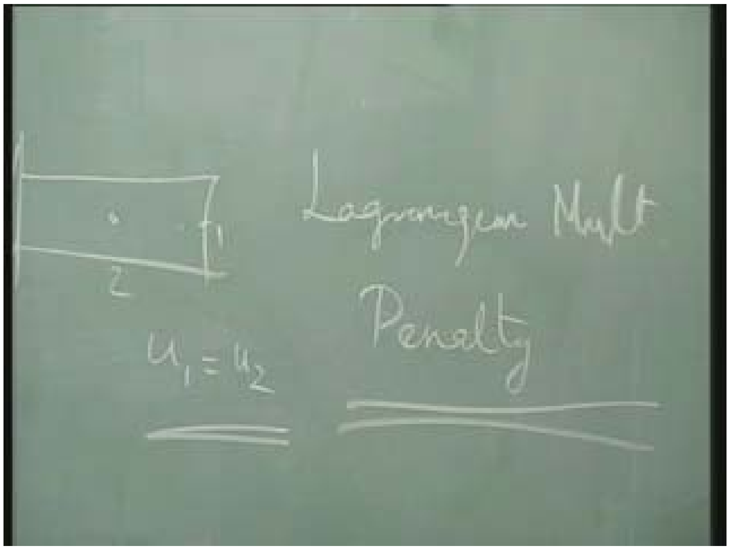
See, when we say axis distortion there are lots of disadvantages by considering symmetry. Suppose there is an axis like this, how does the deformation, distortion take place? Does it become like this, in which case there is no problem. On the other hand, many people what they do is to hold this and start welding this alone, in which case the distortion may be like this. Then you cannot use symmetry. In other words what it means is when you want to use either, that is why I was slowly moving towards a situation where symmetry may not be very nice.

Though geometrically it is symmetric, you have to see whether the distortion that you expect is going to be symmetric or not symmetric. Whether the symmetry is induced or the unsymmetric deformation is induced by certain conditions that you pose on this piece. Suppose, you are going to hold this in place and weld this freely, this freely, then definitely there will be distortion. This is a very important question. In other words, you know, it need not be with respect to only pipe. In other words, geometric similarities or symmetries do not mean that you can apply symmetry boundary condition.

What causes symmetry or when do you apply symmetry? You apply symmetry when you are sure that the physical behaviour of the structure, you should have some understanding. I mean some commonsense, even commonsense understanding would make sure that about the symmetry plane, about the symmetry plane on either side, on either side of the symmetry plane, the deformations would be symmetrical. Is that clear? This is a choice which has to be made by you as an engineer. Sometimes it may be that it is not symmetrical and this kind of problems also has to be handled. Another important behaviour that is available in many softwares is what is called as MPC, multi point constraints.

What are multi point constraints? You can make two nodes to behave or to behave in such a fashion or be in such a fashion that they would deform by the same extent.

(Refer Slide Time: 21:13)



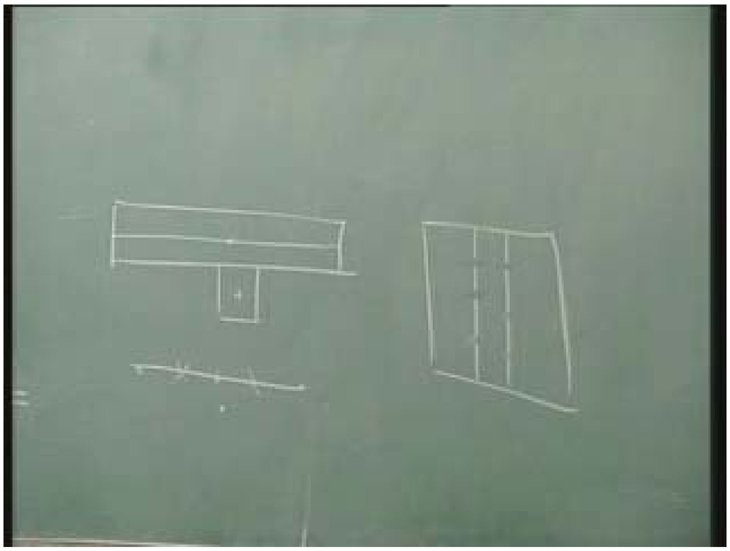
In other words, suppose I say I have a beam, say for example I put two nodes. It is possible to write down a multi point constraint between these two nodes, which means that the displacement of say node 1 say u_1 is equal to u_2 and so on. It is possible to write down constraint equations between the degrees of freedom of different nodes. Is that clear? It is possible to write down constraint equations between the degrees of freedom. What is constraint equation? You can say that u_1 is equal to u_2. This is a constraint, which means that the part which joins u_1 and u_2 is very rigid. It does not deform. Then you write down a constraint equation.

These constraint equations are then taken into the main our main potential function through what are called as Lagrangian multipliers. There are two techniques to handle this. They are Lagrangian multipliers or penalty functions. Any constraints are handled by means of Lagrangian multipliers or penalty approach. We will not digress here, because I want to come back and look at a few more things. But nevertheless, in a later class, we will explain what is Lagrange multipliers and penalty approach and how you can put down MPC's. MPC's are very, very useful techniques.

At one point of time, you want to tie two nodes. Then it is important that you put a MPC. For example let us i me let Let us look at another example. There, you are going to use it. How you are going to use it is different. If you are using a package, then you need not worry about how to use Lagrangian multiplier and penalty. But nevertheless, it is important that you know what they mean. That we will look at in a later class and let us see how else are you going to use this kind of thing?

Let us see that I have a, let us say that I have a sheet or a plate.

(Refer Slide Time: 23:40)



On the plate, I have some reinforcing bar. On the plate, I have some reinforcing bars. For example, in the figure that you are going to see now, you will see a coach, railway coach and this railway coach has a number of reinforcements. We will see that in a minute, we will see that in a minute. But before that, let us look at the theory. Let us look at the theory. This is the plate and this is the reinforcement beam. That is the plate and the reinforcement beam. Let us see how we are going to model these two things. There are number of, for example, the railway coach will have number of reinforcing beams. What will be the type of element, say for example, you will use for this? Plate or a shell element, say 4 noded shell element. This 4 noded shell element would be such that it would pass through the centre here.

Now, let us have a look at the beam. So, I will use a beam element. In the top view, this would look like this. Though I am not following the conventions of drawing, it does not matter that it would look like this. I mean you will not see it. I know, smile in some of these guys, I should put it at the top and so on. See, anyway that it looks like a dotted line and I hope you understand that it is below this. Now, I want to replace this by means of a shell element and a beam element. Where would I do it? Where would I do it for a beam element?

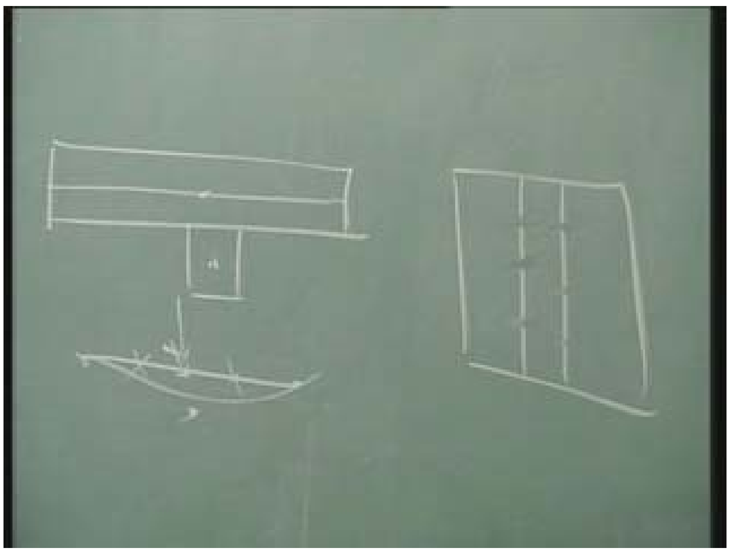
I have to do this at the centre or the centroid of the beam. When I reduce the situation to a modelling situation, what would I ultimately have? I will have one plate element like this and one beam element like this. This is the view perpendicular to the board. Is that clear? Now I have, I am in trouble. May be one more suggestion you may give. Divide it into two plate elements, so that you will have a node here that is two plate elements. This is one plate element and another plate element, so, I will have two plate elements. Let us see how you have to model this? Before that, let us look at the coach now, to understand this picture more clearly.

(Refer Slide Time: 26:46)

Now, you see the coach there. You see that there are number of vertical bars there. At the end of the coach, you see that there are number of vertical bars. This is a solid model. You see that red side wall. Now, all of you are familiar with the Indian coach, that red side wall and when you peep inside this, actually these vertical pillars extend to the side walls as well. You can see them as small yellow lines there. If you just look at it carefully, you will see in the side walls some small lines. Nevertheless, what it means is that the side wall sheets are reinforced by means of beams, vertical beams which run throughout from the roof onto the under carriage of the floor. They are reinforced. The situation is exactly similar to what we have been seeing here.

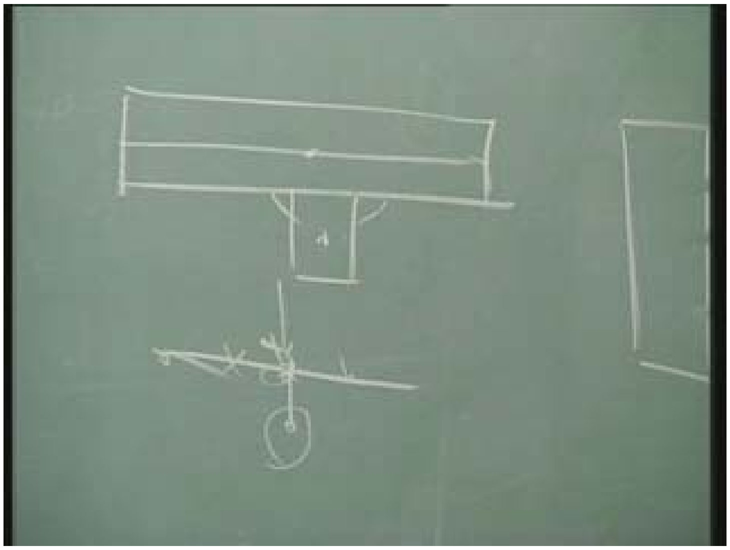
This is something like the side wall and this is something like a beam. Now, you can tell me that why you want only one plate element? Why not you put two plate elements? I will have a node here, I will have a node here. But my beam is still standing away from my plate. In actuality, there is a load transfer between the two and in fact they are reinforcements. Is it clear? On the other hand, when I plot like this or when I put the elements, then they do not go together. Do you see the issue? If at all now I apply a load here, what will happen? The beam will not have any effect. The fellow will be standing in the air and now they will start, this fellow will start deforming like this.

(Refer Slide Time: 28:39)



Is it clear? I have to now define the rules of the game at this place.

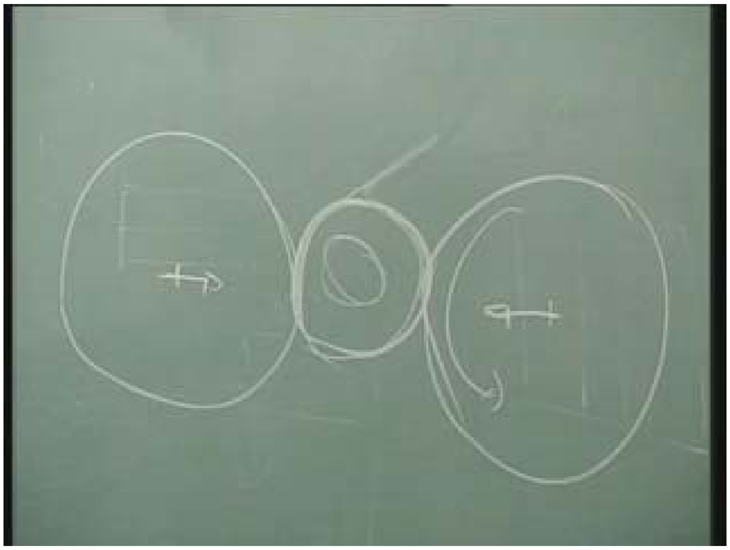
(Refer Slide Time: 28:54)



I have to say that look it is not like that and that this node and this node here, they go together. They are actually tied, because I would have welded it or done some, you know something has to be done here, say may be a number of spot welds or welding which could have done, which could have been done here. Whatever, however you do, they have to behave as one piece. That is the whole idea. You may say that look this node and this node are actually tied. They are the same. They should behave in the same fashion; then only will I get the reinforcement of it. Then I write an MPC between these two.

Situations like this warrant, warrant the introduction of multi point constraints. This is a small digression, but nevertheless, since we came up modelling issue, it is important that we realise that it is not very easy to put down or the moral of the story is that it is not very easy to put down a boundary condition. If you want to put down the boundary condition, then you should be very clear as to physically that you understand the picture completely. Then only we can go ahead and put and in order to model, you have aids like this, like multi point constraints. I mean, since we are talking about a modelling issue, I think, I should tell you one more problem. Let us just continue this, because we have not talked much about modelling issues. We have talked about elements and I have to talk about axi-symmetric element, which I gave you as an exercise in the last class; I mean, we will talk about that as well. I have a small problem, which is very interesting and let us see how you can model this problem.

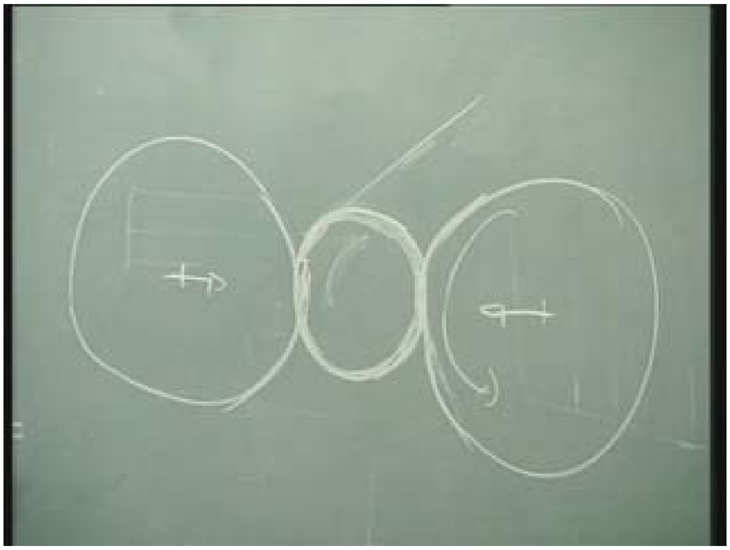
(Refer Slide Time: 31:04)



There is a cylindrical, say, piece; a cylindrical piece, you are looking through the board or in other words the cylindrical piece extends like that. You are looking at it from one end. This piece is made up of powder metallurgy, it is made up of PM process. This piece has a density which is say 90% of the theoretical density, say 90% of 7.8. It so happens that the piece is in service, the cylinder is in service at such a position that the stress levels at the surfaces are high and the stress level inside is very low. Say for example, that this piece is used where surface stresses are high. In other words, this piece warrants a surface densification.

If I can increase the density only along the surface, I can use a powder metallurgy component straight away. Powder metallurgy components have some advantages because, they are near net shaped. You need not machine it and the cost may be much lower in some cases; not in every case, but in some cases and so on. What I wanted to do is to now roll this piece. How am I going to roll it? I put it, for example, something like centralised grinding; I put this, this roll I put it. Say, this is the centre of the rolls, rolls are very rigid. These are the centre of the rolls. Now, I apply pressure or force and start rotating the rolls. What happens? Let me redraw that clearly.

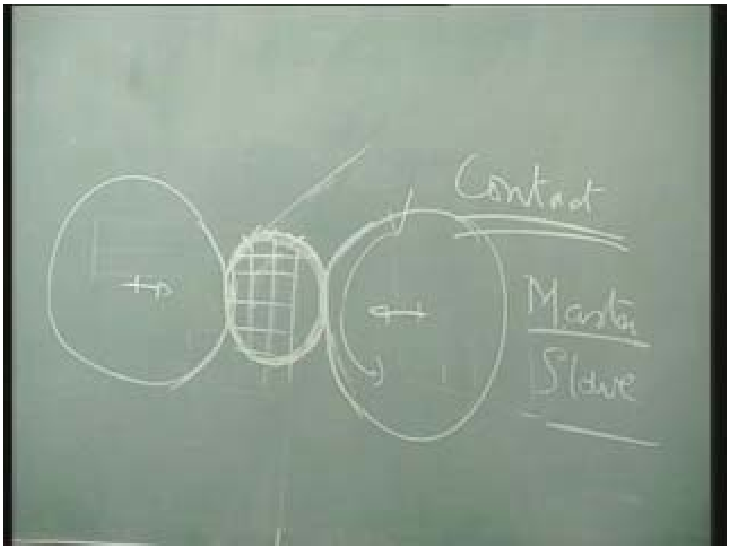
(Refer Slide Time: 33:28)



This is the piece. What happens? The piece first gets squeezed. In these locations, very near this, they first get densified, because it will be compression, there will be compression and compression is good to densify them or compact; whatever pores are there, it will go and close the pores. Now, I start rotating it. As I rotate it, we will see the results in the next class, but let us first understand this. As I rotate this, what will happen? This piece will start also rotating. As I rotate like this, the piece will start rotating in the opposite direction. When it starts rotating, what will happen? As it gets past, a layer of compacted material will form. That is the process.

I have lot of issues here. I want to find out what is the load to be applied, because if I apply too much load, then the shear bands are formed and there is failure and all that. I want to model this using finite element analysis, this manufacturing process. How do I model this process, is the question? First of all, what is the type of element we have to use in this process? What is the type of element you think we will use or what is the type of analysis? It is a long cylinder. Do you use shell? Because, it is a long cylinder please note that it is a long cylinder and the cylinder compresses. So plane, fantastic. So, you will use a plane strain, so that you will do a two dimensional analysis. If the diameter and the length of the cylinders are comparable, they are almost the same, then you may have to use a 3D analysis itself. Since it is a long cylinder, say for example, I can use a plane strain or you can choose a plane strain element.

(Refer Slide Time: 35:52)



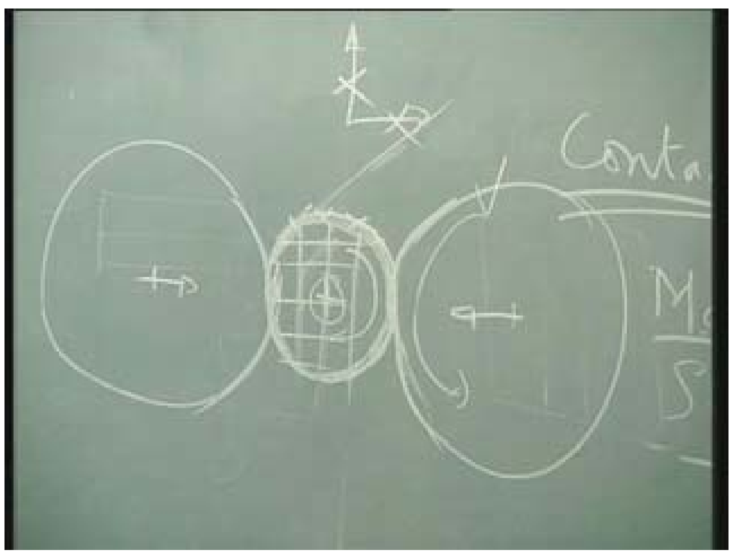
This is now discretized; this body is discretized by means of plane strain element. I had already commented that the two rolls are rigid. This roll and this roll are rigid, which means that I can use what is called as a contact element. I can use what is called as a contact element. Contact elements are very, very important. I have been telling you quite often that they are very important in practice. Especially if you want to do manufacturing simulation, contact elements are important. Many times even in a regular design processes you may have to use contact elements or else the results may not be very accurate. You can get a ball park figure, but it may not be very accurate. But in this particular case, we have to use a contact element.

Contact elements can be defined between say two deforming bodies or between a rigid body and a deforming body, between a rigid body and a deforming body. The contact elements have what are called as master nodes and slave nodes or master surface and a slave node sets. These are the two things that you define. The slave nodes are the nodes which belong to the deformable body or when the stiffness is lower of the two bodies, then we choose those nodes which belong to that body to be the slave nodes. The nodes here, all these nodes sitting on the surface, are the slave nodes and this happens to be the master surface.

An algorithm is written to see that the nodes do not penetrate the master; the slave nodes do not penetrate the master. We define this as a contact element between the two or we define a contact element between this rigid and this deformable body. I have to apply force here. Can I, apply force directly here? Can I apply just force directly here, without holding anything? Why do you think I cannot apply? I am going to rotate it. I mean, imagine that I am going to rotate it. No, direction of the force I have given already. So, what do you do?

Fine; so, what you are saying is that the centre node here, the node which sits at the centre can be fixed, because that is not going to move in the x and the y direction. I can fix it in the x and the y direction. Is that clear? Very good; then what else do I do? Just can I apply the load now? Theoretically you can apply the load, you know, without problems, because only thing now, is it free of, by the way is it free of rigid body motion?

(Refer Slide Time: 39:50)

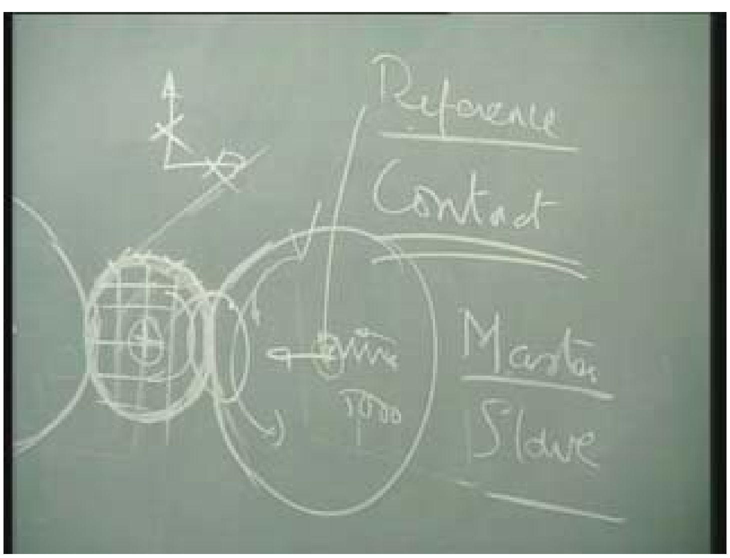


I fix x and y of this node. See this node here, both x and y, both of them are not allowed. Usually, you know, you read in text books unfortunately the sentences are not very clear. So, many students think that now I have fixed, I have two degrees of freedom. I have fixed the node in the x and the y, so, I should not have any singularity problem. That is what people think, usually that is what students think. Is it true here? No; very good. Why? Yeah, that is beautiful. It is capable of rotating. See, anytime when you put a boundary condition, when you want to remove singularity, do not blindly go by the rule that I have fixed in the x direction one node, I have fixed in the y direction one node and so I have fixed in both the directions and hence I am very safe. No.

You should go and look at whether rigid body motion, rigid body velocity or displacements are possible, inspite of you going and fixing these nodes. Now, you have fixed these nodes. As he correctly said, it may start rotating. See, there are two things. It is not again you should look at whether it is going to rotate under my load or not. It is not that kind of issue, because the singularity is not removed by not applying a load in that direction. It is not like that. You should say whatever be the load that is applied, it should not rotate. That is how you should look at it, when you put a boundary condition. When I fix this, I will have a problem. Just fix it, because it will start rotating.

Fortunately, if I properly put boundary conditions here, the contact would be able to remove the singularity problem. The contact will improve and there will not be a singularity problem. So, you can drive this whole problem by contact itself, but there are number of tricks to it. What we do now is that, we have to have, we have to place this complete system in a much more stronger footing or robust footing I would say; robust, within quotes, numerically robust footing, than what it is now.

(Refer Slide Time: 42:50)

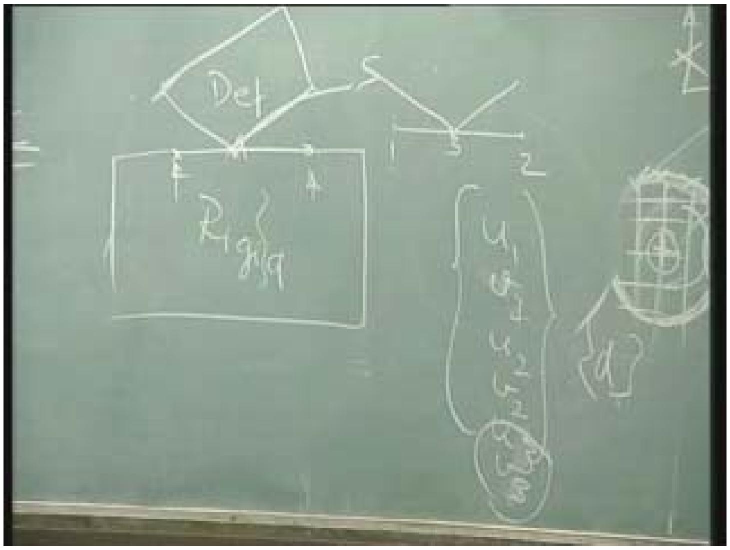


What we do is, we put one more node here we put a spring element there, give some arbitrary k value say about 5000 or whatever it is, some arbitrary k value and if I want to apply say 1000 Newtons, I give a displacement here of 1 by 5, such that the centre here realises whatever load I want to give. So, what happens? Centre here immediately realises a load. This is called as the reference node. That node is called as the reference node. You will come across this if you use any package, the reference node with respect to this master surface. Master surfaces are defined by means of a reference node and the reference node will realise this load that you have applied and he will go and be in contact with the body and singularity will also be removed, because this contact element would be considered, I mean this would be considered completely as a system with the contact element taking care of your singularity, because you will be putting some k n normal stiffness, tangential stiffness and so on. You will not have any problem and when you start rotating, if the friction is good enough, this fellow will start rotating as well.

To summarise, what we are trying to say is that there are two things that are important. At some point of time you may have to use MPC's, multi point constraints, when two nodes behave in the same fashion or move together or in certain times you may have to use a more sophisticated contact analysis or contact elements have to be put and the contact elements are defined by what are called as master surfaces and slave node sets. Master surface can be deformable like two gears when we make them …… and rotate, both of them are deformable or deforming in which case you can say that it is deformable. Deformable model is what we are interested in, in which case one of them is made the master. But, let us stick to the problem here like this, where there is one deforming body and one rigid body and we define a master and slave. See, many softwares ask which is the master? Then you say that this is the master set or this rigid surface is my master. It also asks for what is your slave set? Then you say that these nodes here are my slave set.

Point number two, when you use contact, you define a master surface or master node set and a slave node set. Point number three that this contact algorithm develops a stiffness matrix. Maybe, it is worthwhile going slightly deeper into it. How does it develop a stiffness matrix?

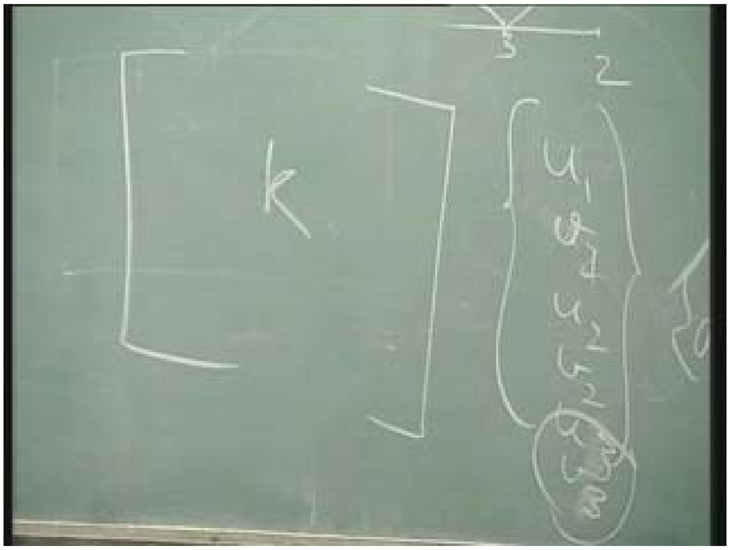
(Refer Slide Time: 46:30)



Suppose, I have two nodes which define, say a master surface, though the nodes here are not deformable, the nodes are sitting on the surface of the body, say a block or whatever it is. In this case, it may be sitting on the circle and I have a slave node that is the body say which I am analysing, a part of the body which I am analysing. This is the slave node, this is the deformable body, this is the rigid body; that is the slave node, s node, these two are master nodes. Then, I develop an element which connects these three or in other words, my d matrix, small d matrix, for a contact element is, if I call this as 1, it consists of these three sets. If I call this as 1 2 and 3, my d matrix here consists of u_1 sorry, v_1 u_2 v_2 u_s v_s or u_3, sorry u_3 v_3 I have put already 3, where 3 is nothing but the slave node. Is that clear?

So, a contact element consists of 3 nodes, but they are not separately defined nodes. Note this carefully, they are not separately defined nodes. They belong to the two bodies, two of them belong to what is called as the master body and one belongs to the slave body and if you put them together you will see that the d vector, the d vector which we usually write, we used to write u_1 v_1 u_2 v_2 u_3 v_3 u_4 v_4. I have a d vector here and this d vector consists of u_1 v_1 u_2 v_2 u_2 v_3, sorry u_3 and v_3, to which is attached a k matrix; to which is attached a k matrix.

(Refer Slide Time: 49:02)



We will not go into the derivation of k matrix. It is very complex, but nevertheless, I want you to understand to use, to model a contact situation, so to which it is attached a k matrix. We will continue with this discussion, because these are some of the tools which we pick up and before we go to the next step of non-linear problems, it is important that we realise that these are some of the tools which are very important. But, they change the complexion of the problem itself to a non-linear problem, which I am going to do in the next class and we will discuss more closely the contact algorithm in the next class.