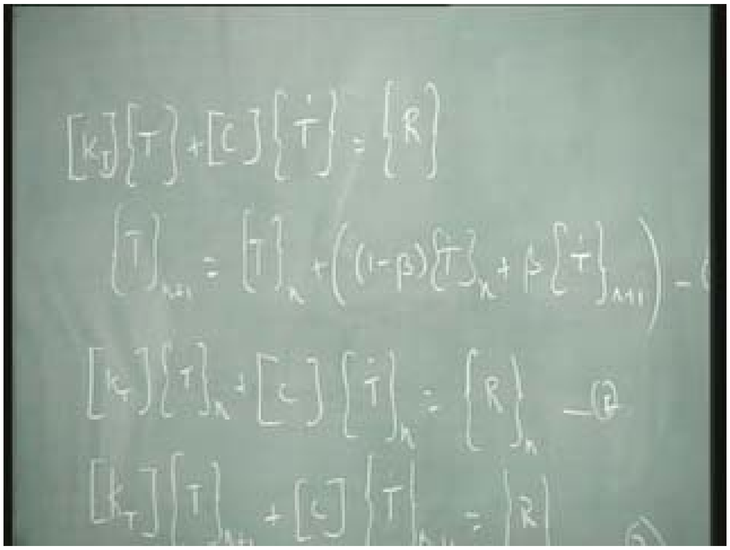
**Introduction to Finite Element Method**
**Dr. R. Krishnakumar**
**Department of Mechanical Engineering**
**Indian Institute of Technology, Madras**

**Lecture - 27**

In the last class, we were discussing about the temperature distribution. I hope all of you understood till this, this step. We had arrived at for a transient heat transfer, what would be the equation that you would solve.

(Refer Slide Time: 1:19)



$$[K_T]\{T\} + [C]\{\dot{T}\} = \{R\}$$

$$\{T\}_{n+1} = \{T\}_n + \left((1-\beta)\{\dot{T}\}_n + \beta\{\dot{T}\}_{n+1}\right) - ($$

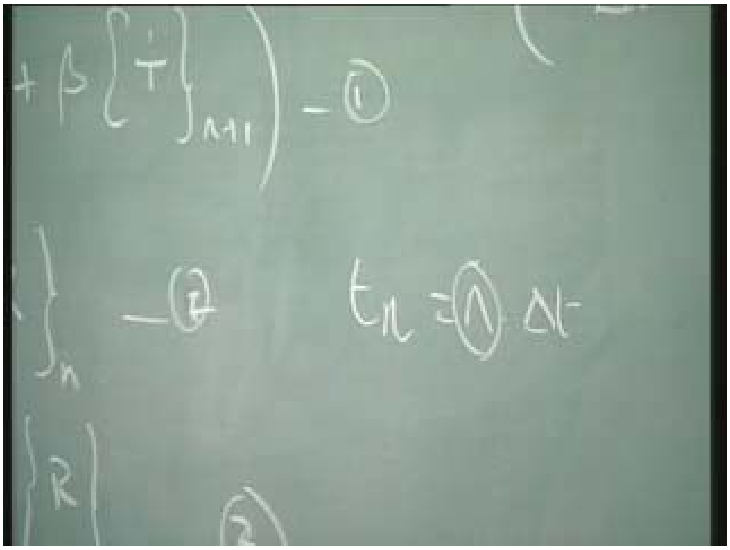
$$[K_T]\{T\}_n + [C]\{\dot{T}\}_n = \{R\}_n \quad - ②$$

$$[K_T]\{T\}_{n+1} + [C]\{\dot{T}\}_{n+1} = \{R\}$$

We had also discussed that this equation can be solved by means of finite difference scheme. Actually this is finite difference; people, who are familiar with finite difference approach for numerical analysis, would immediately realize that what we are looking at is similar to many of the schemes what you have seen; may be people would have called this as midpoint rule or trapezoidal rule and so on. In other words, essentially you can summarize at this point of time that you had discretized and used finite element analysis in the space, in space in 2D or 3D and you are marching ahead in time using a finite difference scheme. Actually you are looking at 4 different angles. One is $x_1$ $x_2$ $x_3$ as well as the time.

Please note that we had already realized that when I write like this, this is an initial boundary value problem. This can be very easily reduced to a steady state equation. How do I do that? Just remove this chap here, so, I can just straightaway reduce this to a steady state equation. How am I going to solve or what are these steps? These things we had already written in the last class. Let us see how I am going to solve this? Please note that I am just emphasizing, just telling you that we had started with time t is equal to zero, we have a time step delta t; we keep marching, all those things. I hope you are clear.

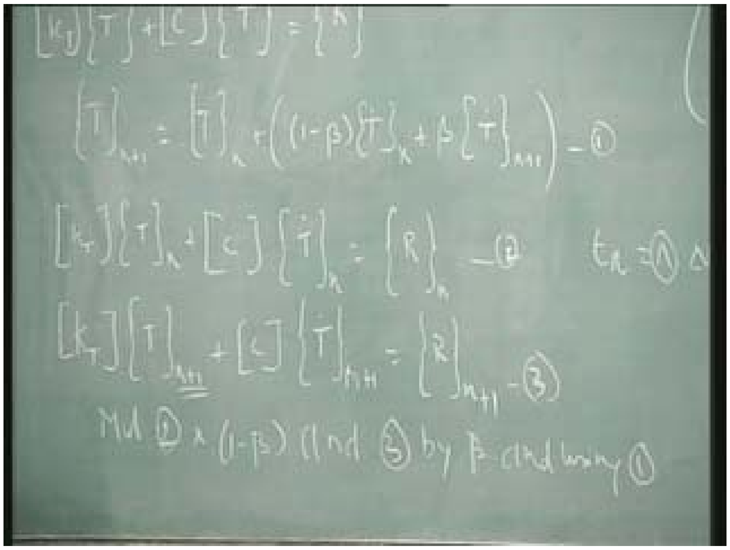
(Refer Slide Time: 2:58)



So, time at any time step, we usually denote it by means of a subscript say $T_n$. If it is a constant time step, then $T_n$ is equal to n into delta t where n is the number of increment of the time step. Wherever we use the subscript $T_n$ it means that we are at the step n delta t. What is that we are trying to do? As the first step, before we actually derive the ultimate equation, as the first step, note this that we are writing down the first equation for the time or for the step n. In other words, we are putting the subscripts n for t as well as for T dot n, as well as the right hand side. I hope you remember what R means and what C means and all that or else you can refer to it. Then for n+1, for n+1th step, I am marching from nth to step to n+1th step. That means time $t_{n+1}$ is equal to $T_n$ plus delta t. I can write down the same equation for n+1th step like this. Is that clear? The same equation, I can write it down. Now, I have three equations, 1 2 and 3.

What is that I am going to get? I going to now combine these three equations and get one simple simultaneous equation using which I am going to proceed as I had done the previous case.

(Refer Slide Time: 4:41)



$[K_T]\{T\} + [C]\{\dot{T}\} = \{R\}$

$\{T\}_{n+1} = \{T\}_n + ((1-\beta)\{\dot{T}\}_n + \beta\{\dot{T}\}_{n+1})$ — ①

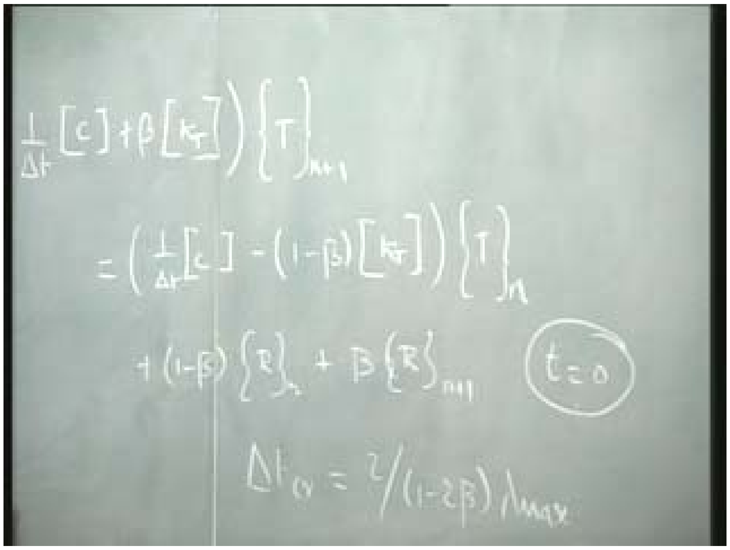
$[K_T]\{T\}_n + [C]\{\dot{T}\}_n = \{R\}_n$ — ②   $t_n$ = ① Δ

$[K_T]\{T\}_{\underline{n+1}} + [C]\{\dot{T}\}_{n+1} = \{R\}_{n+1}$ — ③

Mul ② × (1-β) and ③ by β and using ①

What is that I do? I multiply this second equation by 1 minus beta, the third equation by beta. I multiply these two, add them together and use the first equation; very simple. Simple algebra, there is nothing there, just multiply this like any other you do; any other simultaneous equation. I multiply that, apply the first equation. What is the result of this algebraic step? The result of algebraic step is this equation. Look at that equation for a minute, look at that equation.

(Refer Slide Time: 5:22)



$$\frac{1}{\Delta t}[C] + \beta[K_T]\Big)\{T\}_{n+1}$$

$$= \left(\frac{1}{\Delta t}[C] - (1-\beta)[K_T]\right)\{T\}_n$$

$$+ (1-\beta)\{R\}_n + \beta\{R\}_{n+1} \qquad \boxed{t=0}$$

$$\Delta t_{cr} = 2/(1-2\beta)\lambda_{max}$$

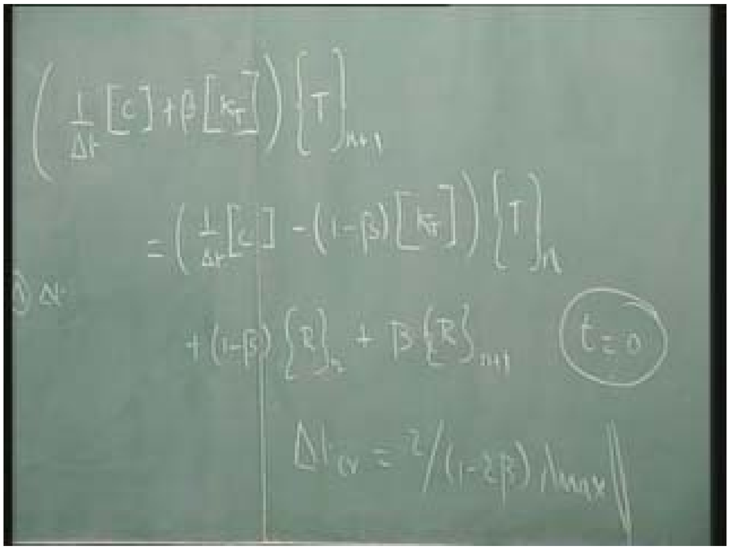
What is that you get out? Can you comment on it? Can you see, can you say that what has happened by applying this finite difference scheme? Can you see what has happened? Can you see that left hand side and right hand side? What I mean is comment means look at the left hand side and right hand side. For the time being, let us assume that the thermal properties are constant. It can vary with time. Then, you have to go to some nonlinear aspects and all that. Let us not worry about it right now, but let us just look at it. Can you comment on this equation?

In other words, this is essentially a simultaneous equation. That is all. Knowing the left hand side, this part of the left hand side, the unknown is $T_{n+1}$ and the right hand side quantities are already know to me. In other words, it is exactly like my very first equation in finite element kd or ku is equal to r, it is very similar to that. K is now a combination of C and my thermal K and the rest of it are ones which I had, which I know how to calculate. Is it clear? How do I start this problem? I have 2 1 start up is there, that is the initial condition. I start with what is the initial condition at time t is equal to zero. I start with there, decide a delta t, decide a delta t and then keep marching; keep marching, calculating C, $K_T$ and so on and then solve this equation to get $T_{n+1}$.

Please note that you have to apply the boundary conditions. If there is a temperature that is specified you have to properly apply that and so on. That is very similar to what we had done for the displacement case. If there is a boundary condition for temperature, you have to apply that, keep adjustments to the left hand side and the right hand side and so on. Is that clear? That has to be done, please do not forget that. In other words, you have to apply both the initial condition as well as the boundary condition. Is that clear?

We already talked about two things. One is the oscillation and the other is what we call as stability. We said that the solution may become unstable for cases where beta is less than half in which case you have to get a critical delta t. You have to operate only below that delta t.

(Refer Slide Time: 8:16)



$$\left(\frac{1}{\Delta t}[C]+\beta[K_T]\right)\{T\}_{n+1}$$

$$=\left(\frac{1}{\Delta t}[C]-(1-\beta)[K_T]\right)\{T\}_n$$

$$+(1-\beta)\{R\}_n+\beta\{R\}_{n+1} \qquad \boxed{t=0}$$

$$\Delta t_{cr}=2/(1-2\beta)\lambda_{max}$$

The critical delta t is given by this equation, where lambda max is the maximum Eigen value of the problem $K_T$ into T is equal to lambda CT, lambda K, sorry $K_T$ minus C T or in other words, K and C, you treat K and C like m and k and then you can write down that as the Eigen value problem. Let us not worry about the Eigen value problem right now. Anyway nevertheless, what I want to state is that this lambda max has a role to play and delta critical is important, when beta is less than half. But, usually when we use beta is equal to half, the trapezoidal rule then, this issue does not come away. So far, it is quite clear.

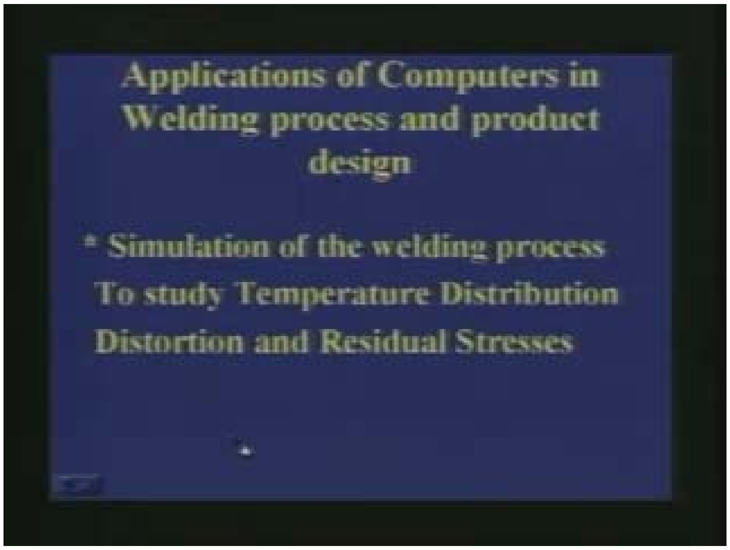
Now, I am going to give you an assignment which I will explain in the next class, but I want you to have a look at that and write it down. We will discuss it, if there is any issue, in the next class. What I am going to do is that we have not discussed in depth axi-symmetric element. We have done plane strain, plane stress and solid. Now, I want to introduce axi-symmetric element slightly more, to a much deeper extent than what we have studied previously. I want you to develop a B matrix and so on, $K_T$ and so on for an axi-symmetric element; attempt it. We have already seen what an axi-symmetric element is. There is an axis of symmetry and that you are going to rotate it and so on. I want you to develop this for an axi-symmetric element, heat transfer; also extend it for our displacement problems as well. So, my assignment for you is, read or try to develop axi-symmetric element, axi-symmetric, of course isoparametric element.

(Refer Slide Time: 10:35)

We have not dealt with it in depth. So, please look at that. The procedure is the exactly the same or the equation, the set of equations that you are going to deal with is exactly the same as that of plane strain and plane stress, but there is one r there, radius, which comes into picture and so on. Remember that we had looked at it when we discussed, where did we discuss? Strain displacement matrix; early, in one of the earlier classes we had discussed this, but we have not looked at it again after we had put down isoparametric element formulation. I would like you to have a look at that axi-symmetric element. I will give you time, so, tomorrow we will discuss more about axi-symmetric element and its application here. I just reserve it, because I want you to think about it. I do not want to give you the solution straight away.

So far, I hope there is no question on this. We will give, we will look at certain examples now. Welding, as I had been harping on it for quite a while is an important manufacturing process where we are going to apply whatever we have studied right now. We are going to apply more than that and that will lead us to the next lecture on nonlinear finite element. But, before we go to that let us now look at some of the examples that we can get from welding. We will just quickly go through what you can get out of it and why you do manufacturing simulation and so on. May be, you have heard about it before itself, I have just given you. Hence we are going to quickly run through it, but now the purpose of repeating it is because, now with all this background, the theoretical back ground, your understanding of the problem itself will be better now. Hence let us start with the first slide and let us see what we get out of it.

(Refer Slide Time: 12:37)

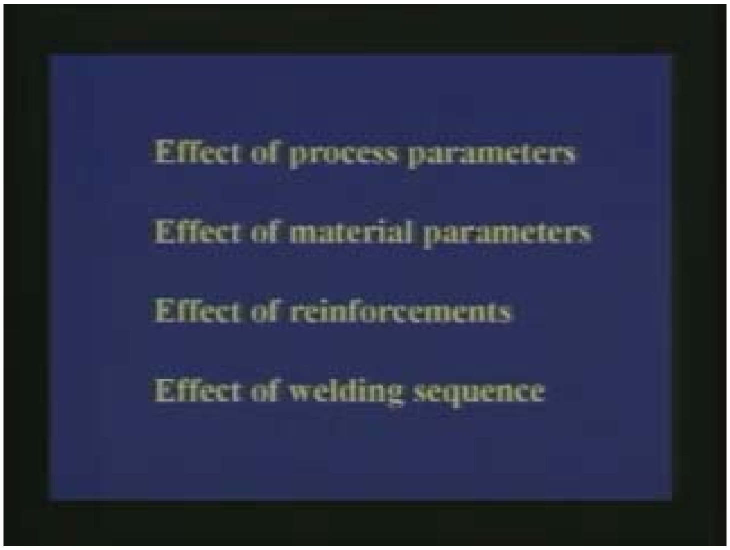


Applications of Computers in Welding process and product design

* Simulation of the welding process
To study Temperature Distribution
Distortion and Residual Stresses

What we are seeing now is the application of computers in welding process. Let us look at the next thing. We are going to study the temperature as well as the residual stresses.

(Refer Slide Time: 12:49)



Effect of process parameters

Effect of material parameters

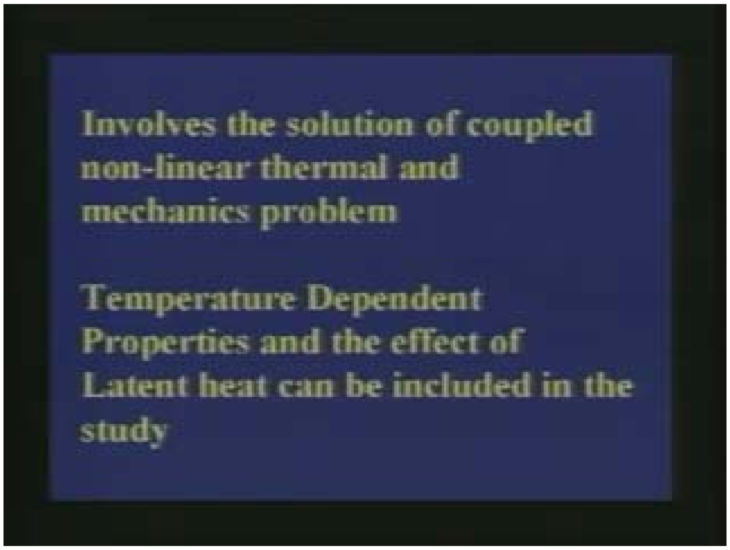
Effect of reinforcements

Effect of welding sequence

What are the things that we are going to study or why do we study this welding problem? We can determine what effect of process parameters is. Say for example, the speed with which we are going to move the arc; you can study for example what would be the effect of say the voltage, current what you apply and so on. Then you can also study what the effect of the materials is? Material parameters, you know some of the materials are more difficult to weld and so on. So, we can have an effect on the material parameters. We can also look at what is the effect of reinforcement.

You can look at the slide again. You can see that you can look at the effect of reinforcement. In other words, many times we reinforce, for example the sheet metal work which we had seen earlier for our coach, has certain beam type reinforcements, the sheet is reinforced. It is also possible to study this kind of effect of reinforcement on the deformations by complete finite element analysis and let us have a look at it and we can also study the welding sequence. How do you sequence this welding? There are different ways in which people do this welding and that also it is possible to study.

(Refer Slide Time: 14:08)



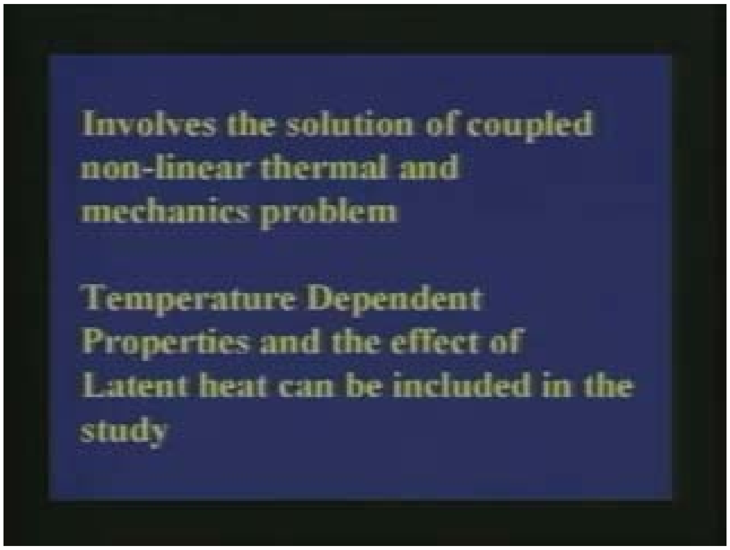
Involves the solution of coupled non-linear thermal and mechanics problem

Temperature Dependent Properties and the effect of Latent heat can be included in the study

Let us look at the next slide and see what we can further know or involved in welding simulation. Basically, I have been telling that this welding simulation is a coupled problem. We have to solve both the thermal problem as well as the mechanics problem. Now, this problem has to be done sequentially. What is meant by sequentially? You know, probably all of you now understand what I really mean by sequential? You see that from whatever theory we have done, whatever theory we have done, we see that we have to march in time. We take t is equal to zero, we go to $t_1$ which is $t_0$ plus delta t, $t_2$ so on. So, we go to the solution in increments. What is that we do? What is meant by this sequential? We first solve this problem of temperatures at say $t_1$, then stop there, determine what the displacement is due to say thermal expansion, if there is a load due to that load. Then find out what the stress and strain is. First thermal problem, then mechanics problem, then again I go to the next step t is equal to $t_2$ that is $t_1$ to delta t. Then again solve for the temperature problem. Is it clear?

Then, again look at the mechanics problem. Many people misunderstand this sequential; it is not that we solve the complete problem, temperature problem, completely and take only the last temperature and then look at what the time step is. We cannot do it, because many of the problems what we are looking at, especially when you are looking at residual stresses, may involve plastic strains. Plastic strains being history dependent and so on we are going to study that shortly, it is not possible that you just take the final temperature and initial temperature especially a problem like welding. So, you have to do it sequentially. You solve the temperature, come back, look at the mechanics, again solve the temperature, mechanics and so on.

(Refer Slide Time: 16:37)



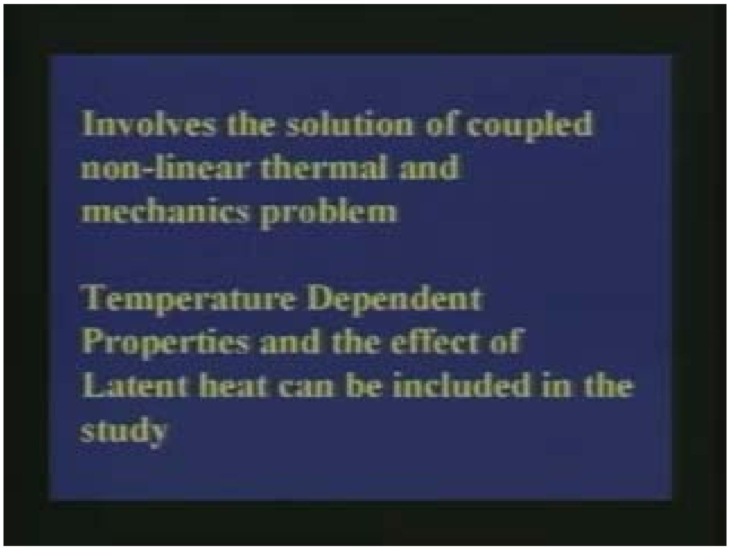
Involves the solution of coupled non-linear thermal and mechanics problem

Temperature Dependent Properties and the effect of Latent heat can be included in the study

Let us now again have a look at the slide and see the other important thing that you see. You see that we have introduced the temperature dependent properties and the effect of latent heat. What does it mean? It means that the temperature that we are talking about is a, sorry properties that we talking about, what are the properties that we are talking about? Remember we got it from $K_T$ C and so on. What are the properties that, yeah K rho C P and so on all these things are also a function of temperature. Not only that when we saw mechanics problems, when we solve mechanics problems, especially with welding distortions, residual stresses and so on, then what is involved there? Sigma$_y$, yield strength and other mechanical properties, may be E. These fellows are also functions of temperature. It is important to realize that whatever we are looking for or whatever we are seeing are all functions of temperatures.

Let us go back and have a look at that slide again. There is one more thing that you see there.

(Refer Slide Time: 17:53)



Involves the solution of coupled non-linear thermal and mechanics problem

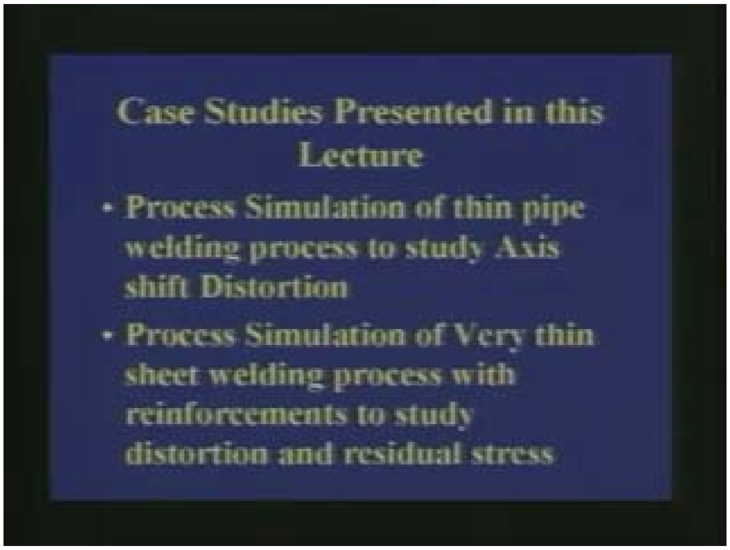
Temperature Dependent Properties and the effect of Latent heat can be included in the study

What is it? Latent heat; there are number of ways in which you can treat latent heat. It can be treated as q or the heat input or it can be treated as a variation in C P and so on. There are latent heat models that are available. I hope if you understand right now that latent heat has to be treated, you will be very careful, because you can say that for example; I know one of the ways in which you can do is, what is latent heat? Yeah, hidden heat. What happens during say melting or solidification, say for example what happens during solidification?

Without temperature change there is an increase. This is for pure metals, but it varies from one material to the other. You can treat this as some sort of either a heat sink or heat source depending upon what you are doing, solidification or melting or solidification and so on. So, it can be treated as a heat source or you can adjust the latent heat to be for example an increase in Cp values and so on. There are different latent heat models. In other words, by adjusting the thermal properties, latent heat can be taken into account.

Let us see the next slide.

(Refer Slide Time: 19:13)

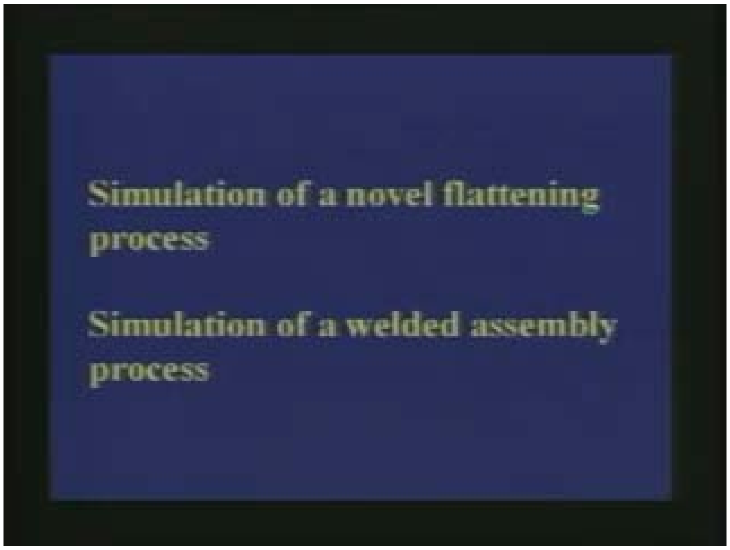


Case Studies Presented in this Lecture

- Process Simulation of thin pipe welding process to study Axis shift Distortion
- Process Simulation of Very thin sheet welding process with reinforcements to study distortion and residual stress

But, what I want to emphasize is that it is possible to take into account latent heat. Let us look at two case studies. Let us look at the first case study which involves the welding of thin pipes, what we call as axis distortion during thin pipes. Let us first look at the first case study. Let us look at the next slide.

(Refer Slide Time: 19:37)

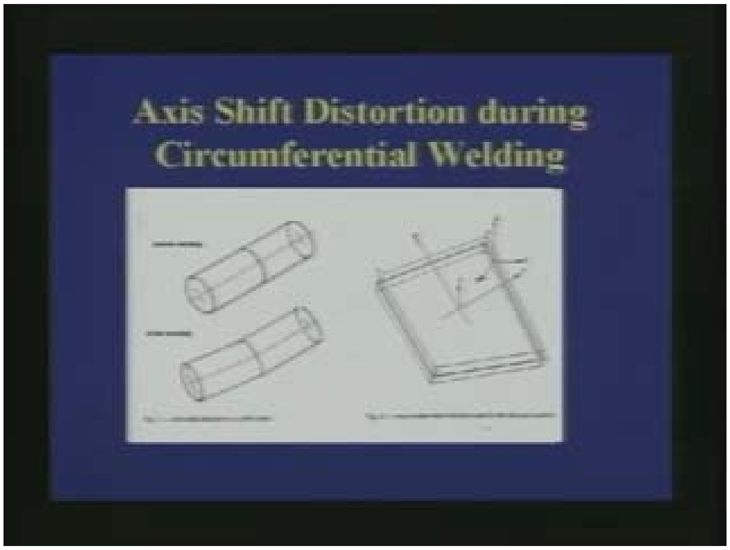


Simulation of a novel flattening process

Simulation of a welded assembly process

We will also study the flattening process and so on. I think that we have already done it, so, we will not do it again. But let us look at the first case study, which we will start from the next few slides.

(Refer Slide Time: 19:53)



Axis Shift Distortion during Circumferential Welding

Yeah, so this is the problem. The problem is defined there. You see that there are two pipes. These two pipes have to be welded; circumferentially these two pipes are welded. Is it clear? Have a look at that. There are two pipes which have to be welded. The second figure actually shows the distortion. They are thin pipes and both of them are to be welded. Where do you apply this? These pipes are for example applied or this welding of these pipes becomes important when you want to carry oil or gas or whatever it is. Oil gas and all these things for example are carried over very, very long distances. For example, in India we are talking about gas, you know, pipe lines from Bangladesh and so on and say for example in Russia, these pipe lines are laid for thousands of kilometers in their Siberian region and so on. In Australia, this has become a very important problem, where they transport oil over these pipe lines.

The distortion of this pipe line becomes important. Why, because if the distortion keeps increasing as you keep adding these pipes, the axis’s are not the same. The axis keeps shifting.

(Refer Slide Time: 21:19)

So, instead of giving a straight pipe like this, you will, sorry like this, you will get a pipe like this. Then laying becomes a problem. So, you have to study the distortions, see whether the distortions can be first understood, why it happens and then adjust it. See, always when you know why distortions happen, what is the cost, it is easy to understand and then take corrective action. Is that clear? Let us now look at that problem.

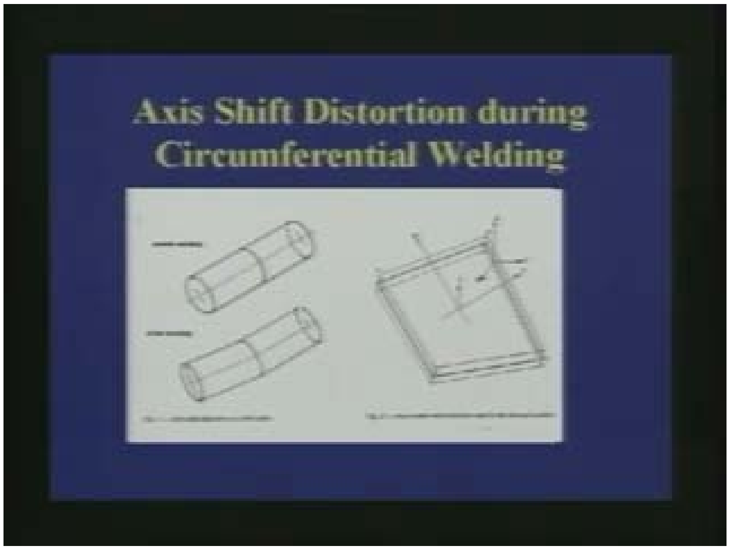
(Refer Slide Time: 21:47)

Axis Shift Distortion during Circumferential Welding

The right hand side shows the type of element that we have used. What is the problem now? I have to do sequentially two problems. One is I have to of course predict the temperature as well as solve the mechanics problems. What do I mean by the mechanics problem? Please note, I have been using this word mechanics. Please note that because of temperature changes, temperature gradients, there are thermal expansions. We have already indicated that, remember that, when we derived one of our first lessons on stiffness matrices and so on we saw that there is a strain term. If there is a strain term, how to handle, the initial strain term, epsilon$_0$? Go back and see that notes. You will see that we had already indicated that it is possible to handle initial strains and these, the coefficient of thermal expansion and this temperature variation, temperature gradients, result in an initial strain epsilon$_0$. So, that is what we are going to handle.

The loading is through this, for example initial strains. But, if you have any other type of loading, may be you can weld it under pressure or something like that, then that also has to be added to the loading condition for the mechanics problem. Is that clear? What is the element that we have used? Have a look at that.

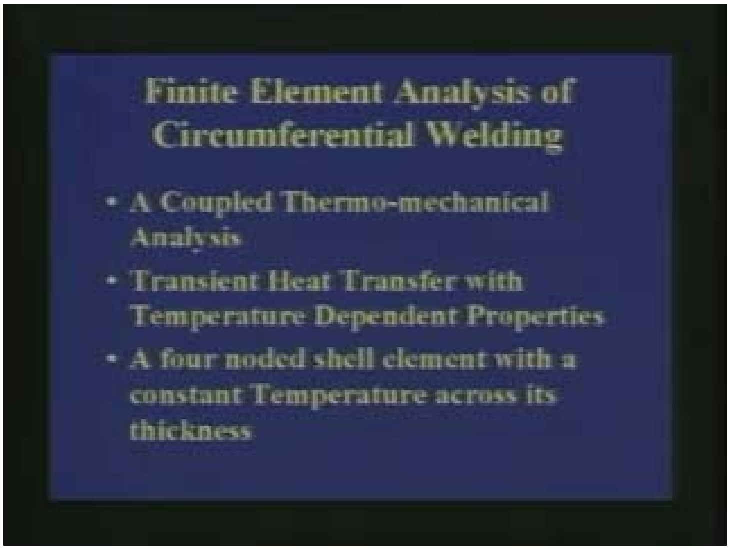
(Refer Slide Time: 23:30)



Axis Shift Distortion during Circumferential Welding

You see that we have used a shell element, a 4 noded, what we call as deep generated shell element is what has been used, obviously, because this pipe as you can see it has a shell behavior, shell type of behavior and hence we have used a shell element. Now, let us look at the next slide.

(Refer Slide Time: 23:54)

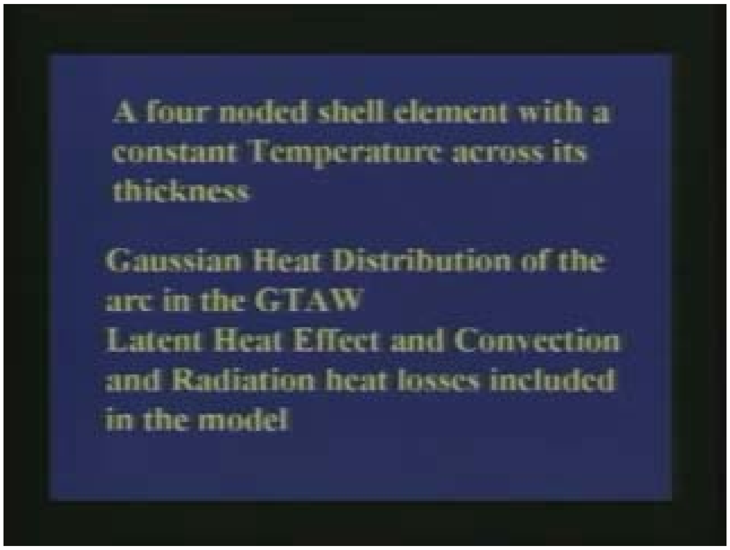


## Finite Element Analysis of Circumferential Welding

- A Coupled Thermo-mechanical Analysis
- Transient Heat Transfer with Temperature Dependent Properties
- A four noded shell element with a constant Temperature across its thickness

Let us look at the properties. It already says that you have to do a transient; we have already seen that, a 4 noded shell element and the next slide shows you something very interesting that the latent heat, convection and also radiation can be taken into account.

(Refer Slide Time: 24:15)

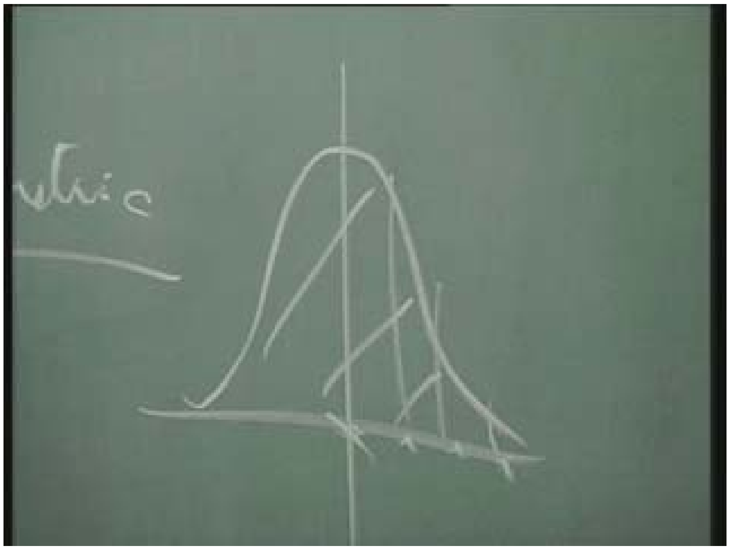


A four noded shell element with a constant Temperature across its thickness

Gaussian Heat Distribution of the arc in the GTAW
Latent Heat Effect and Convection and Radiation heat losses included in the model

Sometimes it so happens that when the temperatures are large near the welding zone, the radiations also has to be taken into account. But, before that there is what is called as a Gaussian heat distribution for the welding process that has been put. In other words, when you weld you give a heat input.

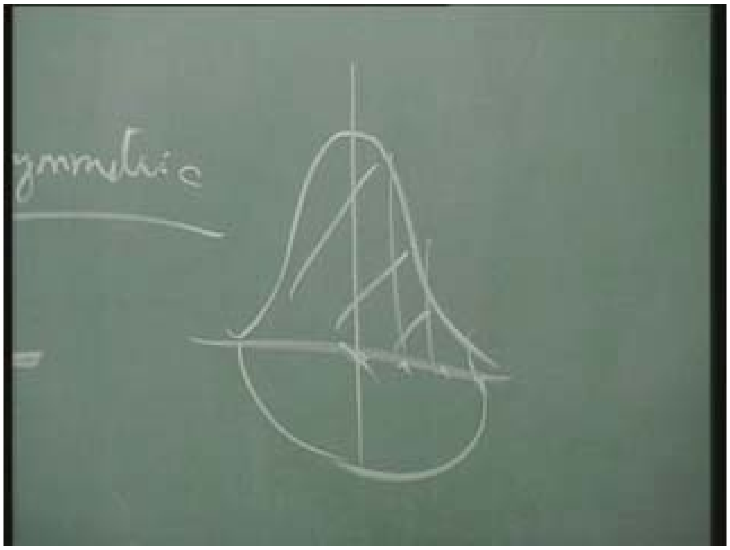
(Refer Slide Time: 24:48)



The heat input can be given in such a fashion that it can be distributed something like this about a center. The heat input is distributed in a Gaussian fashion. In other words, the area under this Gaussian curve is equal to the total heat input that you give. Is that clear? In other words, what it means is that this is the total area. If there are nodes, number of nodes, you can see that these are nodes there. I am cutting a section. Actually the arc is moving perpendicular to the board; it is moving perpendicular to it. As it moves perpendicular to it, a number of nodes come into contact. Infact in the other direction, it becomes a bell shaped. In 3D, it becomes bell shaped, in 2D it looks something like this.

This Gaussian distribution of heat energy starts moving. It starts moving. so, different nodes It is a moving heat source, so different nodes come into contact with this Gaussian distribution with respect to time and the distribution itself gives a very good picture of how heat will go into the material. If you are not so concerned about it, of course, you can put it at only one node at the centre all of them, but if you are not concerned about heat distribution very near the weld pool, then you can, it is possible to concentrate them, the whole of them only at one node. But, in this case we are actually not modeling weld pool. There are lots of other issues in weld pool modeling.

(Refer Slide Time: 26:36)

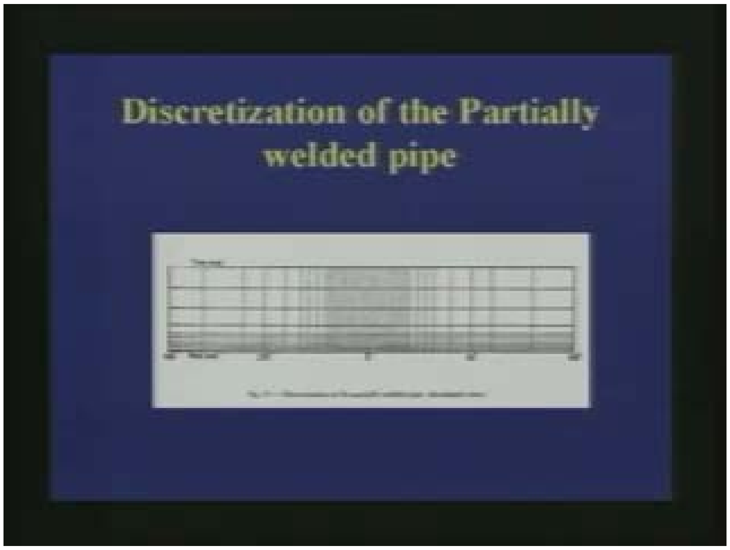


There is going to be material that is going to be melted below this, below the surface. Is that clear? We are not going to be concerned with the convection, for example, convection effects that may take place in this weld pool and so on and all those things we are not considering. We are considering this heat input. Of course we are also considering the latent heat as I had mentioned before. What is this and why this becomes important? This becomes important because practically the heat source cannot be at one point. We are trying to convert, with the best things that are available, the heat source, the heat input into a distribution which can nicely fit into the mathematical aspects of our finite element as well.

There are other issues like for example, what the efficiency of the process is and so on. I think a welding engineer would be able to tell you as to what the type of efficiency is or whether the power that you draw that goes, how much percentage of it goes? May be about 40% of the power is what goes inside and so on. So, it depends upon the efficiency of the process. What does it tell us? It tells us a very important story that on one aspect there is mathematics, there is finite element. on the other aspect, There are other aspects to this process simulation which depends upon the process. So, you need to have a process knowledge, say for example, you should know what the power is, what the efficiency of the arc is, what the velocity is with which the arc can move. It has to move, but what is the velocity with which it can move and so on, as well as the material properties. What are the material properties that you are looking at, all these things become important. That is why manufacturing process simulation is a very interesting and very intense process.

On one hand, you have to have the knowledge of the process itself. What the type of welding is, then what is the type of process variables that you are looking at, then what is the type of materials that you are looking at and then coming to finite element analysis and see how we can apply all these aspects to finite element analysis. Let us have look at the next slide.

(Refer Slide Time: 29:03)

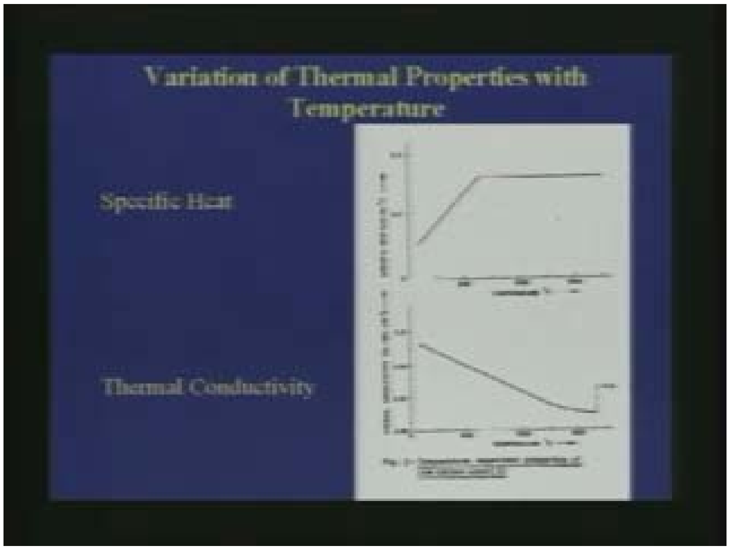


Discretization of the Partially welded pipe

This is the discretization of the welding that we see. This is a thin shell process. Actually, you can see very fine mesh along the regions where it is being welded. At two places where it has been welded you see that it has a very fine mesh, because the welding has both circumferential direction, I mean as well as if necessary you can also study welding in the other directions as well.

Now, let us look at the next slide and see how the variation of properties are taken into account.

(Refer Slide Time: 29:45)



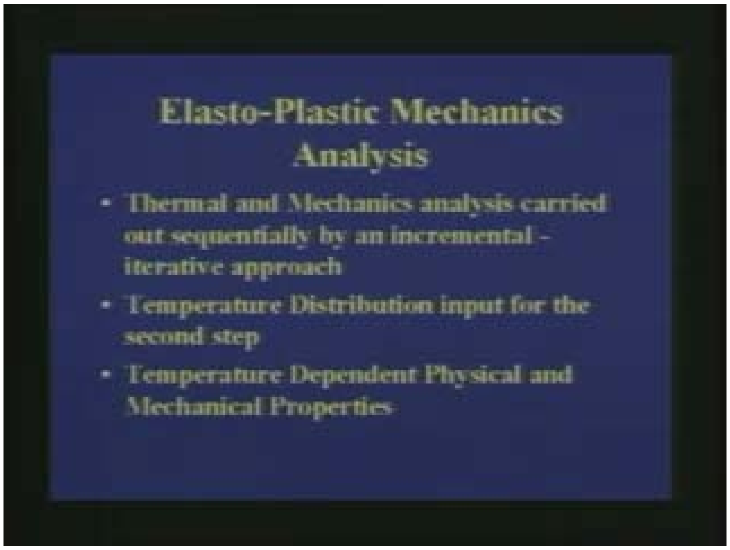
Variation of Thermal Properties with Temperature

Specific Heat

Thermal Conductivity

Look at the variation of properties with respect to temperature. That is also been studied. Let us look at the next one.

(Refer Slide Time: 29:52)

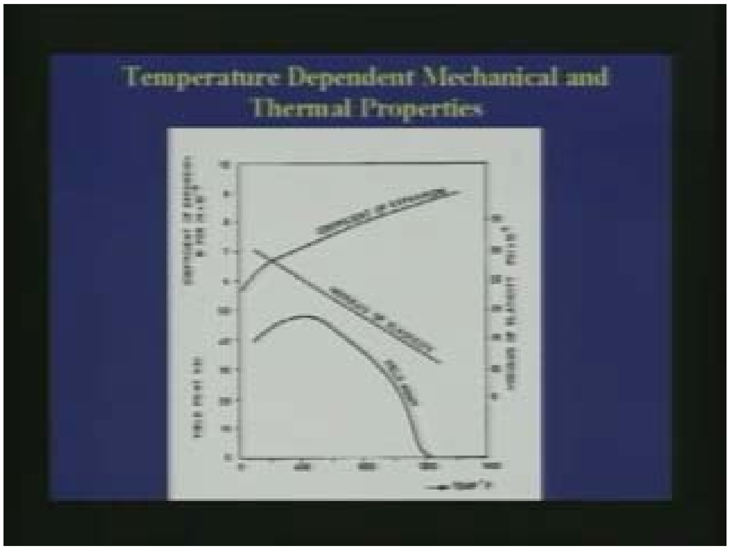


Elasto-Plastic Mechanics Analysis

- Thermal and Mechanics analysis carried out sequentially by an incremental - iterative approach
- Temperature Distribution input for the second step
- Temperature Dependent Physical and Mechanical Properties

What we are going to attempt here for mechanics problem is what is called as an elasto-plastic analysis. After we finsh the study, we will take up elasto-plastic analysis, elasto-plastic analysis, but till now, the only game is that when you want to do similation, you have to look at also, after all these things, properties also and so on you have to also have a mesh design in a such a fashion that you have fine meshing places which is of interest to you and courser mesh as you go outside the region and so on. We are going to study how to do an elasto-plastic analysis, what is meant by elasto-plastic analysis, that we are going to study after the set of things and then let us have a look at the next slide.

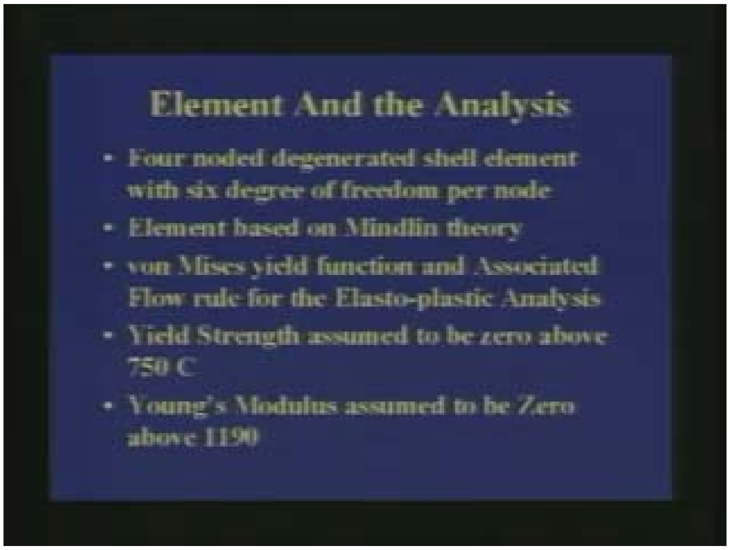
(Refer Slide Time: 30:45)



Temperature Dependent Mechanical and Thermal Properties

Look at that, temperature dependent thermal properties. You can see how different mechanical properties vary with respect to the temperatures. Any question? Let us look at the next slide.

(Refer Slide Time: 31:09)



## Element And the Analysis

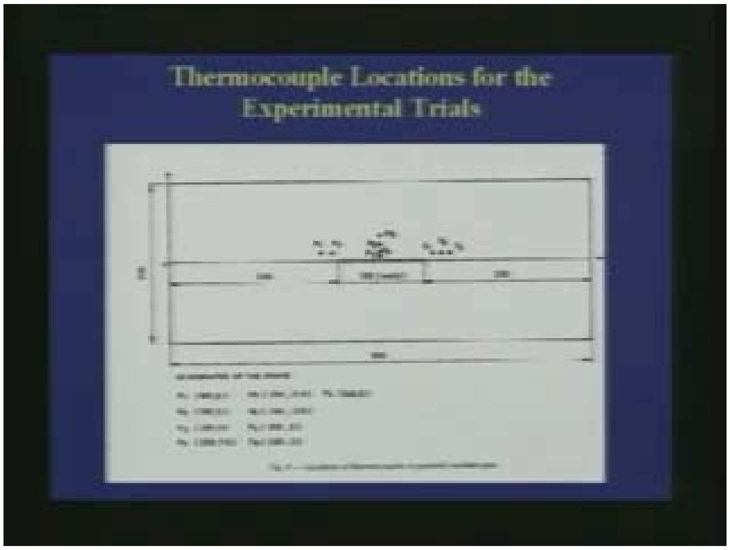
- Four noded degenerated shell element with six degree of freedom per node
- Element based on Mindlin theory
- von Mises yield function and Associated Flow rule for the Elasto-plastic Analysis
- Yield Strength assumed to be zero above 750 C
- Young's Modulus assumed to be Zero above 1190

We had already talked about the type of element that we have used. These elements, later in the course may be after 3 or 4 lectures, we will talk more about the shell element, may be after 3 or 4 lectures, but right now, keep in mind that this is a shell element based on Mindlin theory. There are two theories, Kirchhoff theory and Mindlin theory. We will see what this Mindlin theory is. For the time being you can assume that when the shell becomes thicker, some of the assumptions that you make on shear deformation of the shell or in other words plane sections remain plane, for a plane or plate?, it may not remain the same. Then, the plane sections need not remain the same. So, you have certain adjustments that have to be made and there are theories to make these adjustments. We will study in a nutshell about these things later in the course may be after about 3 or 4 lectures.

Right now, it is important to understand physically that you have chosen a shell element. You know why, because, we are dealing a pipe. One of the best elements; that you can see immediately looking at the configuration, geometry and looking at how it is going to behave, you automatically say that you choose a shell element. This shell element what we have chosen, is a 4 noded shell and the temperature or the thermal properties or thermal the way the thermal analysis has been done is very similar to what we have discussed so far that the heat transfer from the faces of the element also happens and if you remember we had put down two terms. Remember that we had put down two terms for or to take into account the heat transfer from the face of the element as well. Remember that that also we did. So, that also has been taken into account. Let us see, let us look at what happens with this elasto-plastic analysis?

Let us look at the next slide.

(Refer Slide Time: 33:20)



Thermocouple Locations for the Experimental Trials

This is a work which was started in collaboration with an industry and what is also important is just not predict temperature. Predicting temperature is important, but also we have to verify it. It is not that every time we run a finite element program, we have to verify it. It is not possible, because we are using finite element program for designing. So, we may have to run finite element program even before we start manufacturing the process or sorry manufacturing the product and hence it is important that at one point of time we verify and see whether whatever results we have or the model we have is correct.

Please note this carefully. This is one thing which is usually bugging lot of people in the industry as well that every time you run a program, have you verified your result. You know, this is a question that may be, many of you even in your viva-voce or thesis presentation, people would have asked. Have you verified your results? This is some sort of a catch 22 situation. I want to emphasize this. This is some sort of a catch 22 situation. What is it? Now, I want to use finite element analysis for design. That means that my product does not exist. If I can verify whatever results I have, I need not do finite element analysis. I can as well do the component or the product and then test it experimentally and be done with it. Why would I go through all these mathematics or all through, all the software and hardware and all those kind of things to do it?

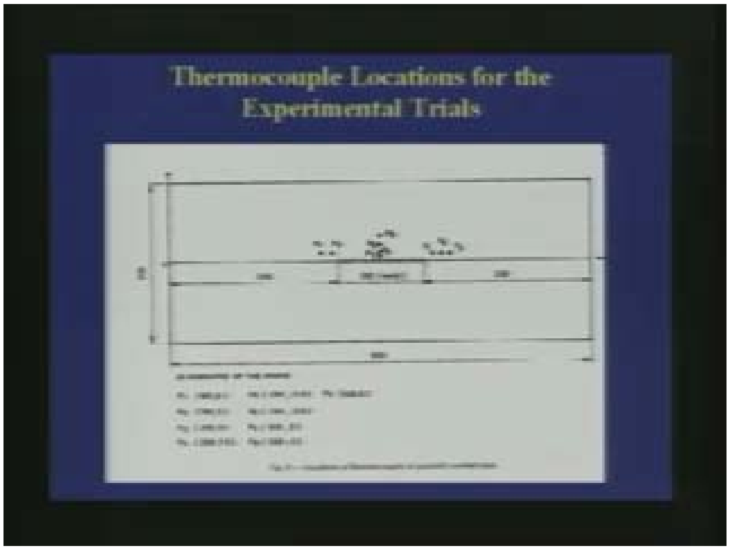
Then, it brings us to a very important question that what is the reliability of finite element analysis, because many people still do not believe in the finite element results. That is why they keep asking this question, have you verified your simulation? I am sure lot of you has been asked this kind of question. Have you verified whatever you have done, simulation? Actually the question has a different meaning, which people do not understand. Even people who ask this question, they do not really understand why they are asking this. Basically because, the meaning is that it is not a question of suspecting finite element analysis. The meaning is that you have made some assumption; the assumptions are say properties, thermal properties. May be you have made some assumption on the heat input, say for example, Gaussian distribution heat input, I have assumed. Maybe I have made an assumption as to how I can treat latent heat, total heat that is say evolved from an element I am distributing it to say 4 nodes as q. I have made a small assumption there. Total heat that is generated in an element I am distributing it to 4 nodes. This kind of assumptions that you make, are they correct, are they valid or not, that is the question; not finite element is right or wrong.

How do you justify this? You just do only one experiment. For example, we have done one experiment for this kind of thing. Once I show that look, I have done an experiment, I have done finite element analysis and whatever assumptions I have made are quite good within a reasonable limit, may be 15% error, 15% is very normal I will come to that in a minute; once I do this kind of experiments and show that whatever finite element I have done, agrees very well with this experiment, I can put a full stop to further experimentation. Then what I say is look, I have already done this analysis, I have already got these experimental results and they have matched quite well. So, from tomorrow onwards, whenever I want to do a new, say welding simulation, I need not worry about this. I will just go ahead and do the simulation, then optimize my process. This is as far as these kind processes are concerned.

Infact, experimental verification I would are required for simulating manufacturing process, because so many things are involved. The problem is very complex as you would have seen. On other hand, if you want to just do a stress analysis, simple stress analysis even involving things like contact, things like contact, you need not, I do not think you need to do every time a check, whether the result what I have got is right or wrong. I think that is a waste of time. As long as you know finite element analysis, of course I am not talking about mistakes you make; instead of shell element you choose plane strain element or something like that, then it is a different story. As long as you know finite element analysis and as long as you apply it correctly, I do not think there is any need for you to go and justify what you have done is right or wrong, there is no need for it. But in these cases, because we have made so many assumptions, we go and look at the result. Infact you would be surprised or may be many of you may know it that many of the nuclear components, nuclear pressure vessel components today, are subjected to finite element analysis, before they are put into nuclear pressure vessels. That is the type of strength or the integrity that is placed on finite element analysis.

Let us now have a look at the results.

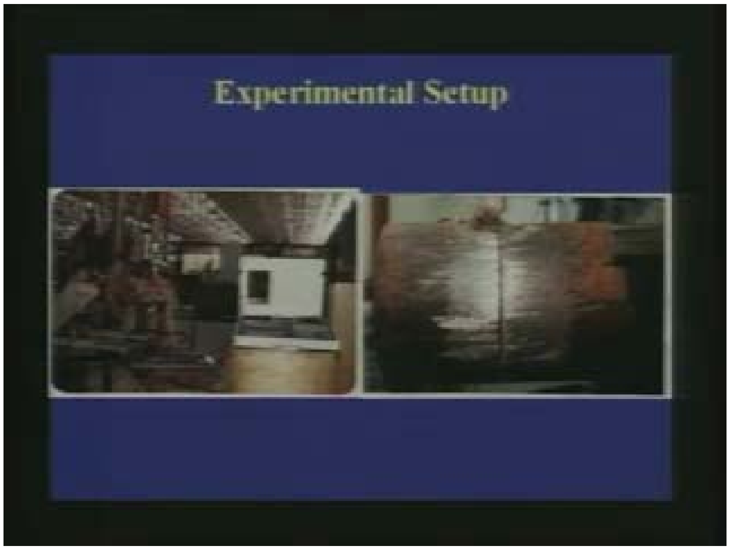
(Refer Slide Time: 38:59)



Thermocouple Locations for the Experimental Trials

That is where you know, thermocouples were placed and temperatures were measured. Let us look at the next graph.

(Refer Slide Time: 39:07)

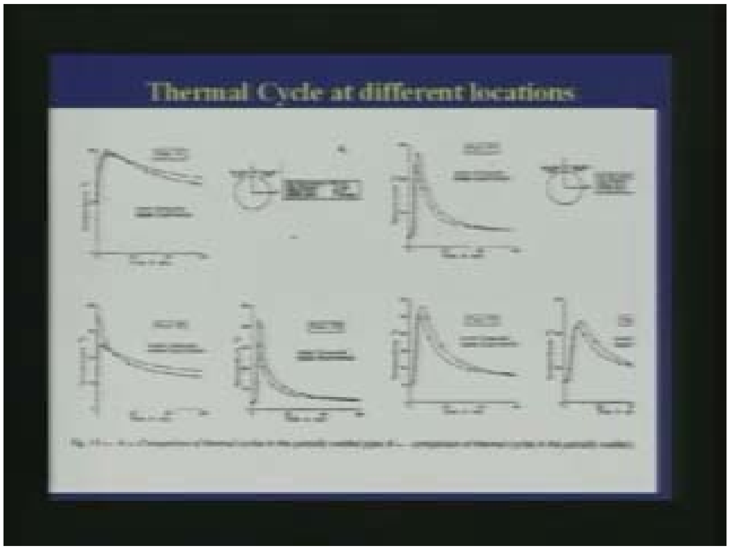


Experimental Setup

That is the experimental set up. This slide show the experimental set up. Very detailed experiments were conducted. You can see the pipes together and you can see that there are number of thermocouples that are there and you can see that these thermocouples are monitored with respect to time and the welding now taking place. Is that clear? At different places all these thermocouples are located and what is that you are going to do? Now, you are going to run the analysis, you are going to run this test, see whether at the same locations where you had placed thermocouples, what are the results? Is that clear? What the temperature is, whether it agrees with whatever you have measured or not with respect to time and so on.

Let us look at next graph and then that will tell you what happens?

(Refer Slide Time: 40:00)



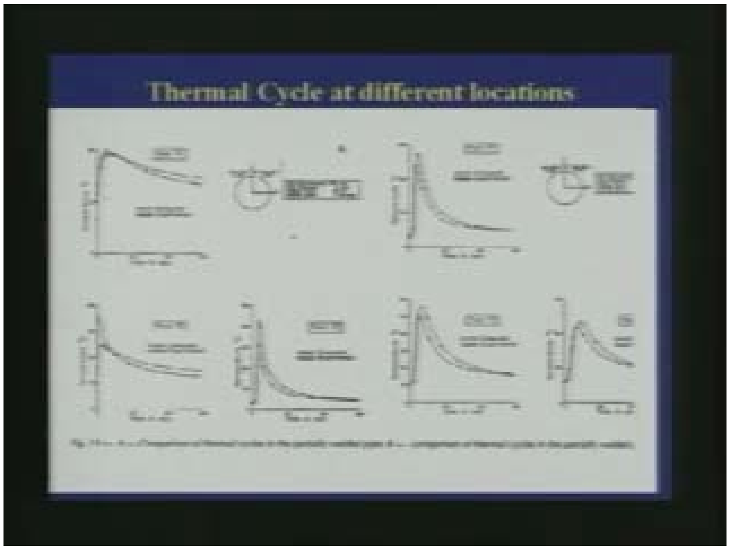
Thermal Cycle at different locations

That is the results. I am sure it is very clear. There are two graphs. You can see that the agreement is quite close between the experimental and the finite element analysis. Not that they are 100%, it is impossible that the two graphs would be 100% the same, but the results are quite very close and as I told you, we are looking at about 15% error. Now, this term error, it is important that you understand what we mean by error? In this case, the error may not be due to finite element. See, error may be there in finite element analysis, especially if you are running a linear elastic analysis, due to the selection of the element.

We have already seen that you may have to go for smaller and smaller elements. So, error may be there from a finite element sense due the element choice. But here, when we say error what we really mean is the properties that we have selected, the type of boundary conditions we have put and those kinds of things; error due to modeling. These errors cannot be eliminated even if I make the mesh finer. There are two errors. One is finite element error which can be reduced by making the mesh finer, better choice of elements and so on and the other error is due to this kind of what we call as model. So, the error may be high or low depending upon your intelligence or understanding the process itself. Whenever we talk about error, we talk about combined error, both the finite element error and the assumptions that you have made to simulate the process. On the whole about 15% to 20% personally I feel, will be very happy and that is the type of results we have been getting when compared to the actual graph.

Let us have a look at that again, the same graph, let us have a look at that again.

(Refer Slide Time: 42:05)

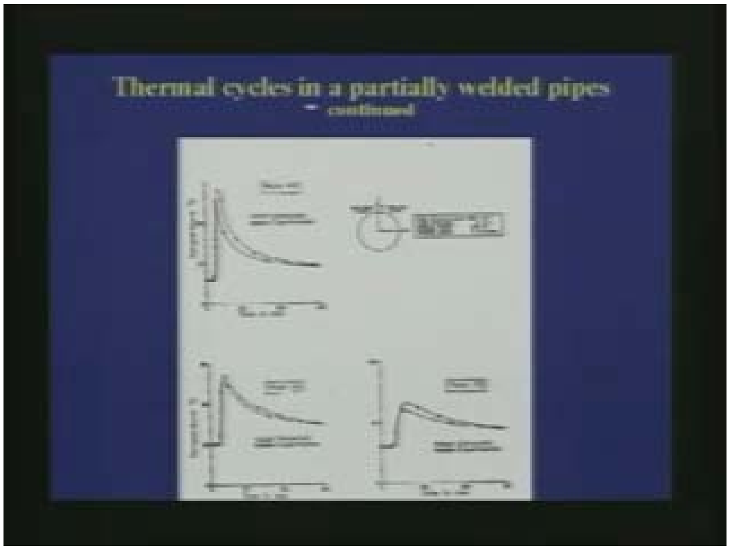


Thermal Cycle at different locations

You will see that for many of these things or many of the positions, there is a good agreement between the finite element analysis and the experimental result. Is there any question, is it clear? All the graphs indicate or each graph indicates different positions and there is a time verses temperature. The x-axis is time, y-axis is the temperature and how the temperature varies with respect to time at different positions of the tube as it is welded, is what you see in this graph.

Let us look at the next slide.

(Refer Slide Time: 42:46)

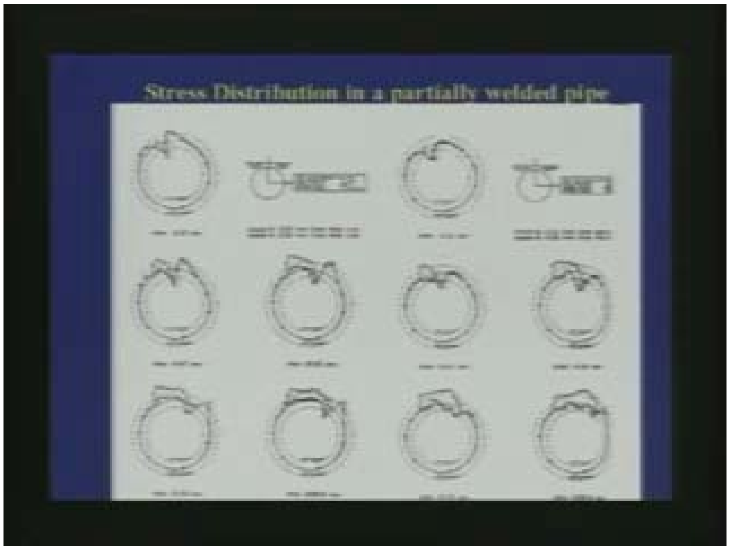


Thermal cycles in a partially welded pipes

– continued

There are two types of analysis that was done. Let us not go into the details of it. This is for one type of analysis, how things vary with respect to time. We see that again they are very close and next one, next slide.

(Refer Slide Time: 43:04)



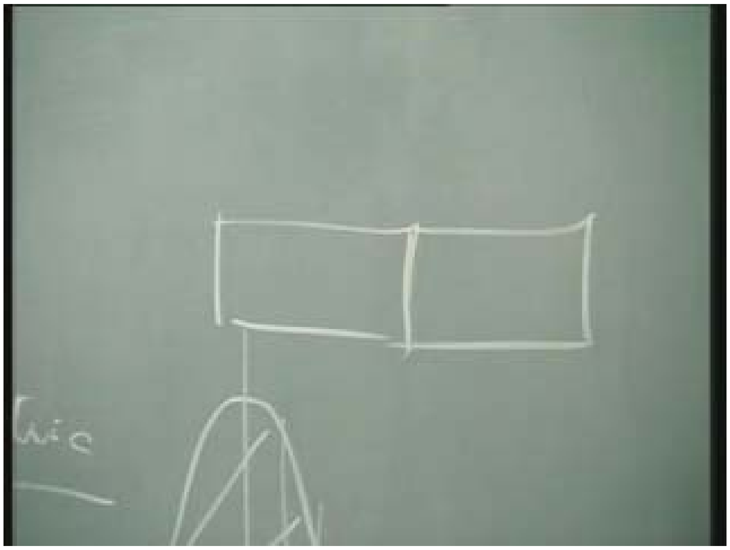
Stress Distribution in a partially welded pipe

This slide tells you the stress distribution, how the stress distribution is obtained during analysis or when we do the analysis for this kind of coupled problems. Is that clear? I mean, I am not going to discuss this result in more detail, because we are not interested in this pipe problem. But what we are interested is that this kind of problems can be done. How we do it is what is of interest to us. That is what I had discussed now. If there is any question on whatever we have done, let us see, let me answer. Is there any question?

I had not discussed deliberately two or three things. I thought there will be some questions on it. What are the types of boundary conditions that you apply? I have not discussed this. In this pipe, you can assume that there is symmetry at the centre. It is possible, but it also depends upon the way the welding takes place. If it is symmetrically welded, then there is no problem. You can assume that there is symmetry about the centre. There is heat transfer from the sides and note that you have to give boundary conditions also for the mechanics problem. This boundary conditions depend upon the fixture. If you are going to clamp it at some place, then you have to say that the displacements at those positions are zero. If you are leaving it free in certain positions during welding, then that has to be left free and so on. Please note that the boundary conditions of the problem for mechanics is also very important and that also has to be given beforehand. Is that clear?

I am going to give you a small exercise. I am not going to discuss this; tomorrow I am going to discuss it. For this pipe problem, temperature is no big problem. You know, it is very straight forward. What is the type of boundary condition that you will give in order to do the problem?

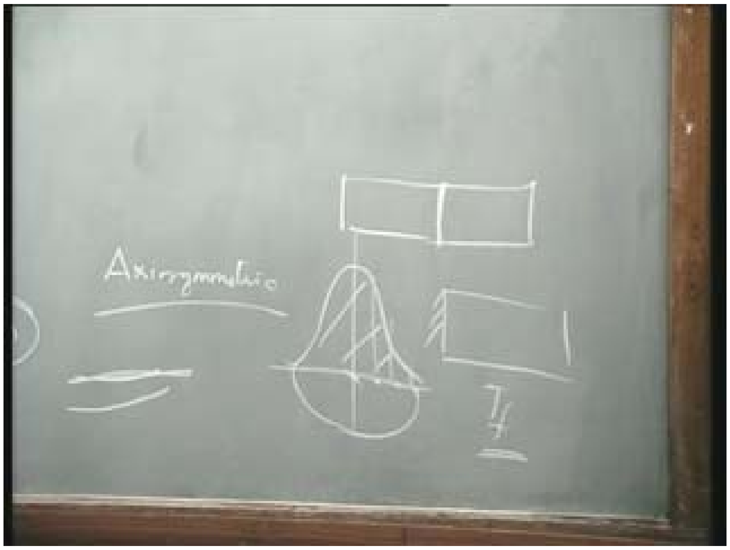
(Refer Slide Time: 45:40)



I have two parts of the pipe which is now welded? There are two exercises that have been done. One is complete welding and other is partial welding, just to check some results. What is the boundary condition that you will give or how will you do this analysis for the mechanics problem? What I have essentially done is, you can do it; what is the type of analysis you have done and what is the type of element that you have to choose, all those things I had already discussed. But I have not given the boundary conditions. I would like you to think about it and tell what is the type of boundary condition that you can give?

Let me see whether someone can throw light, right now. Is it possible? We have another two minutes. Is it possible for someone, how do we analyze this problem? Can someone throw light on this? What are the thermal boundary conditions? You want to first discuss the thermal boundary conditions. Yes, the heat input is along this line and it is possible at this stage. Can you assume symmetry? So it is enough if you look at one pipe? Let us say that we are just welding it around straight away and so it is possible that we assume symmetry. We will say that we will take this pipe.

(Refer Slide Time: 47:29)



Now, what are the assumptions that we make in this pipe for thermal boundary? You want to discuss the thermal boundary condition, so, what are the things that we make? We assume that there is convection from the surface, now, convection from the surface, both inside as well as outside. The inside, we know that the heat transfer will not be as sufficient as outside. Why? Though both of them are natural convection, the H value of inside and H value of outside can be different or will be different. It is important that we distinguish between inside the pipe and outside the pipe; inside the pipe and outside the pipe or two surfaces of the shell element inside surface and outside surface; that we have to distinguish.

There is one, convection heat transfer coefficient. The other is we can give a radiation; radiation heat transfer can be given. The radiation depends upon what? T power 4, so, you have to give Stefan–Boltzmann constant and so on, and emissivity and so emissivity becomes important. You should have some data on radiation heat transfer. Whatever you have studied for example in your heat transfer course on emissivity and other things can be used in order to determine this epsilon bar as we call it. So, convection heat transfer from inside and outside and if necessary radiation heat transfer condition as well, then what else do we apply?

There is no heat transfer in that region. We say that initially both of them are touching and that there is no heat transfer in that region, because we have a symmetry about this line, symmetry about this line, so, no heat transfer in this position. We do not anywhere mention the temperature as heat transfer boundary condition, but what we mention is the temperature of the fluid outside $T_f$, outside.

We will discuss this again in the next class along with the other boundary conditions for mechanics as well. We will meet in the next class to discuss this, but meanwhile I would like you to look at axi-symmetric element also. We will discuss this as well as the elasto-plastic algorithm that we would be interested in to solve such problems.