Indian Institute of Science
Bangalore

NP-TEL
National Programme on
Technology Enhanced Learning

<u>Course Title</u>

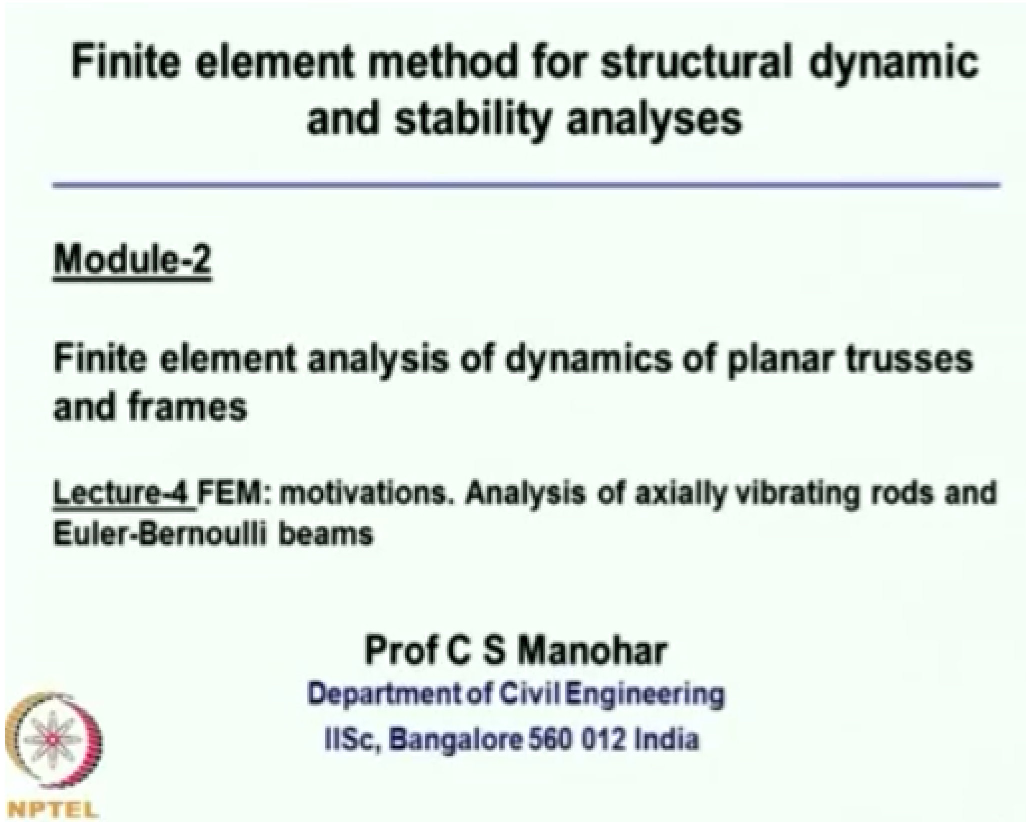
Finite element method for structural dynamic And stability analyses

Lecture – 04
FEM: motivations, Analyses of axially vibrating rods and Euler-Bernoulli beams

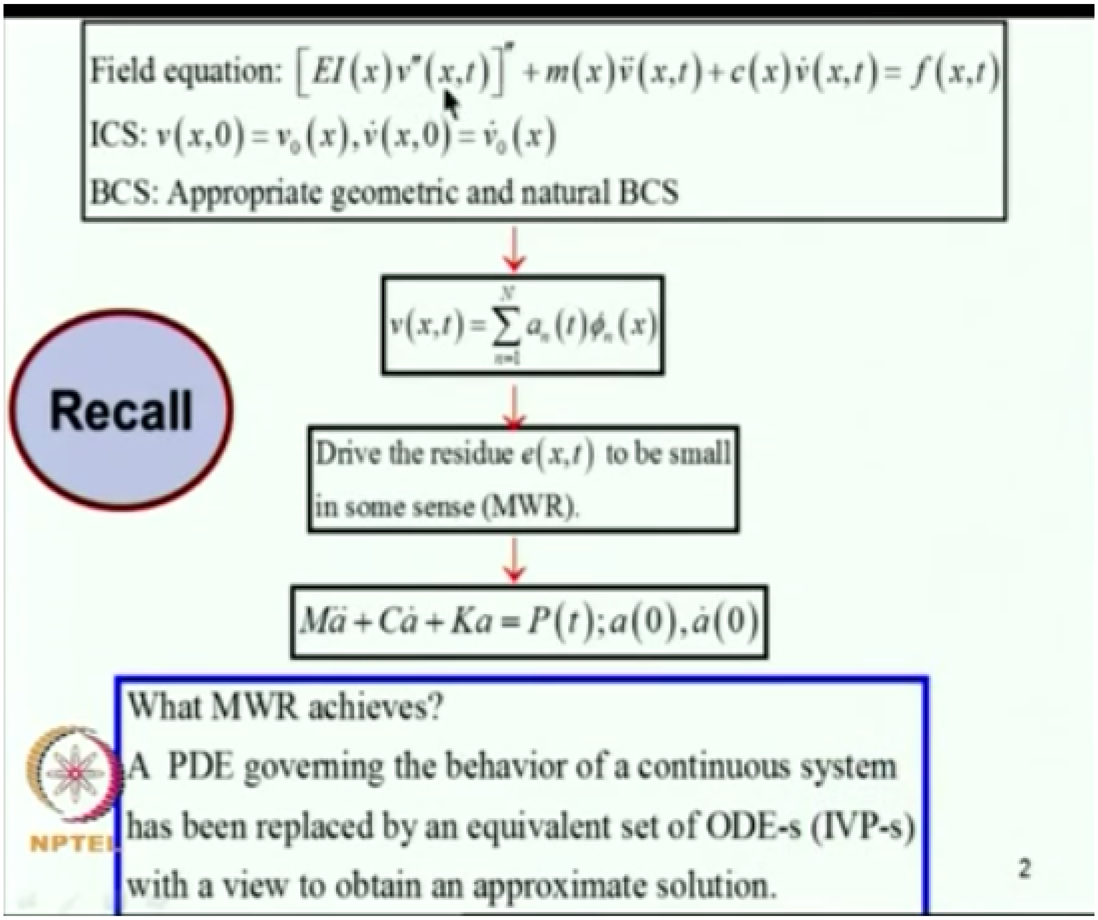
By
Prof. CS Manohar
Professor
Department of Civil Engineering

Indian Institute of Science,
Bangalore-560 012
India

We have been discussing approximate methods for analyzing structural dynamic problems as a build-up to the development of idea of finite element method.



So in today's lecture, we will start talking about finite element method what are the basic motivations and start analyzing some simple problems.



So let's first quickly recall what we have been doing so with reference to a problem of beam vibration, this is the governing equation for Euler-Bernoulli beam with specified initial conditions and appropriate geometric and natural boundary conditions. So the general format of finding approximate solution is to seek the unknown dependent variable in terms of a series like this where Phi N(x) are known as trial functions which are to be chosen by the analyst and AN(t) are the unknown generalized coordinates to be determined through an analysis.

So the basic idea here is to we will substitute this assumed solution into the governing equation and we obtain an error or a residue term, and the whole idea of developing approximate methods is to drive some measure of that residue to 0 or as small as possible and use that criteria to select this generalized coordinates, so this class of methods are known as Method of Weighted Residuals, so they lead to a discretized set of equations here A is N cross 1, vector MCK are all N cross N matrices, P(t) is the generalized force which is N cross 1, we also discussed how to obtain initial conditions A(0) and A dot(0) in terms of specified initial displacement profile and velocity profile.

Now basically what does method of weighted residuals achieve, a partial differential equation governing the system behavior is now replaced by a set of ordinary differential equations with a view to obtain an approximate solution, and that approximate solution is encapsulated in this representation.

$$v(x,t)=\sum_{n=1}^{N} a_n(t)\phi_n(x)$$

Least squares $\int_0^L \frac{\partial e(x,t)}{\partial a_n} e(x,t)\,dx = 0$ for $n = 1,2,\cdots,N$

Collocation $\int_0^L \delta(x-x_n)\,e(x,t)\,dx = 0$ for $n = 1,2,\cdots,N$

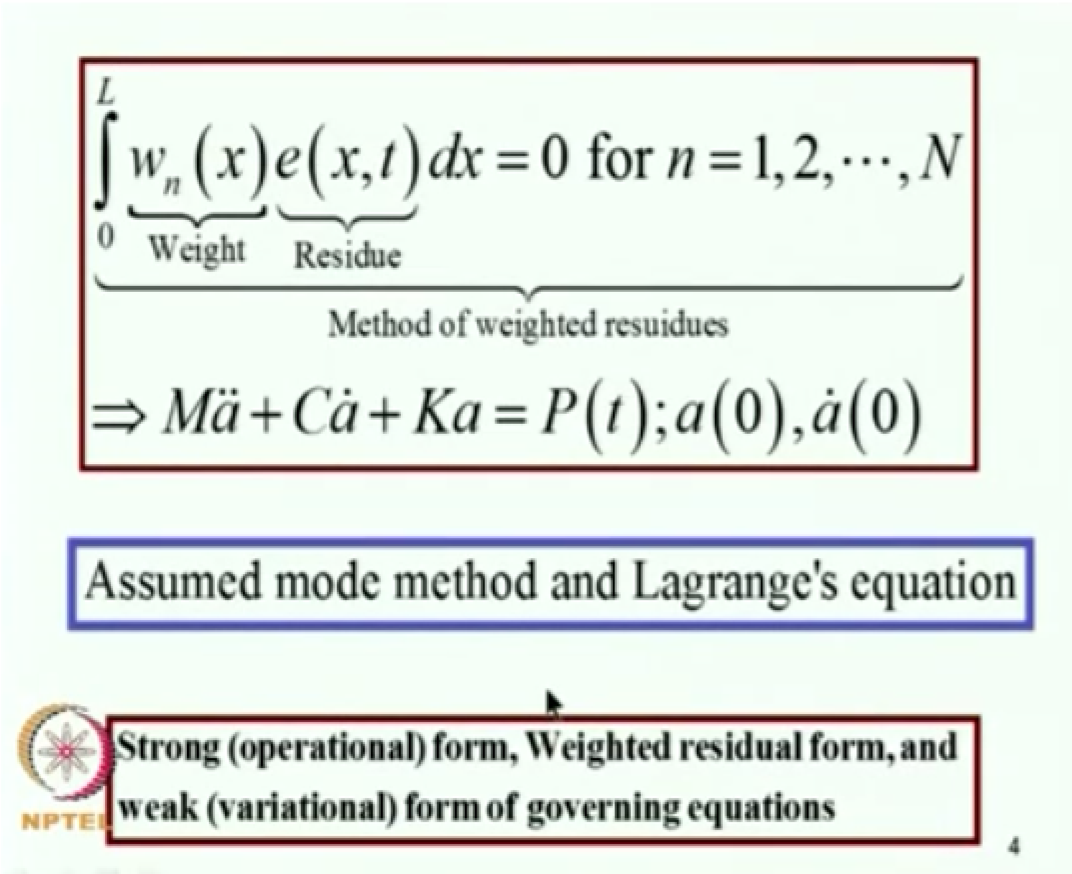
Galerkin $\int_0^L \phi_n(x)\,e(x,t)\,dx = 0$ for $n = 1,2,\cdots,N$

Subdomain collocation $\int_0^L \left\{U(x-x_{n-1})-U(x-x_n)\right\} e(x,t)\,dx = 0$ for $n = 1,2,\cdots,N$

Petrov-Galerkin $\int_0^L \psi_n(x)\,e(x,t)\,dx = 0$ for $n = 1,2,\cdots,N$

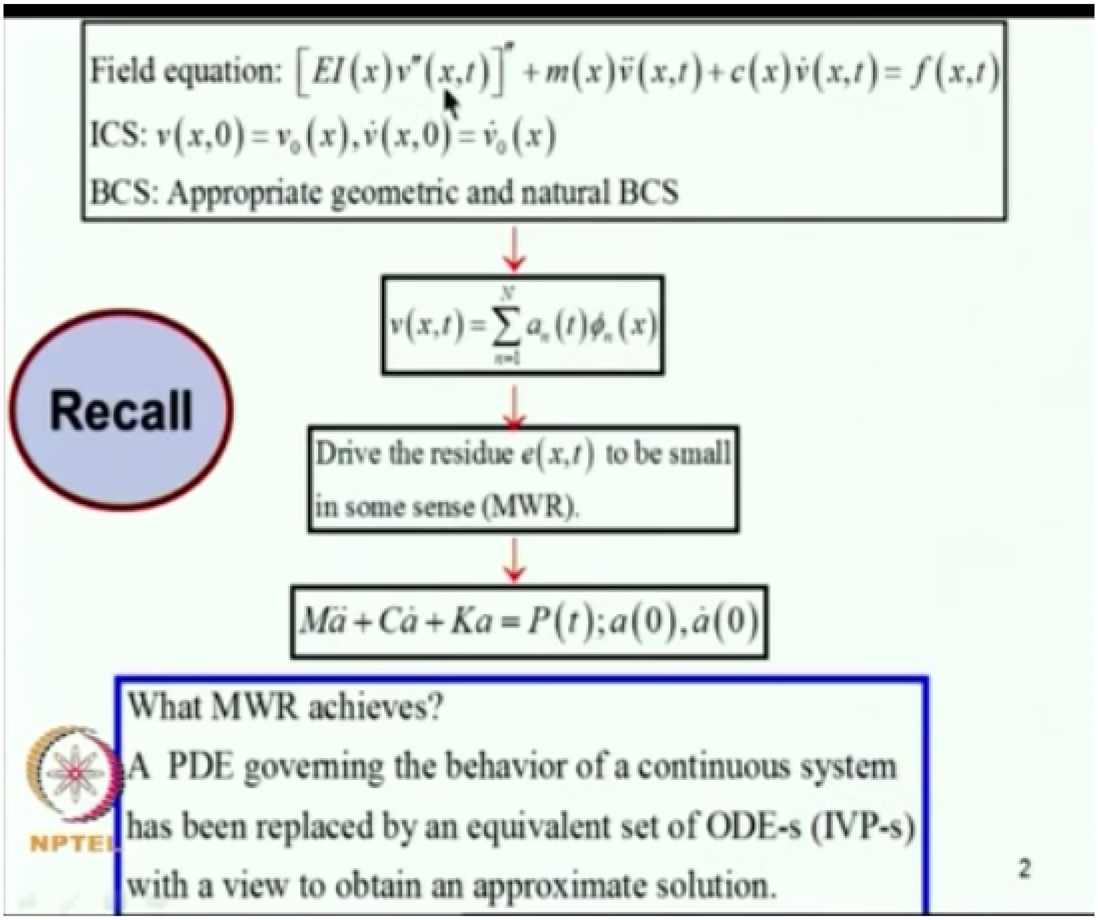
3

So we discussed several criteria for minimization of the residue, some measure of residue so we discussed method of least squares, collocation method, Galerkin method, subdomain collocation method, Petrov-Galerkin method etcetera, in all these methods the format of implementing the method of weighted residuals share certain common features, if we look at the way we obtain equations for AN in least squares, approximation we minimize the total mean square error and we get a residue term multiplied by another function, so here also in collocation also we get the residue term multiplied by a weight function, so this is common to all these methods, so the format is therefore we have the residue term and we select N weight

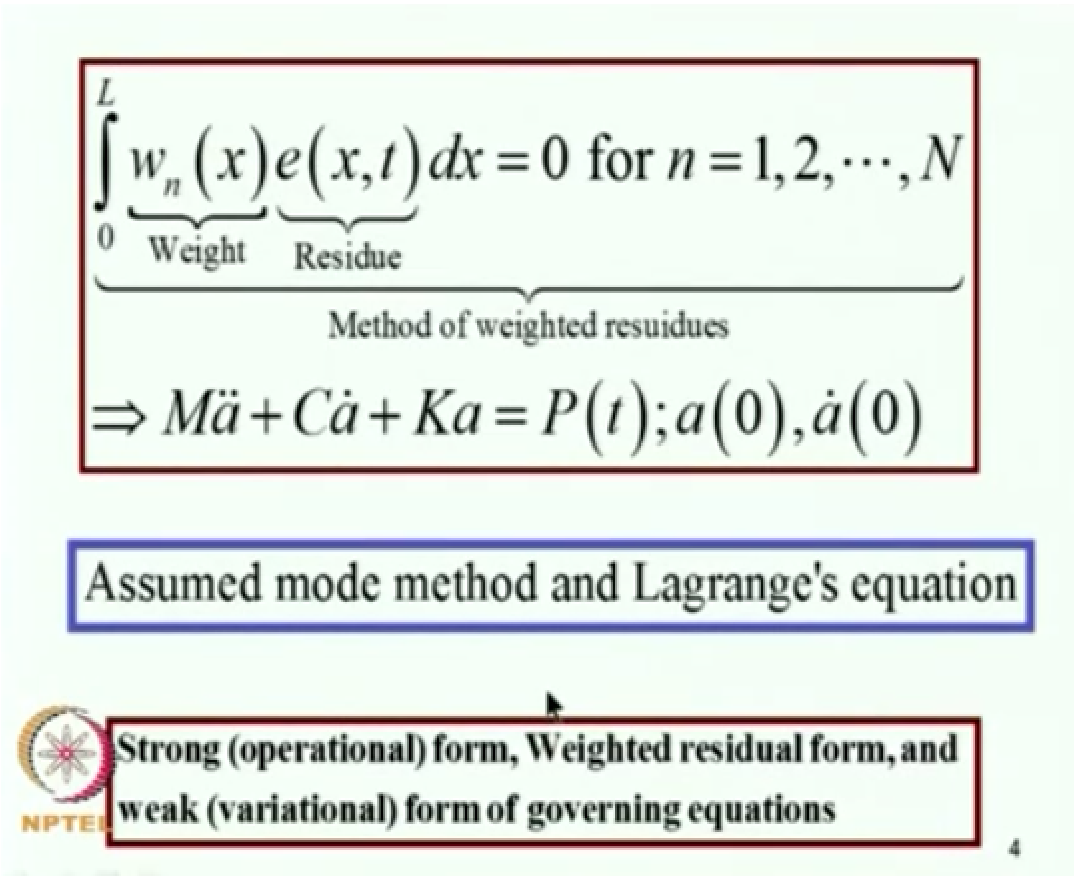


functions and write these equation for N = 1 to capital N thereby leading to a set of ordinary differential equations for the unknown generalized coordinates.

Now the implementation of this method therefore requires choice to be made on two quantities, namely trial functions and the weight functions, so if we look at the residue which will be equal to this term minus F(x,t)



it has fourth derivatives, fourth derivative terms present in this example, so consequently the trial functions in this example need to be such that they are differentiable over to fourth order,



but whereas the weight function does not, we need not impose any such requirements on the weight function. Now this format is known as weighted residual format or the weighted residual statement for the given problem, the original partial differential equation is called a strong format of the equation, in the so called weak formulation what we do is, we want to now recon the sense of continuity demand on the trial function and pass on some of that to the weight function, so that is achieved by integrating this equation in parts, by parts thereby what happens is the continuity requirements on trial functions come down and whereas on the weight function it goes up and the requirements will be shared equally between the weight functions and trial functions which the user anyway has to choose, so that approach is known as the so-called weak formulation of the governing equations.

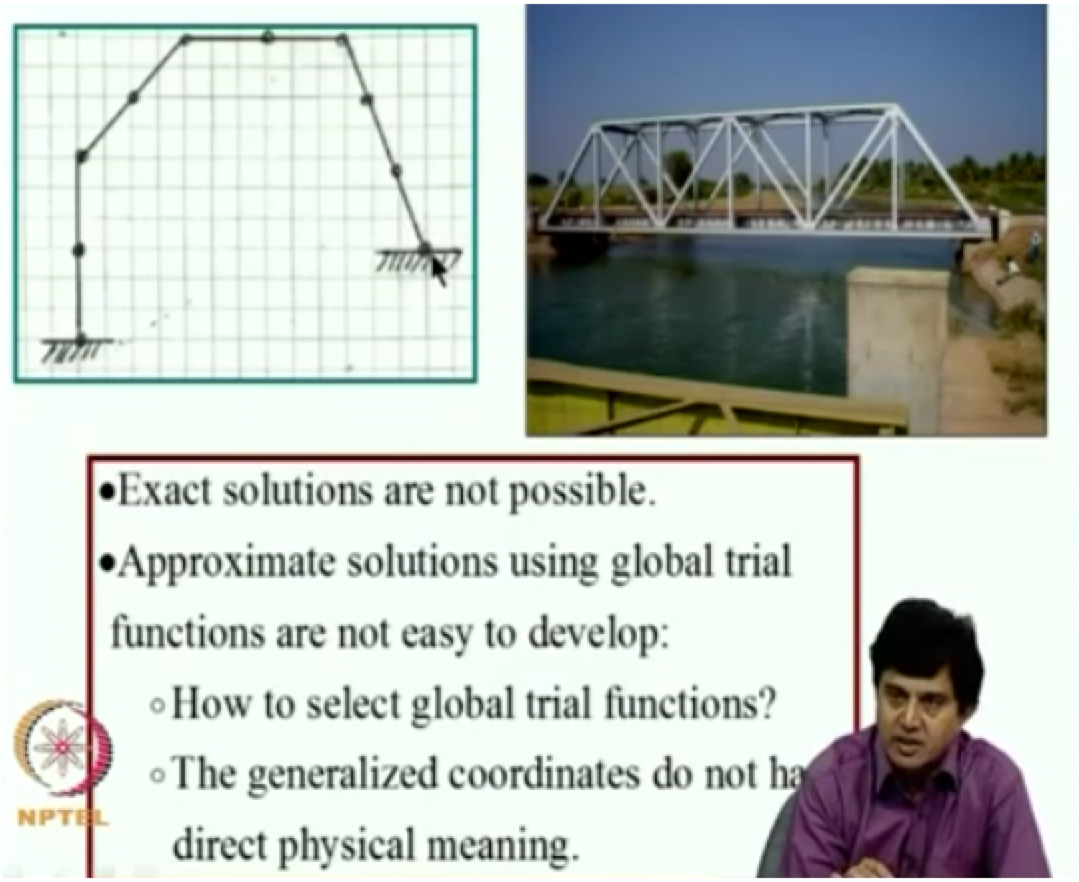
We also discussed another method known as assumed mode method which is suitable for structural dynamic applications, you can assume the solution as in this form and directly work with the governing Lagrange's equation to obtain the governing equations for A(t). So we have discussed all these methods, and what are the limitations of these methods?

Limitations

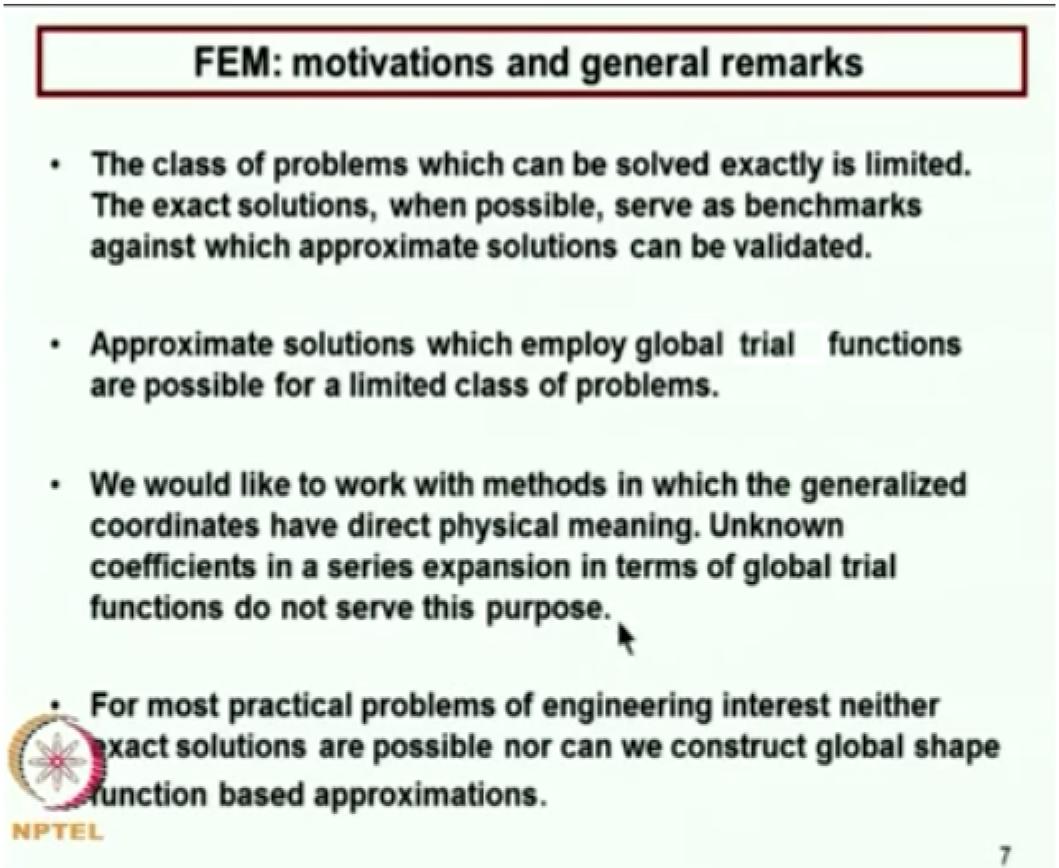
$$v(x,t) = \sum_{n=1}^{N} a_n(t)\phi_n(x)$$

- $\phi_n(x), n = 1, 2, \cdots, N$ are global in nature, that is, they are valid for all $x \in [0, L]$.
- Constructing these functions for simple geometries is relatively easy. Not so when geometries become more complicated.
- $a_n(t), n = 1, 2, \cdots, N$ are generalized coordinates and they do not have direct physical meaning.

Now in all these methods we are representing solution in this form and this Phi N(x) are global in nature, that is they are valid for all X belonging to 0 to L, so for simple line geometries as we have been discussing in the example so far the choice of this trial functions is reasonably straightforward, we can make those choices with reasonable effort, but however if you are dealing with more complicated geometries or structures built up of line elements and so on and so forth, developing global shape functions is not a easy exercise. Another problem with this representation is these generalized coordinates AN(t) N= 1 to N do not have a direct physical meaning, and we would like to deal with generalized coordinates if we could wear these coordinates have some direct physical meaning.

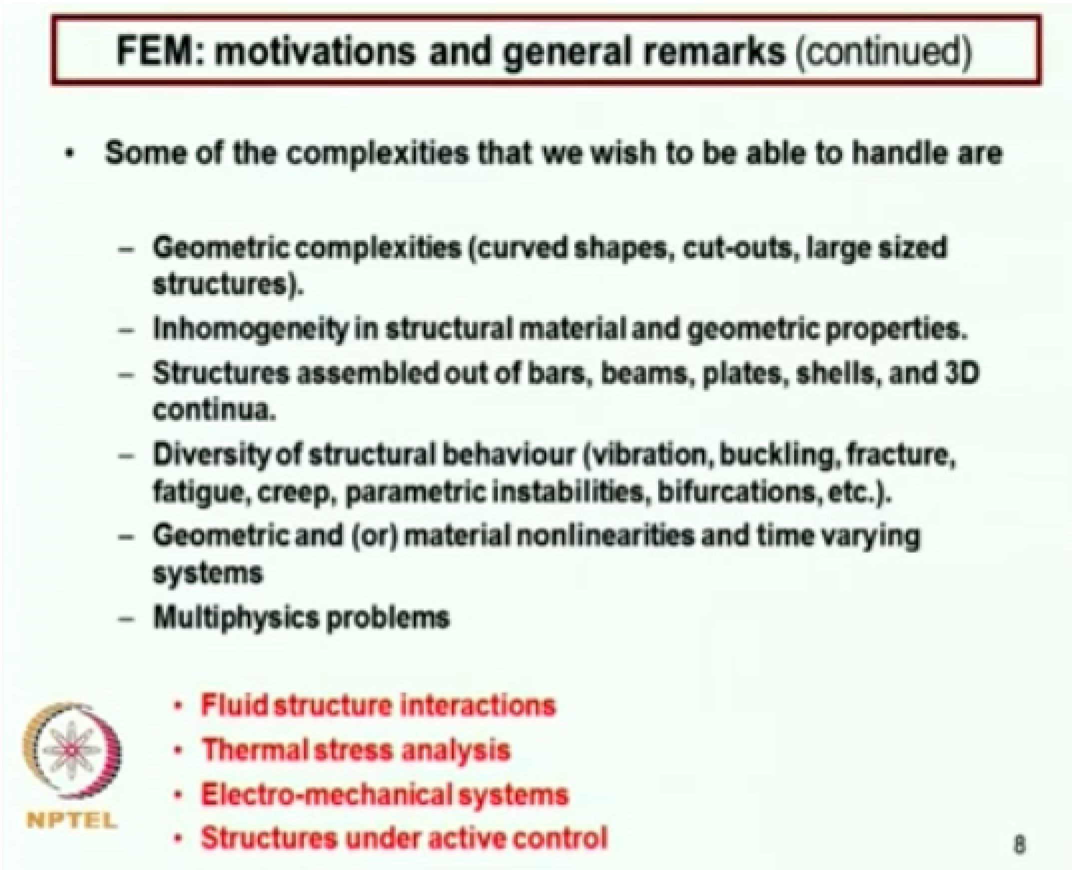


Now so to elaborate on what I said if you consider a truss, a three dimensional truss structure like this or a frame like this, it's very clear that developing an exact solution for dynamics of this type of structure using, by writing the governing partial differential equations for every member and establishing what happens at joints etcetera, etcetera, you are unlikely to succeed, even for this simple example it will not be a straightforward exercise, so exact solutions are difficult, if not impossible. Now approximate solutions using global trial functions are again difficult because it is very difficult to set up trial functions for this type of system, similarly for this type of, even for this simpler system it is very difficult to set up the globally valid trial functions. As I again said the generalized coordinates even in such these representations do not have direct physical meaning.



Now the finite element method basically aims to overcome these limitations, so again let us now start by making some observations and some general remarks, what are the motivations for developing finite element method and what are the generic features before we get into specific details, so let me again summarize, so the class of problems which can be solved exactly is limited of course the exact solution have their own value when they are possible they serve as benchmarks against which approximate solutions can be validated therefore study of exact solutions has a place in our studies.

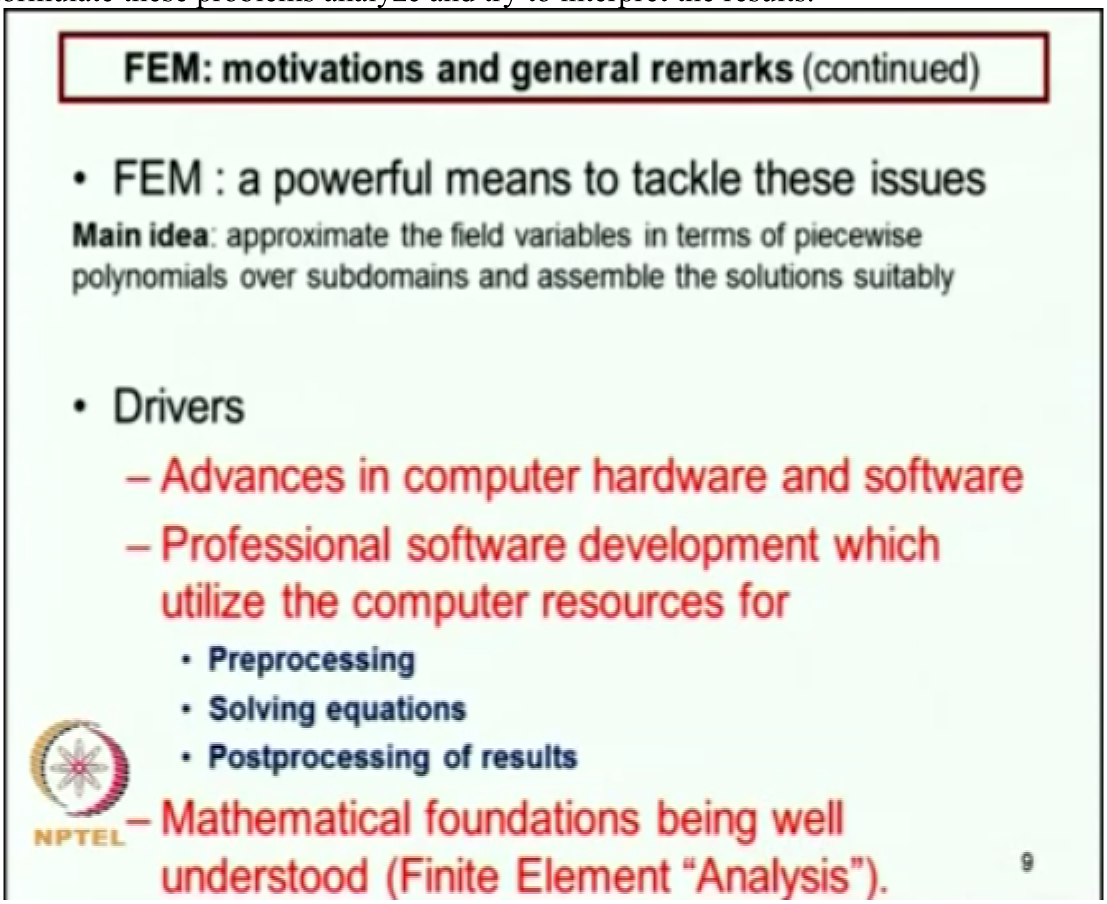
On the other hand approximate solutions which imply global shape functions are possible for a limited class of problems, again we would like to work with methods in which the generalized coordinates have direct physical meaning, unknown coefficients in a series expansion in terms of global trial functions do not serve this purpose, for most practical problems of engineering interest neither exact solutions are possible nor can we construct global shape function based on these approximation, so these are some of the difficulties.



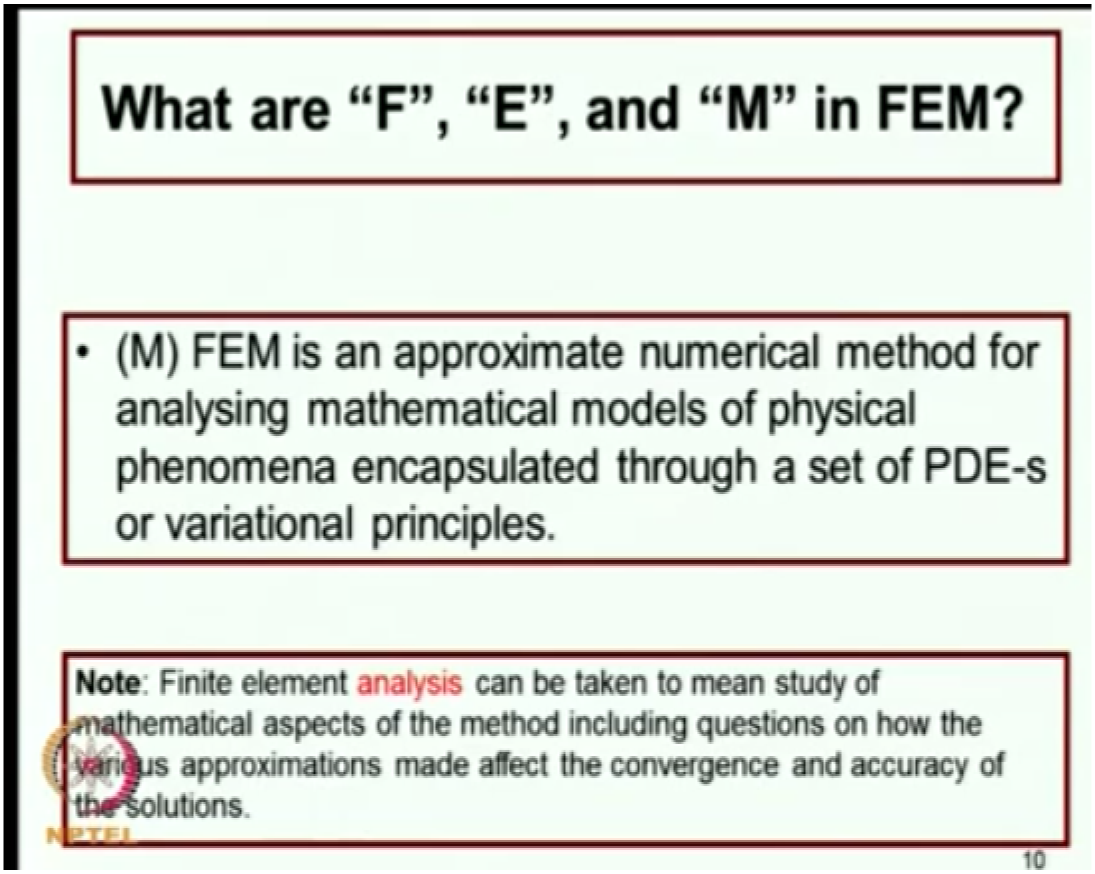
Moreover we would like to deal with certain complexities in our studies which could handle, which could include one or more of this, for example we would like to handle geometric complexities curved shapes, cutouts, large sized structures, for example nuclear power plants, cooling towers, bridges, long span bridges, and so on and so forth with foundations and piers and superstructure all of them treated together, so geometric complexity is a basic difficulty that we use to handle. Similarly inhomogeneity and structural material and geometric properties, say members need not be prismatic, thickness could vary, material properties could vary and so on and so forth, then we may even deal with structures assembled out of simple structural elements like bars, beam, plate, shells and 3D continua, for example a bridge structure will have soil, piers, superstructure and so on and so forth, so different parts of the structure could be modeled using different structural elements so but we want to treat the entire assembly as a single structure.

Then we want to deal with diversity of structural behavior for example static stress analysis, vibration analysis, buckling, fractured, fatigue, creep, parametric instabilities, bifurcations and so on and so forth, so when we deal with practical problems we should have handle on dealing with some of these issues. Another important issue is we would like to be able to deal with geometric or material nonlinearities and time varying systems we'll come to that examples of all this in due course. There are many problems where more than one physical process is involved, for example in fluid structure interactions, like pipes conveying fluids or liquid storage tanks subjected to earthquake so on and so forth we would like to model the interaction between structure and the fluid.

In thermal stress analysis which is needed, for example in analysis of structures under fire, we need to combine heat transfer analysis with stress analysis and in electromechanical systems like smart systems and things like that the field variables could include certain electrical variables in addition to mechanic variables. Similarly structures under active control, for example in buildings where we have active control mechanisms in place against earthquake loads, so how do we deal with this variety of problems, we need a general framework where we can formulate these problems analyze and try to interpret the results.



Now the finite element method is basically a powerful means to tackle some of these issues, the main idea of finite element method is that the field variable defined over a given domain is approximated in terms of piecewise polynomials or subdomains and we assemble the solutions that we develop on subdomains suitably to get the solution for the entire domain, so the use of piecewise polynomials lie at the heart of finite element method. Now the finite element method has developed significantly in recent years, the main drivers for this is advances in computer hardware and software, and we have no professional softwares for using finite element method as a tool and which has three important components, it is not only that we use computers to solve the governing equations but we also use the computers to even to formulate the problem, set up the you know mathematical model for a given structure, by taking into account geometric details, constitutive laws and so on and so forth, and then arrive at the governing equation that needs to be solved. The equations that need to be solved could be a set of algebraic equations or a set of coupled ordinary differential equations it is not that we use computers only to solve those equations. Similarly after we solve these equations we use again the computers to process the results, so it is not that we obtain solutions at all points on the structure in our analysis, but we need to know what could be the response at different points of the structure, and how to display the results so that engineering decisions can be made, so computers are being used extensively to deal with all these wide variety of problems and this contributes to the wide success of the method. On the other hand the mathematical foundations of the subject is also being well understood and now study of finite element analysis forms a major theme in mathematics research.



So let me ask this question, finite element method we have abbreviation FEM what does this, what do these three alphabets mean? What is F in finite element method that is what is finite in finite element method, what is element in a finite element method and what is this method in the first place? So we will address this problem of what is the M, finite element method is an approximate numerical method for analyzing mathematical models of physical phenomena encapsulated through a set of partial differential equations or variational principles, so it is an approximate numerical method basically to treat partial differential equations or problems that can be formulated using certain variational principles.

Now we also have the phrase finite element analysis, now this can be taken to mean study of mathematical aspects of the method including questions on how the various approximations may affect the convergence and accuracy of the solutions, so this word analysis implies addressing these questions, whereas method implies using the tools to solve engineering problems with a proper understanding of how the method has been developed in the first place.

We generally work in Cartesian coordinate system.

Let $u(\tilde{x},t)$ be the $s \times 1$ vector of dependent field variables to be determined. Ω= domain of interest.

FEM treats $\{u(\tilde{x}_i,t)\}_{i=1}^{N}$ as unknowns and develops approximation to $u(\tilde{x},t)$ in terms of $\{u(\tilde{x}_i,t)\}_{i=1}^{N}$. The method employs piecewise polynomials to achieve this.

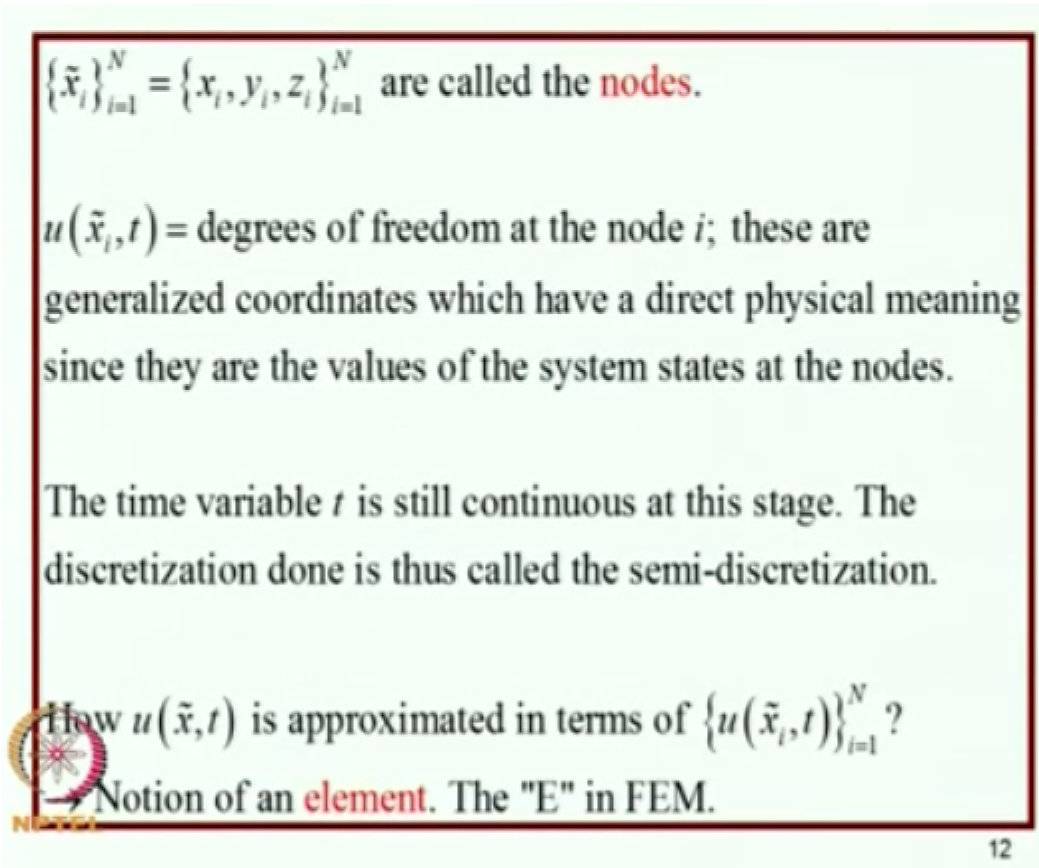
Thus the problem of approximating the infinite number of unknowns, that is, $u(\tilde{x},t)\ \forall \tilde{x} \in \Omega \,\&\, t \geq 0$, is converted into the problem of determining a finite set of of functions $\{u(\tilde{x}_i,t)\}_{i=1}^{N}$.

This is the "F" in FEM.

11

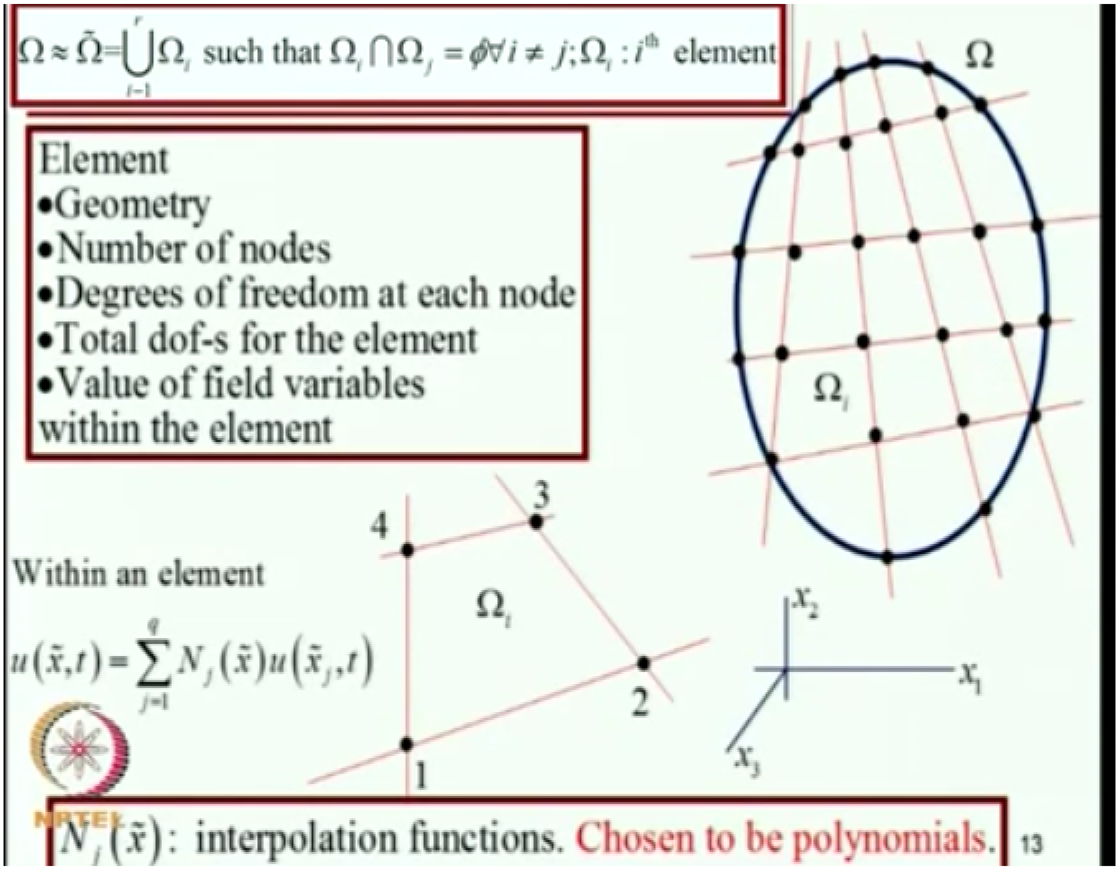
Now to explain what is finite in finite element method, we'll start with some observations we generally work in Cartesian coordinate system when we apply finite element method, we typically deal with vectors and tensors, like displacements and stresses and strains and so etcetera, therefore we need a coordinate system, so we use Cartesian coordinate system.

Now a field variable U(x tilde, t) let us say that is the S cross 1 vector of dependent field variables which we need to determine and omega is the domain of interest. So X tilde is X1, X2, X3 which is a Cartesian space. Now what finite element method does is it treats, we don't take the field variable U(x tilde, t) as they unknown, what we do is we select certain points XI tilde, and the value of the field variables at these points, if there are N number of them, these are treated as unknowns, and we aim to determine this and after knowing this we will develop an approximation to the field variables in terms of these N variables, so the method employs and also to develop these solutions, I mean for this purpose the method employs piecewise polynomials to achieve that, that is again a very important aspect of the method. So therefore the problem of approximating the infinite number of unknowns that is U(x tilde, t) for all X tilde belonging to Omega and for all T greater than equal to 0 is now converted into the problem of determining a finite set of functions U(x tilde, t), I running from 1 to N, so this is what makes the name finite in finite element method, this is what this aspect of the solution contributes to this nomenclature.

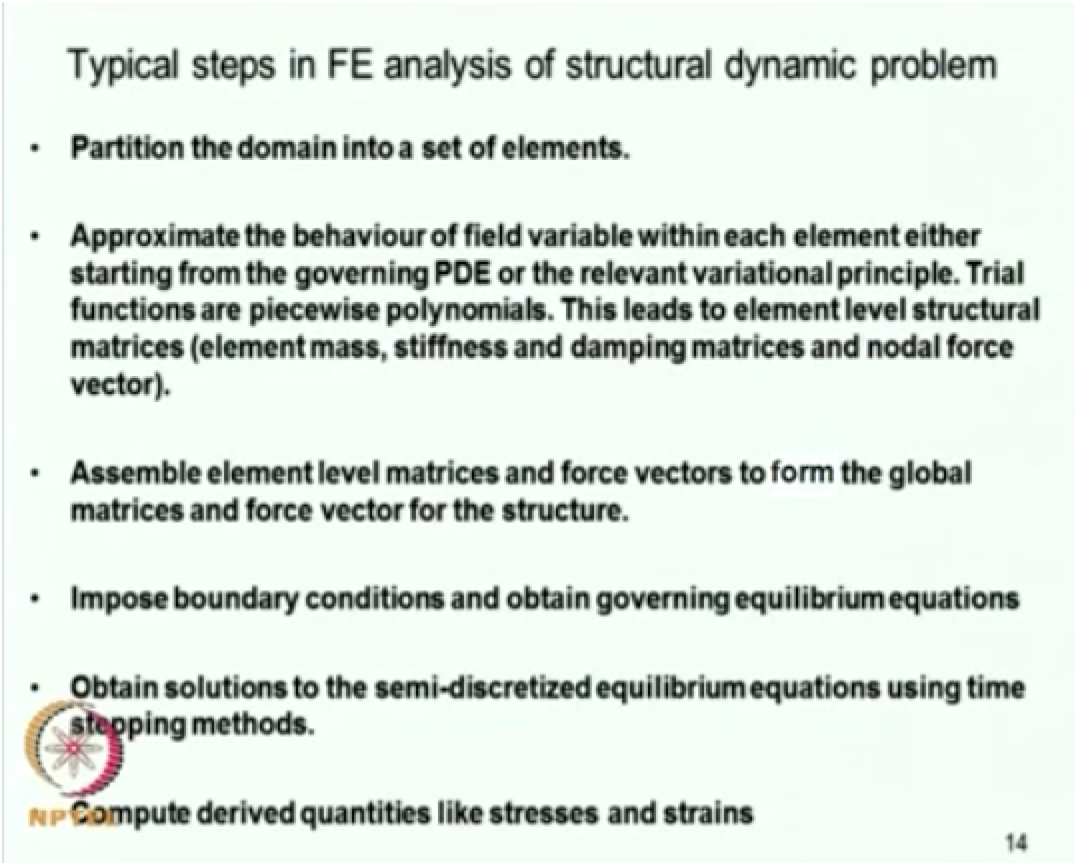


Now the points in the domain where we, the field variables are taken as unknowns are known as nodes, now we also call this as the value of the field variable at these points or degrees of freedom at the node I, these are the generalized coordinates which have typically have a direct physical meaning because they are the values of the field variable at the nodes, they have an immediate physical meaning because the field variable itself has a physical meaning for a given problem.

Now in this discussion the time variable T is still continuous at this stage, I have not talked about discretizing time, so we call this discretization as semi-discretization, only spatial variables are discretized, the time is still treated as a continuous variable. So now the question is how this U(x tilde, t) is approximated in terms of these degrees of freedom, this leads to the notion of element, that the E in the finite element method, what does that mean? So you consider a domain omega, suppose I partition this omega into a set of I equal to 1 to R as shown



here, this is one I-th element, we say that these omega eyes whose union contributes to the domain and we expect that there is no intersection between these sets, that means Omega I intersection Omega G is a null set for all I node = J and Omega I is known as the I-th element, so this element itself now need to be characterized in terms of geometry, and how many nodes it has? I have shown for each element here for this type of element three nodes, but whereas if you look at this there are three nodes, then how many field variables are treated as unknowns at a given node that depends on problems as we will shortly see, and what is the total number of unknown coordinates that is assigned to each element, that is known as total degrees of freedom for the element. Then value of the field variables within the element, how do we determine? Suppose I know value of the field variable at the nodes, how do I get the value of the field variable within an element, so these are some of the issues that we need to address when we talk about an element, so what we do is suppose if you consider this element I name these 4 points as nodes 1, 2, 3, 4, and what we do is within an element we approximate the field variable by interpolating the value of the field variables at these 4 points, so we write this as J = 1 to Q, Q is 4 in this illustration, and this is the value of the field variable at 1, 2, 3, 4, and these are known as the interpolation functions. So NJ(x tilde) interpolation functions, what we do is, we choose these interpolation functions to be polynomials, the nature of this choice, the paramount in determining the quality of the solutions we get, so this is a very vital aspect of implementing the method.



So what are the typical steps in finite element analysis of say structural dynamic problem, so what we do a partition the domain into a set of elements, now this domain omega actually we may not be able to represent exactly in a finite element model, for example it has covered edges so in our modeling even the domain is replaced by an approximation, so those details will come to as an when we encounter curved boundaries and things like that, so even these curved edges need to be suitably approximated, it is not that the given domain omega with all its complexities is exactly replicated in our model, so that also should be emphasized so that we partition the domain into a set of elements, then we approximate the behavior of the field variable within each element either starting from the governing partial differential equation or the relevant variational principle if it is available for the given problem, we use trial functions as I said trial functions are piecewise polynomials this leads to element level structural matrices typically in dynamics we get element, mass, stiffness, and damping matrices, and vector of nodal forces.

Next we have now partitioned the domain and analyze each bit like this, now the next job is we should put back all this and form the buildup structure, so we assemble element level matrices and force vectors to form the global matrices and force vector for the structure, this is assembling. Then we look at the boundary conditions to be imposed, right after we impose the boundary conditions we obtain the governing set of equilibrium equations for the problem, typically in this application there will be a set of ordinary coupled, second order coupled ordinary differential equations, they could be nonlinear if the underlying problem is nonlinear otherwise it could be linear.

Now for these set of ordinary differential equations we now address the question of discretizing the time, so we typically use time stepping methods explicit or implicit depending on the details of the problem and this completes the determination of the field variables at the nodes. Then we compute derived quantities like stresses and strains, if displacements are the degrees of freedom which in most of the applications is the case then we need to derive strains stresses at the nodal values, at the nodal node point as well as in the interior, okay, so that completes the solution. We need to consider several other points.

Several other points to consider

- $\Omega \approx \Omega = \bigcup_{i=1}^{r} \Omega_i$ such that $\Omega_i \cap \Omega_j = \phi \forall i \neq j; \Omega_i : i^{\text{th}}$ element

Also defines division of geometry.

Mesh : Collection of elements which replaces the domain Ω in the simulation work.

- Inter-element behavior of field variables: communication only through value of state variables through nodes (and not through lines or surfaces).
- Evaluation of the derived quantities (e.g., strains and stresses) within an element and their behavior across element boundaries.
- Selection of number of elements, order of polynomials, step size in time integration and the convergence of the solution $(h - p - \Delta t)$

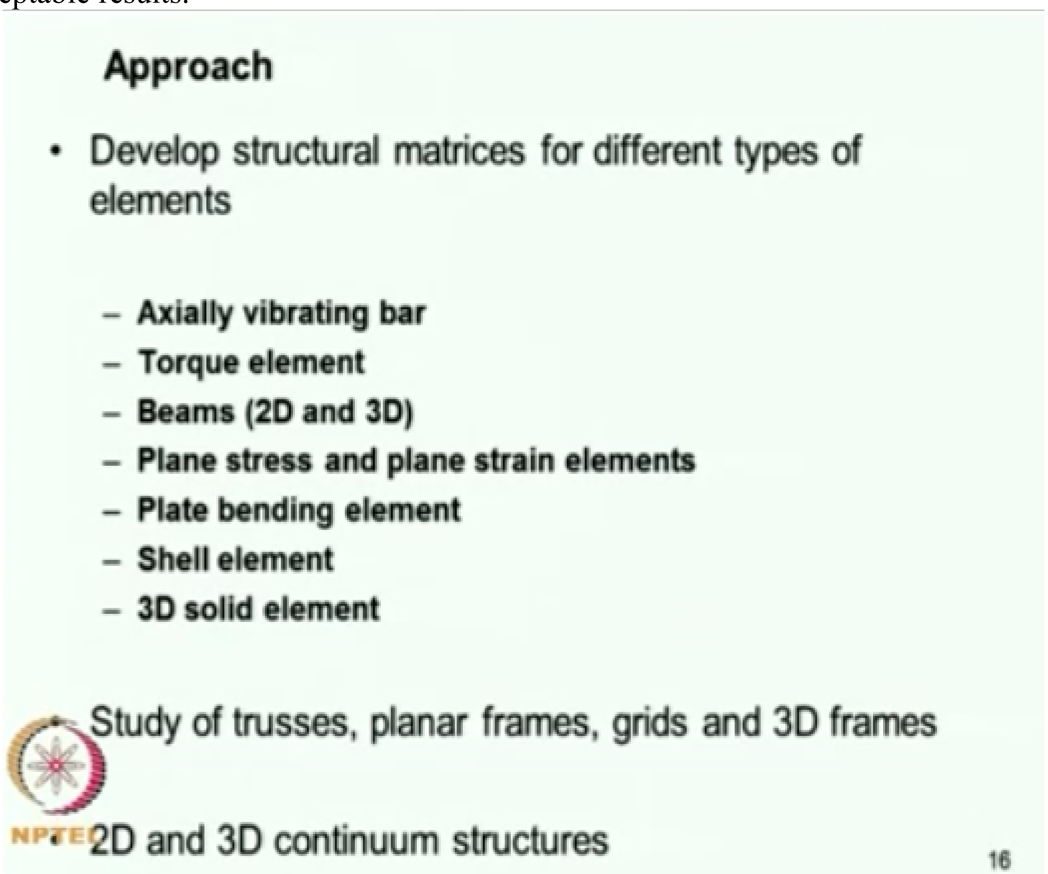
15

I'll just make a brief mention of these points and we'll return to some of these issues as we go along in the course, so we have said that we have defined element through this description so the given domain is partitioned into a set of R sub domains which are you know mutually exclusive and collectively exhaustive and we call omega as I-th element, this also defines the division of geometry.

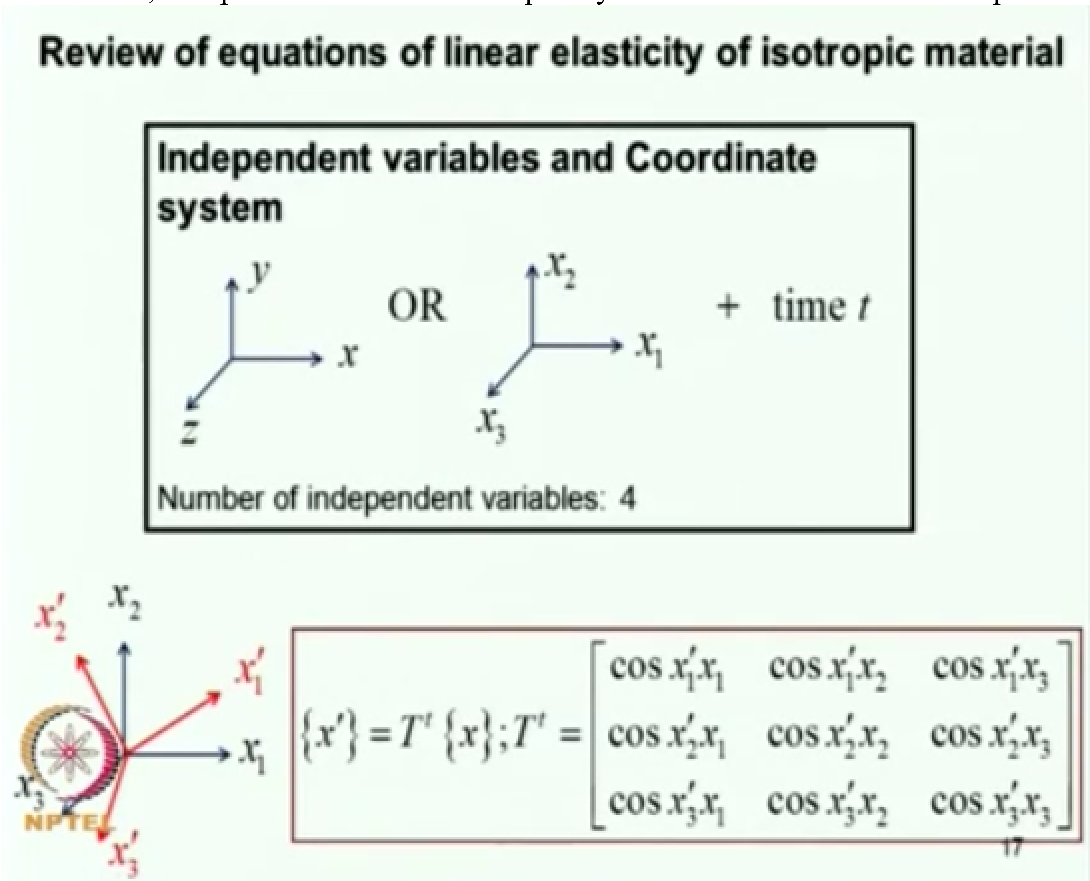
Now the word mesh is used to denote the collection of elements which replaces a domain omega in the simulation work, now we need to address question on inter-element behavior of field variables, so we should note that the communication between two distinct element is only through the value of the field variables at the nodes, is only through the points, common shared points, we don't talk about interaction through lines or surfaces, it's only through point even in two-dimensional, three-dimensional problems, because in two-dimensional, in a line element the two elements are connected through a point, but in a plane element two-dimensional element there are two neighboring elements can be connected through lines, and in solid element three-dimensional element the neighboring elements are connected through surfaces, but the interaction is always through nodes that are the points.

Then how do we evaluate the derived quantities, for example strains and stresses within an element, and how do these quantities behave across the element boundaries so this is an interesting question we will see that there are some difficulties in modeling this and this is one of the concern that we should be aware of.

Then we should look at quality of the answers obtained, so there are many decisions that we have made, how many elements that we should take, how many element that means, how fine should this partition be, right, that we call it as H, H indicates in some sense the size of the element, and within an element what order of polynomial we should use to interpolate the field variables, that determines P, and Delta T is a time step that we need to use in integrating the semi discretized equations, so the accuracy of the solution finally depends on a combination of H P and Delta T, right, so we need to understand how to make these choices to ensure that we get acceptable results.

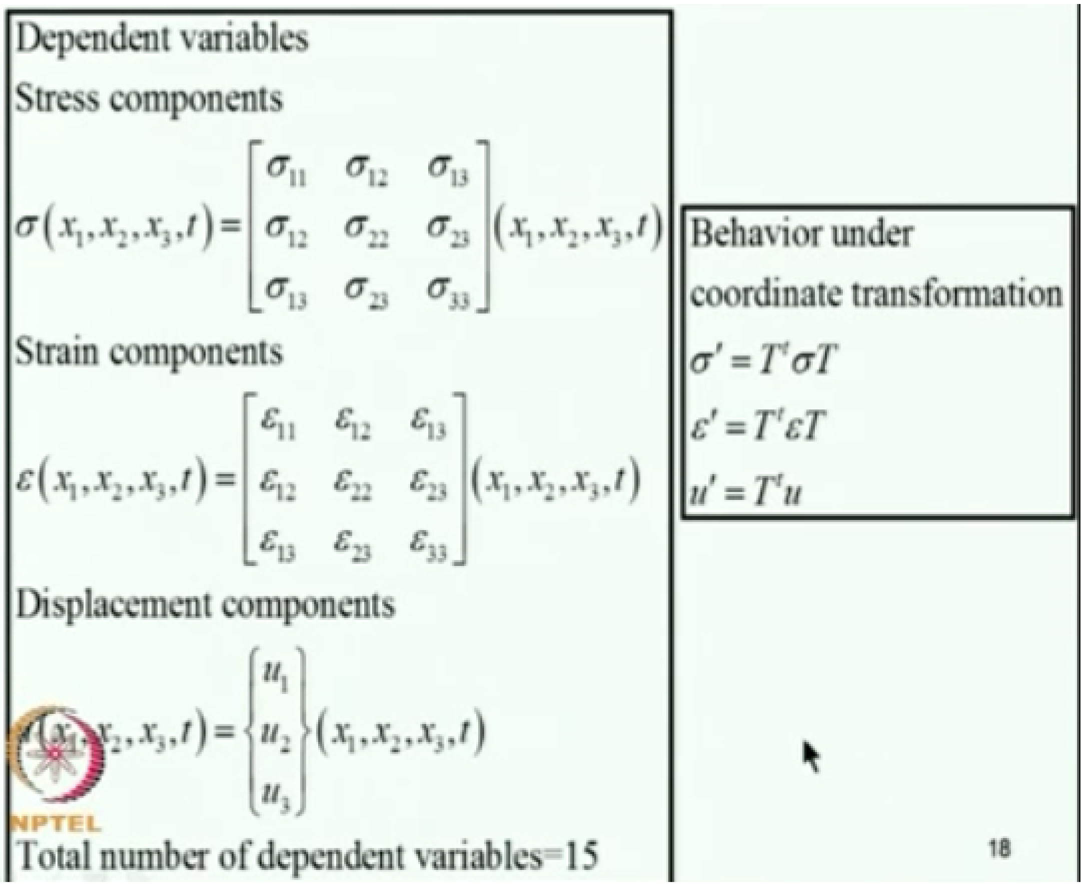


So now to develop this discussion on finite element method the approach that we will take is we start by developing structural matrices for different types of elements like axially vibrating bar, the torque element, two dimensional and three dimensional beams, then plane stress and plane strain elements, plate bending element, shell element, 3D solid element, and so on and so forth, so for each of these we will develop character, we will characterize the behavior of a single element and then we will address question of how to assemble the formulations at an element level and study structure such a trusses, planar frames, grids, and 3D frames, and also certain two-dimensional and three-dimensional continuum structures, so this is a broad outline of at least the next few lectures, so we'll start somewhere, we will start with addressing questions you know listed here, as a prelude to that we will quickly review some notations and equations from



linear elasticity for isotropic material. So the independent variables and the coordinates of the system we use X, Y, Z or X1, X2, X3 and time T, so there are four independent variables in a three-dimensional problem of elasticity, one is time and three are spatial coordinates.

Now we can make a coordinate transformation, so we can have one coordinate X1, X2, X3 a coordinate system, another one X1 prime, X2 prime, X3 prime and we would like to know how to transform quantities described in X1, X2, X3 to the new coordinate X1 prime, X2 prime and X3 prime, the coordinate transformation matrix for a position vector is given by this well-known transformation matrix, this is an orthogonal matrix, we will be using some of this quite often. What are the dependent variables in elasticity problem?



The state of stress at a point is given by the stress matrix which is 3 by 3 symmetric matrix and each component will be a function of X1, X2, X3 and time T. Then we also have strain components which is again a tensorial quantity and it is described by another 3 by 3 matrix which is again symmetric and which is again function of the 4 independent variables, displacement component, there are 3 displacement components U1, U2, U3 and they are again functions of the spatial and time coordinates. So the total number of dependent variables is 15, independent variables are 4. Now how does these, how do these dependent variables behave under coordinate transformation? Sigma stress and strain are tensorial quantities, so they undergo transformation through these rules, where T is the transformation matrix between the coordinate system, so sigma prime is the stress, state of stress at a point in the transform coordinate, new coordinate system and this is the sigma is the state of stress with respect to a reference coordinate system, similarly strain and this is a displacement, displacement is a vector this is how it translates. Scalar quantities are invariant with respect to coordinate transformation.

Number of unknowns=15

Numer of equations

Equillibrium: 3

Strain-displacement: 6

Stress-strain: 6

Equillibrium equations

$$\frac{\partial \sigma_{11}}{\partial x_1}+\frac{\partial \sigma_{21}}{\partial x_2}+\frac{\partial \sigma_{31}}{\partial x_3}+X_1=\rho \ddot{u}_1$$

$$\frac{\partial \sigma_{12}}{\partial x_1}+\frac{\partial \sigma_{22}}{\partial x_2}+\frac{\partial \sigma_{32}}{\partial x_3}+X_2=\rho \ddot{u}_2$$

$$\frac{\partial \sigma_{13}}{\partial x_1}+\frac{\partial \sigma_{23}}{\partial x_2}+\frac{\partial \sigma_{33}}{\partial x_3}+X_3=\rho \ddot{u}_3$$

$$\sigma_{ij}=\sigma_{ji};i,j=1,2,3$$

Strain-displacement equations

$$\varepsilon_{11}=\frac{\partial u_1}{\partial x_1};\varepsilon_{22}=\frac{\partial u_2}{\partial x_2};\varepsilon_{33}=\frac{\partial u_3}{\partial x_3};$$

$$\varepsilon_{12}=\frac{1}{2}\left(\frac{\partial u_1}{\partial x_2}+\frac{\partial u_2}{\partial x_1}\right);$$

$$\varepsilon_{13}=\frac{1}{2}\left(\frac{\partial u_1}{\partial x_3}+\frac{\partial u_3}{\partial x_1}\right);$$

$$\varepsilon_{23}=\frac{1}{2}\left(\frac{\partial u_2}{\partial x_3}+\frac{\partial u_3}{\partial x_2}\right);$$

$$\varepsilon_{ij}=\varepsilon_{ji};i,j=1,2,3$$

19

So now we have 15 unknowns and we need to now generate the 15 equations to tackle this problem as we know there are 3 equilibrium equations, 6 strain displacement relations and 6 stress strain relations, so I have listed these equations here for a quick reference, the equilibrium equations have terms involving gradients of stresses and also this capital X1, X2, X3 are the body forces, there are forces which are proportional to the volume, and rho is the density U1 double dot, U2 double dot, these terms on the right hand side represent the inertial forces, okay, so sigma IJ is same as sigma JI, therefore there will be 3 equilibrium equations, the strain displacement relations are given through this set of formulae, again strain epsilon IJ is epsilon JI these are normal strains, these are shear strains, the three normal strains and three shear strains.

Representation of the constitutive laws-I

$$e = \varepsilon_{11} + \varepsilon_{22} + \varepsilon_{33}$$

$$\sigma_{11} = \lambda e + 2G\varepsilon_{11}$$

$$\sigma_{22} = \lambda e + 2G\varepsilon_{22}$$

$$\sigma_{33} = \lambda e + 2G\varepsilon_{33}$$

$$\sigma_{12} = 2G\varepsilon_{12}$$

$$\sigma_{13} = 2G\varepsilon_{13}$$

$$\sigma_{23} = 2G\varepsilon_{23}$$

$$E = \frac{G(3\lambda + 2G)}{\lambda + G}$$

$$\nu = \frac{\lambda}{2(G+\lambda)}$$

Representation of the constitutive laws-II

$$I_1 = \sigma_{11} + \sigma_{22} + \sigma_{33}$$

$$\varepsilon_{11} = \frac{1}{E}\left[(1+\nu)\sigma_{11} - \nu I_1\right]$$

$$\varepsilon_{22} = \frac{1}{E}\left[(1+\nu)\sigma_{22} - \nu I_1\right]$$

$$\varepsilon_{33} = \frac{1}{E}\left[(1+\nu)\sigma_{33} - \nu I_1\right]$$

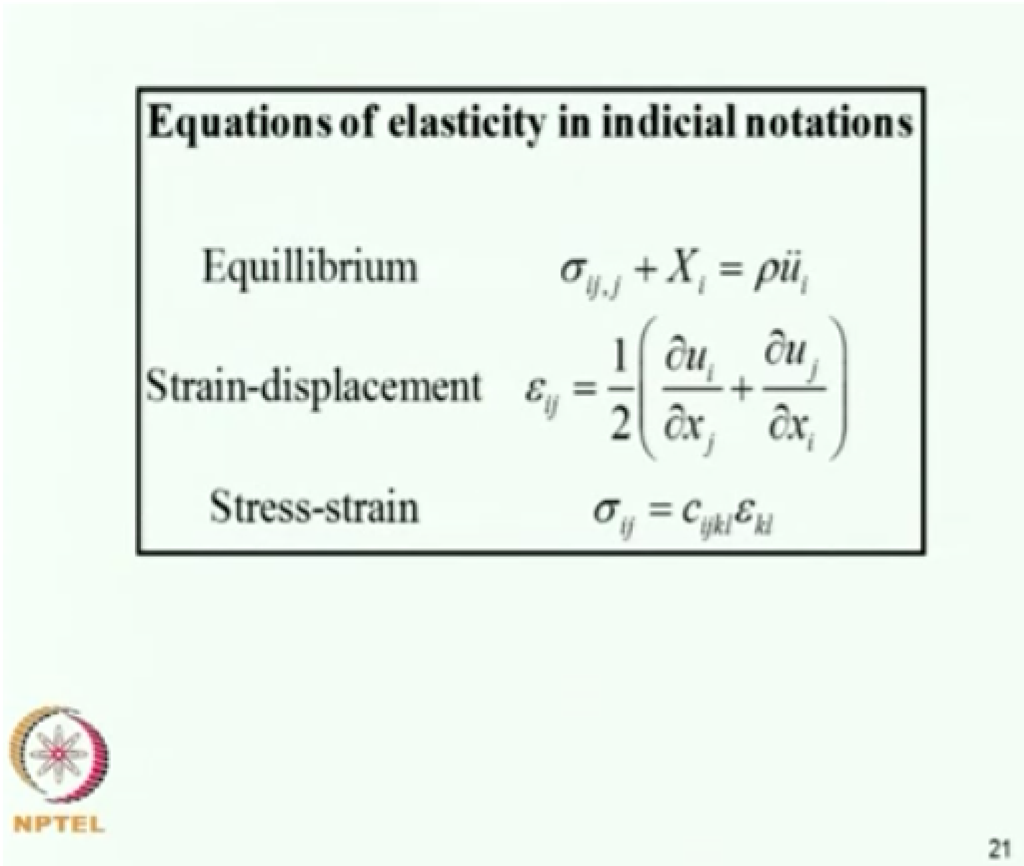
$$\varepsilon_{12} = \frac{1}{2G}\sigma_{12} = \frac{1+\nu}{E}\sigma_{12}$$

$$\varepsilon_{13} = \frac{1}{2G}\sigma_{13} = \frac{1+\nu}{E}\sigma_{13}$$

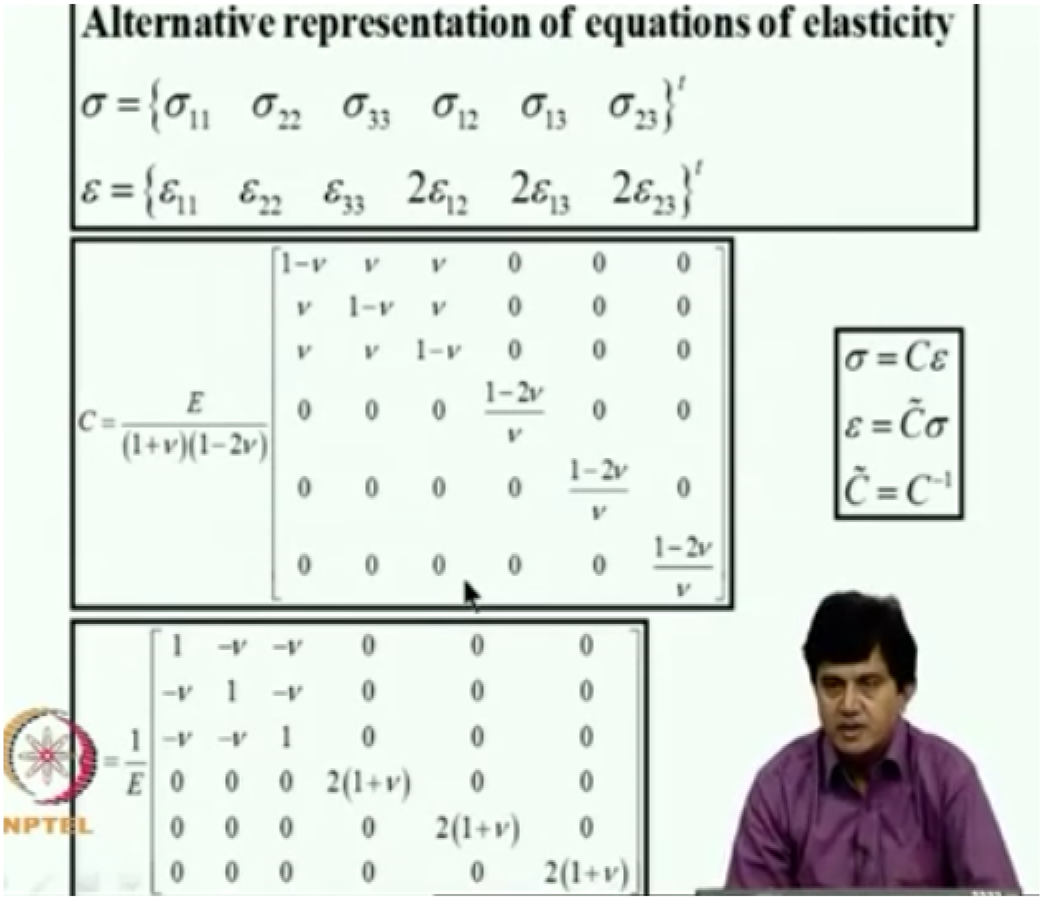
$$\varepsilon_{23} = \frac{1}{2G}\sigma_{23} = \frac{1+\nu}{E}\sigma_{23}$$

20

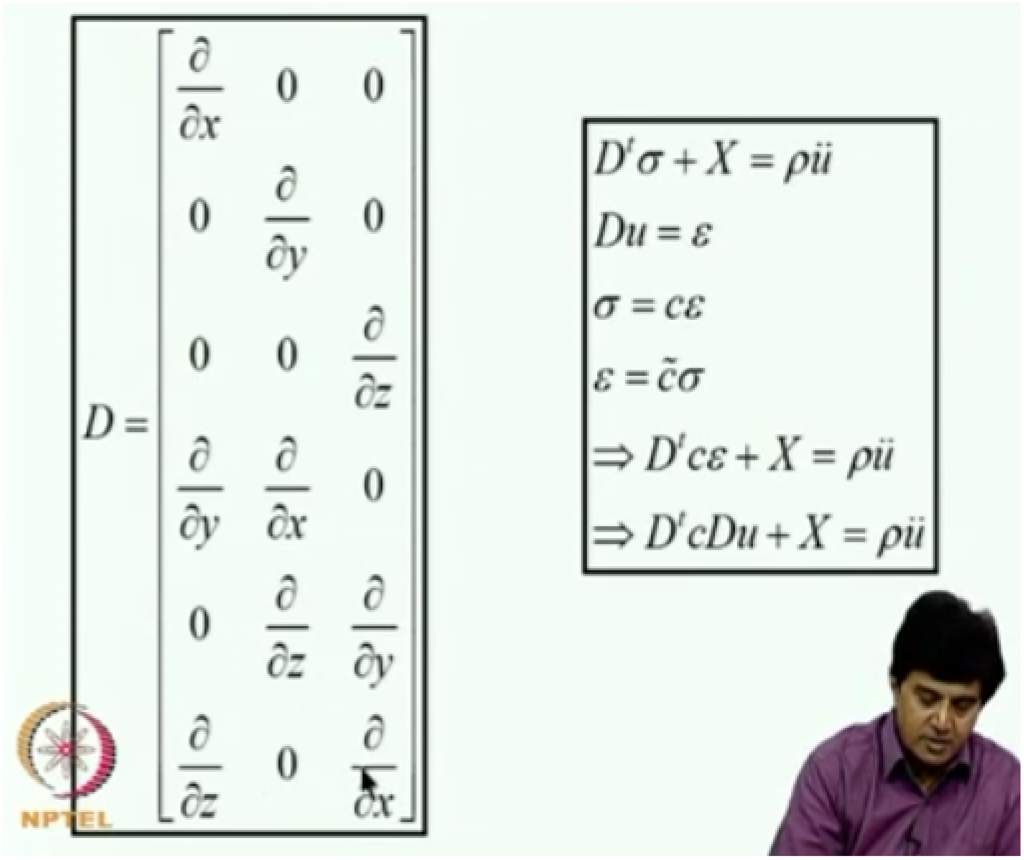
Now for a Hokkien material the stress and strain are linearly related so we can show that there will be two independent constant in terms of lame’s constants the constitutive laws can be given like this, the way the stresses are related to the strains is through this, the E is the sum of the normal strains, and in terms of that we have this constitutive law, these are the stress-strain relations. In terms of, you can also express strains in terms of stresses, this is the first invariant of the stress matrix and we get these relations, again this is in terms of young's modulus and shear modulus. So we could use either this or this depending on the context in which we formulate the problem. If you use indicial notations all these equations can be written in a compact manner, so this is the equilibrium equation, set of equilibrium equations, this is the set of strain displacement relations, and this are the stress-strain relation I, J, K, L etcetera take



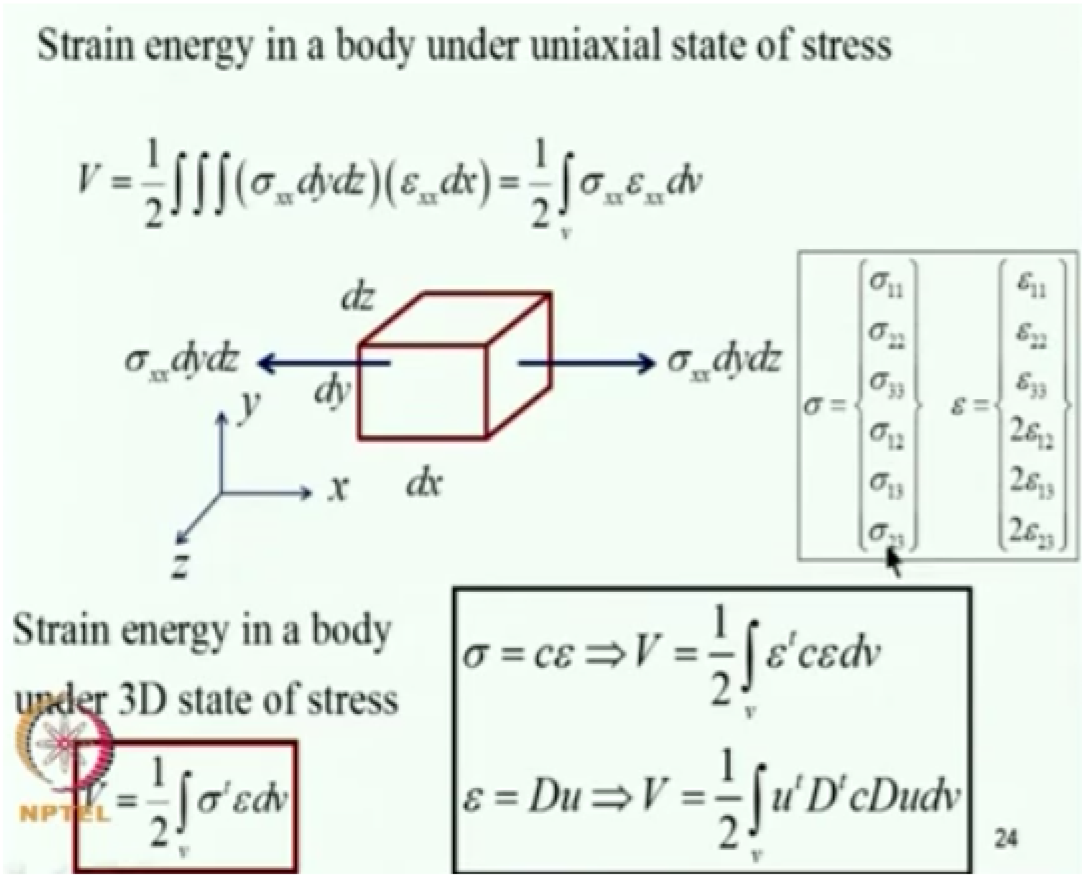
values from 1, 2, and 3, so the repeated indices imply summations, a comma is the differential all those notations are used here.



Now there is yet another way of representing stresses and strains, here we write the state of stress at a point as a 6 cross 1 vector, instead of a 3 by 3 matrix, symmetric matrix we write it as 6 cross 1 vector, and the strain is written in this, so this is not a tensorial quantity anymore, so this is the strain and if we now write the stress-strain relations in this notation this C will be 6 by 6 matrix, because stress is a 6 cross 1 vector, and epsilon is 6 cross 1 vector, so either we can write stress in terms of strain, or strain in terms of stress, and these matrix obey this identity and we have these expressions for the C matrix and C tilde matrix and new is the persons ratio is an young's modulus, so this is for an isotropic material.



Now if we define an operator D as shown here, this is 6 cross 3 matrix operator, we can write the governing equilibrium equations, and in this form D transpose sigma + X is Rho U double dot, now the strain displacement relation using the same matrix can be written as DU is equal to epsilon, now here if I now use the constitutive law sigma is equal to C epsilon or epsilon is C tilde sigma and suppose if I substitute here so I will write for sigma CE, then this equation becomes D transpose C epsilon + X = Rho U double dot. Now for epsilon I will write, for epsilon I will write now DU, so if I write that I get this as the equations, governing equations where the equilibrium equations, stress-strain relations and constitutive laws are all been captured, and this is the final equations are written in terms of displacement components, okay. So D is this operator, C is the constitutive matrix or constitutive laws, and this is the body force, vector of body forces.



Now we will also need general approach to derive the strain energy in a body under a multi axial state of stress, so we will start by discussing how to derive strain in a uniaxial state of stress, suppose at a point in the structure we choose an infinitesimal element DX, DY, DZ or volume DX, DY, DZ, and if sigma XX is the only nonzero stress component, the forces on this face is given by sigma XX, DY, DZ.

Now the work done by these forces on the deformation can be computed over this entire volume, the force is sigma XX, DY, DZ and the displacement is strain into DX, this is this and 1/2 because the load is applied the deformation take gradually and we can write this as sigma XX, epsilon XX DV integral or we multiplied by 1/2. Now if the body is in a multiple state of stress that means all stress components are acting simultaneously then this logic can be extended and the equation for strain energy or the entire volume can be written in terms of the stress vector and strain vector in this form, okay.

So now if we substitute the constitutive law and use the strain displacement relation, the strain energy can be expressed in terms of the displacement components the operator D and the constitutive matrix of constitutive you know coefficients in constitutive law, so this is the expression for strain energy in a 3D solid under multiple multi state of stress, so this is strain

$$V(t)=\frac{1}{2}\int_v u^t D^t c D u\, dv$$

$$T(t)=\frac{1}{2}\int_v \dot{u}^t \rho \dot{u}\, dv$$

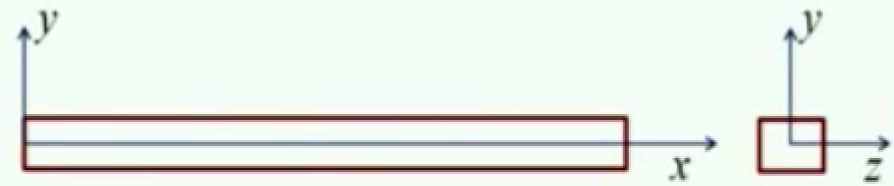
$$L=T(t)-V(t)$$

$$A=\frac{1}{2}\int_{t_1}^{t_2}\int_v [\dot{u}^t \rho \dot{u} - u^t D^t c D u]\, dv\, dt$$

25

energy, this is the kinetic energy, right, so the Lagrangian is this and we can define the action integral, so if we now apply the you know select DU so that action integral is minimized now we recover the equations of elasticity, okay, so this is a general framework for treating three-dimensional elasticity problems.

Axially vibrating rod element



AE, m, l, c

$\sigma_{xx}(x,y,z,t) = \sigma_{xx}(x,t)$; all other stress components are zero.

$$\varepsilon(x,t) = \frac{1}{E}\sigma_{xx}(x,t); u(x,y,z,t) = u(x,t)$$

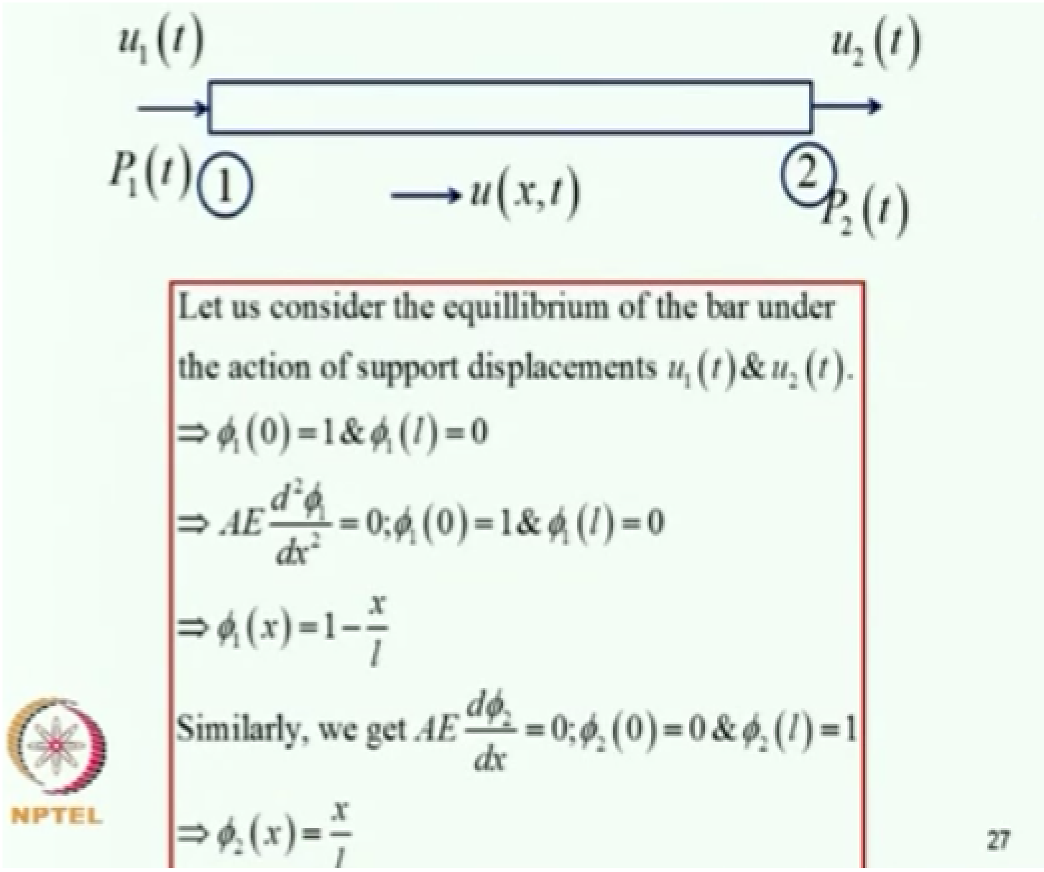
$$V = \int_0^l \frac{1}{2} AE\left(\frac{\partial u}{\partial x}\right)^2 dx; T = \int_0^l \frac{1}{2} m\left(\frac{\partial u}{\partial t}\right)^2 dx$$

$$A = \int_{t_1}^{t_2} \left\{ \int_0^l \frac{1}{2} m\left(\frac{\partial u}{\partial t}\right)^2 dx - \int_0^l \frac{1}{2} AE\left(\frac{\partial u}{\partial x}\right)^2 dx \right\} dt$$

NPTEL

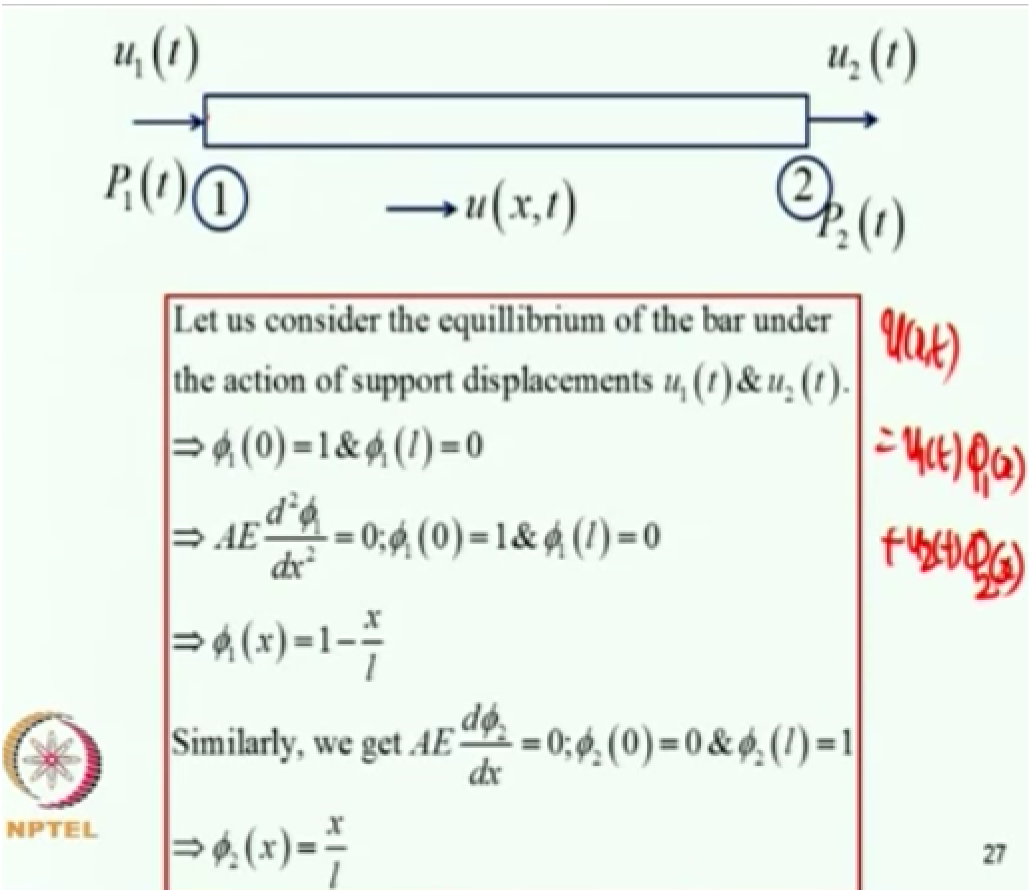
26

Now as a special case we will start now talking about simple elements will start with discussion on actually vibrating rod element, so I consider a rod element of length L axial rigidity AE damping C, and M is mass per unit length, so this is a cross-section and the coordinate system X is along the length of the bar, Y is as shown, and Z is shown here. Now this rod will be in a state of uniaxial state of stress, therefore the state of stress is given by sigma XX (x,t) that means all other stress components are 0, so this is standard assumption that we make in modeling actually vibrating rods, so strain is consequently given by this and the displacement is only function of X and T. Now if I use that now in the expression for strain energy we will get now the strain energy expression as this, kinetic energy is this, and the action integral is this.

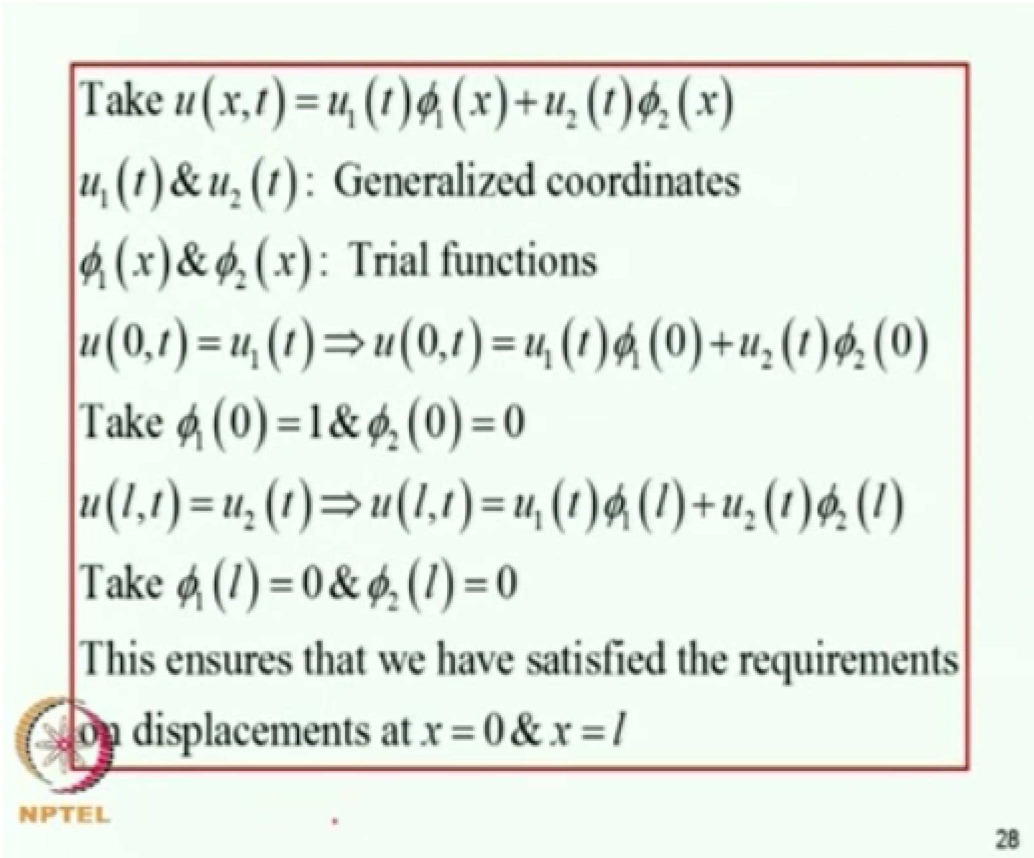


Now we want to now develop the finite element approximation to handle this element, so what we do is, we define these two ends as nodes so that I have shown as these numbers in a circle 1 and 2 are nodes, U1(t) and U2(t) are the values of the field variable, U(x,t) at node 1 and node 2 respectively, P1(t) and P2(t) are the stress result ends at one end.

Now we will consider the equilibrium of the bar under the action of support displacements U1 and U2, so suppose Phi 1 is the trial function that we are going to you know work with, we need Phi 1(0) that means what we are doing is we are going to approximate the field variable in terms of the value of the field variable at the nodes, that means we are interpolating,

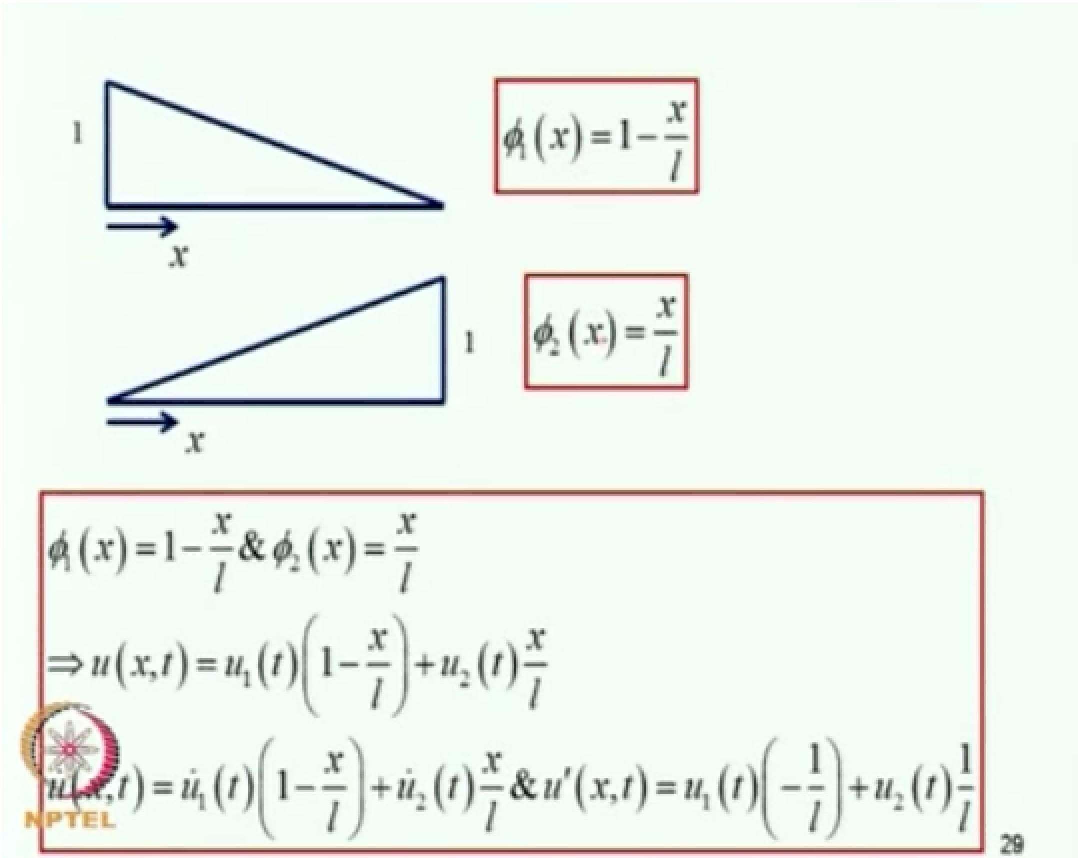


suppose I want a state of stress here, the field variable here, I will interpolate the values at these two nodes and this is a representation, and Phi 1(x) and Phi 2(x) are the trial functions. Now how to select these trial functions, that is the question we're asking, and what we do is we will assume Phi 1(0) is 1 and Phi 1(l) is 0, this is the equation under these displacements and we get Phi 1(x) is 1 - X by L. Similarly we get for the other displacements Phi 2(0) is 0, Phi 2(l) is 1, we get Phi 2(x) is X square, that means we are basically using linear interpolation functions. So what we are doing is we are taking U(x,t) as U1(t) into Phi 1(x) + U2(t) into Phi 2(x), the U1(t) and U2(t) are the generalized coordinates, we call Phi 1(x) and Phi 2(x) as trial functions.



So now U(0,t) is U1(t) because at X = 0 they prescribed displacement is U1(t), so if you use that U(0,t) is U1(t) Phi 1(0) +U2(t) Phi 2(0), so consequently I get Phi 1(0) is 1, Phi 2(0) is 0. Next U(l,t) is U2(t), so if I impose that condition I will get Phi 1(l) is 0 Phi 2(l) is 0. Now this ensures that if we select now Phi 1 and Phi 2 so that these four conditions are satisfied, this ensures that we have satisfied the requirements on displacements at X = 0 and X = L.

Now the previous slide I discussed on how to select this Phi 1 and Phi 2 under the required conditions that is this one and this one. So now we are ready with the interpolation functions which are 1 - X / L and X / L and then approximation for the field variable as shown here.



Now I can find the velocity field by differentiating this with respect to the time I will get U1 dot (t) into this U2 dot(t) into Phi 2(x), similarly I can get the derivative with respect to X this will be this. Now I can substitute this into my expression for strain energy and kinetic energy, the strain energy is given by 1/2 AE Dou U by Dou X whole square so if I now substitute that Dou U / Dou X as you have seen here it is U2 - U1 / L it is independent of X so I can quickly evaluate this.

$$V = \int_0^l \frac{1}{2} AE \left(\frac{\partial u}{\partial x} \right)^2 dx = \frac{AE}{2l} (u_2 - u_1)^2$$

$$T = \int_0^l \frac{1}{2} m \left\{ \dot{u}_1(t) \left(1 - \frac{x}{l} \right) + \dot{u}_2(t) \frac{x}{l} \right\}^2 dx = \frac{ml}{6} \left(\dot{u}_1^2 + \dot{u}_1 \dot{u}_2 + \dot{u}_2^2 \right)$$

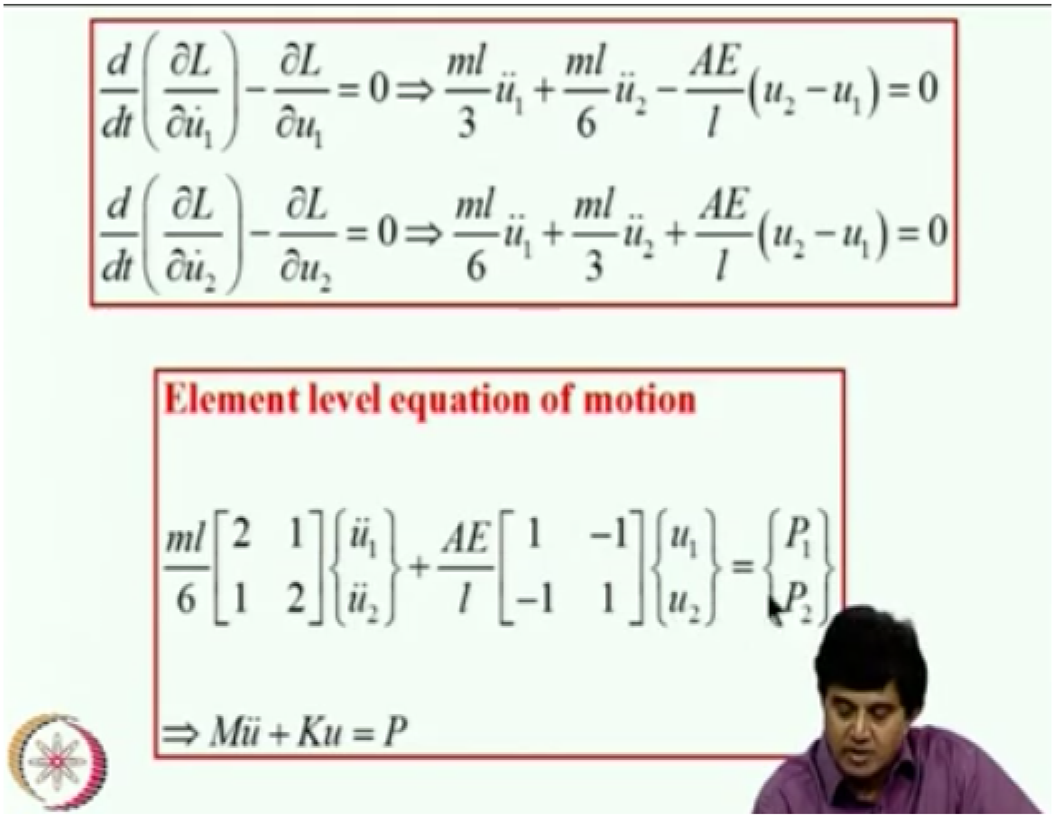
$$L = T - V = \frac{ml}{6} \left(\dot{u}_1^2 + \dot{u}_1 \dot{u}_2 + \dot{u}_2^2 \right) - \frac{AE}{2l} (u_2 - u_1)^2$$

$$\frac{\partial L}{\partial \dot{u}_1} = \frac{ml}{6} (2\dot{u}_1 + \dot{u}_2); \frac{d}{dt} \left(\frac{\partial L}{\partial \dot{u}_1} \right) = \frac{ml}{6} (2\ddot{u}_1 + \ddot{u}_2)$$

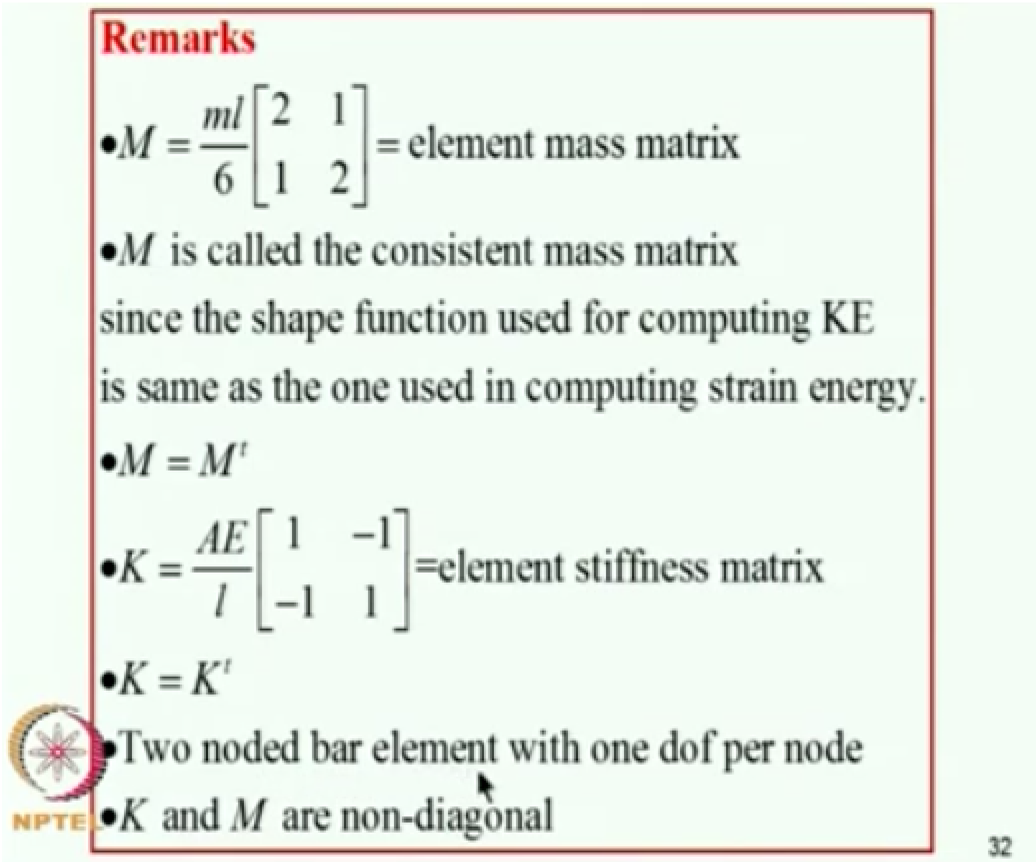
$$\frac{\partial L}{\partial \dot{u}_2} = \frac{ml}{6} (2\dot{u}_2 + \dot{u}_1); \frac{d}{dt} \left(\frac{\partial L}{\partial \dot{u}_2} \right) = \frac{ml}{6} (2\ddot{u}_2 + \ddot{u}_1)$$

$$\frac{\partial L}{\partial u_1} = \frac{AE}{l} (u_2 - u_1); \frac{\partial L}{\partial u_1} = -\frac{AE}{l} (u_2 - u_1)$$

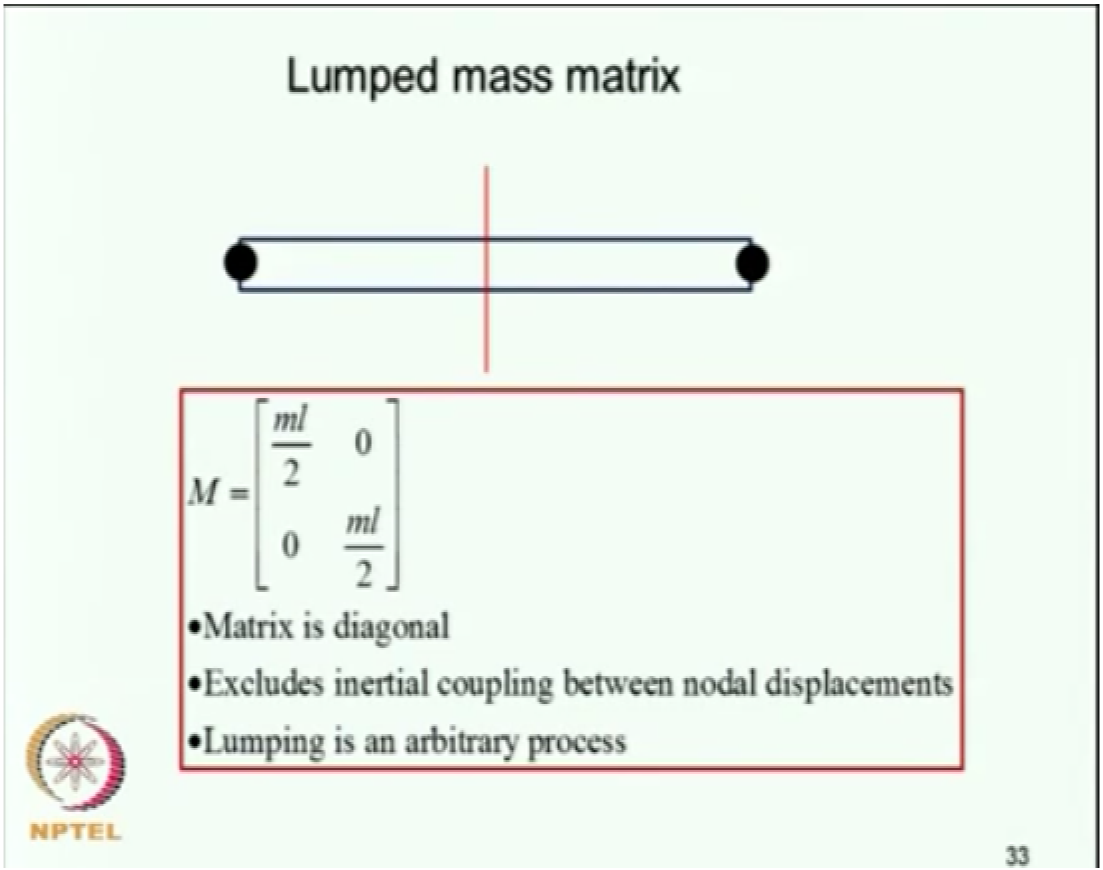
Next kinetic energy this is 1/2 M U dot square so U dot is U1(t) 1 - X / L U2(t) X / L whole square DX, now these are polynomials, simple polynomials so I can quickly integrate this, so if I do this I get this as the kinetic energy in terms of the two generalized coordinates, and this is the strain energy in terms of the generalized coordinates, so now I can form the Lagrangian, the Lagrangian is T – V, I get this, there are two degrees of freedom and I need to run the Lagrange's equation for U1 and U2 so that involves finding gradient with respect to U1 dot and U2 dot and substituting these terms we can easily derive all this and we get the equation



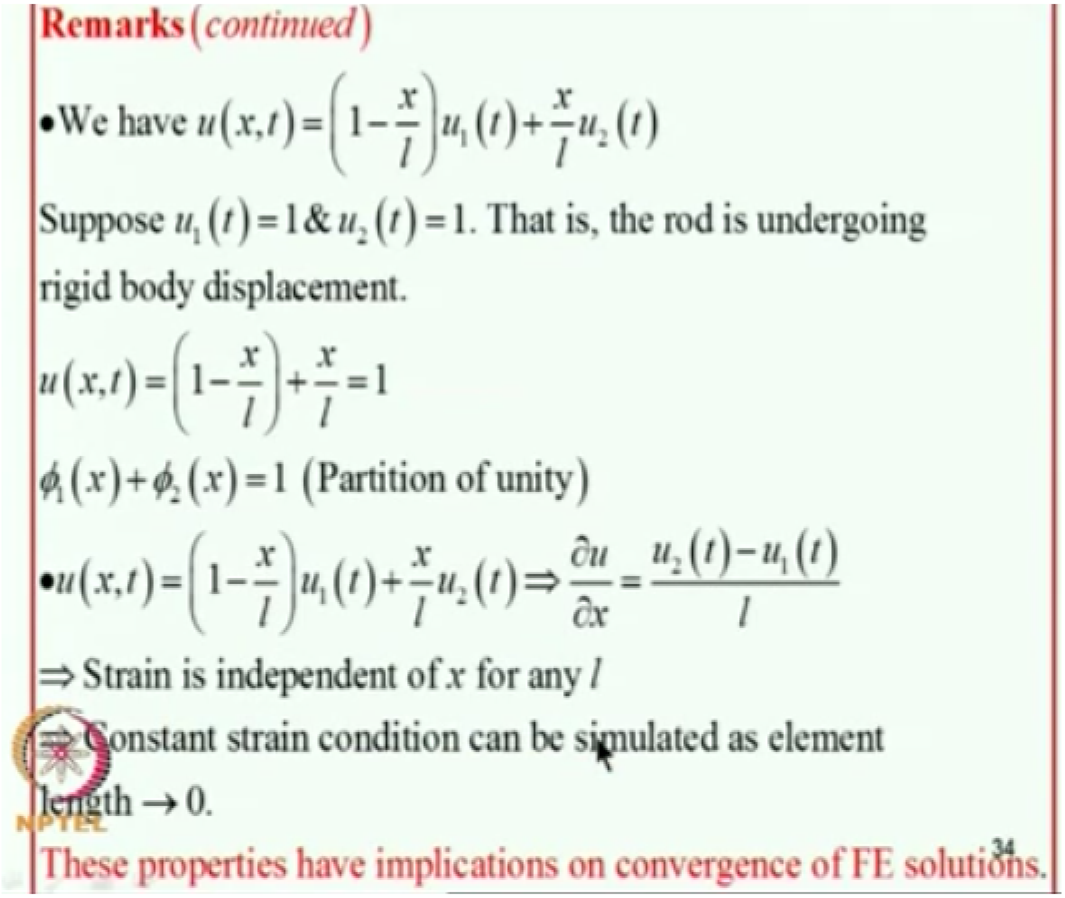
governing U1 to be this, equation governing U2 to be this, and now I can recast this in a matrix form I can write an acceleration vector U1 double dot + U2 double dot and this gets multiplied by the so-called mass matrix, and U1 and U2 which is a displacement vector which gets multiplied by the stiffness matrix, so I will come to this force vector in due course but we can believe that these are the nodal forces, so I get now the equation, element level equation of motion the form MU double dot + KU = P.



Now we can make few observations this M which is ML / 6 into this is known as element mass matrix, we also call this as the consistent mass matrix, why, what is the genesis of the phrase consistent here? It is consistent mass matrix because the shape function that we have used for computing the kinetic energy is same as the one used in computing strain energy, so that means in that sense it is consistent, it also implies that there is an alternative way of finding mass matrix which is not consistent I will come to that shortly and you can see that mass matrix is symmetric, and this quantity K AE/L into this matrix is an element stiffness matrix, it is symmetric and we call this element that we have developed as a two-noded bar element with one degree of freedom per node and two degree of freedom for the element, this K and M we should also notice that they are non-diagonal they are not diagonal matrices, so the nodal degrees of freedom are coupled in this representation.

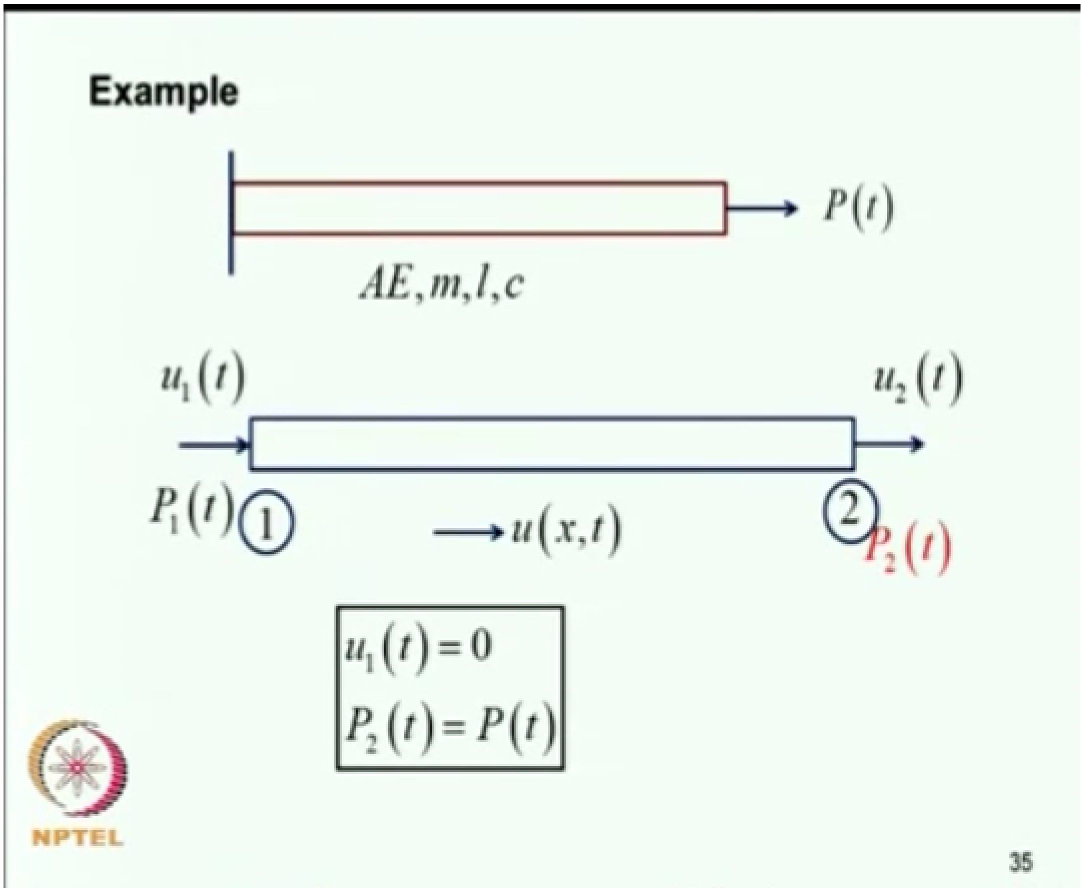


Now I talked about consistent mass matrix and an alternative way of deriving mass matrix used to use heuristic arguments and in an element half the mass is lumped at one end, and half the mass is lumped at the other end, if we do that we get mass matrix to be diagonal and this is called lumped mass matrix, this is diagonal but it excludes the inertial coupling between the nodal displacements and the earlier formulation showed that these quantities are not 0, so obviously that coupling has not been considered in this formulation, lumping is to some extent an arbitrary process in this case it may seem intuitively acceptable but when it comes to elements where there will be flexure as we will soon see lumping this mass matrix may not be straight forward.



Now we can make some more observations we have now this representation U(x,t) is 1 - X /L U1(t) + X / L U2(t), suppose both the nodes are given equal displacement of unity, that is the rod is undergoing a rigid body displacement in that case what should be U(x,t) it should be 1 because there should be no elastic deformation of the bar and that is consistent we do get that, so this means Phi 1(x) + Phi 2(x) is 1 and we call this property as partition of unity.

Now let us again take a look at the assumed representation for the field variable, now let us, this displacement let me compute the strain we get strained to be independent of X, okay so strain is independent of X for any L so that means constant strain condition can be simulated as element length goes to 0, so in a finite element model as element becomes smaller we expect that there won't be any strain variation within an element and that is simulated correctly here, so these two properties are of they have serious implications on convergence of finite element solutions and at some later stage we will revisit these issues again, but at this stage we can make this observations.



Now we are ready to tackle some quick examples so suppose there is a rod which is clamped at this end and it carries an axial load like this, so suppose I model this using one element we see that this bar is clamped here therefore U1(t) is 0 and at this end I am applying your external force P(t) therefore P2(t) should be P(t). Now if I put that into my governing equation for the element I get for U1 double dot is 0, U1 is 0, and at P1 that means at this end there will be an unknown reaction, okay, so that is R(t) which is not known on the other hand there is an applied force that remains as P(t) and U2 is the field variable, U2(t) is not known, so there are essentially two equations for two unknowns, one equation is for the unknown nodal displacement U2(t), other equation is for the unknown reaction at node 1,

$$\frac{ml}{6}\begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix}\begin{Bmatrix} \ddot{u}_1 = 0 \\ \ddot{u}_2 \end{Bmatrix} + \frac{AE}{l}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 = 0 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} P_1 = R(t) \\ P(t) \end{Bmatrix}$$

Two unknowns: $u_2(t)$ & $P_1 = R(t)$

$$\Rightarrow \frac{ml}{6}\ddot{u}_2 - \frac{AE}{l}u_2 = R(t)$$

$$\frac{ml}{3}\ddot{u}_2 + \frac{AE}{l}u_2 = P(t)$$

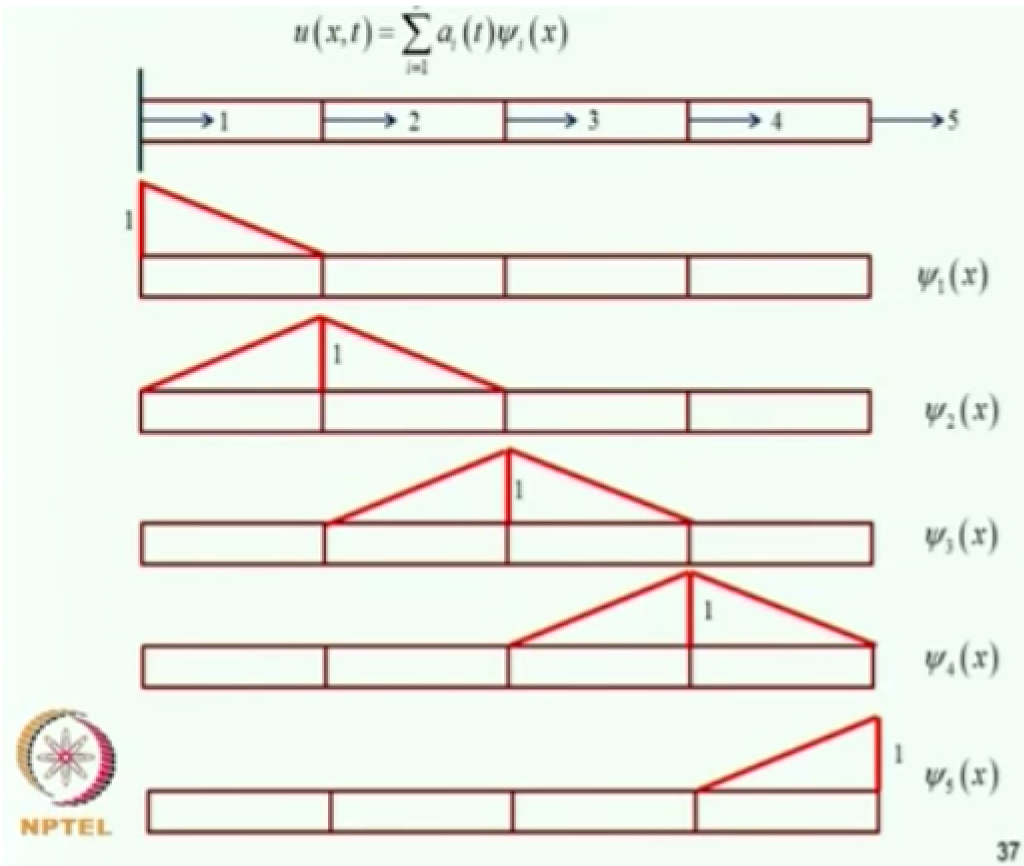
$$\ddot{u}_2 + \omega^2 u_2 = \frac{3P(t)}{ml};\ \omega = \sqrt{\frac{3AE}{ml^2}}$$

$$u(x,t) = \left(\frac{x}{l}\right)u_2(t)$$

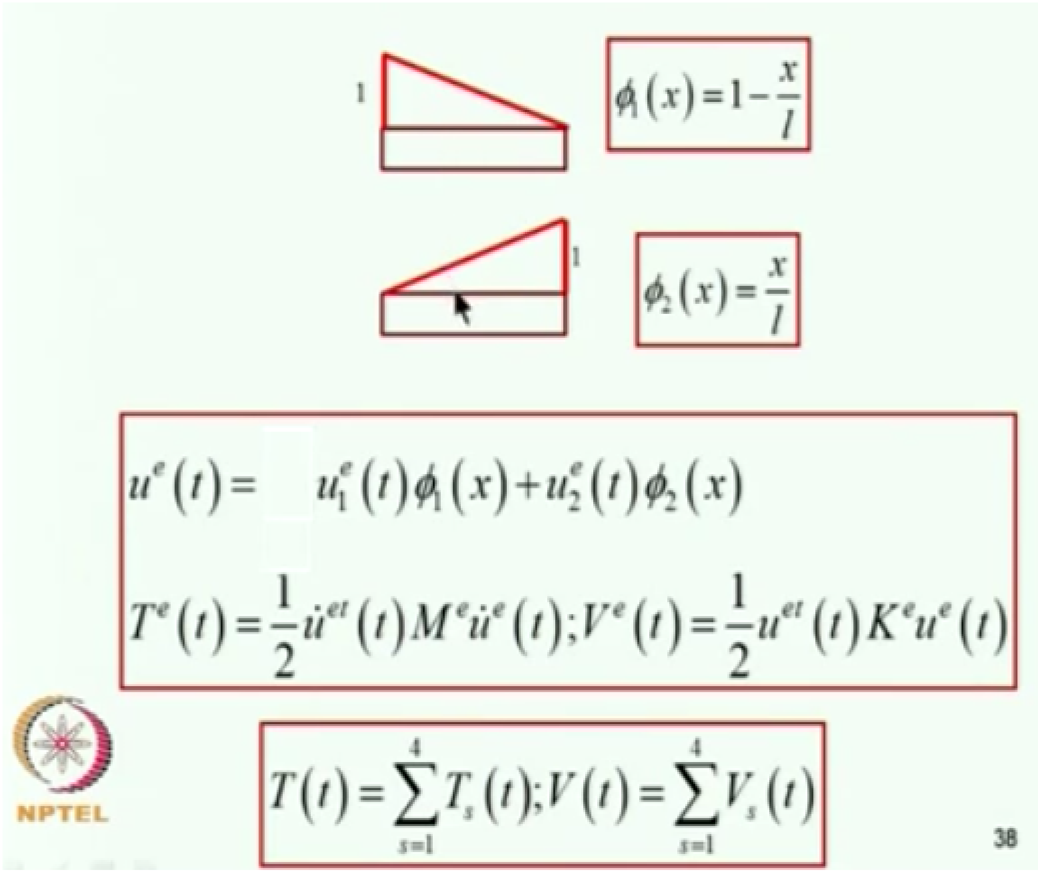
$$\varepsilon_{xx}(x,t) = \frac{u_2(t)}{l};\sigma_{xx}(x,t) = E\frac{u_2(t)}{l};\text{Axial thrust} = \frac{AE}{l}u_2(t)$$

36

so now if you write the second of these equations I get the governing equation for the unknown nodal displacement and sorry this is the equation for reaction, the first equation I get reaction in terms of U2. Now the equation for U2 itself can be obtained by considering the second row here and we get this equation, so if you carefully look at the structure of these equations it follows that first you need to find U2(t) using this equation and then substitute back into this to get the reaction, so if you look at this equation now I get, this is a single degree of freedom approximation there is only one generalized coordinate so I get this an approximation to the natural frequency according to this model is this and U(x,t) is X /L U2(t), U2(t) is solution to this, and the strain is given by this, the stress is given by, the axial thrust is given by this, so we have found out now the displacement, strain, stress, stress resultant, and the reaction so this framework enables you to do that.

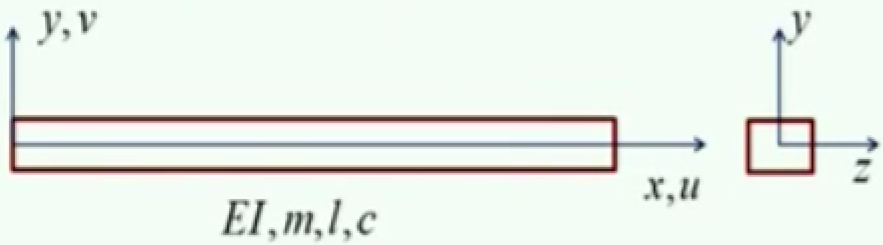


Now let's consider the situation to understand the nature of approximation that we are making it is useful to consider a bar as shown here and suppose I make 5 divisions here, see the statement I made was in finite element method we use piecewise polynomials to approximate the behavior of the field variable for the entire structure, so let us make now 4 sub domains as shown here and within and the U(x,t) is the unknown displacement field all over this, for X varying from 0 to L and this is written like this, so when viewed like this what are the shape function, what are the trial functions? The first trial function in our model will be this, this has nonzero value only for the first element, and it is zero here it is linearly varying within the second element, the second function Phi 2(x) is in terms of these three nodal coordinates and this is linearly varying as shown here, so similarly the other elements are shown here. So what we have done just now as we have considered the behavior of only one element and got the equation of motion for this element, but we need to actually study a built-up structure like this so we need to find equations for A1, A2, A3, A5, now we will come to that in due course but I want you to make a note of some simple things, see what we are doing is at the element level we are using these as a trial functions, but at the built up structure level these are the trial functions, but they are all related.



Now how do we see that suppose I take the element E, the way I am interpolating UE(t) is through these 5 functions it is in terms of the element nodal coordinates and the element level trial functions, so this is the kinetic energy for the element, and this is the potential energy for the element, so the superscript E here refers to the element E. Now that means for each element I can do this exercise of computing the strain energy and kinetic energy using the local interpolation functions and when it comes to computing the total strain energy and the kinetic energy I can add contribution to this from each of these elements as shown here, okay, so this is what we will be doing when it comes to question of assembling the matrices.

Euler-Bernoulli beam element



- Bending in the *x-y* plane
- *x*-axis coincides with the centroidal axis of the beam
- No coupling exists between bending and torsion

$$u(x,y,t) = -y\frac{\partial v}{\partial x}; v \equiv v(x,t)$$

$$\varepsilon_{xx} = \frac{\partial u}{\partial x} = -y\frac{\partial^2 v}{\partial x^2}$$

$$2\varepsilon_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} = -\frac{\partial v}{\partial x} + \frac{\partial v}{\partial x} = 0$$

NPTEL

39

Now we can extend this logic now to problem of Euler Bernoulli beam, so I will begin discussing this and we will continue with this in the next class, so the problem description is as shown here, this is one beam element its characterized in terms of flexural rigidity EI mass per unit length M, length L and C is the damping properties, and this is the X direction in which the displacement field is U, U(x,t) and this is the Y direction where displacement field is V, now we are therefore considering bending in XY plane, we are making an assumption that the X axis coincides with the centroidal axis of the beam, and we also assume that there is no coupling between bending and torsion, this cross section is symmetric therefore there is no coupling between bending and torsion.

So if we now follow the tenets of Euler Bernoulli beam hypothesis U(x,y,t) is given by this because plane sections which are normal to the neutral axis before bending remain plane and normal to the neutral axis after bending, so if you make those assumptions if you use those, if you use that assumption we can write displacement like this. Now V is a function of X,T from this I can get strains, this is the normal strain this is the shear strain, shear strain turns out to be 0 in this model, so I can compute the strain energy that we need to use sigma XX and epsilon XX and if you use this is an integral over volume, so now I will integrate over X and area of cross section separately so using the assumptions that we have made we get for epsilon XX whole square - Y Dou square V by Dou X square whole square and that remains here, and the remaining integral that is integral AY square D, A is nothing but the moment of inertia about the Z axis so I write this equation, so this is a strain energy stored in the beam due to bending.

$$V = \frac{1}{2}\int_{V} \sigma_{xx}\varepsilon_{xx}\,dx$$
$$= \frac{1}{2}\int_0^L\int_A E\varepsilon_{xx}^2\,dx\,dA$$
$$= \frac{1}{2}\int_0^L\int_A E\left(-y\frac{\partial^2 v}{\partial x^2}\right)^2 dx\,dA$$
$$= \frac{1}{2}\int_0^L EI_z\left(\frac{\partial^2 v}{\partial x^2}\right)^2 dx \text{ with } I_z = \int_A y^2\,dA$$

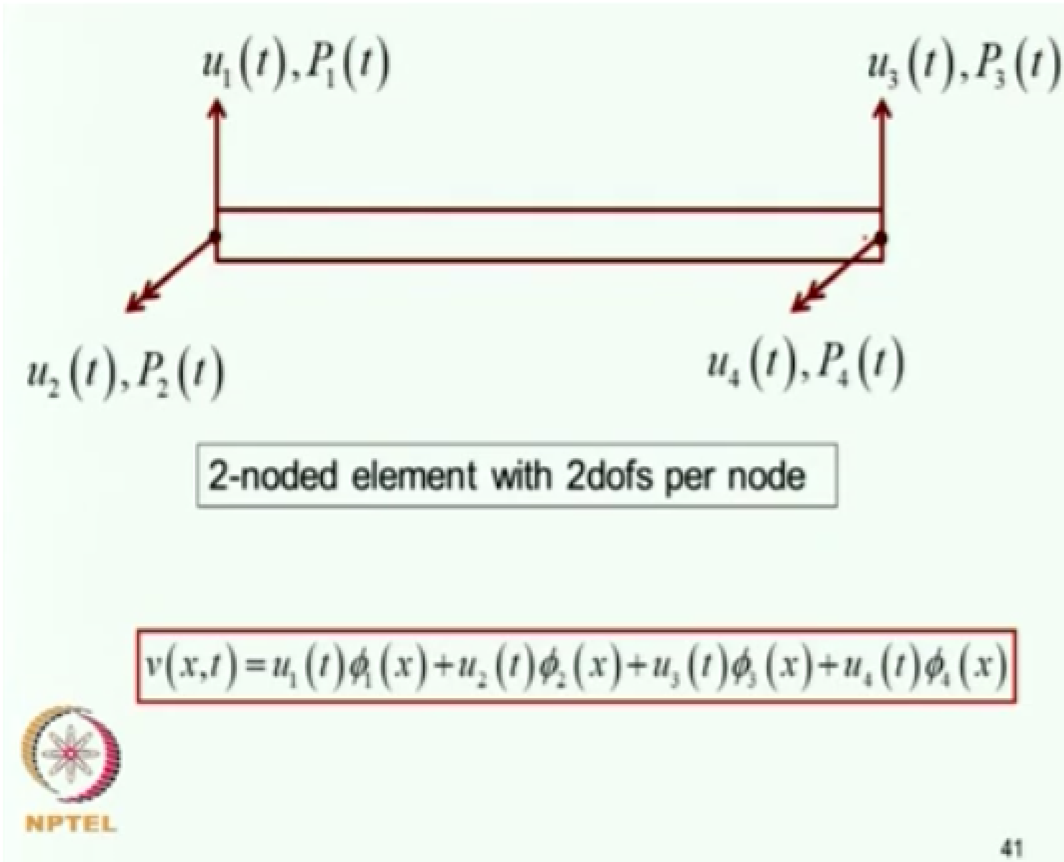
$$T = \frac{1}{2}\int_0^L\int_A \rho\dot{v}^2(x,t)\,dx\,dA$$
$$= \frac{1}{2}\int_0^L m\dot{v}^2(x,t)\,dx \text{ with } m = \int_A \rho\,dA$$

$$L = T(t) - V(t)$$
$$= \frac{1}{2}\int_0^L m\dot{v}^2(x,t)\,dx - \frac{1}{2}\int_0^L EI_z\left(\frac{\partial^2 v}{\partial x^2}\right)^2 dx$$

NPTEL

40

The kinetic energy is integral over volume Rho V dot square DV, and again the DV if I write it as DX into DA, I can write M as integral A Rho DA, this will be the mass per unit length so I will get the familiar equation for the kinetic energy. So Lagrangian for the system is T - V and this is the expression for the Lagrangian.



So in the next class what we will do is, we will consider the beam element to have two nodes and at each node we will consider field variables to be the translation and rotation, this double arrow notation is for rotations, so this is rotation using right-hand rule convention so U2 and U4 are rotations, U1 and U3 are translation, so collectively we call them as displacements, so associated with them there are stress resultants so we assume the field variable which is V(x,t) which is the transverse displacement of the beam in terms of the nodal translations and rotations, and we need to therefore construct a solution we need to construct 4 interpolation functions to achieve this, so we will see how this we can do in the next lecture.

Programme Assistance

Guruprakash P
Dipali K Salokhe

Technical Supervision
B K A N Singh
Gururaj Kadloor

Indian Institute of Science
Bangalore