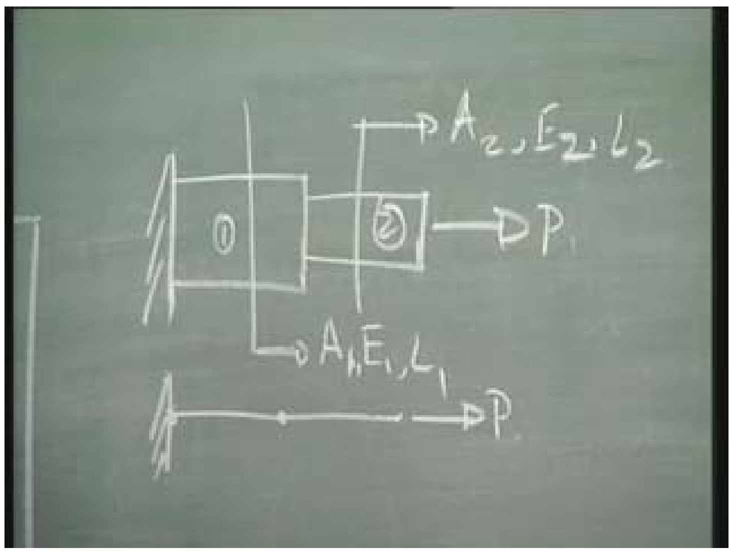
Introduction to Finite Element Method

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

Lecture - 4

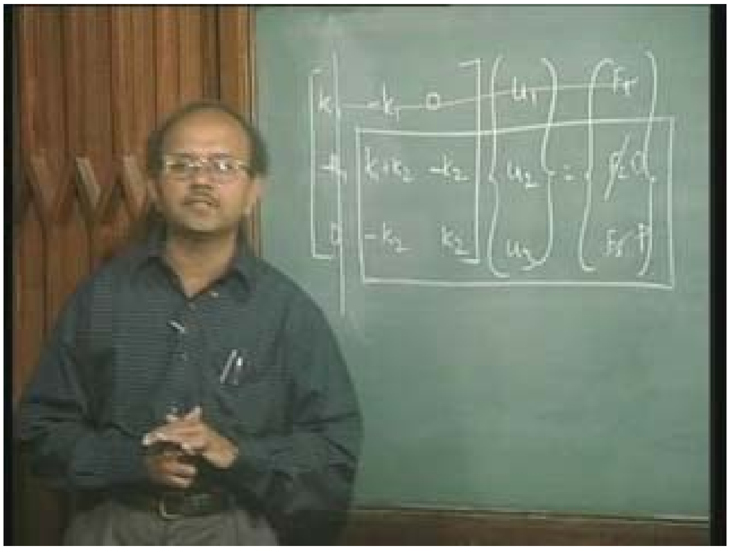
The last class we were looking at a very simple problem. We were trying to do this problem by using the finite element analysis.

(Refer Slide Time: 1:10)



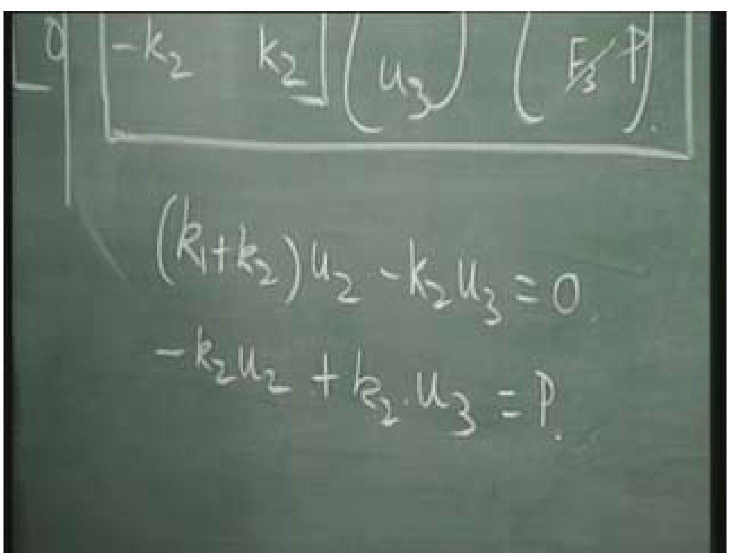
We had done the preliminary work which we will summarize later in this class, but let us pick the threads from where we left. We were here. We had defined the complete matrix which we called as stiffness matrix for this problem.

(Refer Slide Time: 1:27)



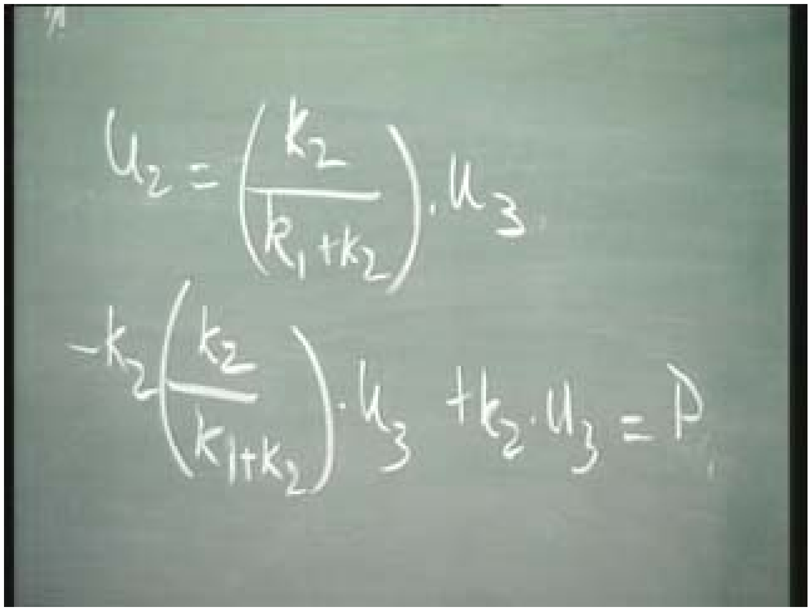
We defined the matrices for the elements and then assembled them and we got to this stage and applied the boundary conditions. It is quite clear from this stage as to how this problem can be solved. It is simple that these two matrix gives a set of simultaneous equations, two equations, simple simultaneous equations which can be written as say K_1 plus K_2 into u_2 minus K_2 into u_3 is equal to zero.

(Refer Slide Time: 2:01)



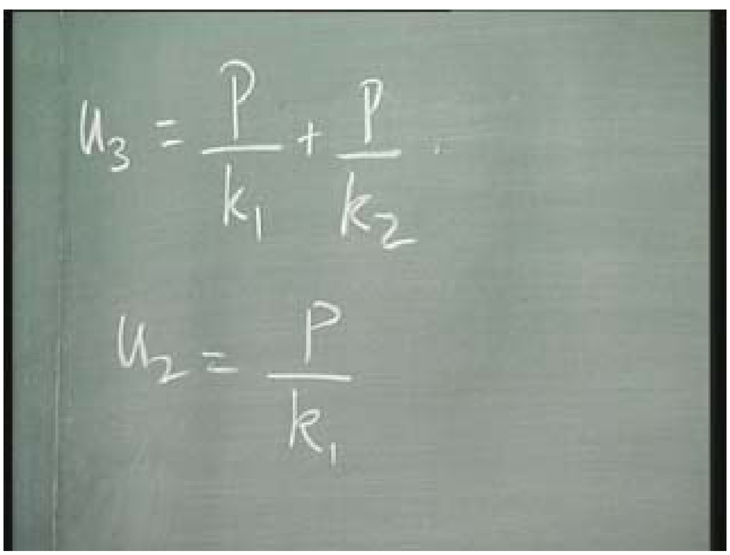
What is the next equation? Minus K_2 into u_2 plus K_2 into u_3 is equal to P. There are two unknowns, u_2 and u_3. It is quite straight forward to solve them.

(Refer Slide Time: 2:25)



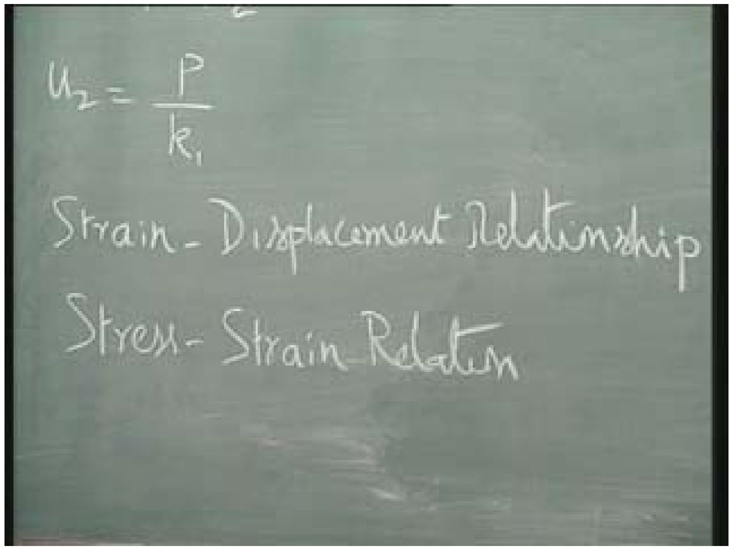
The first equation gives rise to a simple equation in an explicit form by defining u_2 is equal to K_2 divided by K_1 plus K_2 into u_3. Now what we have to do is simple. We have to substitute this value for u_2 in the second equation and then write it down as minus K_2 into K_2 by K_1 plus K_2 into u_3 plus K_2 into u_3 is equal to P.

(Refer Slide Time: 3:21)



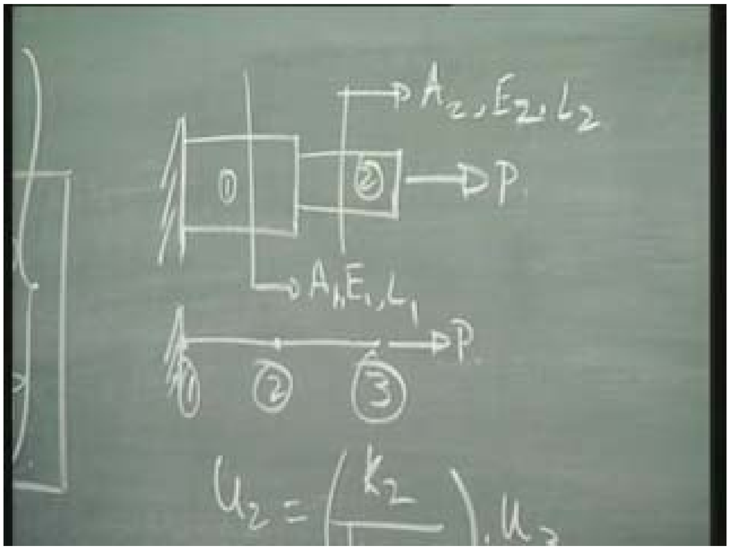
Once this is done, I have to just sort out or determine what is u_3? Solving it is very straight forward. I can write u_3 to be equal to P divided by K_1 plus P divided K_2; P divided by K_1 plus P divided K_2 and u_2 can be written as P divided by K_1. We have determined or we have found out what are the displacements for the system? Once I know the displacements, what is my next step?

(Refer Slide Time: 4:19)



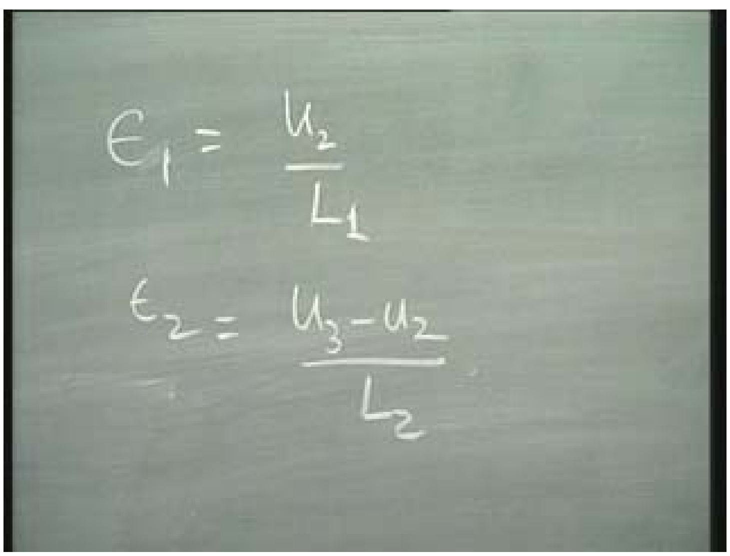
I have to determine the strain because ultimately I am interested in stress. So, what we do is we solve these equations, determine displacement. Then we go to strain through what are called as strain displacement relationship and then ultimately we determine stress through stress strain relation. So, we first determine strain through strain displacement relationship. What would be the strain? There are two strains, $epsilon_1$ and we can say $epsilon_1$ and $epsilon_2$ as the strains that exist in the two bars. In this particular case, we can use a simple formula of change in length by original length. So, I can determine $epsilon_1$. What is $epsilon_1$ in this case? Look at this.

(Refer Slide Time: 5:37)



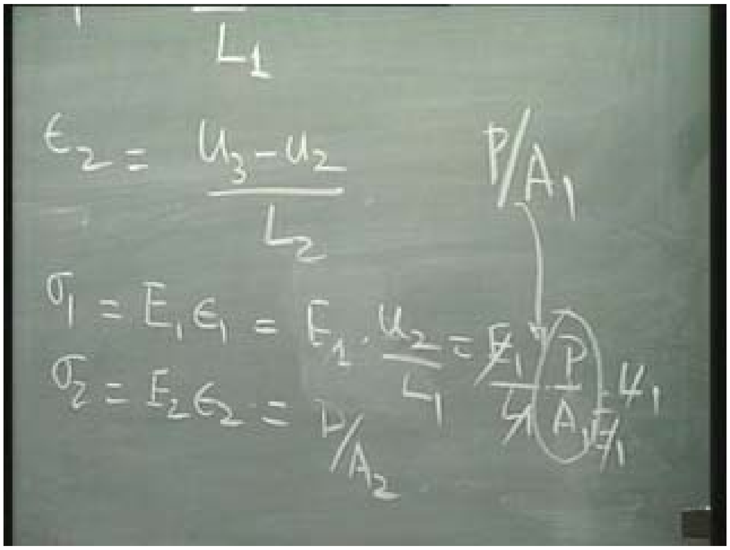
Remember that we had called this as node 1 and node 3. So, what is strain in 1? It is nothing but u_2 divided by L_1.

(Refer Slide Time: 5:54)



From this relationship, I can define the strain in 1 as u_2 divided by L_1. So, I can also determine epsilon$_2$ and epsilon$_2$ is written as, now there are two nodes here, so u_3 minus u_2 divided by L_2; very good. Once I know this, I determine the stress by a simple relationship. Again for this problem

(Refer Slide Time: 6:40)



We have to expand this relationship later for this problem as $sigma_1$ is equal to E_1 $epsilon_1$ and $sigma_2$ is equal to E_2 $epsilon_2$, which on simplifying, by substituting the values for u_2 or u_3 from whatever we have derived here in this case and substituting for K_1 and K_2, a very straight forward substitutions, this is the relationship for u_2 and this is the relationship for u_1.

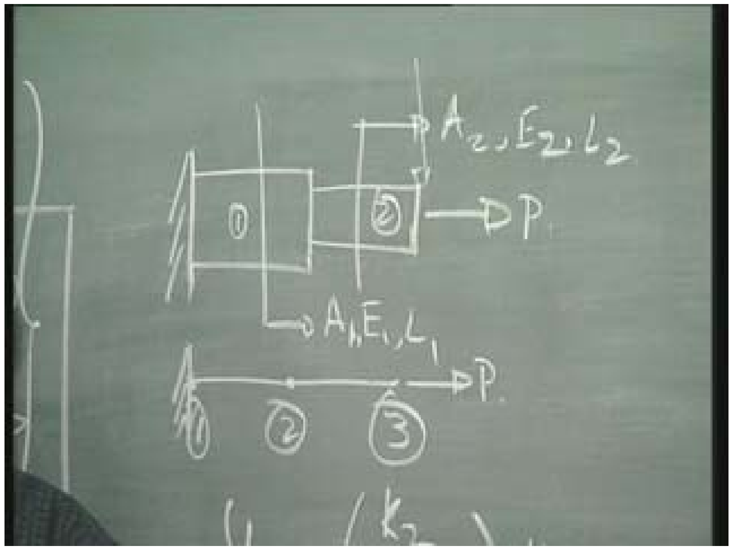
Let me just do only one of the two. The other I can just write down, it is quite straight forward. So, if we do not waste much time on it, remember what K_1 is and what is K_2? K_1 is equal to A_1 E_1 by L_1 and K_2 is equal to A_2 E_2 by L_2. By doing that, it is very easy to show that $sigma_1$ is equal to P by A_1 and you can just substitute it and see it is equal to P by A_2. You can straight away substitute it and you can see that for example if you have a doubt I can say that we can work it out for this; how we got it. Though it is very straight forward, it is useful to look at that quite closely.

Remember that $epsilon_1$ is equal to I am substituting from here; so $epsilon_1$ is equal to u_2 by L_1 and u_2 is equal to P by K_1 and that is equal to E_1 by L_1 into P by K_1, K_1 being A_1 E_1 by L_1. Substituting that A_1 E_1 into L_1, striking them off, I get $sigma_1$ is equal to P by A_1. Same way, you can substitute it, it will take two minutes but you can see that $sigma_2$ is equal to P by A_2. This particular procedure which I will summarize in a minute is in a nutshell what we do in finite element analysis. There are very interesting things we have not talked about. Look at that result.

There is always a misconception, mostly among students, that when I have a bar like this with two different E's, E_1 and E_2, there is always a misconception that when I apply a force to this particular bar, to this bar for example what is here, that since E_2 say for example let us assume that E_2 is greater than E_1, just for an argument. Since E_2 is greater than E_1, the students feel that the strains or displacements are lower; strains are lower and hence stress is lower, when I have here higher E_2. It is absolutely not correct. Why that is, because ultimately we are looking at equilibrium equations. What does this actually indicate, in one sense if these two come out from the equilibrium of the bar? Sigma_1 A_1, if I bring the A_1 to the other side, in a sense if these two come out from the equilibrium of the bar; now, sigma_1 A_1, if I bring the A_1 to the other side, sigma_1 A_1 signifies the internal forces due to stresses; stresses developed due to internal forces and in one hand they are nothing but an equilibrium equation. So, the force, whatever be the material when I apply a force, this force has to be equilibrated by the internal stresses and when they develop, when this internal stresses develop, they develop in such a fashion that they are independent of E_1 and E_2. That is a very important thing to realize, because people usually make a mistake and that is why I have pointed out this particular case.

Now let me summarize. If we look at what all the steps we have gone through, you have gone through quite a few steps, though every step was simple but you have gone through quite a few steps, what is the first step? We looked at the problem and said that this behavior is a simple bar behavior.

(Refer Slide Time: 12:38)



We said that this is simple bar. Please note there is a difference between geometry and its behavior. The same bar if I apply another type of load, for example let us assume that this bar has a circular cross section. The same bar if I apply a load in this direction perpendicular to it, would behave or what would it undergo? It would undergo bending. It would undergo bending and hence the behavior of the body would be like a beam. So, the first thing that we did was to immediately recognize from this problem and with the engineering knowledge we have at the back of our mind, the first thing we said that look this behavior is a bar behavior. In fact I told you that if this is a bar behavior, there is an element called bar element. So, I discretise this using the knowledge of engineering which I had gained and correspondingly looking at a thing which is close to the engineering behavior in this finite element library.

What does it look like or what is the analogy? You remember that the first day we had a problem of determining the area. When I wanted you to determine the area what did you do? You said that look there is a triangle there. I know what is the area of this triangle? Immediately you discretised that body into a triangle. Similarly you know what the area of a circle is and you apply that knowledge; that is what you did that day. You applied that knowledge and you knew that there is a circle whose area is known and so on and this is exactly what you are doing here.

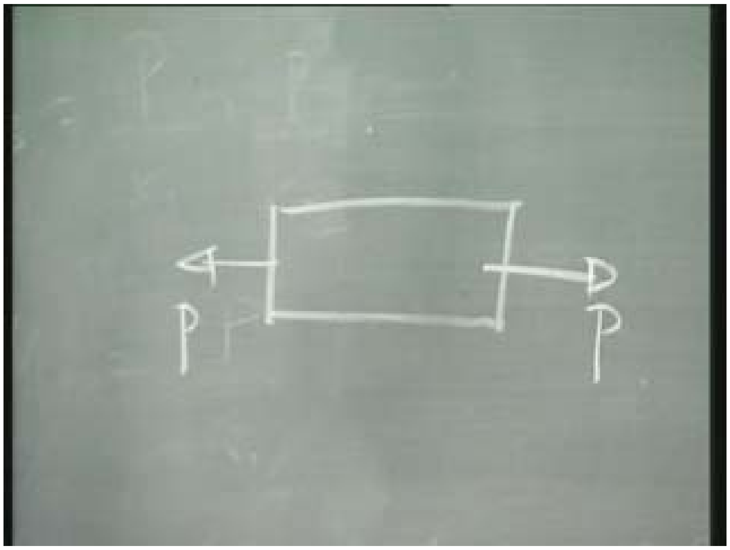
Knowing how the material or the geometry is going to behave under a combination of the geometry as well as loading, you said let us choose bar element first thing. So, after choosing the element, what did you do? You discretised it; you divided it into two elements. Why did you do it? Before you did this, you immediately had an idea of what is stiffness of a bar? That is what we did. What is the stiffness of a bar? Then you realized that there is going to be a term say A_1 E_1 by L_1 which we called as K_1 and A_2 E_2 by L_2. After having realized this you also felt that if I divide this into two elements it is nice, because this guy has one area, this guy has another area. So, I can have two elements so that my calculation of stiffness matrix does not become very messy.

So, you choose this element again looking at geometry and having knowledge of the stiffness matrix or stiffness determination. That is the second step you did; you discretised it. As a third step what did you do? You wrote down what is the stiffness of each of these elements? In that process, you also defined K_1 and K_2. What is the next step you did? You assembled them, you assembled them, you put them together so that I get a global stiffness matrix.

(Refer Slide Time: 16:15)

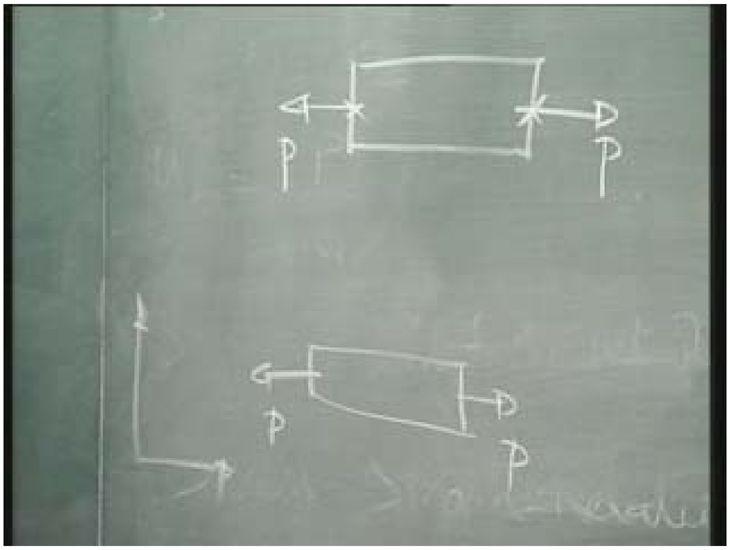
So, you got what is called as global stiffness matrix. Next is the assembly procedure. After having assembled it, you applied boundary condition. This is a very crucial step in finite element modeling, finite element analysis. Let us take a simple example now apart from this to show the importance of boundary conditions which most not only students, but practicing engineers also get confused. The question they ask is, let me keep this figure because we will explain that again.

(Refer Slide Time: 17:15)



The question they ask is suppose I have a bar; let us stick to the bar for time being and if I apply a load say P on either side of the bar which is typical, very simple thing; I take this bar go to some testing machine and usually this is the type of free body diagram you would usually put, simple thing. The question that people ask is where is the boundary condition here? Can I not solve it straight away? Can I not solve it straight away is the question. The other confusion is that look why do you want to apply boundary condition because anyway I have u_1 u_2 and u_3 in this previous problem. Similarly I have u_1 and u_2 in this problem and I have three equations. So, why not I go and solve it? Can I solve this without applying boundary conditions? Because, that is what it looks like from the free body diagram. But unfortunately that is not true. It is Just because you have written down three equations and just because you have three unknowns it does not mean you can solve this equation. because These equations have to be linearly independent, then only it would be possible to solve this problem because you have got these particular equations for the stiffnesses from this the equilibrium, internal equilibrium. So, the solution for this problem as far as say for example displacement is concerned will not be unique. Physically, what does it mean?

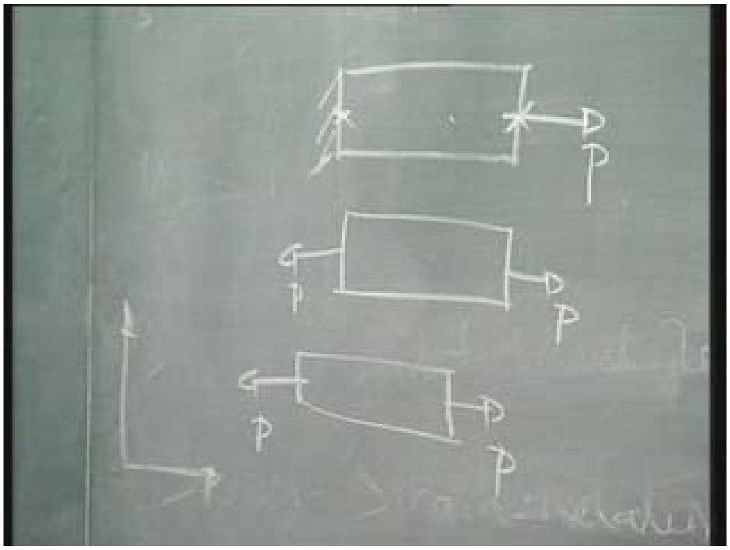
(Refer Slide Time: 19:16)



It means that I have to get the correct positions of these, for example, ends as I apply this by as i apply this bar as I apply the load in this bar. That is not unique; it is something like adding a constant to integration; something like adding a constant to integration. For example, I take this bar and then determine the stresses for this bar in this position. I will get a particular u_1 and u_2 with respect to say coordinate system. On the other hand, if I take the same bar here, same bar and apply the same loads and determine the displacement and stress it has to be the same. But I would get a different value. This can be say at a distance of around 5 units from the origin. This can be at a distance of 3 units from the ……. So, you would not get a unique u or unique solution. In other words a rigid body mode can be applied. There is a rigid body translation, a rigid body translation that is applied here and these kinds of rigid body translation will not affect the result physically as you see it, but will make it impossible to get a unique solution.

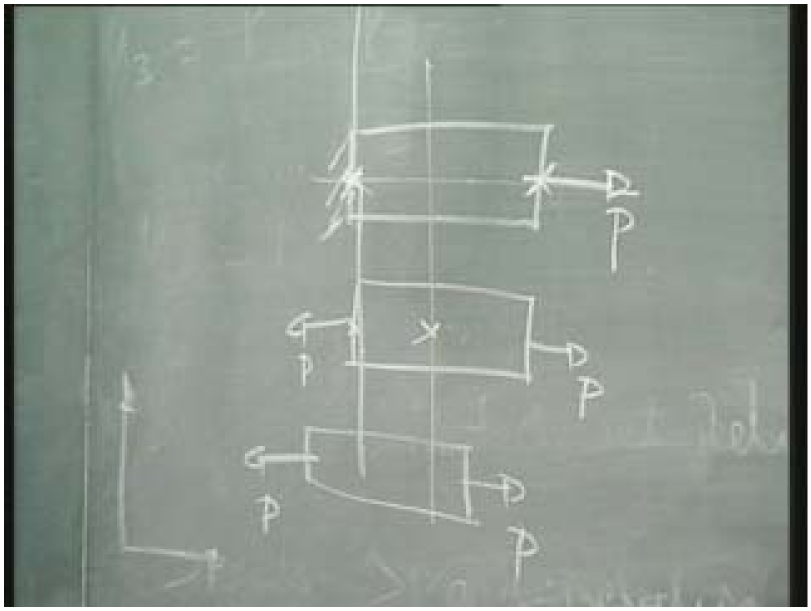
We have to arrest this kind of rigid body motions so that we get a unique solution. Look at that again. Then the question that you may ask is how do I solve this problem? This is free body diagram. We are all people with mechanics background, so, we would like to put the free body diagram and then start this problem. Then how do I solve this problem? Where do I put a boundary condition?

(Refer Slide Time: 21:14)



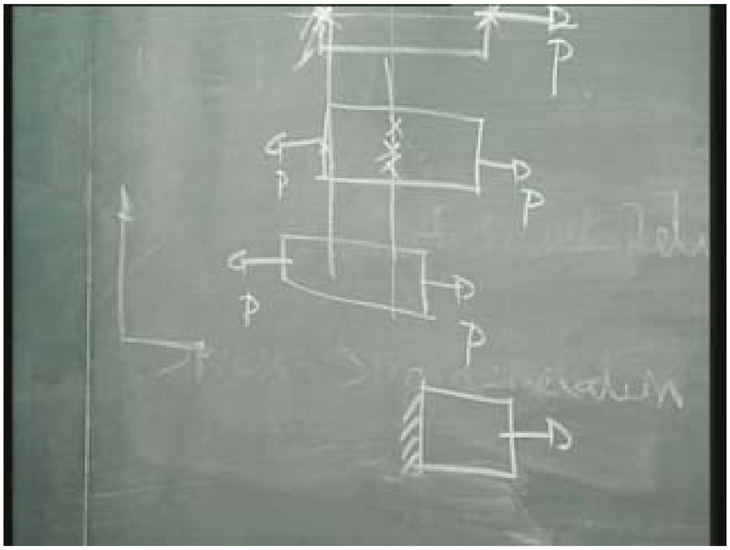
Yes; one solution, as you say that immediately, why not I fix it here and then solve this problem or these two problems from which I started. Are these two problems the same? You feel that both the solutions are the same; you are just fixing it and when I fix it the reaction is anyway going to be P. That is what you are saying and hence these two problems are the same. Yes, to certain extent what you say is right but the purist will not agree to it for a simple reason that if I take a small element there, in this particular problem corresponding element being here, in this particular problem, when I fix it that element will not move. Can you say the same thing of this element? No; this element would start moving. But what you say is right because the stress anyway will not be affected, stress should be the same.

(Refer Slide Time: 22:47)



On one hand in this problem if I plot the displacement right across, it would start from zero. On the other hand the displacement here would start from negative; negative value, if I consider this as the negative x_1 direction, it will start from the negative value. How do you solve this problem? One of the easiest ways to solve this problem is to exploit what is called as symmetry of the problem. What is meant by symmetry? Now let us look at a material particle. Let us sit here in this material particle and see what is going to happen to us as this load is applied; go and sit at this place. When you sit at this place, this guy here to the left is going to pull me and this guy to the right is going to pull me. I am going to have a horrible stress state. Both the guys are going to pull me. They are going to pull me with equal force. What happens? Equal force; you know both the guys are going to pull me. I am going to stand there. Please note again when I am going to stand there, it does not mean that I have no stress. I am stressed; both the guys are pulling me.

(Refer Slide Time: 23:53)

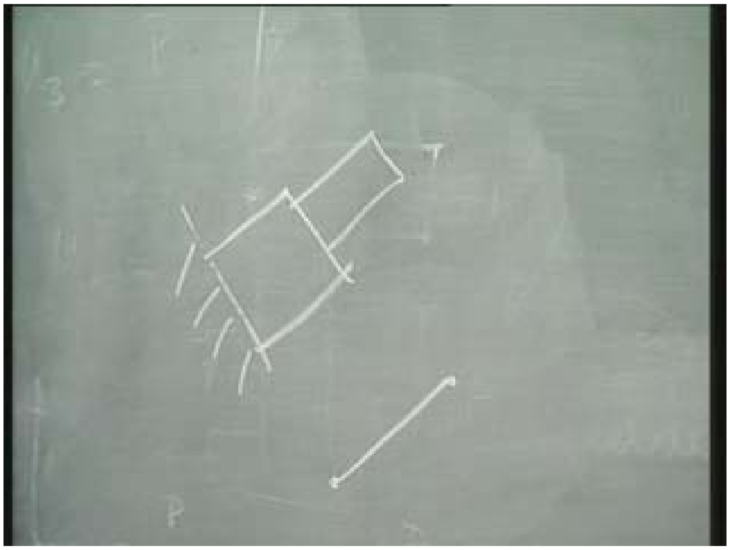


All these material particles which are sitting on this line are not going to have any motion which means that instead of solving this problem I can say take half of the problem. I can take that problem fix this side and then apply a load here and then solve this particular problem. We will come. What is the boundary condition, what is the load to be applied, all these things we will come later; just note that the whole lesson here is not symmetry. Symmetry will come bit later in the course but what is important to realize is that we cannot solve a problem without giving boundary condition. That is the lesson I want to look at from this example. We will talk about symmetry which is slightly more complex later in the course, but note this carefully. Hence boundary conditions are important. Once you apply the boundary conditions you can solve the problem, determine the displacements.

What is the result you are going to get from finite element? The result that you get from this finite element analysis is displacements. You do not get stress; please note you get displacements. Then you go back and through strain displacement relationship, you find out the strain and then through stress strain relationship you determine stress. Is that clear? This in a nutshell completes all the steps that are required in order to solve this problem.

Yes, any questions? Yes, so that is a good question. The first thing is what happens if the bar is inclined?

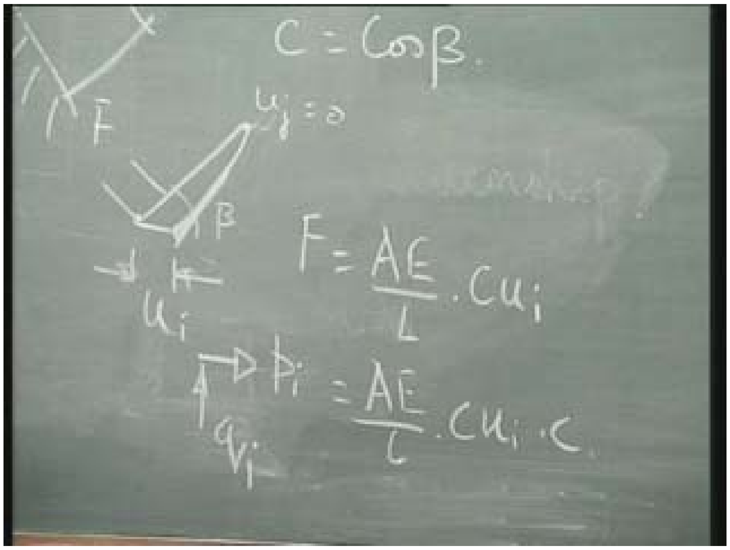
(Refer Slide Time: 26:07)



Suppose I want to solve the same problem, this same problem which is sitting here (Refer Slide Time: 26:16), if I have to solve this same problem with the bar being in an inclined position or for that matter an element being inclined, how do you now solve this problem? Is there going to be any difference in the procedure? This problem becomes important because of two or three reasons or rather most important reason is this. There are number of softwares that are used to do finite element analysis. We will again look at where we use softwares? What is that we use as with respect to different steps that we have seen but immediately these softwares have a tendency to give names. Looking at this they will say three dimensional bar element or generalized bar, three dimensional bar element.

This is the name they would give; three dimensional beam, three dimensional bar. These are the names that are usually given. People get confused because as we had just now talked the bar can take only degree of freedom along its axis. So, what do you mean by 3D or displacement of a node? There are three degrees of freedom for the displacement of the node; that is what they would say. What does it mean or how am I suddenly going to get three displacements? Does it mean that this bar element can also take into account bending? No.

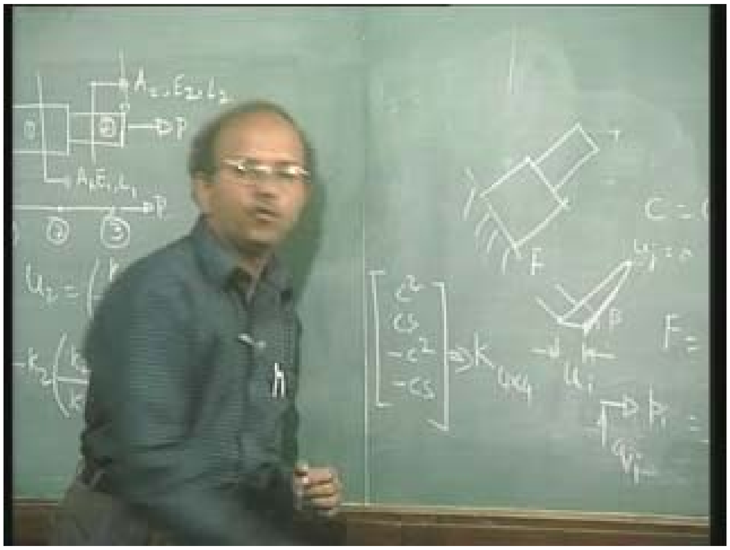
(Refer Slide Time: 28:08)



What essentially it means is that for example if this is the original bar and if this is the displacement and this displacement happens to be say u_i and that displacement happens to be u_j, this displacement, what essentially you do as far as a stiffness matrix is concerned is to resolve these displacements along the direction of the beam or sorry of the bar. If this happens to be, if this angle happens to be beta and let me define C is equal to say cos beta, then if this is going to be u_i then this shortening, keeping say for example let us say u_j is equal to zero so that things become easier to understand, then this shortening of length happens to be what? Cu_i and the force that is developed due to this shortening of length now is to be calculated.

Let me call this force as F that will be same as what? AE by L into Cu_i; if I call that force as F, let me write it here, F is equal to AE by L into Cu_i and this force is now resolved again in the x and y direction which means that if I call this force that is acting in this direction as say p_i and the force that is acting say in this direction as a q_i, then p_i is what? It is equal to AE by L into C square; C square because Cu_i into C. So, this becomes AE by L into Cu_i into C. Please remember C is cos beta. Same way you can get q to be defined as q_i is equal to AE by L Cu_i into, correct, let us call that as sin beta and s or yes? It is possible to develop the stiffness matrix of an inclined bar by following the same steps but by resolving the forces as well as displacement along the axis of the bar. So, my physical behavior of the bar does not change but I change a little, the mathematics that is involved so that I am able to define a more general inclined bar.

(Refer Slide Time: 32:00)

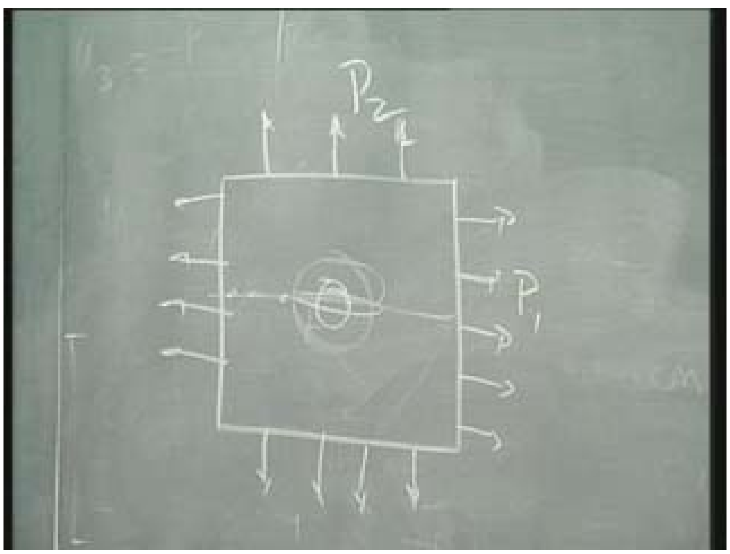


The first column of the stiffness matrix would now look like, the corresponding stiffness matrix would like C square, CS, minus C square, minus CS corresponding to four degrees of freedom. Please note now that we call this as four degrees of freedom, two in the x and the y direction for this node and two in the x and y direction for this node, though physically, in actuality the bar only shortens or lengthens. So this displacement along the axis of the bar has been resolved and that is what you get. So, the stiffness matrix now becomes four by four and this would result in K which is a four by four matrix. So, that is clear. Fine; now, so any other question?

Yeah! Right. See one that is a good question. Again we will discuss that point. Let me repeat that question what he says in a minute. The problem which we have done so far involves only a uniaxial problem, in other words uniaxial stress. Now, there are lots of things which we have to change; the way we are going to think further. What is it that we are going to change or why we should change our thinking? In the uniaxial problem, we did not have much difficulty in understanding stress because stress we said is just E into epsilon, finished it. There is only one stress called sigma and it happens to be or it turns out to be the same as load per unit area P by A_1, P by A_2 and so on.

There are two very simple definitions we used. One is to define strain; just change in length by original length. The other one which we used was just E into epsilon which turns out to be P by A. What happens to a complex situation? Many of the examples which we saw the other day are definitely not in the class of example which we have seen. None of them are one dimensional bar type of elements. All of them were 3D; remember the railway wheel which we saw, it was a 3D problem. Now a three dimensional case where the deformation is not also homogenous in sense that the stress as you had seen varies from one point to another point. How do you define stress? Load per unit area or P by A becomes quite difficult to define. What is P by A? What is that area that we have to take?

(Refer Slide Time: 35:45)

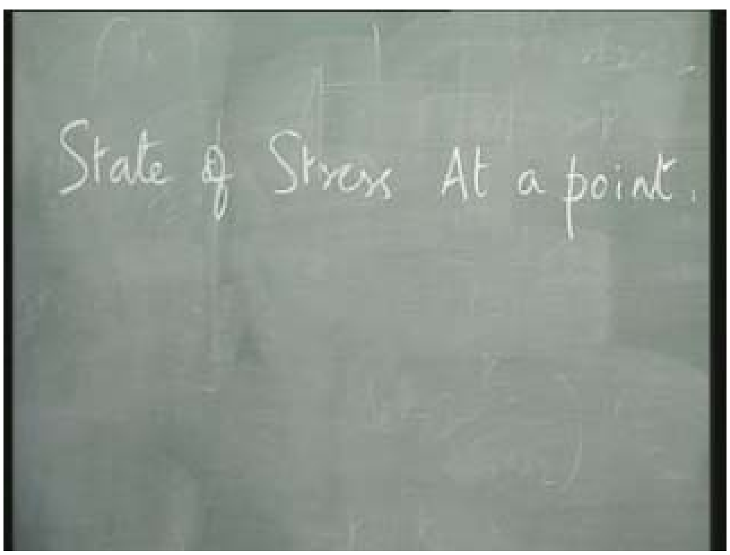


Pardon; can you take projected area? See, the problem here is suppose I have a sheet with the hole and so on and if I start applying loads, what we call as biaxial loads like this and say like this and so on then, we know that the stresses near the hole may be different from the other region. There can be one hole; it can be circular hole or elliptic hole or whatever it is, they are going to be different from other positions. Now in other words, you know probably you have all studied that there is stress concentration region near this hole; probably all of you would say; a very simple answer you would get. What do you mean by stress concentration region? You would immediately say that look when there is a hole in this part then the stresses near this hole is going to be high; is going to be high.

What is this stress? Is it just P by, suppose I say this is say P_2 and this is say P_1. The question that comes to our mind is how do I now define stress in this part? What is A now, P_2 by what A? If you say it is projected area that is what many people say and so the stresses is P_2 by A then you mean to say that at all these regions the stresses would be higher? Then it becomes a problem because immediately you would again come and tell me that no, no it is not like that. Very near the ends the stresses would be higher. In fact when the stresses or when the hole is very sharp, stresses may even reach infinity, the singular point and so on.

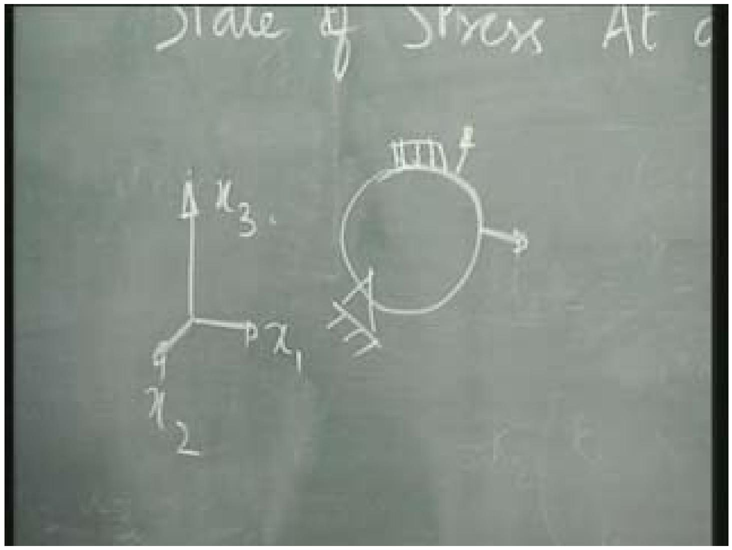
Let us not worry about those things now, but it is important to realize that now the stresses would change as I move away from this corner. If I use one area as you said if I use just the projected area, the normal area, then I would get only one answer. For example in a more complex case of the railway wheel or for that matter the connecting rod or the tyre and so on stresses vary from point to point, from one position to another position. Then what is the area that I am going to define there. Is it just load per unit area? So, we have to put stress in a much more proper perspective and what is it that we are going to do for that? We are going to define what is called as state of stress at a point ultimately.

(Refer Slide Time: 39:05)



But, before we go to state of stress at a point, we have to define first of all what stress is, in a much more rigorous fashion than just load per unit area.

(Refer Slide Time: 39:25)



See, in your earlier classes you would have used a coordinate systems; usually you will call this as x y z or zee. American's call the z as zee; you know why? Because in English z is never defined or never pronounced as z. Take for example z is used in zebra; but, you have z to be, the phonetics is z to be zee. So, the Americans define x y zee. We are not going to get into that complication in language. We are going to define for example these coordinate systems to be x_1 x_2 and x_3.

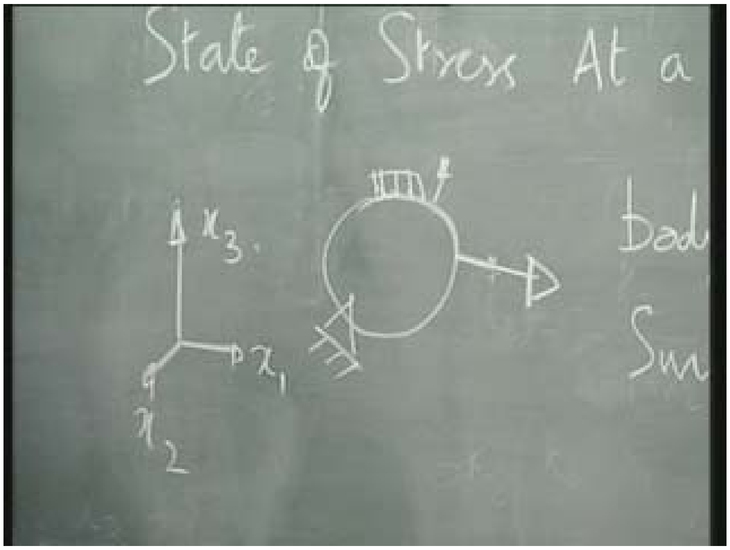
Let us now take a complicated body and then let us say that I apply loads on the body and I have some boundary conditions. Before we go further let us look at the type of loads that happen to be present in engineering components. Loads are due to certain external influence. Right? Loads are due to external influence. What are the types of external influence? Can you classify them?

(Refer Slide Time: 41:05)

Yes, we can classify them into two categories, one which produces what are called as body forces and the other which produces what are called as surface forces. What are body forces and what are surface forces? There are external influences like gravity or centrifugal force which happens to be present in the external or sorry in the total volume of the body. Though they are external influences, they are present in the volume elements of the body. Their influence is ever pervading throughout the volume of the body. Such forces which act on the volume of the body are called as body forces. On the other hand we have what are called as surface forces. The surface forces, for example contact forces, you have a connecting rod; there is a …… pin which is in contact in one end called small end and the crankshaft, the other end and so on. So, there are contact forces. These contact forces, as an example, gives rise to surface forces or when body is in water, it is the force that acts on the surface and so on. So, the forces that act on the body can be classified into body forces and surface forces.

Now, there can be a question.

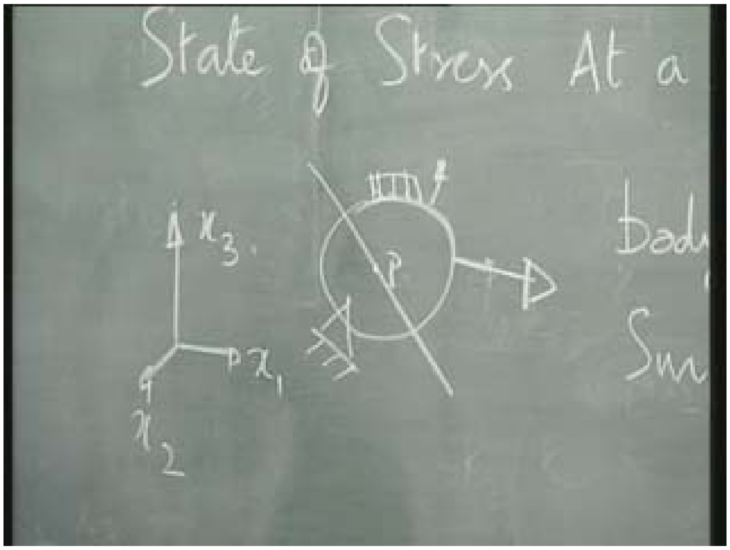
(Refer Slide Time: 42:50)



Usually in all our earlier classes I have applied force, something like this. I have put an arrow and said that look this is the force and probably many of you would have called this force as a concentrated force. Where do I fit in this category? Where do I fit in this concentrated force? Is it body force? Surface forces; correct. So, these forces are surface forces. In actuality there is no such thing as a concentrated force and forces do not act just at one point. They act in a very, very, very small area, infinitesimal area; whatever it is, it acts on an area and that has been modified to be a concentrated force for engineering purposes and hence we have this term concentrated loads or concentrated force and that happens to be a part of the surface forces.

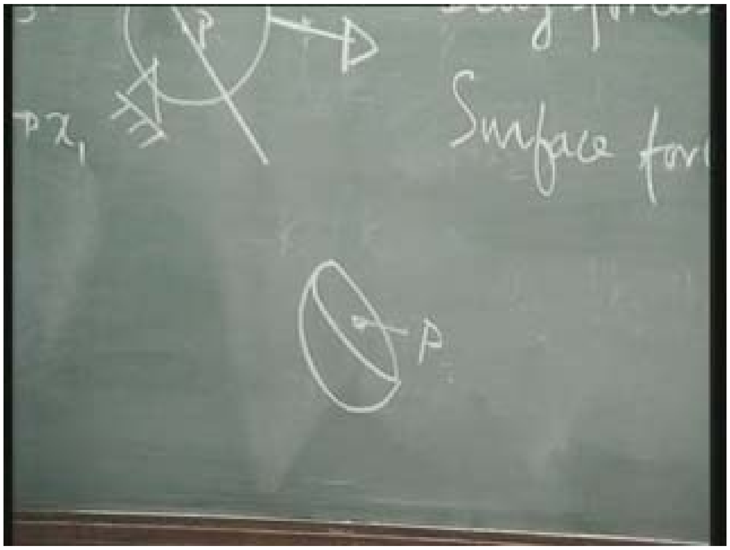
Stresses are produced due to this kind of forces. Then, what is a stress, how do you define a stress?

(Refer Slide Time: 44:12)



First thing that you have to realize is that stress is defined say at a point say P. Stress is defined at a point P. How is it defined at a point P? We will come back to our original definition, but before that what we have to do is to imagine that the body can be split. I am not going to Please note that when I cut the body Before I go further we should realize that when I cut the body it does not mean that I am taking an axe or something and cutting the body and what about all the damage you are going to do to the body? No, we just imagine. So, the first thing is that the stress can be defined at any point P of the body.

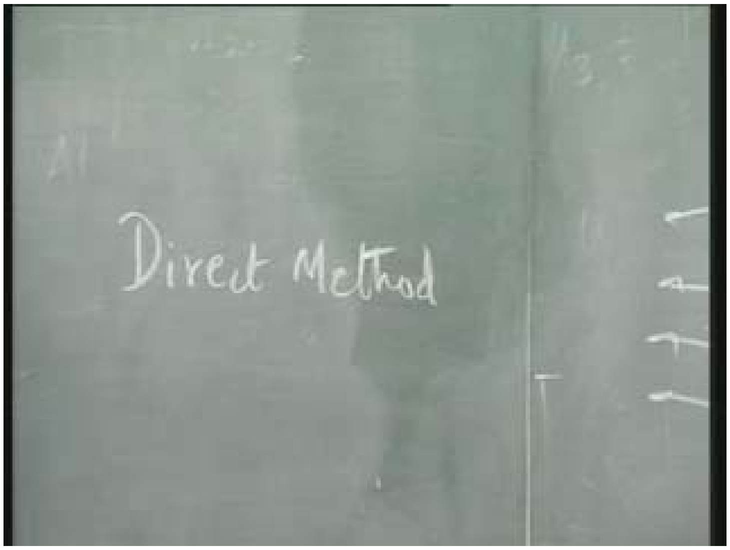
(Refer Slide Time: 45:10)



In order to determine the stress P, I have to cut the body by an imaginary plane. I will say that you want the stress at P, so, let me cut it by an imaginary plane and then take say a part of the body out. Let me take part of the body out. You want me to concentrate on P; yes, let me concentrate on P. This is the point P. So, the next thing I will do is now what is the influence of the other body or the other part of the body which you have cut onto the body which you are now having in your hand and looking at it? Please note the body still has all the forces that are acting. It is not that I have removed everything and cut the body but I have just cut the body, taken it imaginarily and looking at this point, P. We will go through that in the next class; to define what stress is from this point of view in the next class. Before that if there are any questions I will answer.

Yeah, yeah very good. So, that is a next question; that is a very important question. It is good that we stopped here and you asked the question.

(Refer Slide Time: 46:44)



The technique that we used in order to arrive at the stiffness matrix is a direct method. There the question is that is this the only way or in other words if I have a tapered bar, if I have some other shape and if it is not going to be bar, but any other shape how do I get a stiffness matrix. Can I use this kind of same approach? The approach that we used is what is called as a direct method. But this direct method may not work for all cases, but we have to resort to other engineering principles. It may be based on energy theorems, it may be based on virtual work principle and so on. We are going to answer that question later in the course. But in order to do that we have to first understand stress and hence we are now going to look at stress very closely. Once we do that and we understand what is stress, what is strain in a much more general setting then, it would be possible for us to go into more involved theorems, define them clearly and then define stiffness matrices.