Finite Element Method

Prof. C.S. Upadhyay

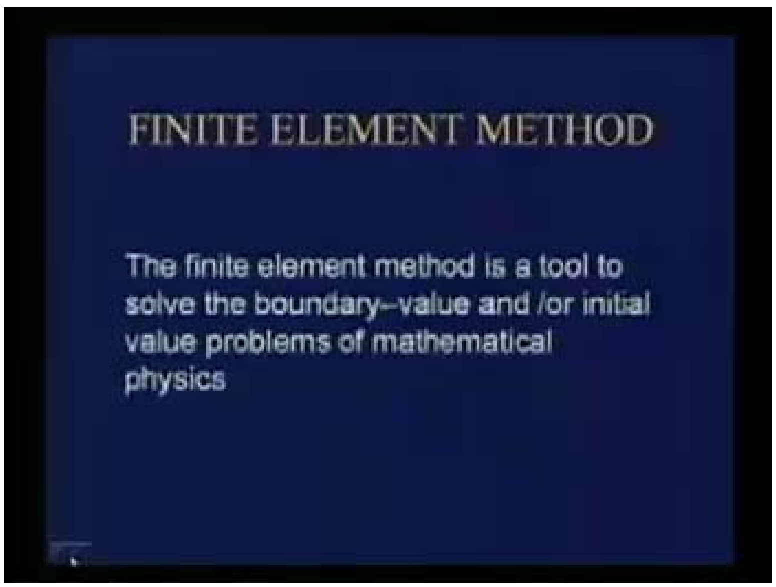
Department of Mechanical Engineering

Indian Institute of Technology, Kanpur

Module - 1 Lecture – 1

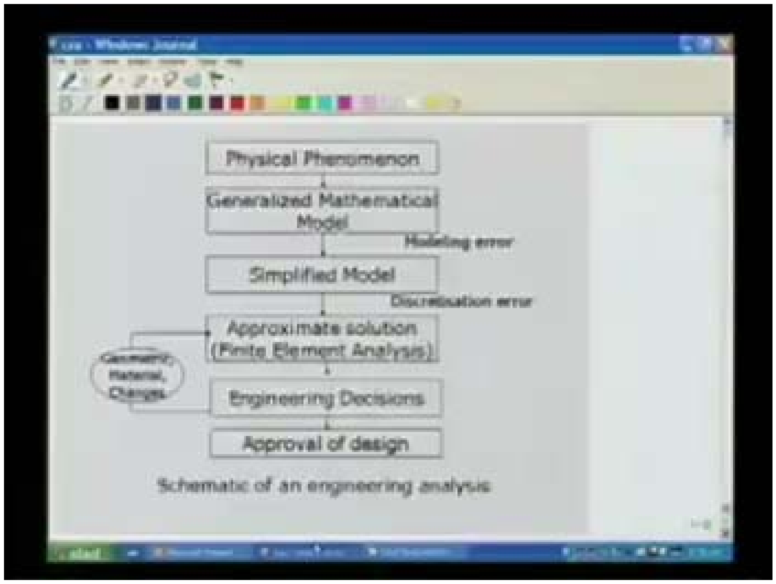
This course is called the Finite Element Method. In this course, we are going to develop a mathematical tool with which we are going to study boundary-value problems and/or initial value problems of mathematical physics. For us engineers, this tool has to be used in all stages of an engineering analysis.

(Refer Slide Time: 00:25)



The physical problems that we face in everyday engineering are many. For example, let us take a physical phenomenon which can be the bending of this pen. This pen is fixed at one end; I am holding it with my fingers and the other end I am applying a load.

(Refer Slide Time: 00:44)

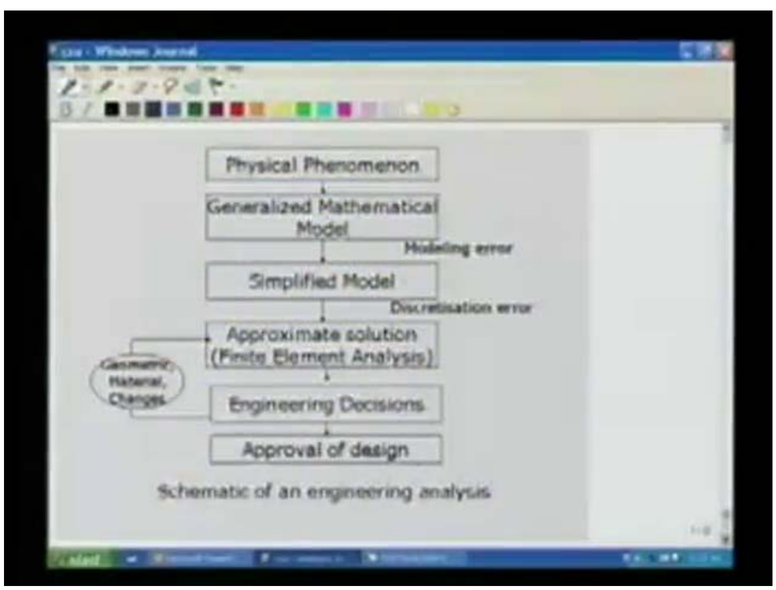


This is a three dimensional object subjected to a loading which I am showing here and under the action of this loading this pen bends. This physical phenomenon is generally modeled using a generalized mathematical model. The generalized mathematical model will incorporate all the essential features of the physical state of the system. Now in principle and in practice, one would not like to solve the most general mathematical model because the problem then becomes almost intractable. So one would like to simplify the problem and pose a so-called simplified mathematical model (Refer Slide Time: 01:47). As soon as we go from the generalized to the simplified mathematical model, we commit something called a 'Modeling Error'. This modeling error should be sufficiently small so that the physics that we are after is reasonably accurately predicted by our simplified model.

Now for the simplified model we have to obtain a solution. Unfortunately, in engineering practice… again let us look at this pen. This is quite a complicated geometry and we cannot obtain the exact solution to this simplified problem in the most general case. So what we would like to do is obtain an approximate solution. A tool to obtain the approximate solution is Finite Element analysis. Now once I have obtained an approximate solution then the analyst has to decide whether that approximate solution is good enough or not. For this, his engineering decisions should be made. If he thinks that the approximate solution is good then he accepts that and then starts post-processing the solution 'i' obtaining the response quantities of interest.

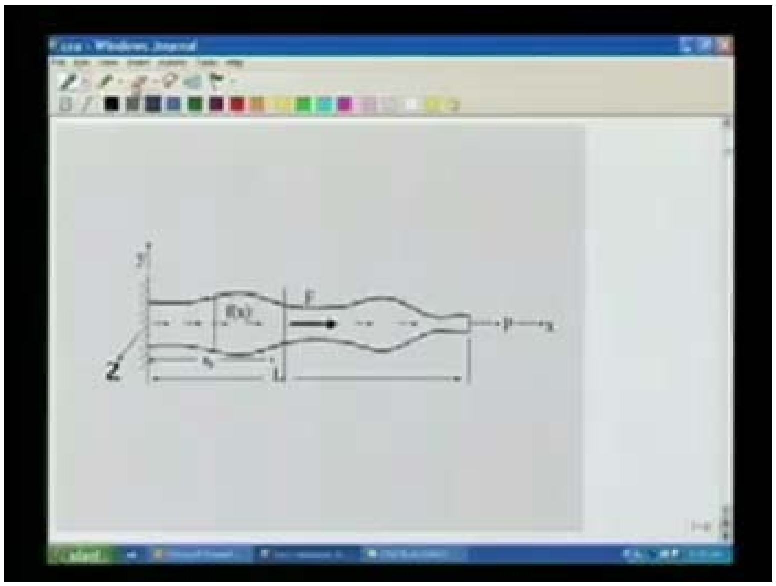
For example, for this beam that is my pen (Refer Slide Time: 03:15), I may be interested in the tip deflection, deflection at this point here, or I may be interested in knowing what the stress distribution in the beam is so that I can decide whether the pen is going to fail at any point of interest. For example, in this case the pen may fail at the root. Once I have obtained these response quantities of interest then as an analyst or a designer I am going to make certain engineering decisions; whether my design is good or I have to make certain modifications. For example, if I see that this pen under the action of this load hardly does anything, that it is absolutely safe, then I may decide to remove material from the pen. That is, I will make geometric changes to the pen. Here I am going to change the geometry or make the pen thinner. In certain cases for example, in aircraft engineering applications, I would like to keep the geometry fixed; for example the wing of an aircraft. Wing is given a certain geometrical shape so that it can develop the required amount of lift. In that case, I cannot change the shape and I find that under the action of aerodynamic loads, which can be represented again like a beam under the action of transverse loads, the wing bends too much; it is unsafe. In that case, I will have to change the materials that I have used in the wing or I will have to in force the material inside the wing. That is I have to add the stiffness inside the wing. After I have done all those things, I will look at my final design and decide as a designer and as an analyst whether this design is acceptable.

(Refer Slide Time: 05:12)



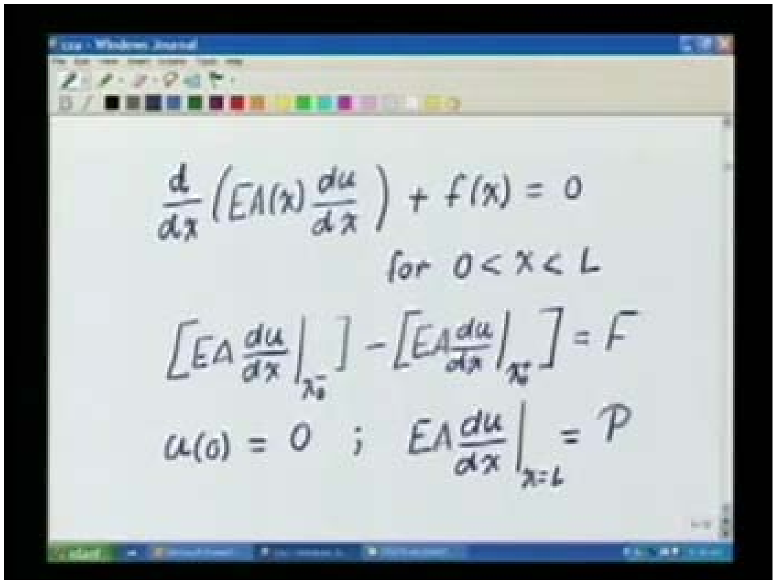
This is an iterative process and the solution obtained using a Finite Element analysis at every step of the design process gives us the response quantities that we are looking for so that I can make my engineering decisions. Once this job is done then I approve the design and send it for prototype fabrication. After the prototype is made, laboratory tests have to be done to see whether what I have done is good or not. If it is not good then I have to go back to the drawing table. So let us now look at a few typical simplified mathematical problems that arise in every day engineering practice.

(Refer Slide Time: 06:03)



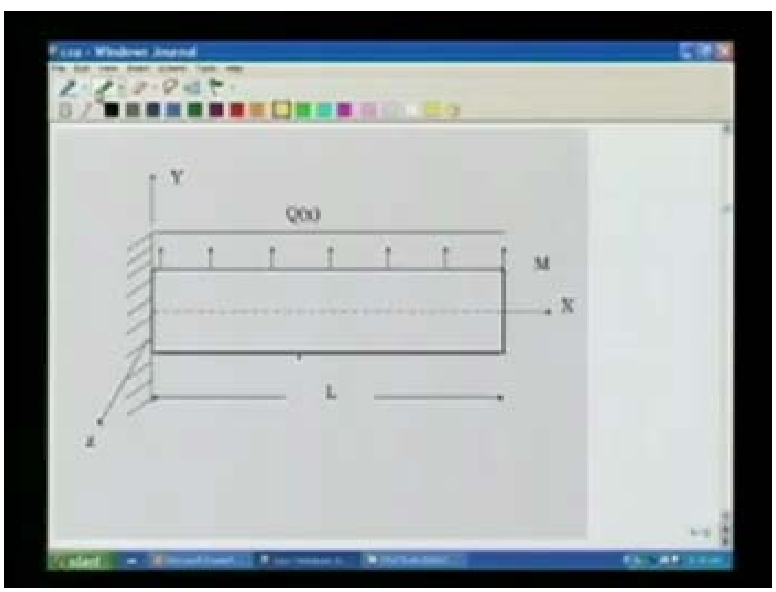
For example, we have this figure here which represents a bar (slender member) with variable cross section, subjected to an actual force which is a distributed body force f(x), a point load at point x(o) of size f and end load e and it is constrained at the point x equal to 0. It is constrained here (Refer Slide Time: 06:36). Under the action of these loads, I would like to now obtain the deformed shape of the bar. I would like to know what happens to this bar. So let us now give the differential equation which corresponds to the state of the bar. Under the action of the load, the bar is going to deform as such: d by dx of EA (x) du by dx plus f (x) is equal to 0 for 0 less then x less than l. That is for every point inside the bar this differential equation has to be satisfied. Okay that is not all when we are talking of a boundary-value problem. We also have to talk about the point x_0 where I have applied a point load. At the point x_0 what happens is there is a jump in the axial force so [EA du/dx at the point x_0 minus] minus [EA du/ dx at the point x_0 plus] is equal to F.

(Refer Slide Time: 08:51)



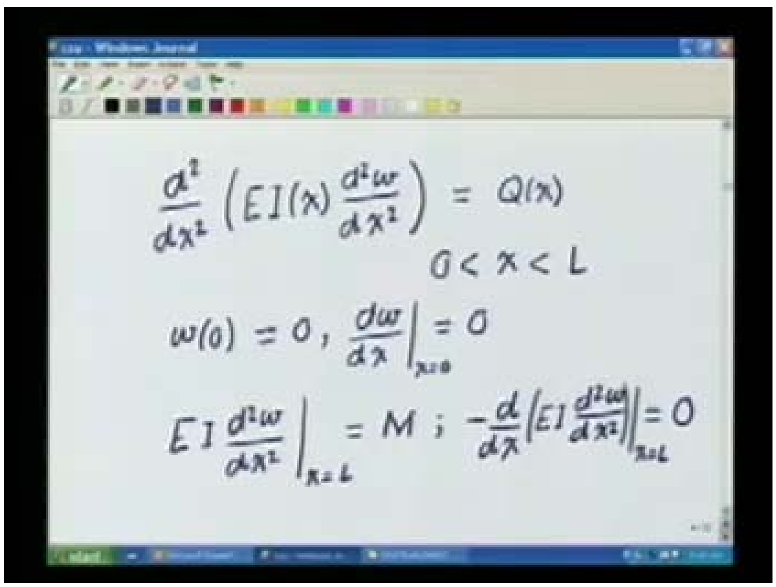
This also has to be satisfied. Further, at the end 0, x equal to 0, I will have my displacement equal to 0 and at the end L, I will have force EA du/dx at end x equal to L is equal to P. This is the complete boundary-value problem corresponding to the bar. Look here, we have a variable cross section. So in this case if I try to solve this problem I cannot obtain the exact solution in my hand. I will have to resort to an approximate solution which will be provided by the Finite Element method.

(Refer Slide Time: 09:15)



Let us now look at another boundary-value problem of interest which corresponds to the pen that I was holding earlier. This is a cantilevered beam subject to a distributed load Q(X) (Refer Slide Time: 09:30); it is a transverse load and at the end bending movement M given by this arrow. This bending movement is acting about Z-axis and this beam under the action of these loads is going to bend. So the deformation of the beam will be given by a corresponding differential equation which is d2 dx squared EI(x) where EI(x) is nothing but the flexural rigidity of the beam at the point x; d2 w dx squared where w is the transverse deformation of the beam, this is equal to Q(x) that is applied. This is true for all points lying between 0 and L.

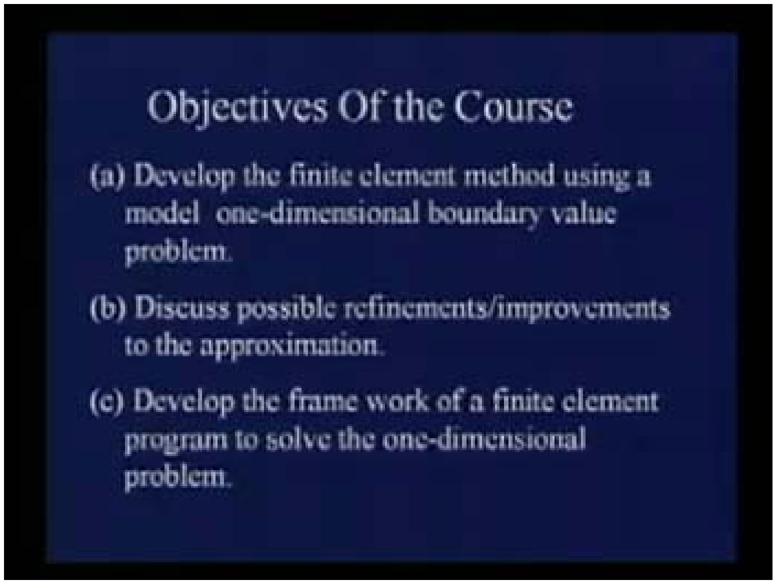
(Refer Slide Time: 11:52)



Now let us look at the boundary conditions. In this case boundary conditions are w at 0 because I have fixed the beam with 0 is equal to 0. Further, because it is a cantilever beam, the rotation at the point 0 is also set to 0 (Refer Slide Time: 11:00). This is the boundary condition at the point x equal to 0. Similarly at the point x equals to L we have an applied bending movement so I will have EI d square w d x squared at the point x equal to L is equal to M. There is no shear force at the end which is free end as far as shear force is concerned. I will have – d dx of EI d2 w d x squared at x equal to L is equal to 0.

Under the action of the given distributed load with the given constraint at the point x equal to 0 and the end bending movement and 0 shear force this beam deflects. This is another simplified problem that we will be interested in. Now we have posed certain simplified boundary-value problems that we will be looking at during the course. Let us also see what the objectives of this course are.

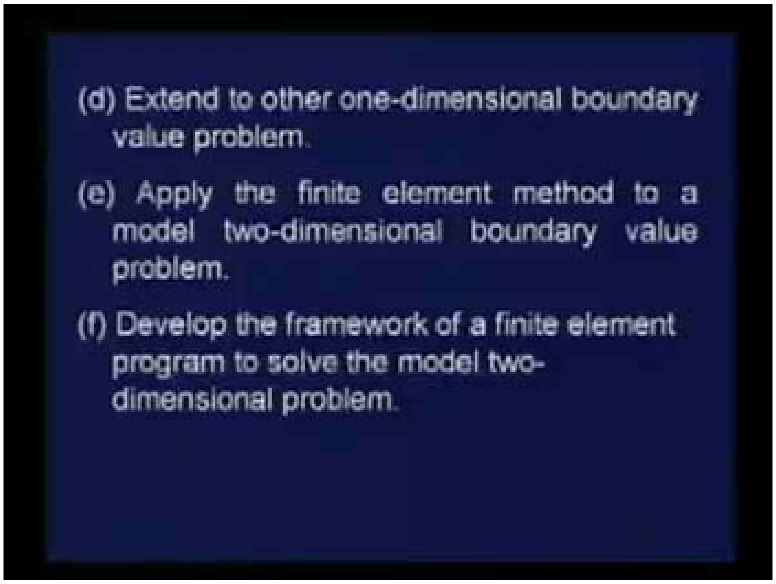
(Refer Slide Time: 12:30)



As far as the objectives are concerned, we are going to go in a step-by-step manner starting with the development of Finite Element method using a one dimensional model problem. For the one-dimensional model problem, we are going to take the variable cross section bar that we have drawn earlier. For this model problem, we are going to pose the Finite Element method or formulation, obtain a solution and we will show how we can refine the solution; that is how we can improve the quality of the approximate solution that we have obtained.

Once you have done that we are going to develop the frame work of a one dimensional Finite Element program which can be programmed by the students and we can actually get the deformation stresses for a bar subjected to various kinds of action loads. This is what we are going to do as the third part of the course. Then we are going to extend whatever we have done to other one dimensional model value problem. For example, we have already drawn the beam and we will look at the beam and we can look at heat conduction problem in a bar or we can look at one-dimensional fluid flow etc. Many problems in that way can be tackled and we will show that you do not have to do significantly different things in order to tackle these problems.

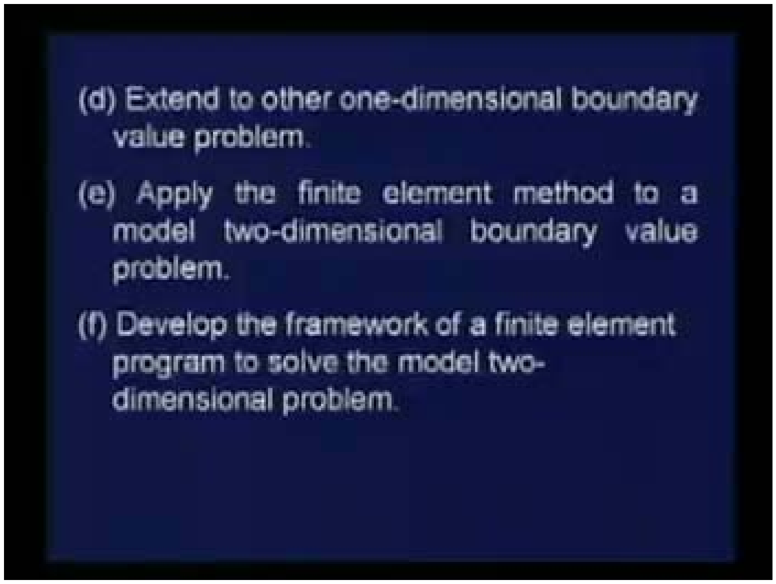
(Refer Slide Time: 13:41)



Once we have consolidated our fundamental knowledge of what the Finite Element method and Finite Element analysis is, we will go to the two dimensional boundary-value problems, where we start with the simplified steady state heat conduction problem for which we will develop the full framework of the Finite Element formulation. From there we will develop a simple, in this case we will not be too detail in what we do, we will develop a simple two dimensional Finite Element program to solve the two dimensional problems. Once we have developed the program, we will extend our formulation to cover problems of planar elasticity. For example, plane strain or the plane stress problem for this we will also outline how to develop the Finite Element program. Once we have finished the two-dimensional problems then we look at certain other special problems which are very important to an engineer.

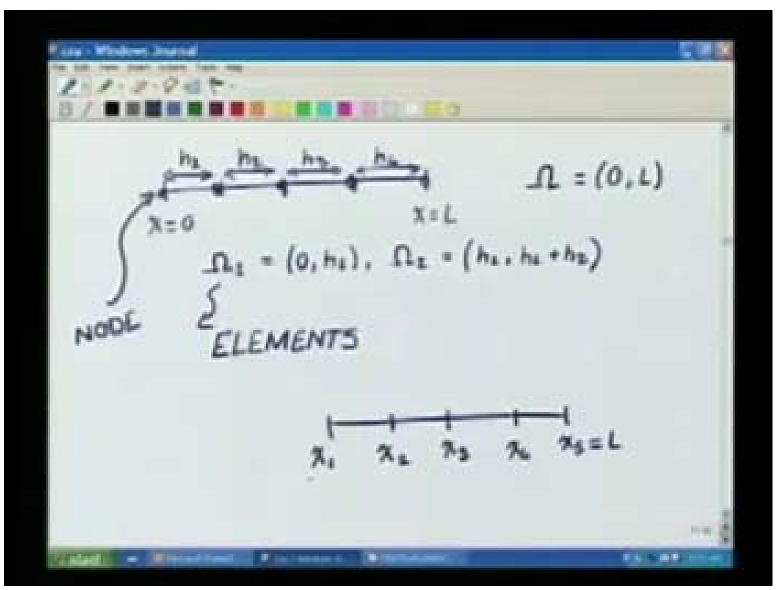
For example, we may be interested in a free vibration problem of a continuous system. Linear vibration for example, is nothing but an Eigen value problem. Or we will be interested in the dynamic response of the bar or a beam or a plate under the action of time varying loads. Then we will have to develop the formulation for a time dependent problem. Further we may be interested in problems which are non-linear in nature. For example, geometric non-linearity can be incorporated in our formulation. So this is essentially what we are going to cover in this course. Once we have outlined the objectives of the course, let us look at the basic steps which are involved in a Finite Element formulation.

(Refer Slide Time: 15:51)



What are the basic steps? We will highlight the basic steps using the one-dimensional problem. Let us take the bar; in that case the bar is a domain extending from the point x equals to 0 to the point x equal to L.

(Refer Slide Time: 19:06)

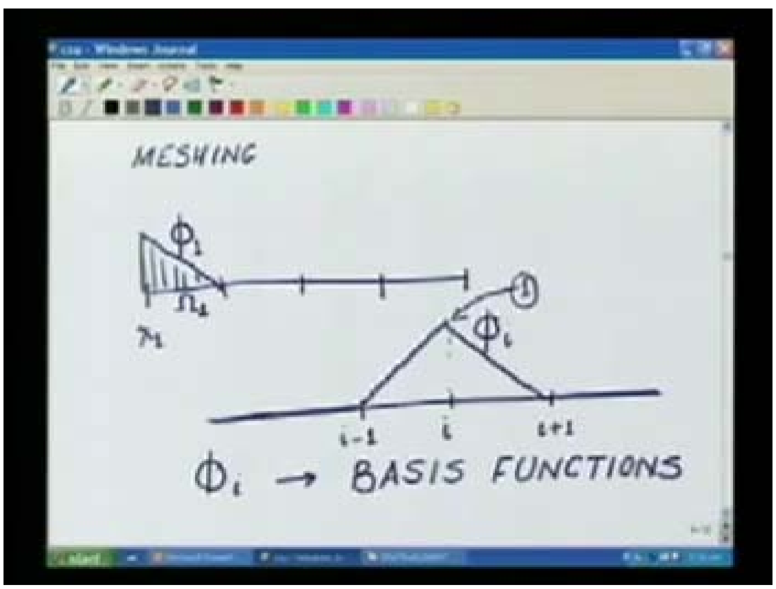


This bar we are going to partition in to smaller pieces by putting these extra points in the domain. These will be I can put them with size h_1 here, h_2, h_3 and h_4 (Refer Slide Time: 17:00).

When we partition the domain which is an interval of size 0 to L the domain we will call by omega which is equal to the interval 0 to L. This domain will be partitioned into smaller sub domains which are omega_1 which is equal to 0 to h_1, omega_2 which is equal to h_1 to h_1 plus h_2 and so on. The smaller sub domains are given a name in the Finite Element analysis. These are called elements. In this case we have four smaller sub domains which together form the full domain. The four elements in this case are omega_1, omega_2, omega_3 and omega_4.

The points which basically form the boundaries of each of these domains are called the Nodes. So this point will be a node. In this case if you see there are four elements and we have five nodes: 1, 2, 3, 4, and 5. We can draw the figure here (Refer Slide Time: 18:50) and have the node corresponding to point x_1, node corresponding to point x_2, node corresponding to point x_3, node corresponding point x_4 and node corresponding to point x_5 which for us here is L and x_1 is 0. Once we have the elements and the nodes, the next step is to now define functions over these elements. By the way, the process of forming the elements and nodes is called Meshing. So that process was meshing. That was the first part of any Finite Element analysis.

(Refer Slide Time: 21:50)

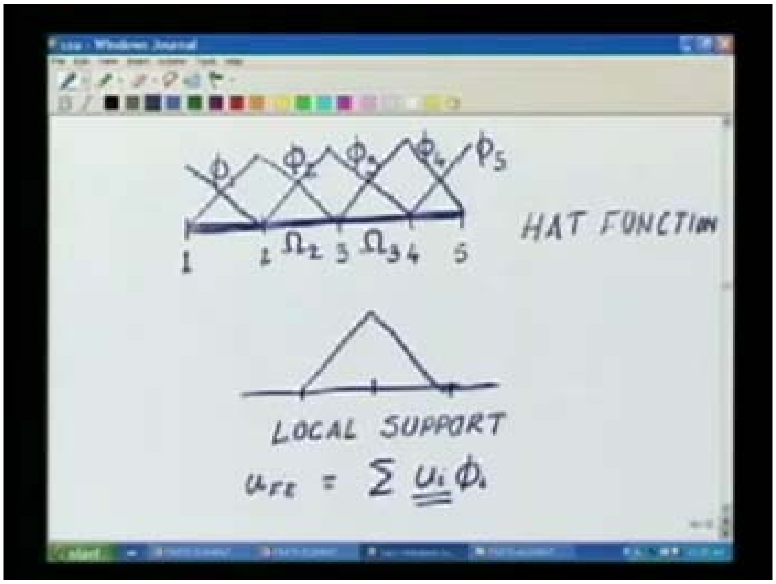


Once I made the mesh, over the nodes I am going to define the functions. How do I define these functions? These functions will be such that; let us take the first node, I will call this function phi_1 which is linear in the element 1 which is linear in omega1 and it has a value 1 at the node 1 and 0 at all other nodes. So what will the function look like? It will look like this (Refer Slide Time: 20:11). I can draw the area under the curve so this is my function phi. What I am going to do is I am going to construct the piecewise polynomial functions which are continuous in the full domain and which vanish at certain nodes and take a value 1 at a particular node.

Let us take any generic node i and its two neighboring nodes i minus 1 and i plus 1. If I now define what is phi corresponding to the node i, then phi will be this function such that it has the value 1 at point i and at point i minus 1 and i plus 1 it is 0 and I am going to extend it to the full domain with the value 0. If I have elements here, this function will have value 0 everywhere else. These functions phi_i corresponding to the nodes, in this case if I go back to my mesh formed, there will be five such functions phi_i. These functions phi_i (Refer Slide Time: 21:36) are called Basis Functions.

Once I have the basis functions, in terms of the basis functions I am going to define my Finite Element solution. The u Finite Element (x) is equal to the sum over I is equal to u_i is equal to 1 to 5 in our case u_i into phi_ix.

(Refer Slide Time: 25:48)

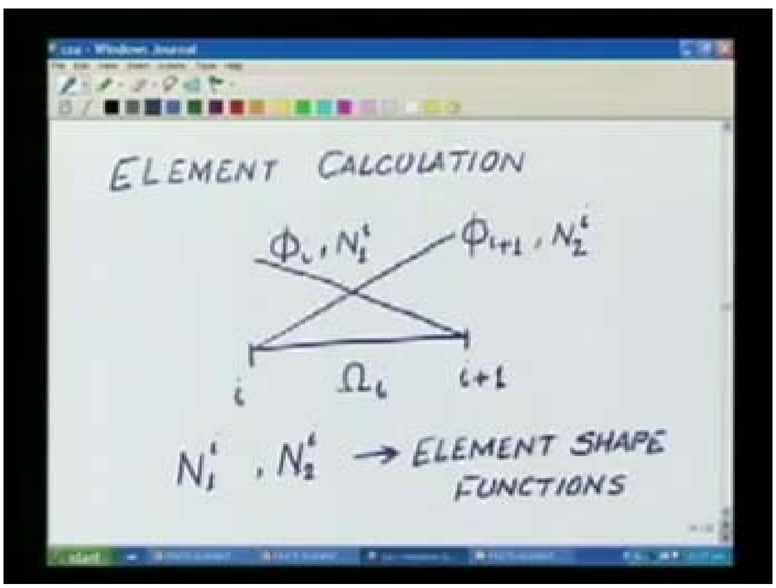


These u_is are the unknown coefficients which we have to obtain using the Finite Element method. Once I have obtained the coefficients u_i then I have the full Finite Element solution for any point in the domain 0 to i.

Let us now take our problem again and draw the various global functions phi_i that we have. This is node 1, node 2, node 3, node 4, node 5 (Refer Slide Time: 23:02). This is phi_1 which extends to 0 everywhere else, this will be phi_2 which will extend to 0 everywhere else, similarly phi_3 will be 0 here, 1 here, phi_4 will be 0 everywhere up to node 3. Beyond node 3 that is in element three, it is non-zero linear. At node 4 it is size 1, at node 5 it comes down to 0. Similarly the last one will be phi_5. In this, phi_5 is extended to 0 beyond node 4. So these are our five functions. These are also called HAT functions.

Now if you see something about the HAT functions what is happening is, each of these functions is non-zero only in certain elements. If you take for example, phi_3 it is non-zero only in element $omega_2$ and element $omega_3$ while it is 0 everywhere else. Such functions which are 0 in most of the domain and non-zero in only a small part of the domain are called functions with Local Support. This is a very important feature of the Finite Element method. In that, for the approximation, we are using functions which are only defined in a small part of the domain and by piecing up by these functions we construct the approximation over the full domain. Okay now let us see how we are going to obtain the coefficients of the Finite Element solutions. If you remember that U_{FE} was summation of u_i phi_i. So we have to still find a way by which we can obtain the u_i. That brings us to the next step of the Finite Element method which is called Element Calculation. This part (Refer Slide Time: 26:12) is Element Calculation.

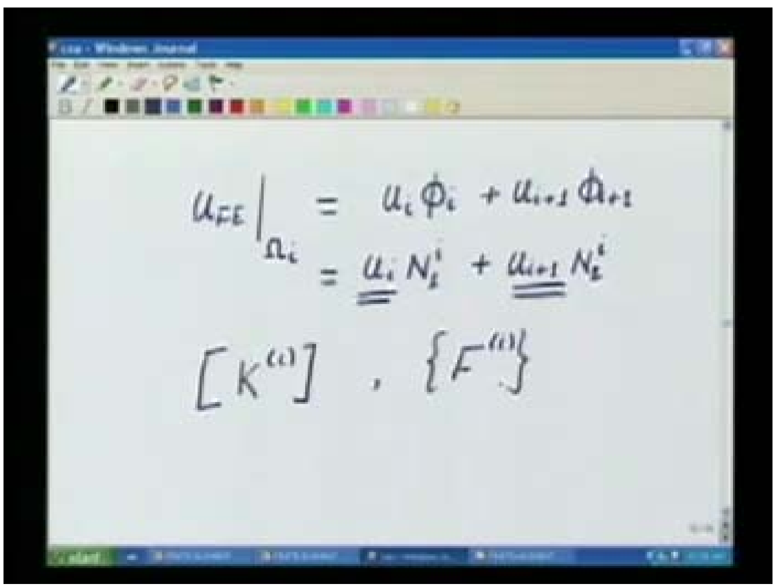
(Refer Slide Time: 27:28)



What is the idea? If you take an element omega$_i$, it will have nodes i and i plus 1. In this element which of the phi$_i$s are going to be non-zero? The phi$_i$s which are going to be non-zero are going to be phi$_i$ and phi$_i$ plus 1. These phi$_i$s in the element are given a name. The name is n_1 of the element i and n_2 of the element i. These n_1 of the element i and n_2 of the element i are called Element Shape Functions.

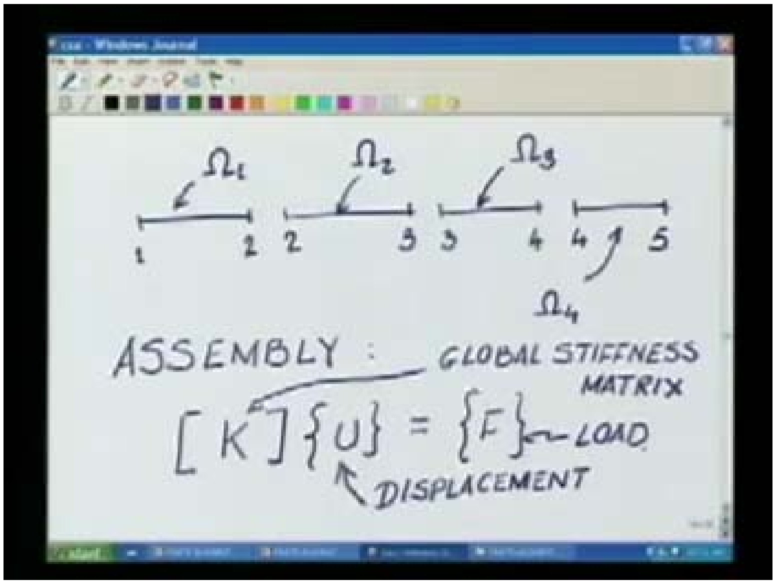
We have to understand that element shape functions are nothing but the part of global basis function phi$_i$s which are non-zero in the element i. This way we have the shape function for each of these elements. What we are going to do next is if I look at the Finite Element solutions, in this element i, u Finite Element in element omega$_i$ is equal to u_i phi$_i$ plus u_i plus 1 phi$_i$ plus 1 which can be rewritten in the terms of the element shape functions as u_i N_1 of i plus u_i plus 1 N_2 of i. As we will see later on the equations from the element corresponding to these unknown coefficient u_i and u_i plus 1 can be obtained through the Finite Element formulations and what we will end up getting at the element level are matrix K_i and so-called load vector F_i.

(Refer Slide Time: 29:00)



For each element, this matrix will be in terms of the two unknown u_i and u_i plus 1 and F_i will also correspond to these u_i and u_i plus 1. So for each element I am going to obtain this set: for example, if I take this problem two equations or contributions of two equations corresponding to the u_i and u_i plus 1. So the next job is the assembly of this equation.

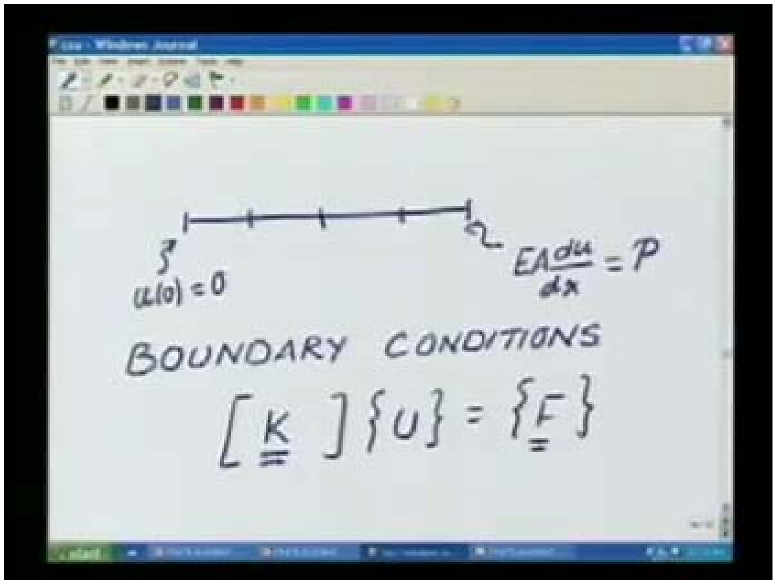
(Refer Slide Time: 31:47)



Once I obtain the element level equations for each of these elements, for the first element I will get the equation corresponding to u_1 and u_2, for the second element I will get the equation corresponding to u_2 and u_3, for the third element I will get the equation corresponding to u_3 and u_4 and for the fourth element I will get the equation corresponding to u_4 and u_5e. Then the question is how we bring these equations together to give us five equations in terms of the five unknown u_1 to u_5. That process is called Assembly.

Here we will add up all the element equations to obtain so-called global matrix K such that K into u where u is vector of size 5: it will have the unknowns u_1, u_2, u_3, u_4 and u_5. This will be equal to vector on the right hand side called F. These things are also given names: K is called the global stiffness matrix, u is called the displacement vector and F is given the name of the load vector. This sounds very much like civil engineering where we have atros or the matrix structure or we can think of this like a spring mark system where K represents the stiffness, u represents the displacement of every marks and F is the vector of the applied forces at the various mass points. It equivalently becomes systems with stiffness K and subjected to an external load F. So once you have done the assembly, our job is not over. If you remember that nodes 1 and 5 have something special specified for them.

(Refer Slide Time: 33:54)

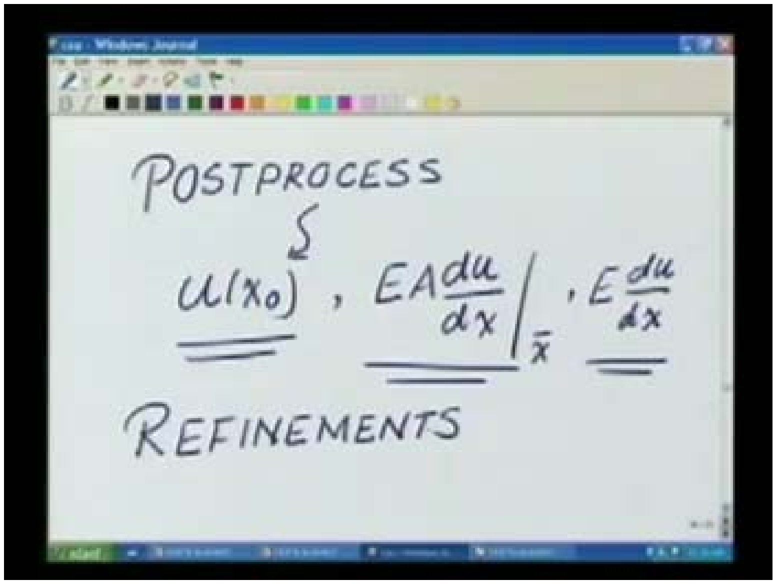


For example, at node 1 I may have the displacement given a particular value; it is given a value 0 and at node 5 I will have the force for example, is equal to the applied force P so this has to also be incorporated in to our equations.

What we do next is called application of boundary conditions. That is we enforce the conditions at the end x equal to 0 and at the end x equal to L for our problem that we have taken. Once we enforce the boundary conditions, it is going to show up in terms of modification of the stiffness matrix and the load vector F. Finally, we have the system of equations corresponding to the five unknowns that we have posed in this problem you can have ten unknown, n unknowns so I have an equation in terms of the n unknowns which we are going to solve and N by N system of equations. For this we have to use a solver such that we can obtain the vector u is equal to K inverse F.

So now what do we have? Now we have these coefficients u_i u_1 to for our case u_5 these are obtained. Once I have these coefficients then I know the Finite Element solution at every point in the domain. Once I know this then what do I do with it? I am going to do oppose processing of the solutions that is I am going to post process Finite Element solutions to obtain what? I may be interested in displacement at a given point.

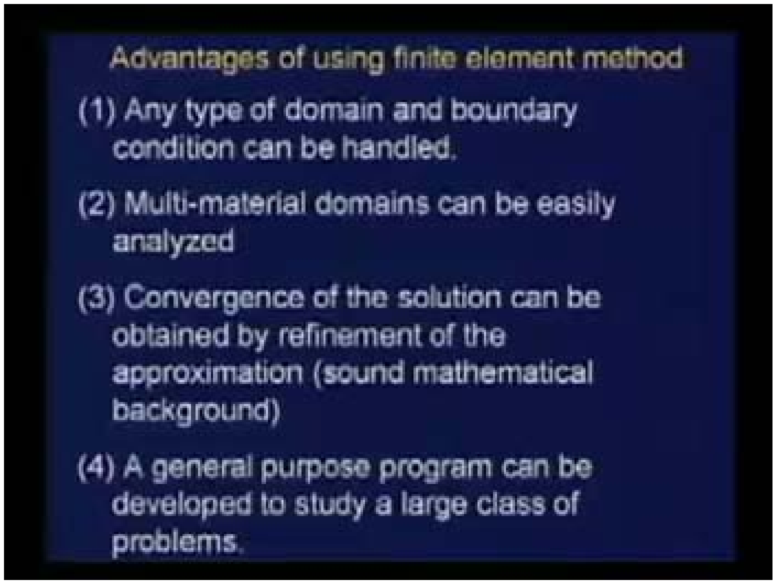
(Refer Slide Time: 36:14)



I may be interested in the actual force at some other points, okay, so on. Or I may be interested in a stress at a point; this depends on what is really required out of the analysis. So once I post process and obtain this solution quantities then I have to decide as an analyst whether what I have obtained is acceptable or not. If it is not acceptable then we have to do refinements in our approximation. After the refinements have been done, then we repeat the same process of obtaining the solutions, obtaining the coefficients u_i and post processing the solutions to get the response quantities of interest and we have to plot them.

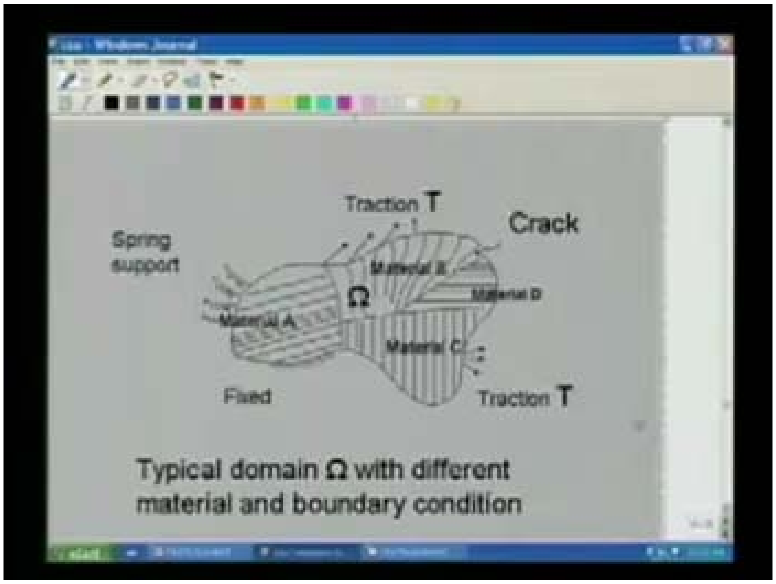
Plotting becomes very important part of any Finite Element analysis we do. The post-processing involves first extracting the quantities of interest and then plotting them in a way that is acceptable to the user. In most commercial codes now a days most of the money is being spent in post processing. This is essentially the various steps we are going to follow in any Finite Element analysis; be it for the analysis for the simple bar problem that we have taken or beam or the plate or a shell or anything that we are interested in. So why do we need to use a Finite Element method? There are so many methods which are available to obtain the approximate solutions to the boundary-value problems that we encounter in engineering or mathematical physics. The answer is that in the Finite Element method there are no constraints with respect to the domain boundary or the boundary conditions that can be handled.

(Refer Slide Time: 37:38)



This is a very important advantage that we get when we use the Finite Element method especially in two or three-dimensional problems. In one dimensional, pretty much all methods can give you the same efficiency but the two and three-dimensional problems it really helps to do the analysis in a very efficient way. Also, we can develop general-purpose codes which can handle any given boundary conditions or domain. So let us take this very complicated domain that I have drawn which resembles a crazy potato made of different materials.

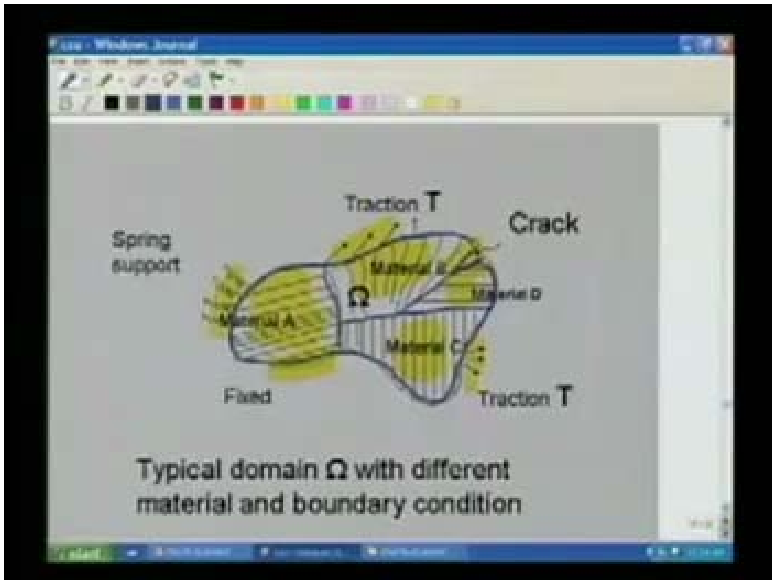
(Refer Slide Time: 38:20)



For example, I may have material A in this region, material B in this region, material C here, material D here. This domain also has a crack at this point, it has a crack here. Then traction boundary conditions are applied here which can be of any type which can be sheer or which can be normal and anything that you wish. And this potato is fixed at this end. While in this region, it is resting on an elastic support; that is it has spring boundary conditions.

If I have to do the analysis of this problem let us say that I would like to do a planar elasticity analysis, then most of the methods that are available to us will have a great amount of difficulty in doing a good job. With the Finite Element method, what we can do is we can very easily mesh the domain piece by piece that is each material domain or sub domain can be meshed separately. Then we bring all those together apply the boundary conditions as we have seen and also take care of the crack and will get a solution. We can also make modifications or improvements to the solution if and when desired by the analyst.

(Refer Slide Time: 40:00)

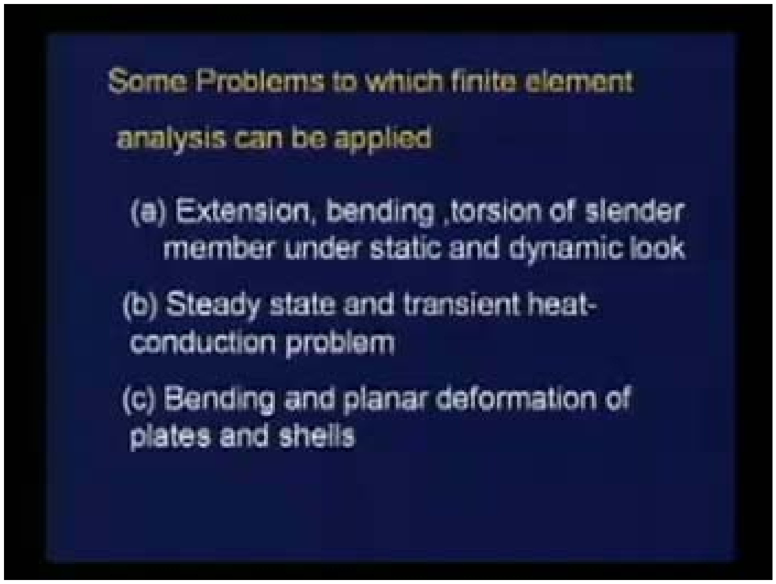


Let us come to the third point as I have shown in the figure that one analysis may not be enough to get a good representation of the response quantities of interest. For example, in the case of a crack domain I would like to have special kind of a mesh or special approximations in order to capture the response quantity accurately. Accurately means they have to be within acceptable engineering tolerances. So convergence of the solutions plays a very important role.

How do I improve my approximation? In terms of the Finite Element method is based on a very sound mathematical background using which we can decide how to improve or obtain our approximations. From commercial point of view, a great advantage of the Finite Element method is that I can write one general purpose code of a program to handle very large class of problems. That is my program does not change if my boundary conditions are changing or if my loading is changing or if my domain is changing, material is changing all these changes can be handled by one program.

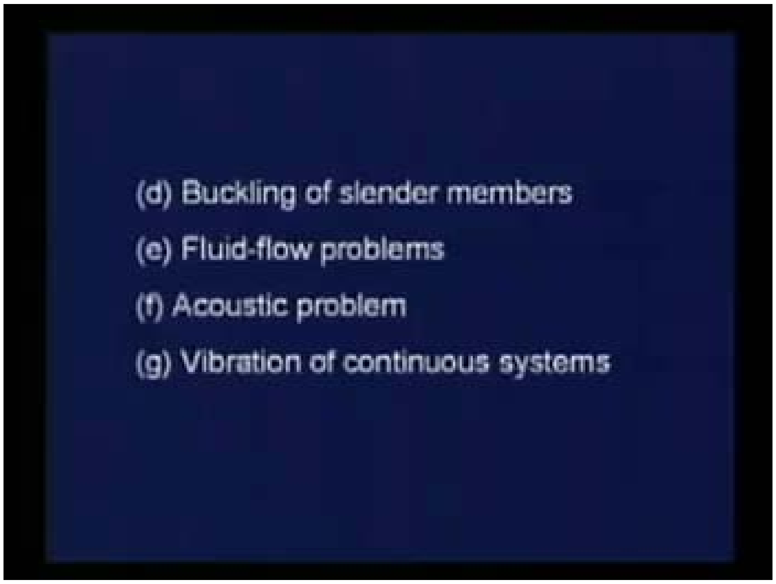
Given these advantages of the Finite Element method, let us now look at some typical engineering analysis problems that may be of interest to us which can be solved using the Finite Element method.

(Refer Slide Time: 41:36)



For example, we have already looked at some of them; i the extension of a bar, bending of a beam, torsion of slender members and the static or dynamic loads. Or I may be interested in study state or transient heat conduction problems. So if it is steady state we are interested in static problems, solving static problems when its transient heat conduction problems then we have to solve the dynamic problems. I may be interested in bending and planar deformation of plates and shells which are very important from the point of view of analysis of pressure vessels and aircraft wings and various other structural members that go either in automobile engineering, aircraft engineering, or even in civil engineering.

(Refer Slide Time: 42:32)



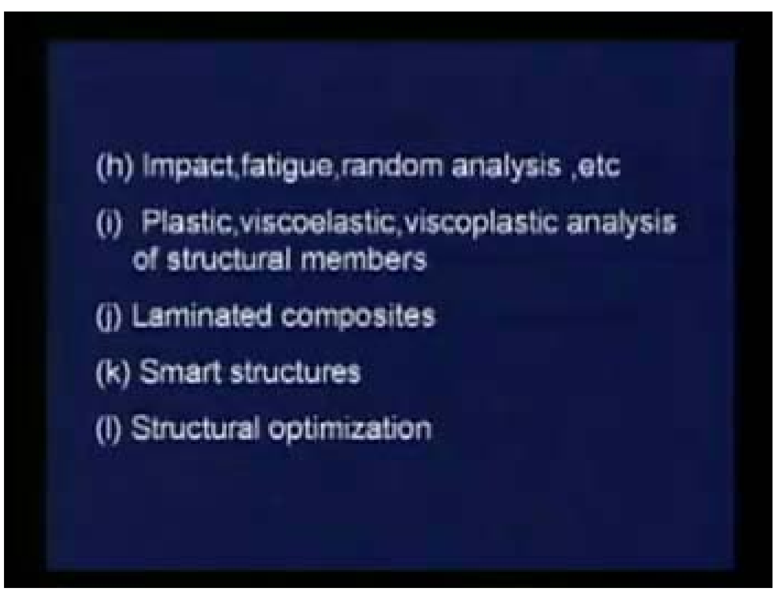
In many problems especially with respect to thin or slender members, there is issue of failure which has to be analyzed along with typical material failure which is Buckling'. So buckling of slender members is of great engineering importance from the point of view of designing of members. Buckling can be studied quite easily using the Finite Element method. We may be interested in analyzing flows; that is fluid flow problems. For example, flow faster if it has to be analyzed, the Finite Element method can be used. Or I may be interested in finding out what happens to cricket ball when it is delivered by a bowler. In those cases the approximate solutions or what we call as numerical simulations can be obtained using an appropriately designed by Finite Element analysis.

I may be interested in acoustic problems. For example, the sound generated by inkjet from a rocket, or the solar waves that are sent by at destroyer to hunt down the summary, all those things can be analyzed using the Finite Element method. I may be interested in vibration of continuous systems again from the flutter point of view or just dynamic analysis point of view or I may be interested in knowing what happens to a structure when it is impacted by the projectiles. This is very important in the current age of warfare where we have to design bunkers which can withstand sophisticated projectiles or bombs that are launched at them. I may be interested in fatigue of structures which is again very important. For example, air craft engineering because when the aircrafts take off, goes in to a flight regime, lands and repeats the same cycle. So all the sorting are essentially repeating cycle and under this repeated loads the aircraft tends to fatigue. So fatigue failures can be analyzed using the Finite Element method. We know that most of the materials that we have do not have one predictable material property because of manufacturing defects, where I am getting the raw material from, what is the raw material, so random analysis is often very useful. Effect of changing loads, changing geometries, everything can be incorporated for that also we can use Finite Element method and we have a special branch of Finite Element methods there called stochastic Finite Element methods.

Similarly I can look at more complicated mathematical models corresponding to the behavior of the structures under large loads which are typically given by plasticity base models of viscose elasticity base models or viscose plastic models. These can all be studied using the Finite Element method. Now we have new materials, new structures which are coming up; for example, laminated composite plates and shells are used heavily in most fast moving vehicles. For example, the light trains, the light cars, the race cars and most of the aircrafts. Our ALH which is advanced light helicopter that India has made and LCA (light compared aircraft) are mostly composite based vehicles, air vehicles.

So the analysis of laminated composite also becomes very important and most of these place where these things are designed and manufactured use Finite Element analysis to give them the preliminary design. Another field which is of great importance as an emerging science and technology field is smart structures. Analysis of smart structures can again be done in a very routine way using Finite Element method.

(Refer Slide Time: 47:00)



Structural optimization which is a key in designing better components light weight components, components which can carry the worst loads that the structure can bear without failing, that comes under the heading of structural optimization. All these problems can be analyzed using Finite Element method and it is being heavily used in engineering industry.

What we will do in the next lecture is we are going to develop the fundamentals of the Finite Element method and the Finite Element formulation using the simple bar problem that we have drawn earlier. Once we develop the basic framework and the structure then we will go ahead and look at the detail analysis using the Finite Element method.