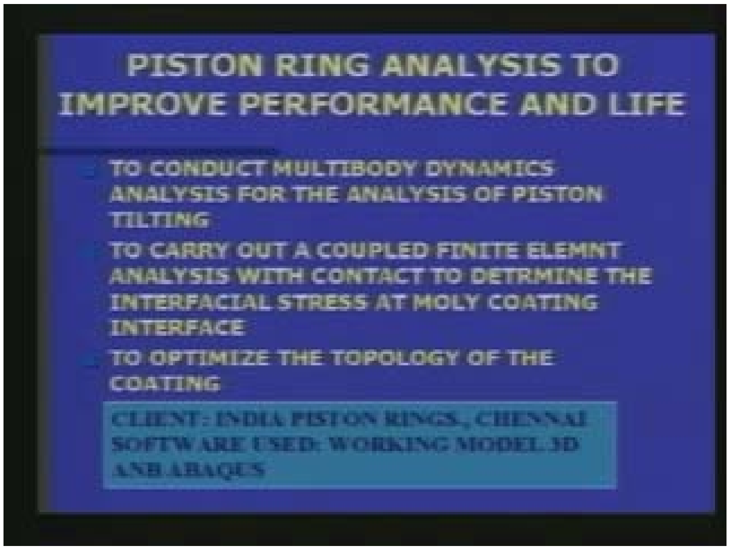
Introduction to Finite Element Method

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

Lecture - 2

In the last class we saw certain examples on the applications of finite element analysis for real life problems. This was done for companies, famous companies as probably you would have seen in our slides before. We are going to continue on these examples. The whole idea here is to give you an overview of where all finite element can be applied in the industry. This will be very useful for us to later conceive a problem, workout a model and look at the results. Let us now see further examples of the application of finite element analysis. This list is not complete, there are lot more applications we are going to talk, but that we are going to do later in the course. But before that let us continue further and look at certain more complex examples rather, that we can do using finite element analysis. Let us start with the first example in the second class.

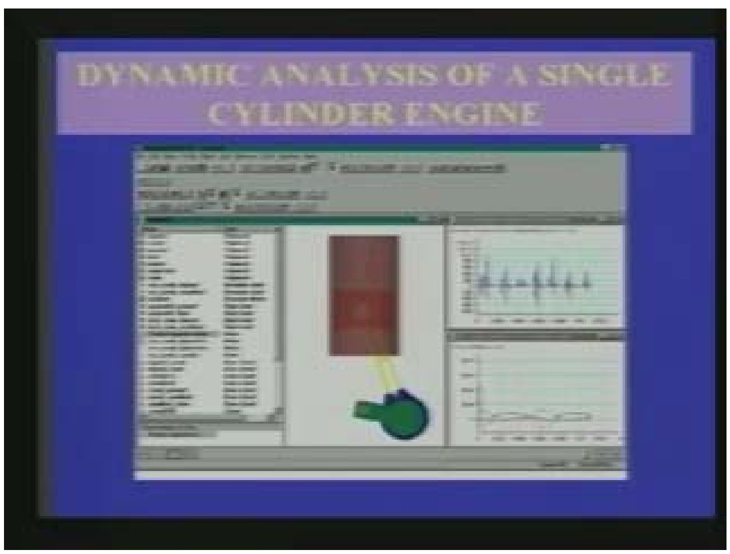
(Refer Slide Time: 2:12)



This example is for the application of finite element analysis in the design of what are called as piston rings. Piston rings though looks very nice and simple and small are not very easy to design due to various aspects. These guys, these piston rings are one of the components which are abused to very great extent in an engine and hence it is important that we understand how the piston ring behaves and what will be the stresses and how the stress analysis can help us to design better rings and so on. Especially the piston ring becomes important because of the fact that we are now talking about Euro I norm, Euro II norm and other very stringent pollution control norms and hence it is important that the ring performs quite well in the engine.

We are going to only have a bird's eye view of what can be done. Lot more can be done, right now we do not have the time or the expertise to go into various aspects. But at least it is important to understand how this technique can be used? Now let us look at how that can be used for the design of piston rings? Let us now look at how we develop the force for the piston ring in our first slide. Let us now look at that slide.

(Refer Slide Time: 3:46)

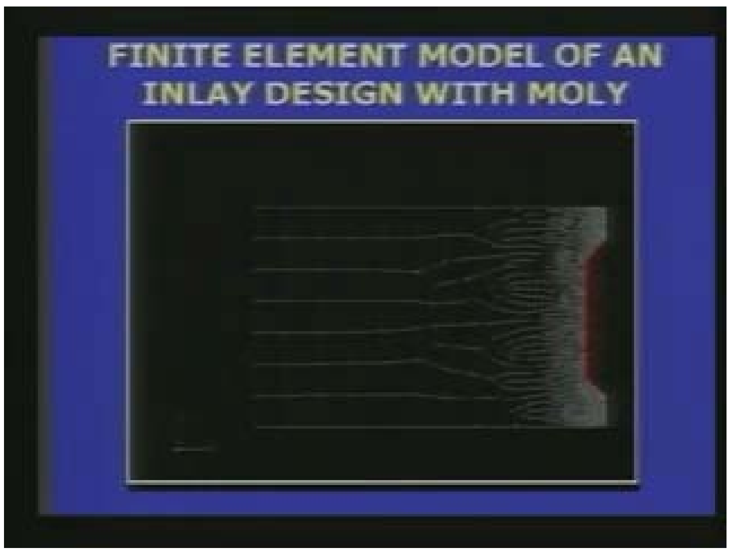


This slide actually gives you an idea as to how forces in a ring can be calculated. You can see that the complete analysis of the dynamics of the engine has been carried out using what is called as multibody dynamics. As the ring sits on the piston and as it goes up and down, the rings actually tilt; they start hitting the sides. The piston starts tilting and along with that there is some sort of a tilt and a flutter for the ring as well. It is important for us to understand the forces that go into the rings. For that we do the multibody dynamic analysis.

We will not talk about multibody dynamic analysis in this course but nevertheless multibody dynamic analysis becomes important for certain components to get the forces. In these multibody components or multibody analysis, most of the time we treat the body as rigid. It is also possible to treat them as flexible but most of the time we can treat it as rigid and get the forces that act on the component. So, we can find out for this particular ring what would be the forces that would act?

Let us look at the next slide and see what it tells us as far as the ring is concerned.

(Refer Slide Time: 5:18)

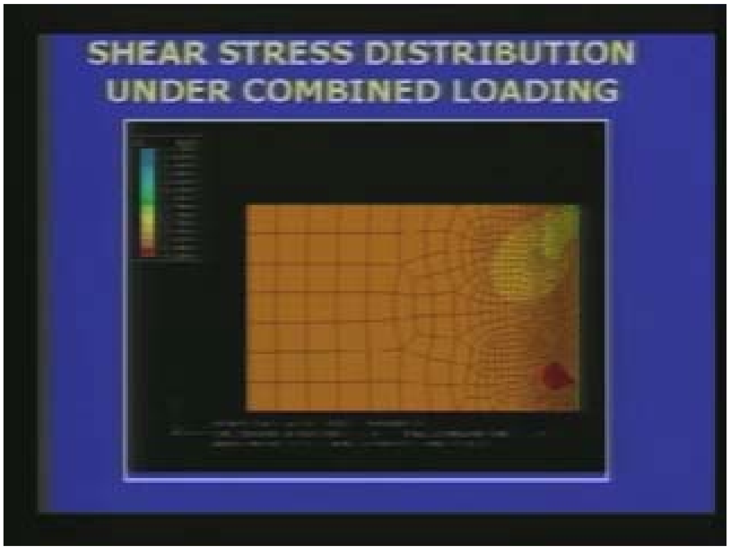


But actually this slide gives us what is called as the finite element model. We are going to discuss more and more about this model later in the course but let us now accept this as a finite element model. We will discuss those lines and what are called as elements which happen to be there towards the end of the class, but nevertheless you see there are two regions. As you see, there are two regions; one region is some sort of a white region and the other there, is a red region. These rings, these piston rings consist of what are called as molybdenum coating. The red region in the ring depicts the molybdenum that has been coated on a steel ring. Ring can be steel, cast iron or whatever it is, but in this case molybdenum is coated onto a steel. This particular design is called as inlay design.

You can also see that or you can understand immediately that it is possible in finite element analysis to get what is called as the combination of materials. Like it is possible to say that we will have steel in one part and we have what is called as molybdenum in the other part and do an analysis together.

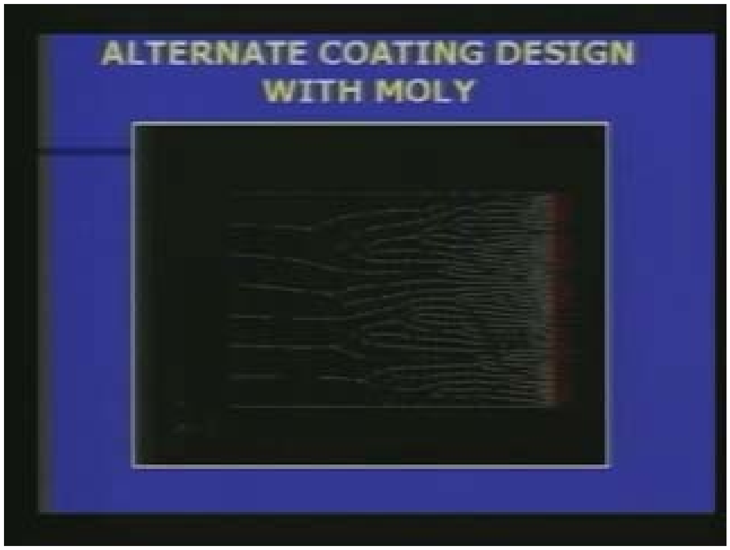
Let us look at the next slide and we will see how the results are?

(Refer Slide Time: 6:44)



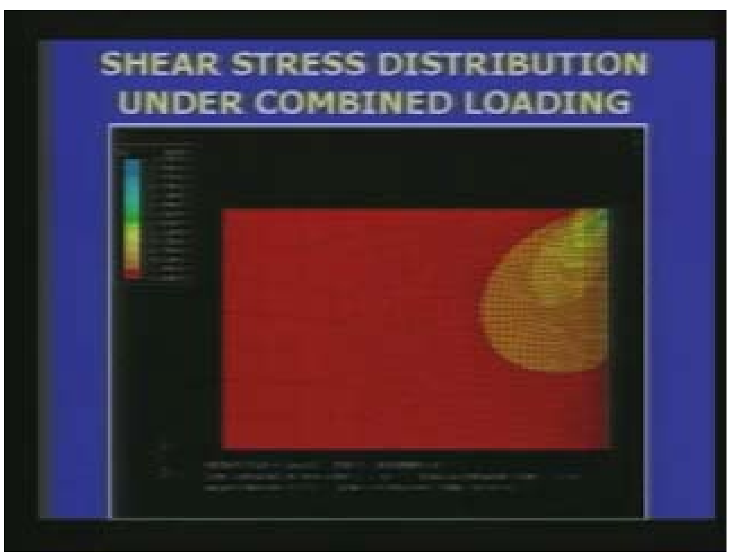
You can see that the molybdenum coating has an effect on the stresses and you can see that there are regions where the stresses are high and there are regions where the stresses are low as well. Let us not worry about what is the stress value, what is the allowable stress value and so on, but it so happens that in this particular design the stresses are high. Let us look at another design and that is depicted in the next slide.

(Refer Slide Time: 7:20)



You can see a second design here and now you see again there is a red region and there is a white region. This red region gives the molybdenum part and the white region gives the steel part. This design is what I would call as a sandwich design where the molybdenum is coated throughout the thickness of the ring. Remember that we are looking at a cross section of a ring and we call this problem as an axi-symmetric problem. Let us look at the results for this problem or for this design.

(Refer Slide Time: 7:55)



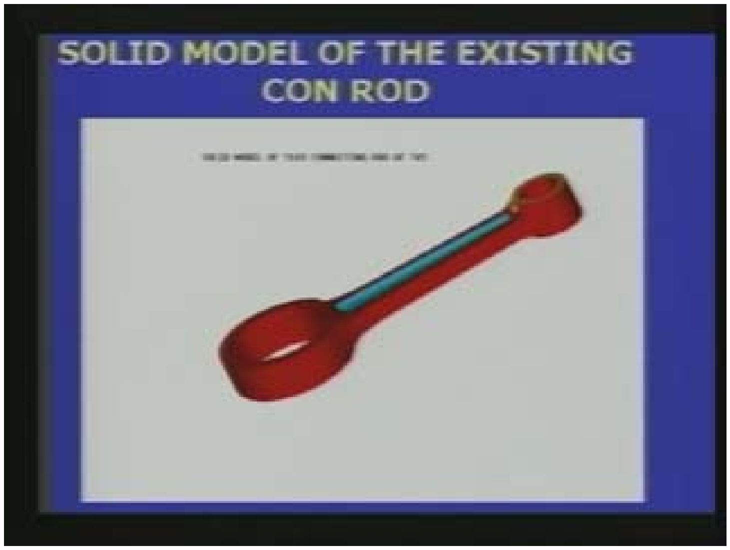
That is the stresses; though you may not be able to clearly see what the stresses are, but I can assure you that stresses happen to be slightly lower than the previous design. That does not mean that this design is good, but it means that for these stresses or for the forces that act on this component which has been derived or arrived at from our previous model of multibody dynamics, this design happens to be better than the previous design. There are lot more issues to rings, but we will stop this example at this stage and may be if time permits we will continue it at a later part of the course.

Let us look at our next problem which is a very, very interesting problem.

(Refer Slide Time: 8:43)

The problem is the design of a connecting rod. Please remember that in the previous problem, the material which we considered was steel as well as molybdenum; a part of the component was steel, a part of the component was molybdenum. Now we are going to consider a material which is porous, which has porosity, whose density is not complete. For example we are going to consider a P/M component, powder metallurgy component, where the normal densities are not attained; not necessarily attained but only about 90% of the density is attained. This is a connecting rod for a two wheeler and now let us look at how we develop this particular component using finite element analysis.

(Refer Slide Time: 9:39)



The next slide gives the solid model of an existing connecting rod for the same engine. It is a very nice picture. You can visualize that it has an I- section and that this connecting rod is performing or it performs very nicely and you can ask me why do you want to change this connecting rod. All the time we always strive to achieve excellence; we want to reduce the weight, we want to reduce the cost and so on. So, from that angle it is now realized that if this forged connecting rod, which is existing connecting rod is replaced by means of a powder metallurgy connecting rod with certain special processes, the cost can be considerably reduced. From that point of view we are redesigning the connecting rod.

The next question that you may ask me is why not I use straight away this connecting rod with a new material or the new process? That may not be possible because the stresses that may result when I use this kind of technique are not conducive for the new material. In other words the stresses may not be withstood by the new material which is a P/M material. Number 2, the process which I am going to have, which I am going to use in order to manufacture this connecting rod also may not be very palatable or may not be very friendly for the existing design. Hence I have to change the design.

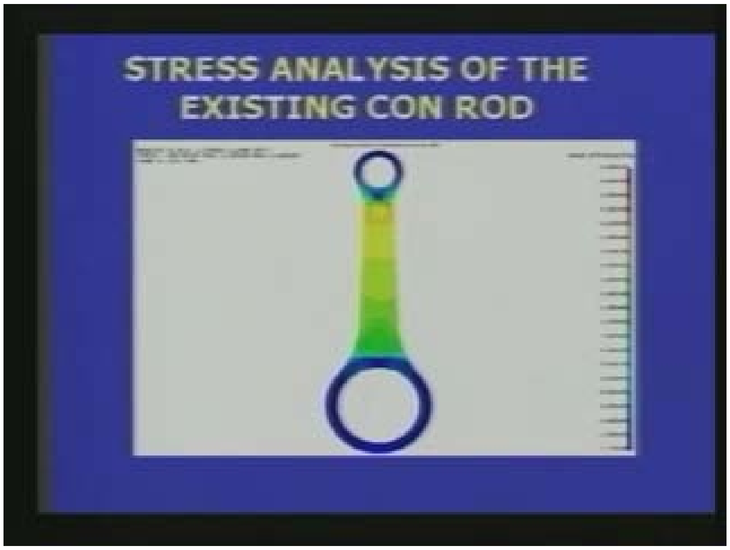
The next question you may ask is why do you want to go through finite element analysis, why not straight away do it by a small hand calculation and start developing certain prototypes and then arrive at an optimum design. There is a problem to it because if you want to arrive at an optimum design, you have to do a number of trials; maybe you may have to do 4 or 5 trials in order to achieve at the correct result. Because certain cases hand calculations may not give you, analytical solutions may not exist, the results that you are looking for and the stresses may not be predicted correctly. So, if you have to do about 5, say examples of 5 prototypes, then you are going to spend a lot of money in order to do this. Hence what we do is we simulate the actual condition or determine the stresses in the actual condition using finite element analysis and then say that look these are the 2 or 3 designs. These are what are called as suboptimal design and I will use now these designs, so reduce my trials to 1 or 2 in order that I can save lot of time as well as money.

Let us look at the next slide and see how exactly we are going to do this.

(Refer Slide Time: 12:36)

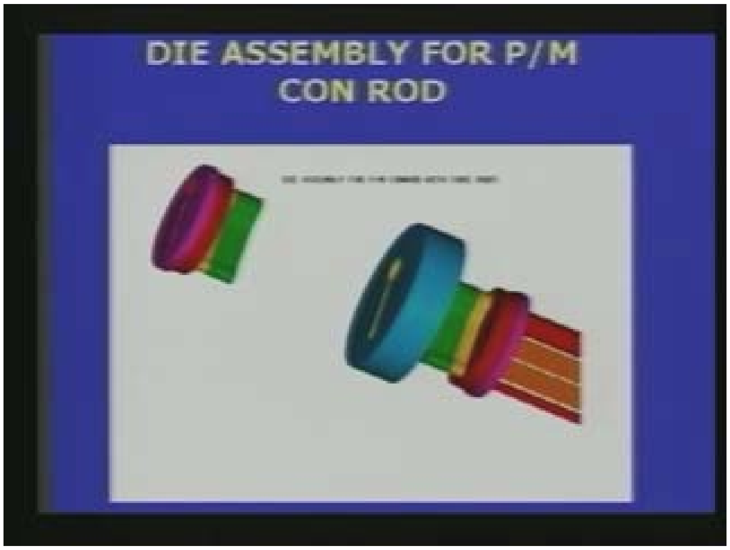
This is the solid model of a new connecting rod; connecting rod which has been designed after a number of trials in the computer, which does not cost me much and the time also required in order to do it, is very small or in other words I can do the whole thing in a matter of about 4 days. Let us now look at the results. The next slide gives me the results of the existing connecting rod.

(Refer Slide Time: 12:59)



This is the stress levels; very nice connecting rod with performing very well, there is no doubt about it. Let us look at the next slide and see the stress analysis of the P/M connecting rod. The stresses have been so adjusted that the maximum stress that exist when the connecting rod is put in operation, put in an engine is less than what could be withstood by this particular material, by this particular P/M material; that has been taken care of. The thicknesses have been optimized and hence it is ready for manufacturing. The manufacturing itself can be continued after this stage, a finite element analysis as shown in the next slide.

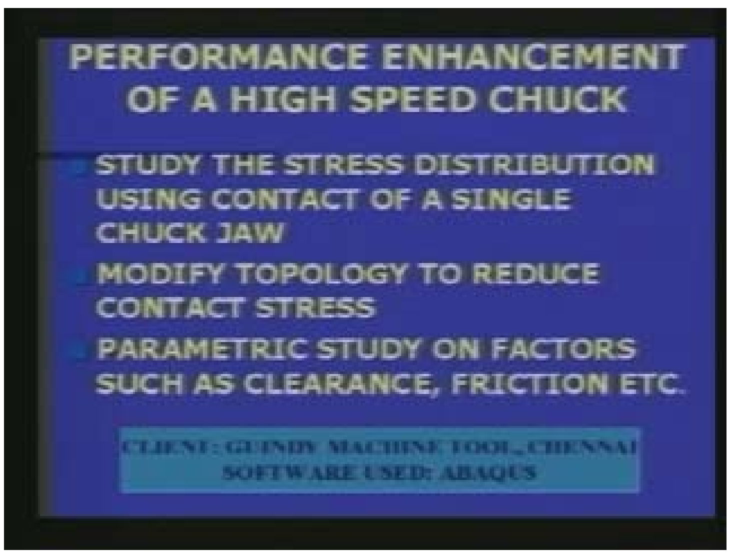
(Refer Slide Time: 13:52)



You can see that it is possible to integrate completely CAD and CAM with finite element analysis sitting in the middle. We started with a solid model, we went ahead, we did finite element analysis and we optimized the shape. Now it is possible for me to push this geometry to CAM and then get the dye assembly for this P/M connecting rod, get the cut location data and machine this and so on. Today the technology is available as an integrated piece to start a design from solid model, visualize it, do analysis and then completely prepare for the manufacturing process. So, that is an interesting example. This piece is under trial.

Let us look at the next, again an interesting example from the field of machine tools.

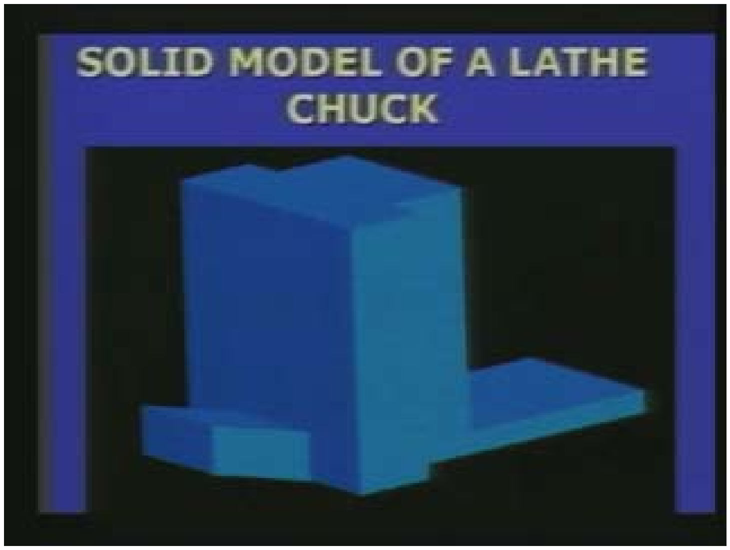
(Refer Slide Time: 14:47)



All of you know that a chuck is a very important component of a lathe. Today, we are looking at very high speeds at which this chuck operate or in other words these are high speed chucks and there are lot of problems of wear of these chucks. When there is wear of this chuck, all of you know that it will not bite or hold the component properly and there are problems of quality of the component and so on and I need not explain all these things to you; you are mechanical engineers you know these things. So, the whole idea here is, is it possible to enhance the performance of a high speed chuck by completely redesigning this chuck.

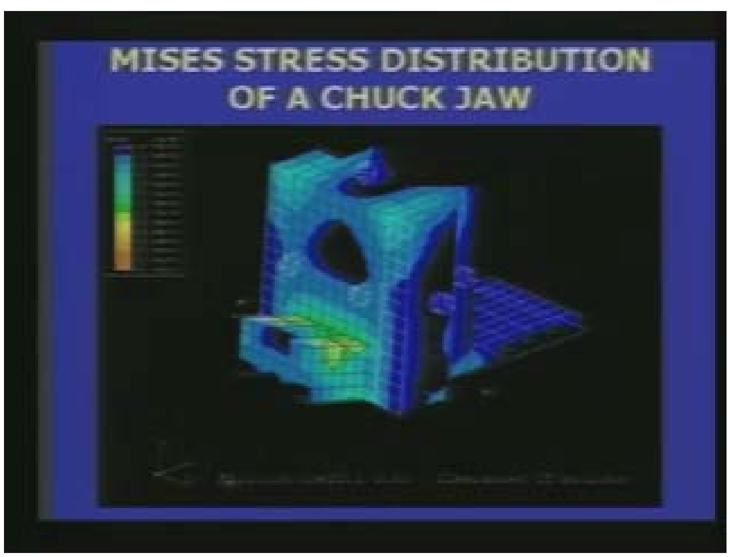
Let us look at the first slide in the series.

(Refer Slide Time: 15:36)



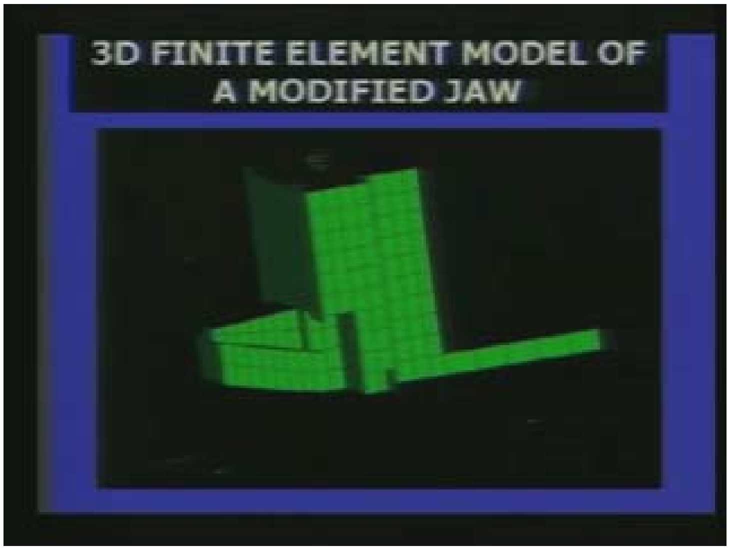
That slide shows you a solid model of an existing lathe chuck. This problem is very interesting and very complicated because the chuck is in contact at number of points to the body of or this chuck jar rather is in contact at number of points to the body of the chuck. This is what is called as the jaw of this chuck and hence my interest here is to find out what are the contact stresses that exist in this jaw?

(Refer Slide Time: 16:11)



The next slide shows you the contact stresses of this chuck jaw as it operate in a chuck. Now, it is possible to look at this stresses and tell whether these stresses are nice enough for you to not to wear or is it going to be subjected to wear. There are lot more issues of wear but nevertheless this gives me a first hand figure whether stresses are going to be critical. It so happens that the stresses happen to be critical and we redesign this chuck and the redesigned chuck looks something like this, which you can see in the next slide.

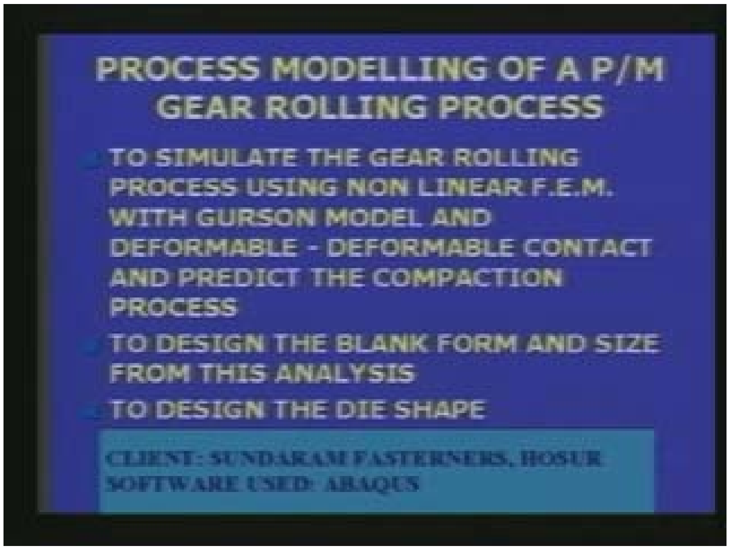
(Refer Slide Time: 16:55)



So, that is the re-modified chuck or the redesigned chuck and the stresses were lower for this particular chuck. So, performance enhancement by modifying a machine tool component is possible using finite element analysis.

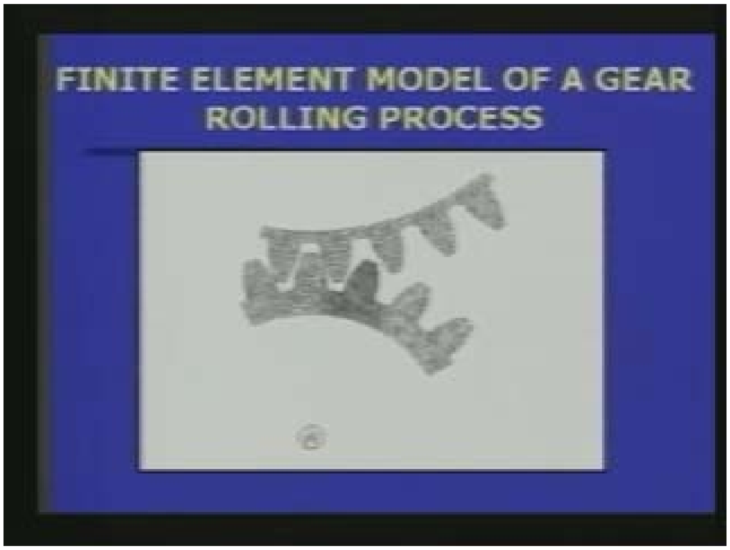
Let us now look at the next example.

(Refer Slide Time: 17:14)



The next example is from the world of manufacturing. In other words next example talks about process modeling of what is called as a P/M gear rolling process. The next slide tells you or shows the model of this, for this process.

(Refer Slide Time: 17:34)



What is that we are doing in this process? All of you know that most of the gears are forged gears and few of them may be cast as well depending upon the component or rather the material. In this particular piece the gear, which is the bottom part is made up of powder metallurgy, it is a P/M gear. Today world over what they do is to look at a P/M component and see whether it is possible to selectively densify those areas where the stresses are higher. For example in this particular gear is it possible to say where the stresses would be higher and is it possible to densify only those areas? For example the flank and the root of the gear may be subjected to high stresses and is it possible to now compact only those places and leaving out the center of the gear? If you do that, that is good enough for us to use a P/M gear for many of the load bearing applications which again means that we can save lot of money.

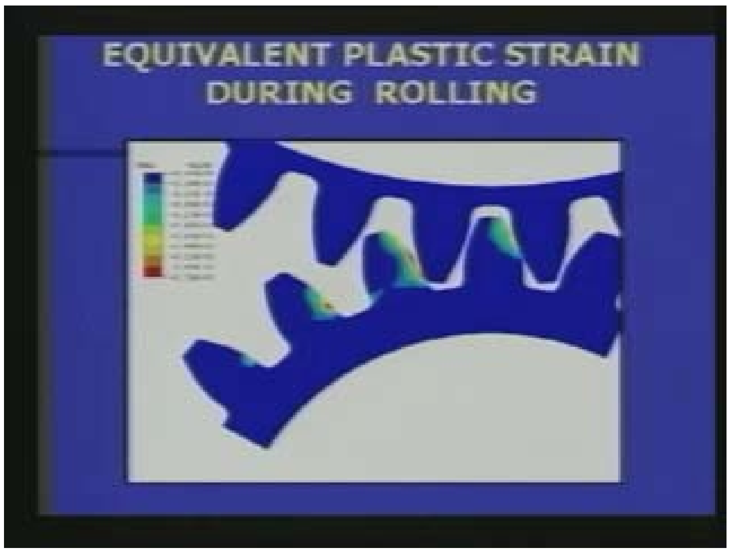
Let us go back and look at that model. You can see from the model that there are two pieces. That model will now show you that there is a bottom piece which is actually the gear and the top piece which is what is called as the dye. The gear actually starts rotating under the influence of the dye and the dye starts compacting. The next slide will show you what happens because of this process? The results are quite clear from the next slide.

Now, what is that we are looking at in this slide? We are going to look at what is the compaction or how efficient has there been as the compaction takes place? In other words what we are going to do is to look at what is called as a relative density of the gear.

(Refer Slide Time: 19:54)

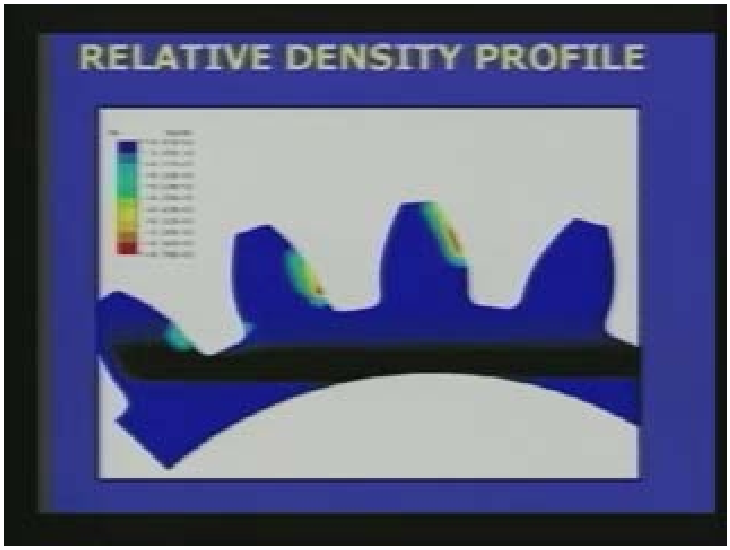
As you can see that in the gear this is the root and that it is important that we have higher densities only in this region and that the densities need not be that very high in this region. So, as the dye rolls, it is, it is a regular rolling process, gear rolling process. As the dye rolls in one direction and then in the other, the material gets densified on the surface to that extent that it can withstand all the loads or all the stresses that would come during the operation or it can withstand the operational loads. It is possible now to design the process using finite element analysis.

(Refer Slide Time: 20:56)



Let us now look at or let us stop for a moment and look at this quite closely. You can see the equivalent plastic strain during rolling. As I was just explaining to you, the top one is the dye and which is considered to be rigid, the bottom is the component. You can see the plastic strains developing which gives rise to what is called as compaction.

(Refer Slide Time: 21:24)



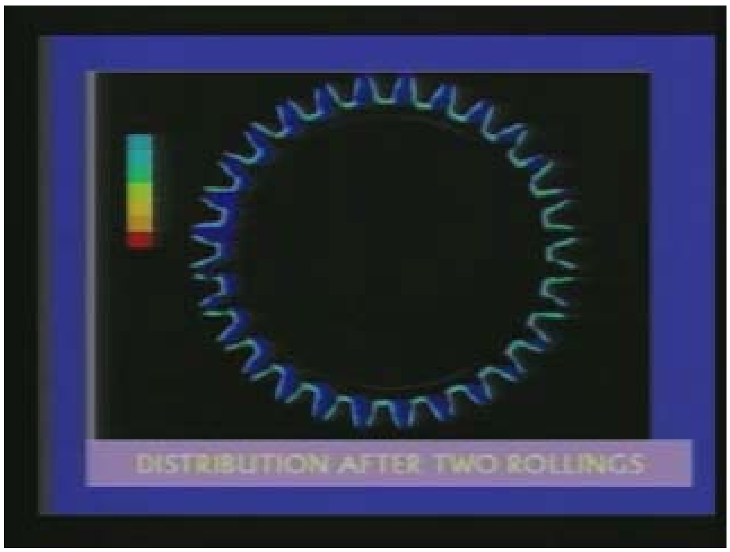
Let us look at the next slide and that gives us a better picture of the relative densities or compaction efficiency of the process which we have been talking about. You can see that as it rolls, the flank slowly gets compacted. Maybe 4 or 5 different cycles are required in order to completely compact the surface. What is that we are trying to do using finite element here? We are trying to say or predict how this compaction takes place in the rolling process? Why is that we are doing, because again here we can reduce number of trials that are required.

This is a very expensive process or the trials may be quite expensive. So, we can reduce both the money as well as the time by doing this kind of simulation and then arriving at the optimum design of both the gear. After all what I have to do is I have to add some material or I have to redesign this gear so that when it gets compacted my profile stays good. It should be good to perform or the profile should be such that the performance of the gear is not affected. So, that is one thing. The next one is how is that I am going to design the dye? Is there modifications that are required to the dye or can it be a regular rolling dye itself.

I can do all those kind of simulations, arrive at what should be the shape and also arrive at what would be the speeds at which I have to rotate and also look at how many number of rollings are required and so, all things can be done beforehand. After all we may follow two processes; one process to just compact only material at the surface and the other what is called as root rolling process where we may be interested to compact what is in the root. In other words we may require special dyes or special shapes of the dye in order to compact material which is existing at this root. So, this kind of geometry optimization in a process is also possible by using finite element analysis.

Let us look at the next slide and see what we get.

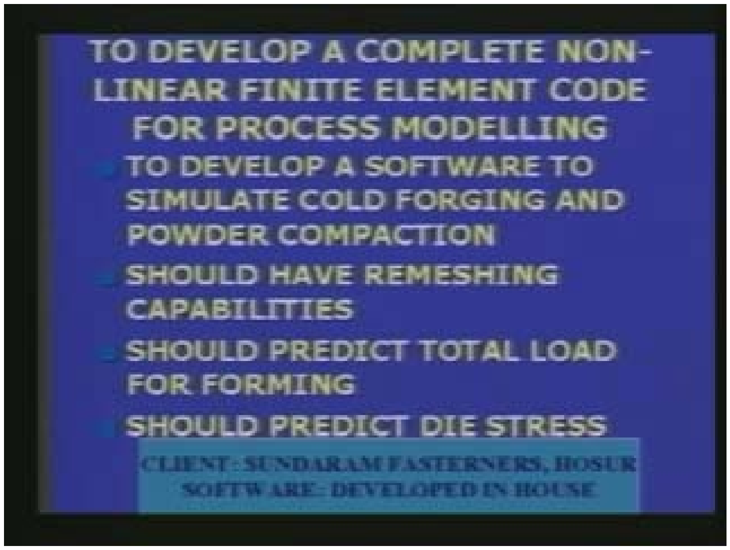
(Refer Slide Time: 23:43)



The process is complete and the distribution after two rounds, we can keep on seeing 2 rounds, 3 rounds and so on; the distribution of this density is complete and that is shown in this particular slide.

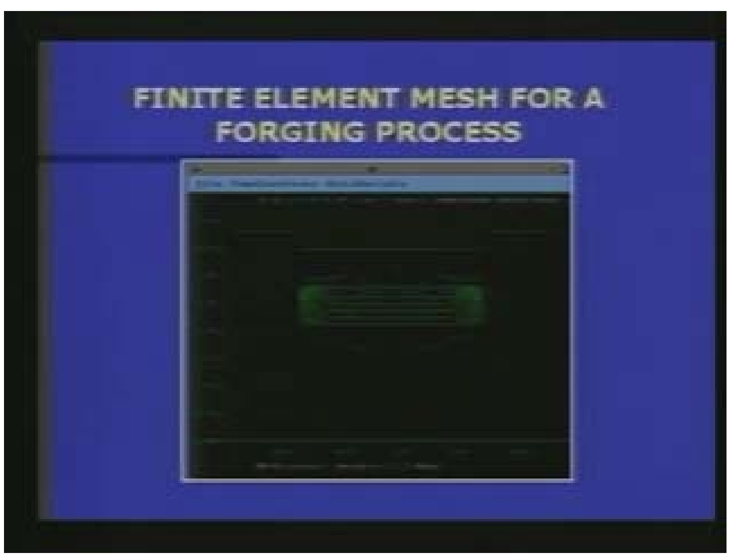
Let us go to the next slide and see the next problem.

(Refer Slide Time: 24:04)



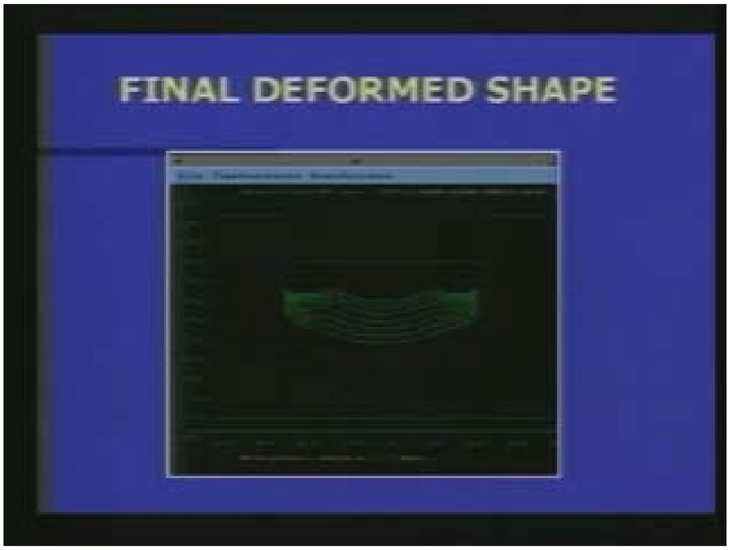
Now, it is not that we are going to or is it possible to simulate only this gear? It is possible to simulate a whole range of manufacturing processes. We are going to study these things in this course as to how to simulate manufacturing processes. In IIT Madras we have developed a non-linear finite element code to simulate forging processes and this is operational in a company and you can see the results of such a code in the next slide.

(Refer Slide Time: 24:41)



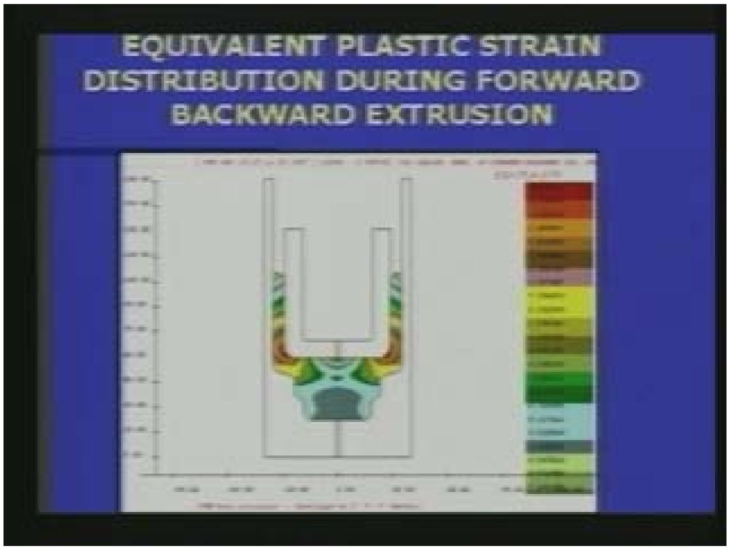
May be slide is not very good but nevertheless it will become clear in the next slide as to what we are trying to achieve.

(Refer Slide Time: 24:50)



We are trying to achieve or we are trying to see whether the filling of a dye is complete and the next slide clearly shows the type of other results that can be obtained from this kind of work.

(Refer Slide Time: 25:04)

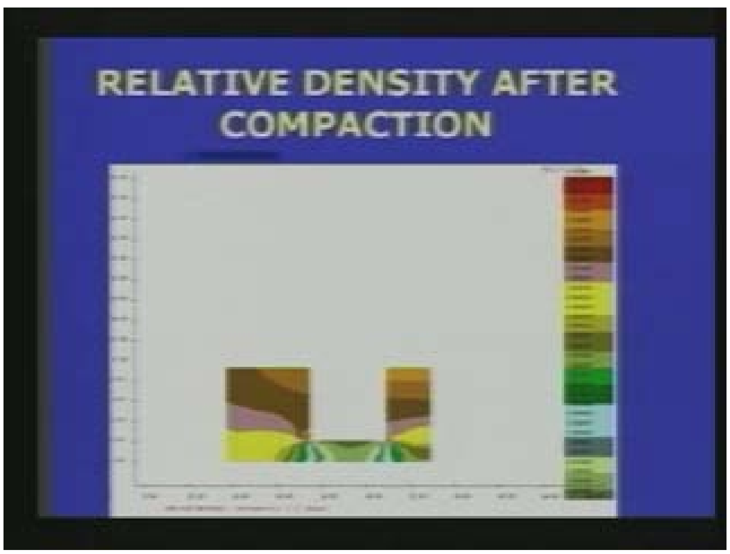


From this slide it is very clear that it is possible to predict plastic strains using finite element analysis. You can see that actually a cup has been formed and the process as you may recognize is what is called as a forward backward extrusion. During this process the slide shows what would be the maximum plastic strains? If you have the data to find out whether maximum plastic strains that are achieved in this process could be withstood by the material, if you have that data, then you can see or you can say whether this process is good; that is number one. In other words, you can also look at the process beforehand, simulate the process beforehand and say whether I am going to achieve this process or not? If not what is to be done?

Number 2 - it may be that the dyes and the punch that are used in the complete manufacture of the dyes are not good enough to withstand the stresses that may be generated during the process of manufacture. It is possible for us to find out what would be the stresses during manufacturing as well. So, it is possible to say whether a punch will break. If it is going to break, then what would be the modifications that are required and whether these modifications will successfully alleviate the problems or remove the problems? All those things can be determined beforehand using finite element analysis.

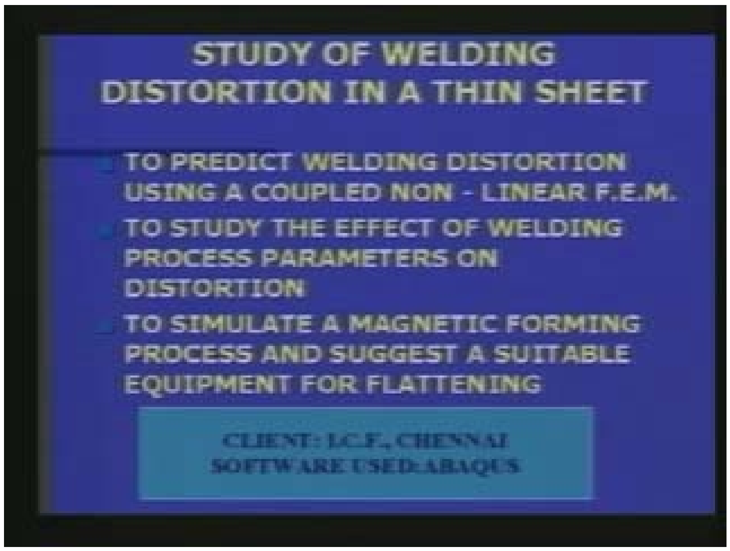
Let us now look at the next example and see what is the lesson it teaches?

(Refer Slide Time: 26:51)



Not only are forgings possible, but whole set of manufacturing processes can be simulated. We are not going to list them now because it is quite a long list. But just to give a variety it is also possible to look at compaction, powder compaction process and see what the density distribution is in a typical component. We will come back to these examples later. We will first have a bird's eye view of where all this process can be achieved. Let us look at the next slide and see what it teaches us?

(Refer Slide Time: 27:27)

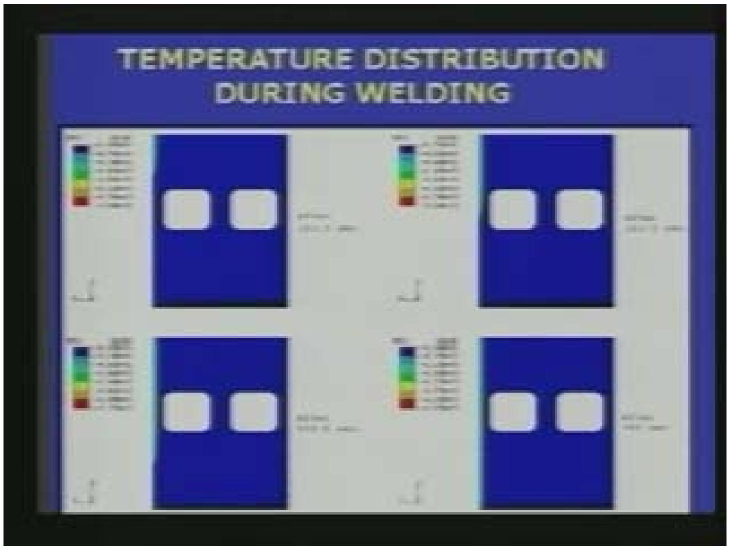


This is another manufacturing process. The manufacturing process that is involved here is the study of welding of thin sheets. All of you know about distortions in welding. What are the things that are involved in welding? There are two things that are, two problems that are important in order to study welding. One is that there is heat transfer; as you weld, there is a weld pool and heat transfer takes place. First of all, you have to do a temperature distribution study or we have to determine the temperatures.

What is the next step? Using this temperature is it possible to solve a mechanics problem and find out what would be the distortions? We are interested in distortions ultimately. These are coupled problems; a heat transfer problem followed by mechanics problem, again a heat transfer problem, mechanics problem and so on. We are going to study this kind of things in this course but nevertheless it is important to understand that finite element can also be applied to study heat transfer problems and coupled problems as well.

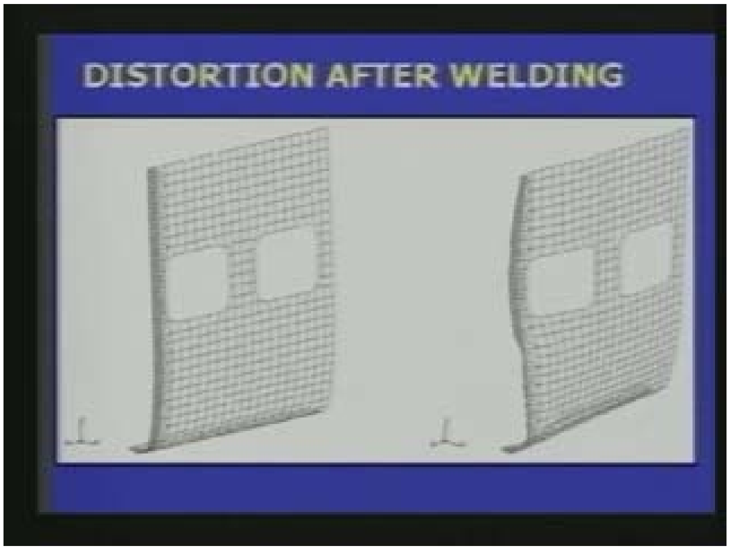
Let us now look at the result of the study.

(Refer Slide Time: 28:45)



You can see the temperature distribution, first of all, during welding; as the torch moves how the temperature gets distributed. You can see very well that it is quite concentrated zone in the zone which is very close to the welding zone. Let us look at the next slide and see what it teaches us.

(Refer Slide Time: 29:06)



This gives the result of distortion after welding or in other words this is the complete, this is the result of the complete study, both heat transfer as well as the mechanics problem; you can see that the sheet is completely distorted. What are we going to do with the sheet? You remember that in the last class we had talked about undulations on the surface of a coach. This coach is made up of this kind of such kind of welded sheets. They are 2 mm sheets very, very thin sheets and hence there are problems of such kind of distortions.

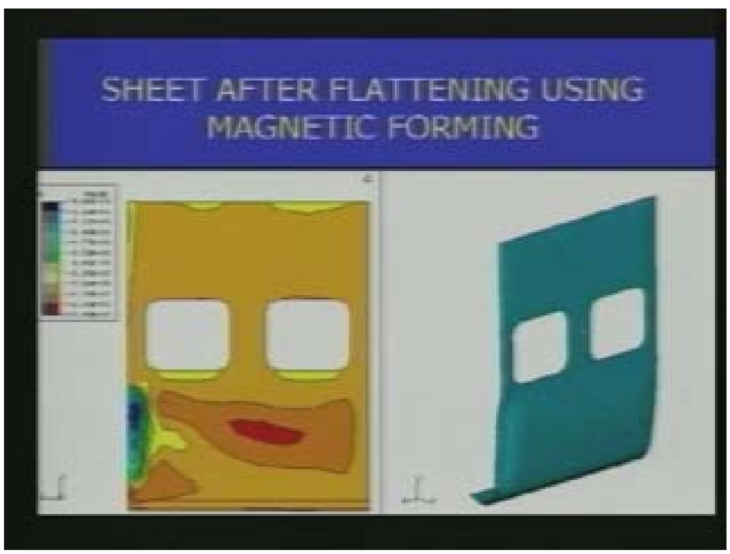
This is what manifests as undulations; apart from the design or the assembly, the coach, it is the fundamental process of manufacturing the coach itself which happens to be welding which gives rise to distortions. How are we going to avoid this or is it possible now to see to it that we can get a straight sheet or straightening as the process, is it possible? Yes, that is possible. World over people whoever manufactures it; whether it is manufactured in Japan or Germany people straighten out this piece. How do they straighten it? That is what is called as a magnetic forming process.

Let us not worry about this process because it is quite involved but electromagnets are used to straighten this piece, make this piece go to a plastic deformation or make this piece go to plastic range and they make it straight. There is a small problem here. These equipments are expensive. So, if I have to get an equipment in order to do this kind of work, then I have to spend quite a lot of money. If you look at this kind of magnetic forming process equipment, there are about five varieties of equipments that exist

What is the hitch here? If I have the first variety say variety one and get it and use it for this kind of straightening process then, if the sheet does not become straight then I lose lot of money. We have to import; in India, we have to import these pieces so, we have to lose lot of money. The question here is can I beforehand find out what is the type of equipment that I have to use in order to straighten? What should be the specification of the equipment in order that I can straighten this piece? Yes, it is possible.

Let us look at the next slide.

(Refer Slide Time: 31:42)

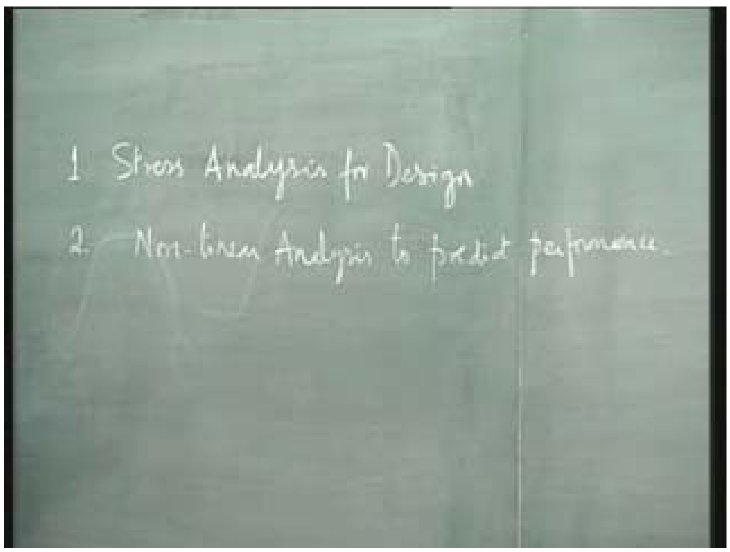


The next slide is the result of such a simulation, finite element simulation and you can see that it is possible to even simulate a magnetic forming process and then look at a straight piece and arrive at what is called as the optimum configuration of the equipment that is used in order to make this kind of flat sheets. Let us pause for a moment; let us get back and see the things that we have seen so far or in other words let us now look at what are the applications of finite element analysis?

Let us get some answers from you. Let us see where all we have applied? Summarize, what are the applications. Let us start from the first example and see what are the areas in which we have applied finite element analysis? Let us get some answers from you.

Yes, the first one is rotor. LP rotor; LP loader is the first example but what is that we got? Yes, stress analysis; in other words it is possible to look at a new design; so for stress analysis for design.

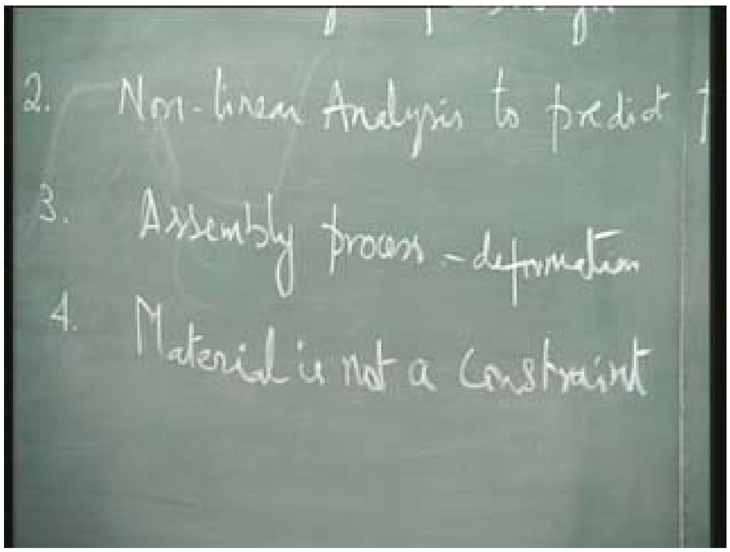
(Refer Slide Time: 33:12)



Quite a few examples; in fact whether it is repair or whether it is a original design whatever it is we can do stress analysis for design. What is the next type of examples that we saw? Non-linear analysis; we did contact analysis. For example for the wheel we did contact analysis to predict failures. It may not be that when we do stress analysis for design as a first step we may be interested in a very complicated problem. Many times we do a simpler analysis but sometimes when we have to investigate failures then we have to do a much more complex and thorough analysis. We have to predict where we have to do much more complex and thorough analysis. So, non-linear analysis to predict, I would say performance; yes, it is that you may argue that why not we do it here itself? Yes, it is possible that you do this kind of analysis right in the beginning. But as a tradition most people do not do a very detailed analysis when we start the design … a performance because 80% to 90% of the components go through with the simple linear stress analysis. But that may not be sufficient when the problem becomes complex and we have trouble in the component hence we have to do a thorough non-linear analysis. This may involve difficult things like contact with what is called as material non-linearities and geometric non-linearities and so on. All these things are possible by using finite element analysis.

What is the third example?

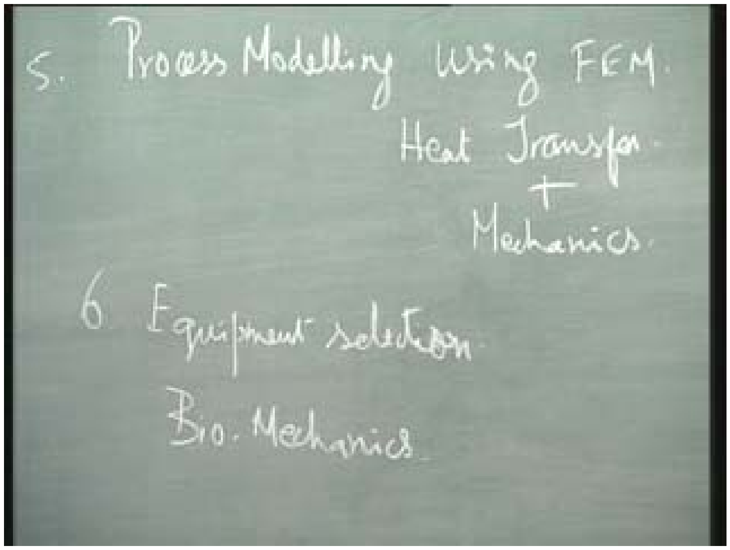
(Refer Slide Time: 35:09)



It is even possible to look at assembly processes. We used it for assembly processes and look at deformation during assembly; so, stress and now deformation assembly process deformation. What is the next issue or what is that we got as the next issue?

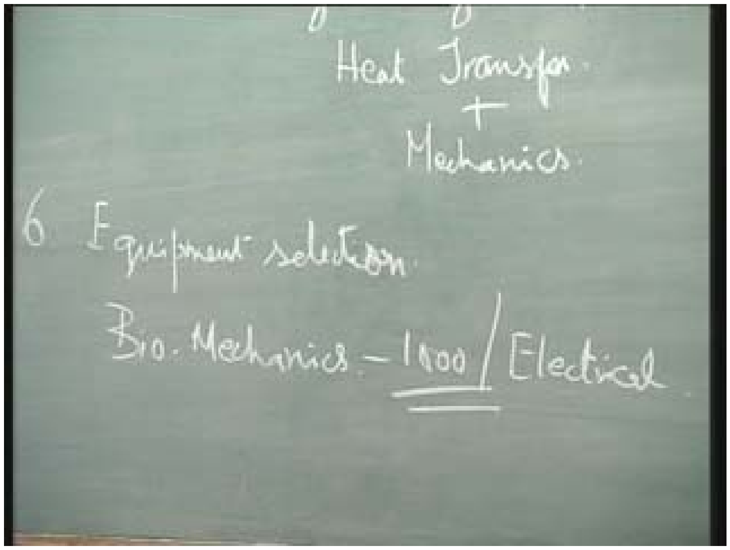
Contact analysis we have already covered here. When you look at say for example piston ring, then automatically you saw that there are two materials. So, material is not a constraint. That is the lesson that we learnt. One part of the component can be of one material and the other can be of another material. So, material is not a constraint. What is the next lesson? Yes

(Refer Slide Time: 36:17)



Process model; so, process modeling using FEM that is the reality. This process modeling consists of both heat transfer sometimes, heat transfer analysis and a mechanics problem or stress analysis and it can be coupled as well. What is the next thing we saw? Is there any other lesson you learnt? Equipment selection; lastly it is possible to use this for equipment selection. This list is no way complete; you know it is not that the list is complete. In fact today finite element analysis is used in biomechanics, extensively used in biomechanics. Unfortunately we will not have time to cover all the applications in this course, but biomechanics is one of the areas where finite element has really taken off.

(Refer Slide Time: 37:55)



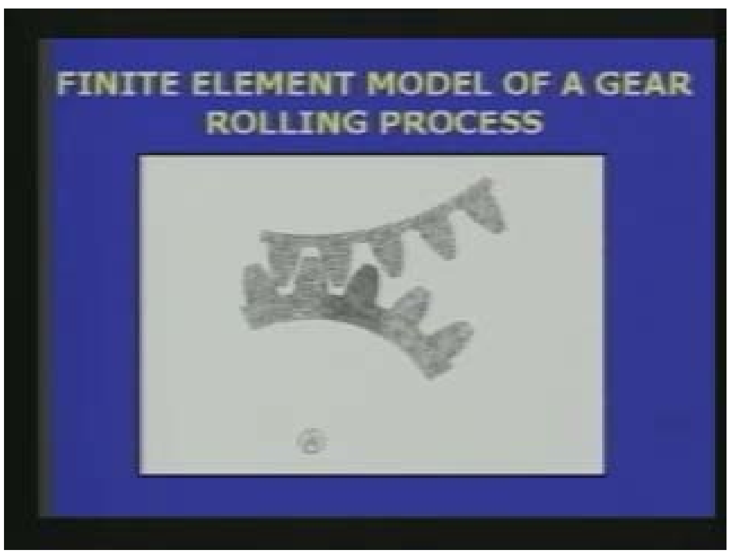
From I think 87 to 97, there have been 1000 papers published on finite element analysis to human bodies, different parts of the human body. There are nearly I think 350 papers; 300 to 350 papers on heart alone, human heart and other things that are important to us or cardiovascular system if I can call it, in that alone there are about 300 to 350 papers. There are extensive finite element studies for dental applications. So, biomechanics is a very important area today for the application of finite element analysis. There are lot of things that can be determined using finite element analysis in biomechanics and if there is time we will see one or two examples, maybe towards the end of the course. But let me warn you that many of these things are very difficult. They are not very straightforward and easy because of the way nature has designed us. The behavior of each and every part in our body is much, much more complex than the material which we use in order to do things.

There is no comparison between how our tissues behaves and say metal behaves; they are totally different and hence the problems that we get in using finite element for biomechanics are much more complex. There are other areas as well. For example in electrical engineering there are lot of applications in electrical engineering. Here again the applications are quite recent and again we are not going to talk more about these things in this course. We will restrict ourselves to design and manufacturing. We will look at only thermal problems, fluid mechanics problem. For example there has been lot of applications of finite element analysis in fluid mechanics; that again will not be covered in this course.

As I told you, finite element is a very, very vast subject. There has been tens and thousands of papers published, so we will restrict ourselves to the fundamental finite element for design and manufacturing. There are some applications of fluid mechanics in manufacturing but because of lack of time in this particular course I will not cover all those aspects. With this background saying that these are the applications of finite element analysis, let us now look at what is involved in finite element analysis.

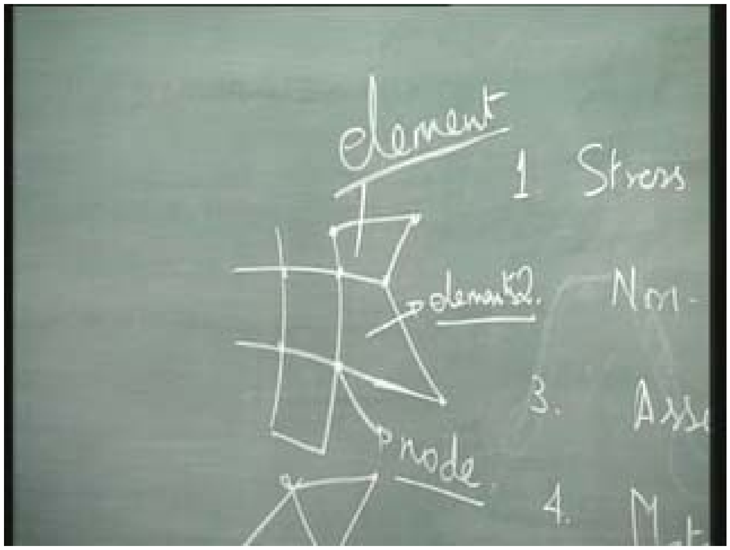
Let us look at the last slide in this particular series.

(Refer Slide Time: 40:50)



Have a close look at that. What does it indicate? What are the things that you see? You see lot of lines or what I would call as network. They intersect at points which are called as nodes. What is that you see or what is that you get or what is the information you get out of seeing that picture?

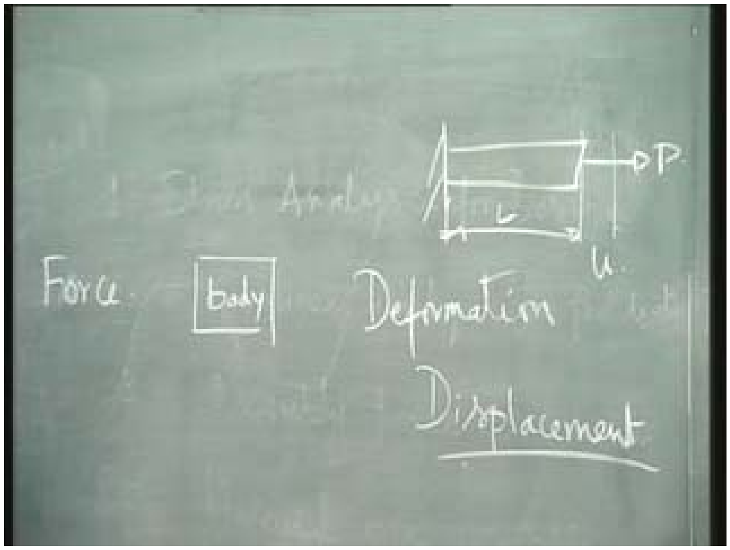
(Refer Slide Time: 41:19)



You see that there are lot of lines like this or in some other figure you would have seen lines like this. These lines meet at certain points and so on; these lines meet at certain points or the bodies which we considered for analysis have been discretized, have been broken down into what are called as elements. These are what are called as elements and this is an element and they are bounded by what are called as nodes which sit at the intersection of these elements. Say for example that is a node, that is a node, that is a node; that is a node and so on.

So, nodes and elements are the ones which are important in finite element analysis or what is that we have done philosophically? A complex component has been broken down into what are called as elements. What are these elements? How does it help us and what are the advantages of using this kind of approach is what we are going to see in this course. Essentially what is that we are looking for in a design problem?

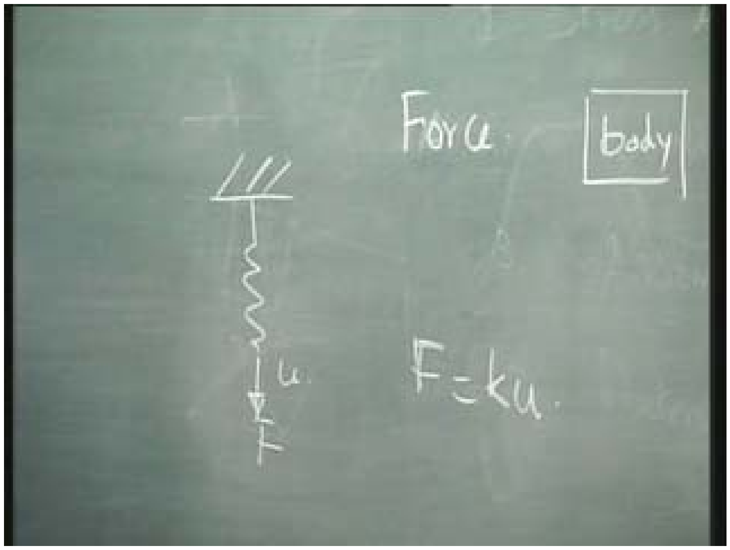
(Refer Slide Time: 43:17)



We have a force and we have a body and we are looking at what are the deformations? So, I want a relationship between force and deformation. Let us call this deformation as simple displacement. There is a difference between displacement and deformation, …. one, let us say that the body gets deformed and we will call this deformed quantity as displacement. If I have a bar and if I apply a load say P the end of the bar gets displaced by a certain amount say u. The body here in this case is the bar. The force that happens to be there on this body is P and what is that you are looking for? You are looking for this displacement, right.

If this is the L or the length of the bar we can easily find out what is the strain what is the stress and so on. What is that we are interested in? We are interested to find out what is this deformation or displacement? As mechanical engineers whenever I say that there is a force on one hand and displacement on the other hand what is it that comes to your mind? Stiffness; beautiful.

(Refer Slide Time: 44:56)

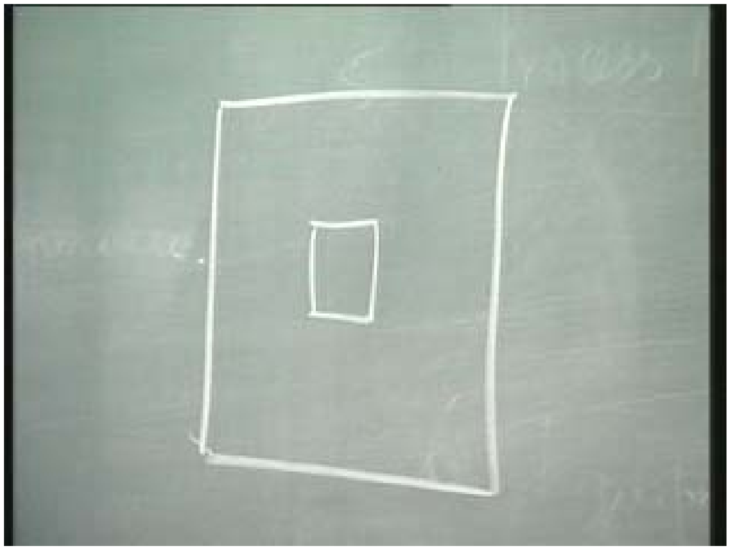


Suppose I have a spring for example and I apply a force here and I ask you what is the displacement? The first question you would ask me is what is the stiffness of the spring or in other words if this is the force and this is the displacement of the end of the spring then F is equal to ku. So, all mechanical engineers or all engineers generally know that this is a very fundamental equation. What is the difficulty in applying this straight away to a much more complex problem like what we have or what we had in these particular examples.

For example can we apply this to a railway wheel or to a side wall and so on? It is not possible, it is not very straightforward. Why? Because here I could immediately say that the spring displacement is measured or is characterized by the displacement of the end of the spring. When I say displacement of the spring immediately you know that I mean how much it is going to displace, this tip, this guy sitting here. On the other hand if you look at a three dimensional body, which we encounter in the actual practice I cannot say by one number that this is the displacement. What does it mean? It means that I just cannot come and tell you that this is the maximum displacement; your interest is just not in one number.

Your interest is in the complete displacement of the component at every point. You would come and tell me the displacement at this point, this point, this point, this point and so on.

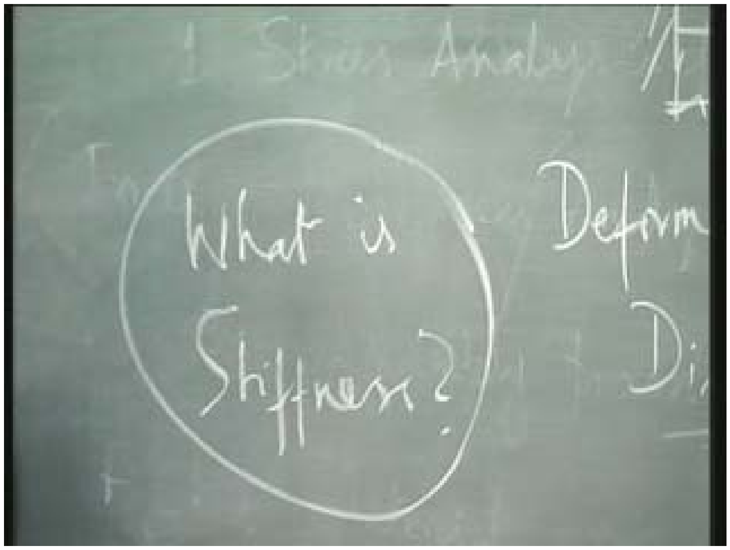
(Refer Slide Time: 46:54)



If I take the complete sheet, you would ask me what the displacement of this complete sheet is. Maybe to make it more realistic let me put a small window there. It is a complete sheet. Not only you will ask me the displacement of complete sheet but you would also be interested in strain distribution throughout the sheet. As we saw in one of our slides that the plastic strains of the forging, for example, was different at different places. It is not that you are only interested in displacement; it is only one small part of the problem. You are interested in strains and strain may not be constant whereas this problem, it is not so and stress in this problem is constant throughout this bar, but here the stresses may not be constant. Most components are such that stresses will not be constant throughout.

How do you now tackle that problem? Yes, this is a nice answer that you have spring and you have a relationship with force and displacement. How do you extend this? Can we develop a concept called stiffness and if we develop a concept called stiffness is it going to be one number? That is the question we are going to answer in the next two classes.

(Refer Slide Time: 48:24)



The next class we look at what is stiffness from a finite element point of view. We will answer that in the next class