Stability of structure

Prof: Sudib Kumar Mishra

Department of Civil Engineering

IIT KANPUR

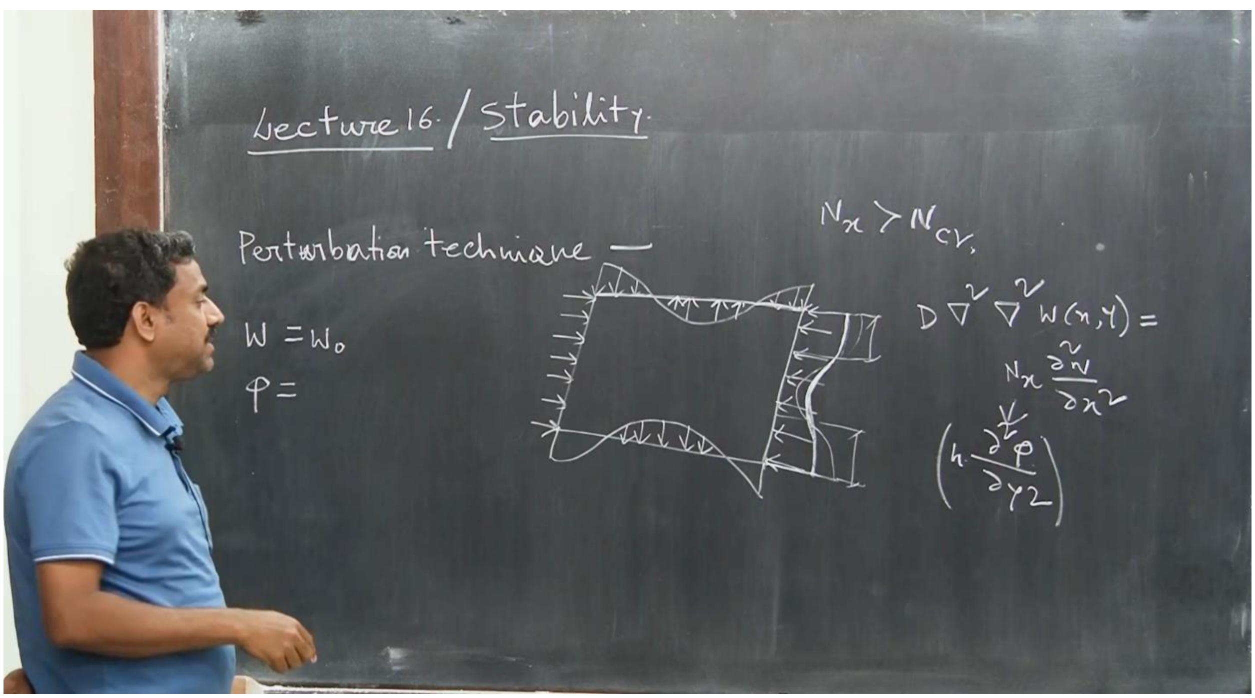
WEEK-08

Lecture 16: Eigen Buckling Analysis of Frame and Dynamic Stability

Welcome to the 16th lecture on structural stability. So, let us briefly recall what we have covered in the previous class. We're discussing postcritical analysis of plates. So post-critical analysis means that we are studying the load-carrying behavior of plates beyond buckling. And then we have seen that, in order to do that, we have to consider the in-plane distribution of forces. So, in-plane forces were no longer treated as constant; instead, we allowed them to vary, thereby allowing them to redistribute, and then we have solved it. The solution was a little ad hoc, but it was provided; it is a very simple solution, a little ad hoc, you know, but it was given by Volmir, and it essentially captured the physics. So, we have successfully demonstrated the redistribution of stresses. We have seen that just on the verge of buckling, it is subjected to uniform compression, right, and in the pre-buckling regime. Also, it is uniform compression, but then, as soon as it buckles, we go increasingly, you know, compress it beyond the critical load. So, here we are going beyond critical. Okay. So, then what happened? this force you know start redistributing right. So, in the end, you know the end portion starts carrying more load, and you know, whereas the middle portion's load basically reduces, right? So, this kind of redistribution occurs, and as you go more and more, you know this will carry more load and this will carry less and less load or something like that. Okay. So, at the end, we may assume that the essential load is carried by the two end strips, right? And from there, we have been told that, There is a concept like equivalence, you know, the weight concept for that has been adopted in American and Canadian codes, okay? equivalent with BE okay and then, what happened along the transverse direction, there is also redistribution occur and there what happened that end strip you know, Earlier in the pre-buckling regime or events on the verge of buckling, there was no compression no forces in the other direction. But, when in the pre-buckling regime, forces in the other direction also develop. Of course, for redistribution to occur, these plates at the end, in the other direction, and the end of the ages in the other direction, must be adequately held in the in-plane direction. You have to okay, that's what otherwise, if it is not restrained adequately, it will not develop; it will not allow redistribution to occur, right? So, that is very important. So, the redistribution of in-plane forces largely depends on, you know, the edge conditions at the upper end, and then the end strips are still carrying compression loads, but in between, they carry tensile forces. Okay. So, that's basically what happens, and then these tensile forces allow the stiffening of the plate. You see that we have all noticed the erosion in stiffness due to in-plane forces, right? When it is under compression, we have seen how it erodes the out-of-plane stiffness. However, tensile force also leads to tensile stiffening. So, that's essentially what happens. So, what we have discussed and the solution we have followed was given by Volmir around the 1960s, right? so, little simple solution, but more rigor solution can also to be obtained. because that equation if you can recall, the equation for plate buckling was

$$\nabla^2\nabla^2 w(x,y) = n_x \frac{d^2 w}{dx^2}$$

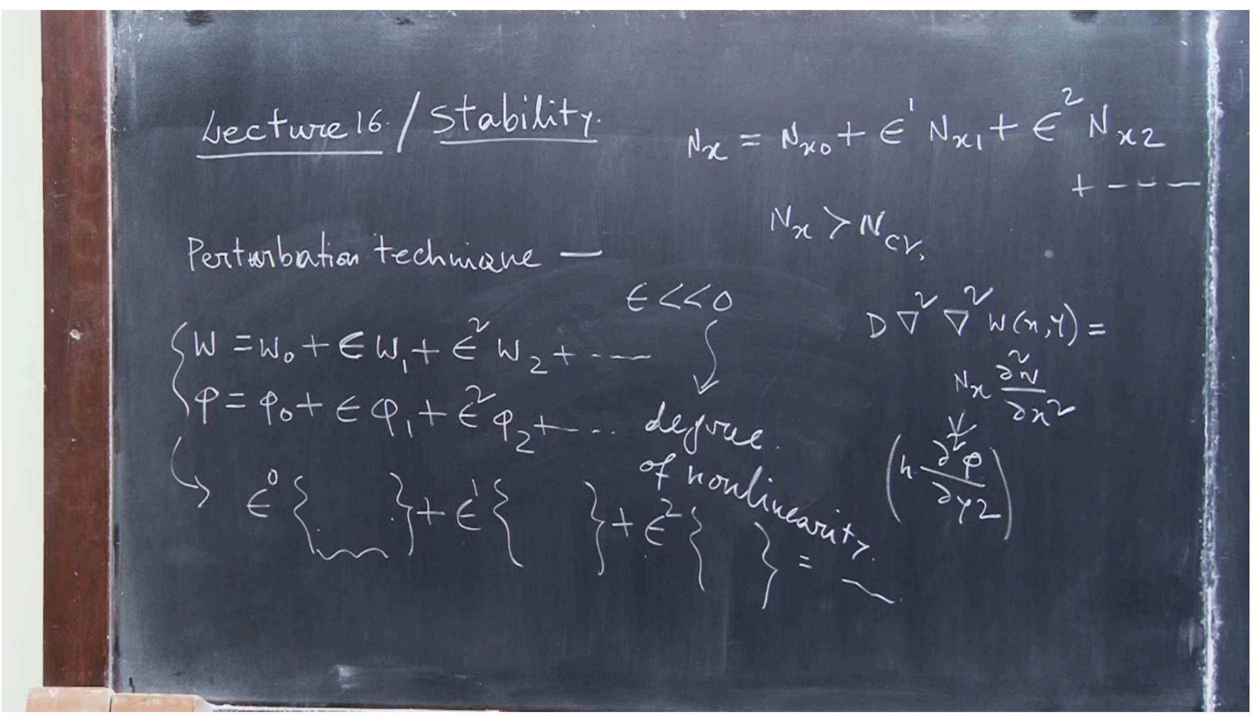
right and now, we took, as we express in terms of the stress function, $\frac{d^2\phi}{dy^2}$, and that is multiplied by the thickness h, right? And then the compatibility equation we have obtained in terms of, you know, w and x, okay? So, there are two equations and two unknowns, w and ϕ, right?



So, here we'll see that this equation is, of course, nonlinear, right? So, nonlinear partial differential equations or nonlinear differential equations can be solved using the perturbation technique, okay? So, the perturbation technique is a method for solving nonlinear equations. What we do in the nonlinear case is assume whatever the field variable is, for example w or ϕ. We assume this: two occur in asymptotic series. Okay. So, what we assume is W expressed using an asymptotic series,

$$W = W_0 + \epsilon W_1 + \epsilon^2 W_2 + \cdots$$

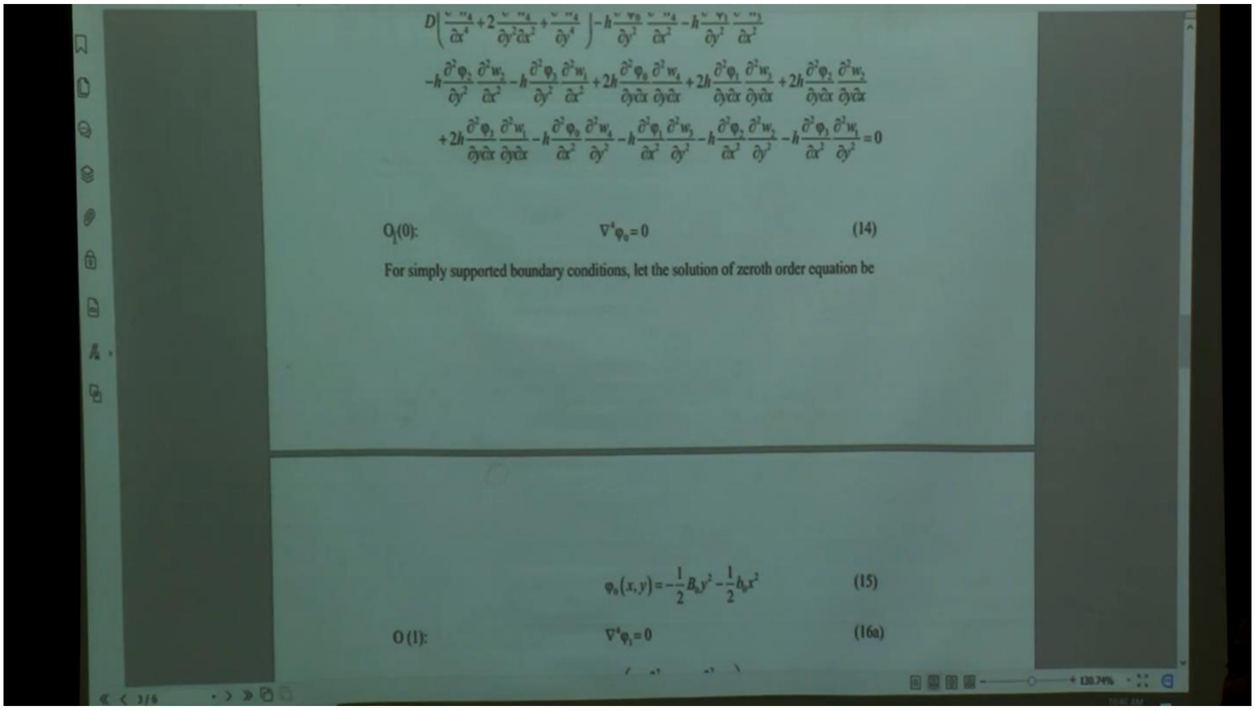
and something like that, okay. So, ϵ is a small parameter that controls the nonlinear effects. ϵ may be explicitly derived from the equation, or it may not; some of this asymptotic expansion is valid when ϵ is small; ϵ is a very small quantity, much less than zero. In case any equations you have, there are several variants of attribution perturbation techniques that are not a problem in nonlinear dynamics, okay? You consider the damping oscillator. So, there you will see that, yes, one linear state, one car linear state perturbation. Because there is not only the field variable, but also the frequency that need to be perturbed as well. So, this is similar to that. So, if you people have little idea about that, then it will be easier to understand. So ϵ is a very small parameter. It basically controls the degree of nonlinearity. It can be directly related to the degree of nonlinearity in the system of linearity.



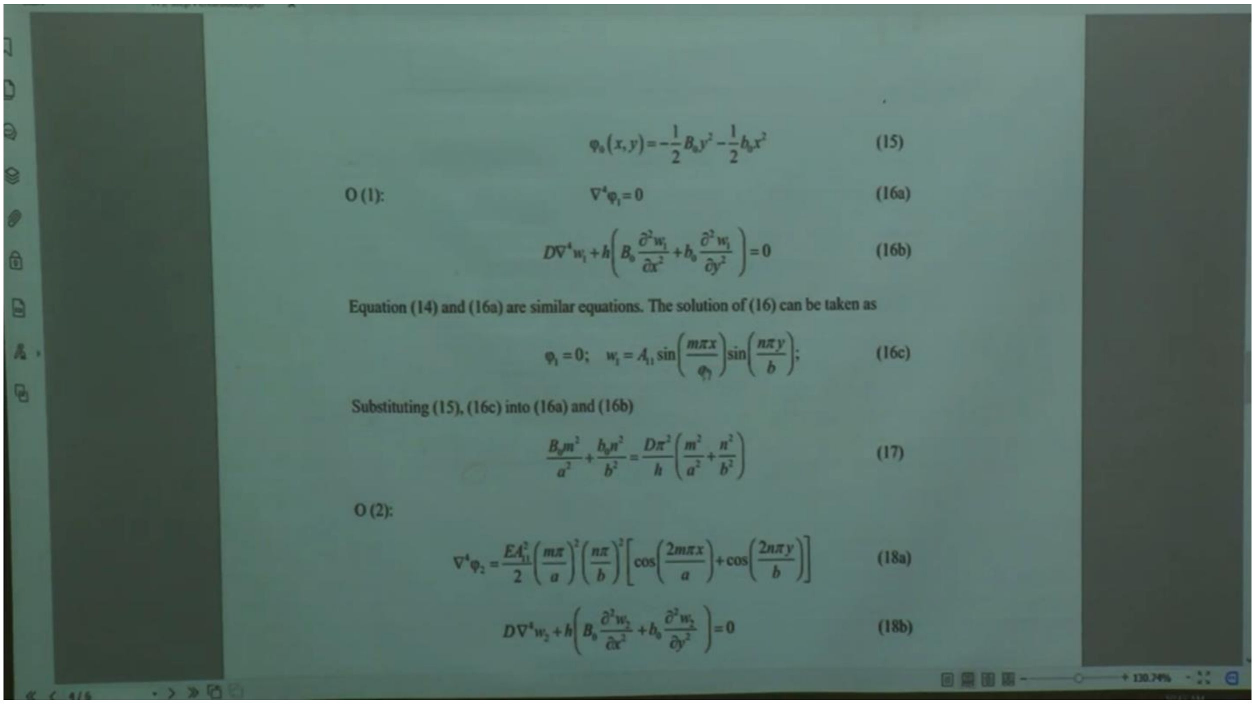
I'm not going into details, but I want to give you a little idea. Similarly, the other field variable can also be expressed as $\phi = \phi_0 + \epsilon\phi_1 + \epsilon^2\phi_2 + \cdots$ Please note that you may wonder whether the series is convergent or divergent. Do not worry about convergence and divergence. These are asymptotic; we don't care about the, you know, but yes, being ϵ small, you know, high terms will increasingly contribute less and less. So, it's that way; it is basically convergent effectively, right? Then we substitute in the differential equation, and then what happened is we equate the terms associated. So, when you take this kind of field variable and substitute it into the governing equation or linear equation, what will you see? Then you basically equate, so you rewrite the equations in terms of something to the power of zero. One equation plus ϵ to the power of one equation plus ϵ to the power of this equation is equal to something, right? The right-hand side also has a similar equation. So, then equate the coefficients associated with this varying power of ϵ. Do you understand? Thereby, you will see a nonlinear equation can be approximated as a series of linearized equations, and then at the end, once you find out w_1 and w_2, you sum them up, and then you will get the solution. Understand that, prior to this, if you don't know what ϵ is, later you can use the boundary conditions to find out what ϵ is. But in many instances, ϵ, you can explicitly determine this and that through scaling of the equation. Now, because we have already expressed the in-plane forces in terms of a stress function. that's what, we can treat ϵ sigma you know the stress function as also fake variable. But please note that, there are perturbation technique in which, we don't require expressing N_x in terms of a stress function; rather, we can directly perturb N_x. So, you will see that N_x can also be expressed as

$$N_x = N_{x0} + \epsilon^1 N_{x1} + \epsilon^2 N_{x2} + \cdots$$

So, this kind of perturbation is also valid, and you do not need to take many terms. This is the first-order equation. This will give a second-order equation. First, this is a zero-order equation. This is first-order. This is second-order. Okay, that way you can also do it. But here I will just show it once, quickly. We'll go through this a little; it involves lots of algebra. That's what I don't do, but I will just show you so that you have a little idea about it. So please look at this screen. I will not go into detail. So, here I'm referring to a two-stage partition because this is a specialized technique, but essentially it will give you the feel. So, we have used it. One of my previous PhD students, who worked on this, derived all of this. Now, all these equations you already know, right? Because we have derived this, right? So, ultimately, we arrived at these governing equations, right? These are the governing equations; right along with that, we get the boundary conditions. So, at this edge WW, the simply supported boundary conditions are right, and then this equation is P. If P is the axial force being applied, right? Then you can integrate over the in-plane forces, over the H; then you will get the equilibrium equation right. And then, for age y0, you can also write this equilibrium boundary condition right. Simply supported, similarly there are no in-plane forces in the y direction. So, integrate and make it zero. But in the other direction, there will be in-plane shear due to the restraint. Now you can also express the unit shortening. You integrate the strain over the length, and then you divide it by the edge length to get the unit shortening per unit width for both the X and Y directions. Right? Then, we assume the solutions W and ϕ. There are two field variables, W and ϕ. Right? We assume ϵ is a small parameter controlling the nonlinearity in the system. So, ϵ to the power of n. Please note that ϵ to the power of n, zero power means anything to the power of zero is 1, right? ϵW_1 plus ϵ does not require you to have a two-term approximation; it is good enough for this, okay? Sometimes you take a three-term approximation; two-term, zero-third order; first order; and second order are good enough. Then you substitute, and then you get the linear edge equation. So, you see these are the linearized equations in terms of ϕ_0 and ϕ_1, then these are the linearized equations of ϕ_2 and this is in terms of ϕ_3. Of course, here it is a homogeneous equation, but then Φ_2 and Φ_3 are in terms of w_1 and w_2. So, you see that if you decide Φ_0, then you can get ϕ_2, and then you have to assume sum w_1. Then you just substitute that, and you will get Φ_2. Assuming w_1 and Φ_2, and something w_2, you will get all these linearized equations, which are easy to solve. Because we know how to solve linearized equations, in fact, nowadays, with the advent of symbolic computation in Maple and Mathematica, you can do these things very effectively. That's what we did, because otherwise the equations become very complicated. I have seen equations, you know, for using perturbation that take, I mean, several hours to do these things, okay. But it's very beautiful, actually, huh? So, then you see the linearized equation. I'm writing all these things, you know, all the equations, okay, in terms of w_1, w_2, w_0, w_1, w_2. So, I will just see that. So, what will this be? This is a zero-order equation. This capital "O" refers to order, okay, zeroth-order equation. You know ∇^2 ∇^2 P 0 0, ∇^4. It's not the correct thing to write, but people write it this way, ∇^2, ∇^2, rather okay two Atlassian. So then, for the solution for the zero order, this can be expressed as f_0 is something like this, okay, then f_1 and then w_1 in the first order equation in terms of, you know, f_1 and w_1.



Okay. Then you will see, these equations you know, the solution will be ϕ_0 is there. So, if you take ϕ_0 and substitute, you will see that ϕ_1 will be for these equations; it can be taken as identically zero. You do not need to have ϕ_1. Okay, that will satisfy this because this becomes identically zero. And then w_1, we can assume some kind of, you know, because it is a linearized equation. sign sign, So, you know by satisfying the boundary conditions assume the sinusoidal right, as you double Fourier series right. And then you substitute this equation and this equation, and you know ϕ_0 in (16B), then you will get the unknown coefficients, right? Similarly, second-order equation. Now you will get the equation for the ϕ_2 stress function; you know the second-order approximation for the stress function, right, and the second-order approximation for the w_2, right? You see another two equations in terms of the Laplacian and the right-hand side; whatever you have obtained, you know whatever is in w_2, or you know whatever term you have obtained from the previous equations. Here once again for 18, you see one of the equations, ϕ_2, that if you solve this equation, because this is a linear equation, it can be expressed in terms of this.



$$\varphi_0(x,y) = -\frac{1}{2}B_0 y^2 - \frac{1}{2}b_0 x^2 \quad (15)$$

O (1):

$$\nabla^4 \varphi_1 = 0 \quad (16a)$$

$$D\nabla^4 w_1 + h\left(B_0 \frac{\partial^2 w_1}{\partial x^2} + b_0 \frac{\partial^2 w_1}{\partial y^2}\right) = 0 \quad (16b)$$

Equation (14) and (16a) are similar equations. The solution of (16) can be taken as

$$\varphi_1 = 0; \quad w_1 = A_{11} \sin\left(\frac{m\pi x}{\varphi_7}\right) \sin\left(\frac{n\pi y}{b}\right); \quad (16c)$$

Substituting (15), (16c) into (16a) and (16b)

$$\frac{B_0 m^2}{a^2} + \frac{b_0 n^2}{b^2} = \frac{D\pi^2}{h}\left(\frac{m^2}{a^2} + \frac{n^2}{b^2}\right) \quad (17)$$

O (2):

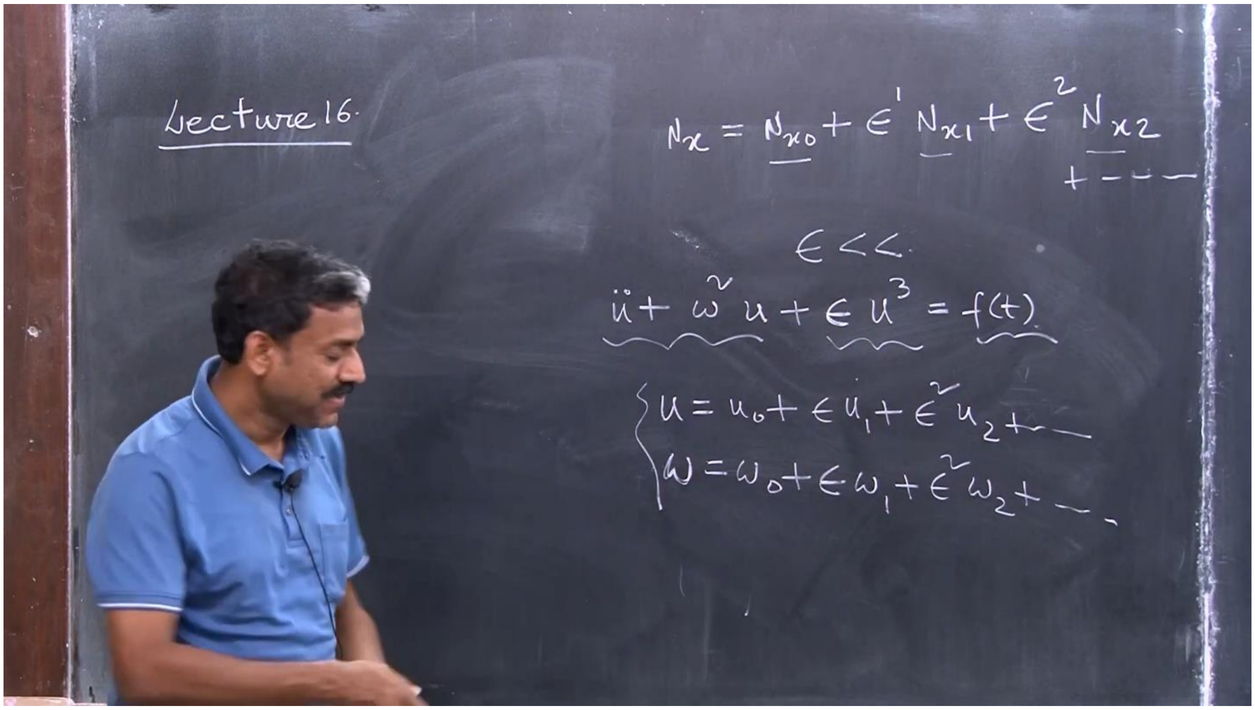
$$\nabla^4 \varphi_2 = \frac{EA_{11}^2}{2}\left(\frac{m\pi}{a}\right)^2 \left(\frac{n\pi}{b}\right)^2 \left[\cos\left(\frac{2m\pi x}{a}\right) + \cos\left(\frac{2n\pi y}{b}\right)\right] \quad (18a)$$

$$D\nabla^4 w_2 + h\left(B_0 \frac{\partial^2 w_2}{\partial x^2} + b_0 \frac{\partial^2 w_2}{\partial y^2}\right) = 0 \quad (18b)$$

But once again, it is found that w_2 can be identically satisfied with this kind of approximation. So, w_2 is zero. Then you substitute this expression into this equation 18B and then into this equation, and from there you will get the unknown coefficients. Okay. Now you take the third-order equation, third order similarly ϕ_3, and then you know w_3. Okay, once again, you see that it is changing; in one approximation, P_1 is zero, and you know W_2 is some expression. In another order of approximation, W is non-zero, but P_3 is zero. Why does this thing happen? I can tell you an analogy because this is happening because of the symmetry and anti-symmetry of the mode, okay? This can be, I mean, I am not going into details, but I can recall when we were doing these things. You know, there is a very beautiful physics behind this, okay? Because some displacement field or some field variable is symmetric, all the standard symmetric terms will be zero. Do you understand? So, that is the reason anyway. Then you know with this kind of progression, we can get all these things and finally we'll get the asymptotic solution. Why asymptotic? Because you know, it is associated with ϵ^0 ϵ^2 and things. You please see that, in ϕ, you see that ϕ is what? ϕ is an even function. Why is it an even function? Because it is appearing as ξ^2, it is even in terms of ξ. You see ξ^0 and ξ^2, whether W contains the terms $\xi\epsilon$ and ϵ^3. So, one basically contains the odd term of ϵ, the stress function, and the even term of the stress function, while the other contains the odd term in terms of ϵ. So, that's a difference between ϕ and w. So, the approximation of some term, some approximation you know ϵ, whatever is associated with ϵ in terms of the same term in

ϕ is equal to zero. So, then this is why we call this the asymptotic solution: because it is coming from the asymptotic series, right? And then whatever unknown coefficients you have, you can enforce the boundary condition. So, in the equilibrium condition, you see that from here you will get the equation; then, if you enforce this boundary condition, you will get P, okay? In sorting, you can also find it out; now, from here, what is ϵ that we don't know? ϵ can be related to deflection; you see that maximum deflection is given by this. If you substitute in this equation, $w_{\max}$ will be; why $w_{\max}$? Because when you find your out-of-plane deflection, you put $x = a/2$, $y = b/2$. $w_{\max}$ is equal to ϵA_{11} plus $\epsilon k a_1^3 a_{31}$, and if you invert this relationship, you can find out ϵa_1 is equal to this. Now, you see what ϵ is. ϵ is nothing but $w_{\max}$ plus something times $w_{\max}^3$ now. So, do you understand from that how ϵ controlled the nonlinearity? If ϵ is directly related to out-of-plane deflection, more out-of-plane deflection will mobilize, causing more nonlinearity. Because it is pushing, it is mobilizing more postcritical redistribution, and you can clearly see that from here, because ϵ is $w_{\max}$ plus something into $w_{\max}^3$. Now $W_{\max}$ is a small quantity. If you write it's a small, the deflection will be much smaller compared to the in-plane forces. So, the smallness of ϵ can also be justified from these things, from expression 29B, right? Clear. So, ϵ is basically a variable that controls the nonlinearity, in this case controlling the degree of post-criticality, if you think about it. So, it can be, I don't know whether it can be considered a kind of order parameter. This concept is used in physics; it's a generalized term. For system behavior, how it changes is important. It can be described as a term that controls nonlinearity, and another thing is that it is small. Smallness can also be justified in many instances, so that is the way. I don't mean I will send this to you; we can go through that you do not essentially work it out. But have, it has with you the concept that how these equations are solved rigorous solution. Because whatever we have done using the Volmir solution was a little simplified and ad hoc, if you want to do research using the Volmir solution, we will never be able to do anything. You have to have a very good reason because science and work in this area have progressed significantly. So, you want to contribute anything new solution, then you have to learn perturbation, and as people in mechanical engineering are very conversant with that, they do it, while in civil engineering, we are a little behind, okay? This concept is not only applicable here, but please note that it is also equally applicable in non-linear dynamics. So, in the case of a Duffing oscillator, when you try to find a close solution, you will see that we use this approximation, okay? I will say what we do, basically. Okay, fine. So, what do you do you know, let me tell you you see. so here the field variables are

w and ϕ, that's what we Perturb like this. But if we don't use this, if we do not use this, you know ϕ, we don't want to express it in terms of the stress function. Then we can directly solve the equation in terms of N_x. And please note that we can have an asymptotic expansion.

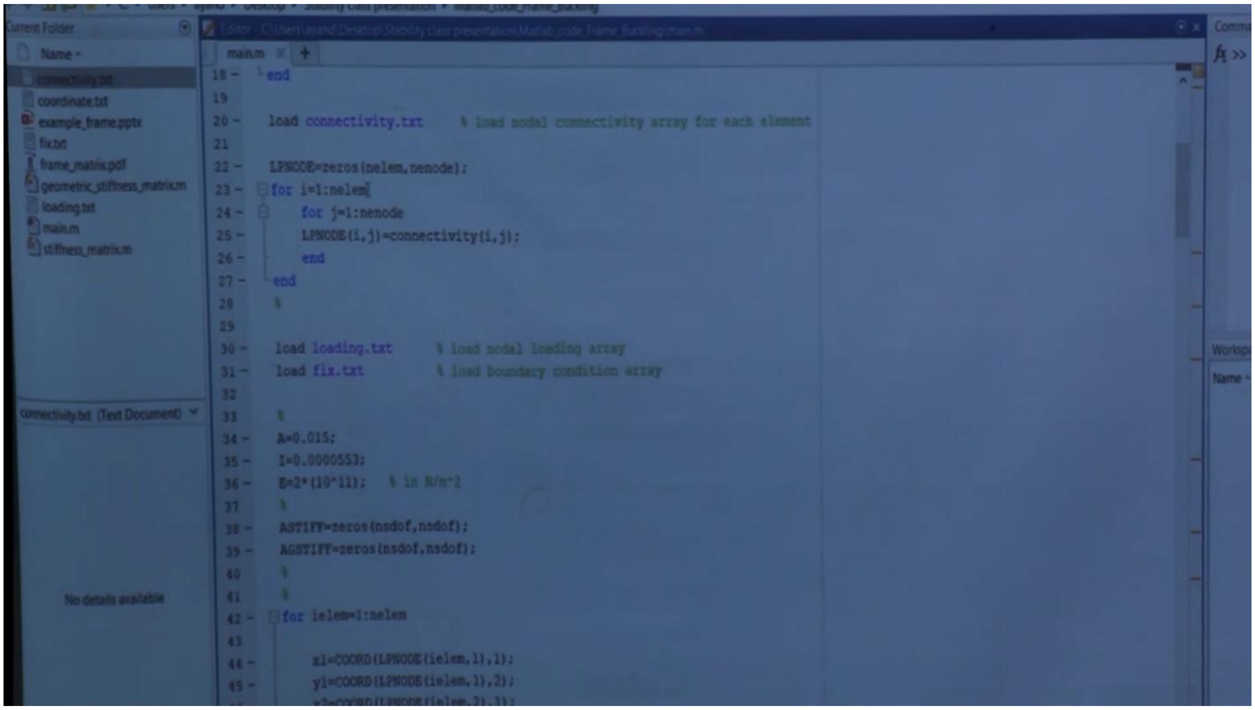


There is literature in which this kind of expansion has been substituted, okay, and there you also solve it. So, you see that the zero-th order equation N_{x0} is essentially what will give you the pre-critical scenario, whatever is happening in the pre-buckling situation. Then N_{x1} N_{x2} is the term that is basically contributing, you know, describing whatever redistribution is occurring, and there is this inflame change occurring in the inflame forces, okay. So, in dynamics, a similar analogy is something like

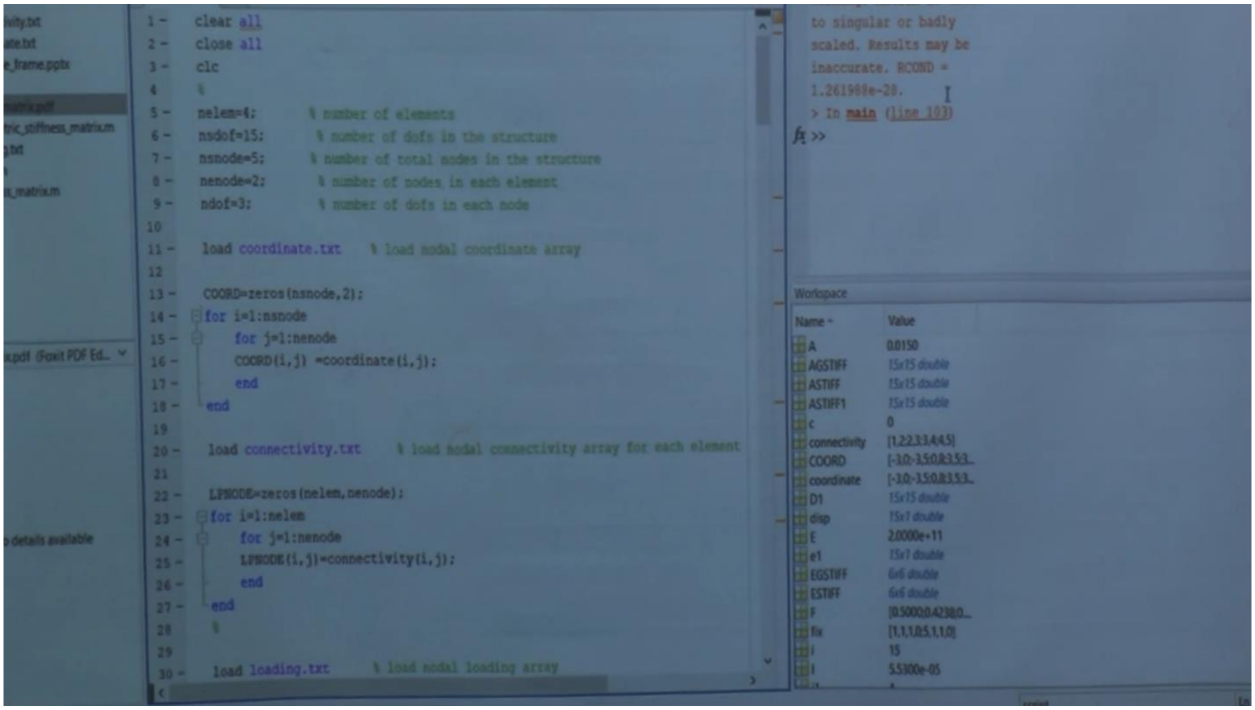
$$\ddot{U} + \omega^2 u + \epsilon u^3 = f(t)$$

right? So, we have learned about these two terms in our dynamics, right? $\ddot{U}$ plus, you know, k is equal to zero. But as soon as there's some nonlinear term that becomes. So, this is called what kind of cubic nonlinearity? This is cubic nonlinearity, and there is a small term ϵ. So, that controls the non-linearity. You see a similarity between these two; there you see that. You see that. So, when you want to solve this equation, this equation is called the Duffing oscillator. This describes the equation of a Duffing oscillator. Many problems, including the one geometric nonlinear problem, can be reduced to a damping oscillator problem: the vibration of nonlinear systems in the presence of geometric nonlinearity. So, U can be expressed as $u_0 + \epsilon u_1 + \epsilon^2 u_2$, something like this. Okay. And you see, if you do and substitute, you will see that you will not get the correct solution. Okay. That will give you a solution that you will not be able to capture. At the same time, you have to perturb ω as well. ω will be $\omega_0 + \epsilon\omega_1 + \epsilon^2\omega_2$ and things like that. And this is called the point car state technique; you have to use the equation you know because its natural frequency will no longer be constant; it will also vary; it's a nonlinear frequency with response frequency that will vary; it will change. So, that change you have to capture. This is called point car instant perturbation, and you can similarly have linearized equations. First zero-th order equation, first order equation, second order equations, and all are linear. So, you can solve them one after another, and you can sum them up. You have an asymptotic expansion for u and also an asymptotic expansion for ω. Okay. Thereby, you can clearly see that this ϵ will be a very small quantity. That control the degree of cubic nonlinearity, in first order say third order term, would you please tell me why there cannot be a second order term U^2? $F = KU$, right? So, the sign needs to be maintained. If you use a square term, that means if you're applying a positive force, negative displacement is occurring; negative stiffness cannot occur. There are devices that exist in structural vibration control or passive control, where you can design a negative stiffness device, but not in physical terms. You cannot have a negative stiffness, right? You can pull it like a spring. There will be positive displacement, and there will be positive restoring forces, right? Anyway, good. So, with this, I will stop the discussion here on plate buckling and post-buckling behavior. Okay, from here, logically we should go to the theory of shell buckling. But then I will not jump to the theory of shell buckling because we will cover that later. Right? Now we are going to concentrate on dynamic stability: how dynamics is important and why dynamic stability is important. Not only are there various forms of dynamic stability that come into real scenarios, but also the treatment itself; we'll come to that later. But before that, there is a small thing I would like to demonstrate to all of you about the Eigen buckling analysis for the frame. I will just demonstrate to you a computer program that I have written and I have shared it with you. So, how that program is written quickly I will go through I will send I will send you by mail. Okay, I will show you the structure. For those of you who are conversant in the finite element method, coding this is very simple, but I will emphasize two important steps here while coding, okay? And that's true; this will be in parity with whatever you do when you perform eigen buckling analysis using commercial software that is a little more advanced, not like STAT, okay? Using abacas or Ansys or things like that huh, which are little advanced software, which is added with advanced feature okay, not like Staad. So, this thing was missing okay. So let me uh show it to you. Okay. Yes. So, please note here. This program is very simple. It has one main routine and then several functions. So, of course you know the command for all of your conversions with MATLAB programming, right? Clear all, because all variables are cleared, and then close all, and then clc. Okay. Yeah. Now see, I'm not a very efficient coder. I mean, I cannot do very efficient coding, but I will essentially give you the steps. So, I am deriving; I'm giving the input. So, how many elements are there? Okay. In this example, we have taken a very simple frame, which is a gable frame, for demonstration, but you can use it for other purposes. So, here I have taken a gable kind of frame. Hm. And you know this is used in industrial settings, right? Gable frame, right? This, and then I am considering this is subject to P. I want to find out the theoretical value of P, right? So, you know, of course, this is symmetric. So, how many, so I have given, I have made discretization; these are all nodes, you know, these are all nodes, and these are all elements. So, the elements are one, two, three, four; these are all elements. The number of elements is four, and the number of structures is how many nodes there are: 1, 2, 3, 4, 5. So, the number of elements is four, right? And the number of ns nodes? Ns nodes mean the number of nodes in the structure, the number of structure nodes, and how many? 1, 2, 3, 4, 5. The number of structural nodes is five; the number of structural degrees of freedom is the number of structures. Each node has what? This is a 2D frame with three degrees of freedom: U, V, and W. Right? Each node has U, V, and W theta. U, V, and θ. Right? So, how many? Five, 15, right? 15. Number of element nodes. Each node has two. These are all two-noded beam column elements. So, the two-node number of nodes in each element means that the number of degrees of freedom in each node is three. This can be changed for anything. So, I will demonstrate with respect to that frame. Now you see that "load coordinate.txt" means you first load a file that contains the coordinates. Huh? What are the coordinates? So, as you see, when I am, you know, can I open this one from here? This is a PPT file; though you see, this is the way it is. You see, this is (0,0). I'm taking minus 3, 0; this is the nodal coordinate, okay? This is basically 3 m, this is 5 m, this is okay, huh, gable frame. I'm closing it, so all these nodal coordinates I am giving as input, right? So, you see that, coordinate.text you know you please see node 1 minus 3 0 node 2 minus 3 5 node 3 0 5 3 clear coordinate. huh nodal coordinate huh then, here is what I am writing basically: I'm just, you know, coordinating the text that I'm transferring with another coordinate array.

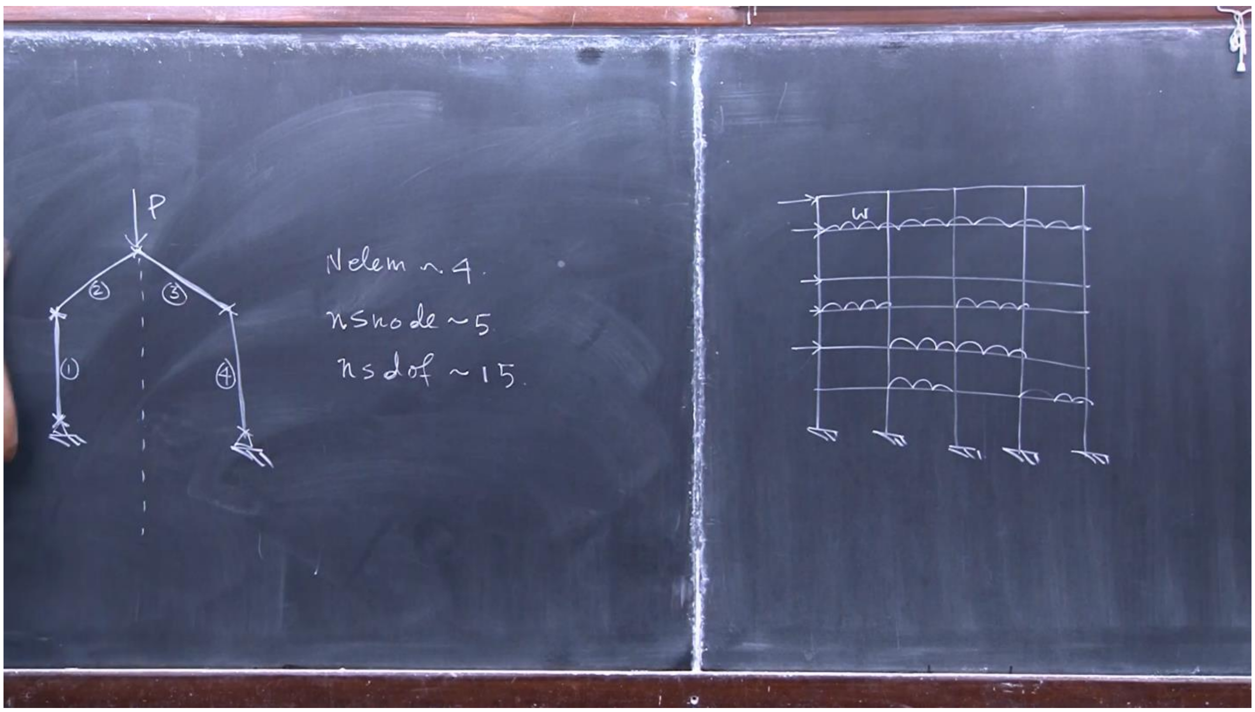
So, for coordinate zeros, I am defining its size. And then I am running a number of structural nodes or a number of element nodes; you know, the number for all structural element nodes. So, each node has, you know, coordinates, right? I am essentially transferring to another, you know, then I'm loading the connectivity, how I can connect each node; each element is connected with two nodes, right? Element one is between one and two, element two is two and three, element three is three, element four is four, and element five is five. So, this is connectivity; then I am basically transferring this connectivity to another variable, LP node. LP node is the nodal connectivity area, which is of dimension any element, any node. Now I am also loading the loading and boundary condition; loading means I have a load on the top at P. So, what do I have to provide? I am, you see, what I mean by the loading I'm giving; I'm defining the node on which the loading is applied: node three. Okay. And there are three degrees of freedom. In each degree of freedom, there can be loading. I have only vertical loading, so that's what I'm giving, right? I don't care about that; I am just assigning a value of one because, from the eigenvalue analysis, I will get the value. So, here I am putting only one unit. Okay, it doesn't really matter. Okay, clear? Now then, I am setting the fixed boundary condition.



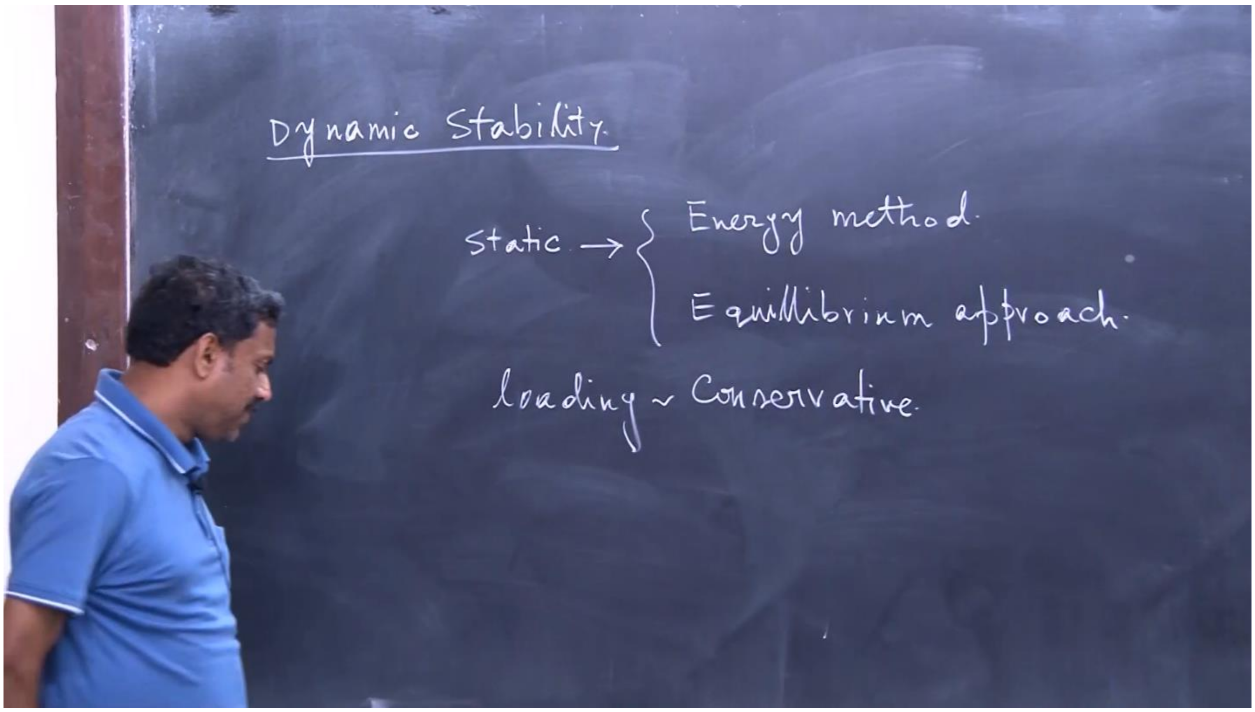
So, node one and node five are basically fixed. And what are the fixed degrees of freedom? U and V are fixed. So, these are all hinged. If you want to make it fixed at the hinge as well, then these zeros should be replaced by one and one. I'm using a binary variable. If it is zero, then that degree of freedom is not restrained. If it is one, then that respective degree of freedom is restrained. So, it's a binary variable. Okay. Clear? Fine. So, then I'm assuming all the areas of the elements are the same. I is equal to the second moment of area and the modulus of elasticity in all SI units is newton per meter squared, right E? So, I am initializing the structural stiffness matrix AST; of course, you see its dimension will be the number of structural degrees of freedom. I'm initializing the geometric stiffness matrix, right? Now, what I will do is find out that the first thing I have to do is a static analysis. Why? Because I need to know, in order to perform the geometric stiffness matrix, I have to find out the axial forces in all the members. So, if there is a vertically downward force at P, you know at the top node, right? Then I have to do a static analysis; that's what you will essentially see in all the software when you do the buckling analysis. That must be preceded by the static analysis, right? Because of the axial force and the relative distribution of the axial force, you should know, right? So, you see that when I'm doing global assembly, you must understand this looping because it is extremely important in all your finite element coding; this is uniform to all finite elements. I have to run a total of five loops; for the assembly, I have to run one loop over element one, element two, and the number of elements. I am finding out that this element is connected with x1, y1, x2, and y2 nodes. This coordinate, and then I am finding the length as the square root of $(x_2 - x_1)^2 + (y_2 - y_1)^2$, then I'm getting $\cos\theta$ and $\sin\theta$, which is the inclination angle of them right. Then I am calling the stiffness matrix, material stiffness matrix in this subroutine. So, material stiffness matrix you know, here you see that material stiffness matrix you see that. These are the 6 by 6 stiffness matrices; I have first populated the upper triangular matrix, and then for symmetry, I'm doing it right. You see, this is the 6x6 matrix and how it looks; I have derived this 6x6 matrix using global coordinates. So, that's why it's a function of $\cos\theta$ and $\sin\theta$. This is the element stiffness matrix, and this is the geometric stiffness matrix. Fine. So, then what do we have to do? You know, after the element, we have to run a node over the node. So, for i equal to one to the number of nodes, LP node I means the first node over which it is connected is LP node. Now you have to recall the nodal connectivity I element, I node, right? So, first, looping is over elements, then looping over nodes, then there will be looping over degrees of freedom, right? But please note that, because we have two degrees of, you know, there will be two indices, I and J, right? Because the stiffness matrix is a matrix, it has two indices. Therefore, the nodal degrees of freedom should appear in this looping twice, as you can see. So, you see, first I will identify the indices, which will fix the row, and then I will have another index that will fix the column. So, that's why you see this: LP node I is equal to this node to which it is connected. Then there is a degree of freedom, no one to. So, when I'm going to each node, there will be a loop over the degrees of freedom, right? So, I know the number of degrees of freedom; then, I am finding out what the structural degrees of freedom are and what the element degrees of freedom are. What will the IS degrees of freedom and the IS degree of freedom be? The global node with which the node is connected. So, LP node I minus 1 into N degrees of freedom plus I degree of freedom. And the element will be I; I node means the node on which the loop is running I minus 1 into N degrees of freedom plus I degrees of freedom. Understand which one is the global index and which one is the local index, element index, and structural index. The structure degree of freedom is when you are picking up the node in the structural node. See, because LP node I is basically you are picking from the nodal connectivity array, right? Similarly, you can also run a loop where J node is equal to 1 to the number of element nodes and LP node J. Now, once you identify the index for the row, we should go to the column index. Then you see, once again, there is a loop over the element, but it is only one. Please note that, right? So, for J node equal to one to the number of element nodes, LP node J is LP node I element, comma J node. Once again for J node one to the number of degrees of freedom. So, J is the degree of freedom. And then you have both of the elements. So, the element stiffness matrix is being added to the initialized structural stiffness matrix, right? So, astf is equal to sty plus est, right? So, each element stiffness matrix you are adding with the initially initialized structural stiffness matrix, right? So end-to-end is a 1 2 3 4 5 loop. It is a one-element loop, two loops for the node, and two loops for the degree of freedom over each node. That is the way it is. Huh? Now I have to enforce the boundary condition. So, once again, you call the, you know, mics that boundary condition that we have specified a node and the respective binary variable for each degree of freedom. You see that for I equal to 1 M to M fix; I fix is equal to if it is one, then I fix is equal to fix I minus 1. You see that you identify which degree of freedom it is connected to and its respective. We have basically used the penalty technique to enforce the boundary condition because we are multiplying; we don't want to disturb the side of the structural stiffness matrix. We want to multiply that degree by a large number; that's what we essentially did in finite element analysis, right? How to enforce the boundary condition without disturbing one way to enforce a homogeneous boundary condition is to eliminate rows and columns. But another way without disturbing is that it disturbs your stiffness matrix. If you don't want to disturb the stiffness matrix, you just multiply it by a large number. That's essentially what we did; you see, you multiply 10^{24}, and similarly, you have to run over all three degrees of freedom, C. Fix I, comma 2 fixed, I, comma 3 fix I, comma 1 is nothing but a node; that's what you see. Fix I, comma 1 is the node over which it is restrained. So, that node minus one into n degrees, because the node which is restrained all other degrees of freedom between which you have to pass, right? So that's what fix I - 1 - 1 into degrees of freedom plus one. Okay. So, you can code it better, but this is the way I can do it; then I find out the eigenvalue. Why did I find what I found out about the eigenvalue? Because if you don't enforce the boundary condition, then find out the eigenvalue, you will get three values that are zero, because all three will give the rigid body mode. So, that's a kind of check on finite element code that you must do correctly. If you do this, you will see no rigid body modes, and rigid body modes will be identified by zero values. So then what I did was basically find out the global load vector and the structural load vector. So once again, I ran a loop from I equal to 1 to M. I found out the loading vector. So, I, I is nothing but loading I minus one in. So, the degree of freedom over which it is connected. Okay. And then I am adding the respective load vector. You see here I have initialized the load vector, and then you do the static analysis. So, then you solve the linear simultaneous equations; the displacement you obtain by solving ast, y nodal displacement is correct. Please note that until now we haven't talked about the geometric stiffness matrix. We are doing a static analysis; after that, once we obtain the static displacement, we can find out what? We can find out the axial force in the member. How do you find out the axial force in the member? once again you identify run a loop over the whole element. So, in each element x1y1 x2y2 length cosθ sinθ and then a by l u2 - u1 plus you know u1 you know u1 - u2 -1 plus v2 - v1 right you know how to find out Axial force in finite element. So here a e by l c u2 -1 plus sinθ v2 - v1 that is the Axial force. So, we have identified u1, v1, u2, and v2 from the displacement. This is the structural displacement. So, all you know is that displacement; therefore, you have to pick up the displacement that corresponds to that node. So, how can you do that? You have the local connectivity area; you see that you have to identify the node over which it is connected: I1 and I2. These are the two nodes over which the element is connected. So, LP node I element, comma 1, LP node I element, comma 2. Do you see that? Then what we have to do, if you want to find out the support reaction, is to relax the boundary condition. So, the respective term that you multiply with the large number, you have to divide it by that. Okay, if you want to, and then you just multiply a step with the displacement, you will get the reaction forces. This is important for nonlinear analysis. Here we are not doing that because when you do the nonlinear analysis, you have to find out the residual forces. So, the external applied force causes a structural reaction. If you substitute then for a perfectly done linear analysis, there will be no unbalanced internal force. External force and general force must be crucial checks in finite element analysis. I don't know whether that was discussed in the finite element course; unless you code it, you will never be able to learn finite element. Let me tell you, you are going to code everything anyway. So, forget about that. Now what I do I will go once again. Now, once you obtain the axial forces in all the members, you run a loop for the global assembly on the geometric stiffness matrix using a similar approach. The problem is that I'm calling it copying and pasting. But you can have the subrouting for global assembly, assembly of stiffness, and call it again and again. Okay. But I just, and then once again, you have to enforce the boundary condition on the geometric stiffness matrix, as well, and then you solve the eigenvalue problem. So, you see that it's a generalized eigenvalue, okay? It is an eigenvalue problem in a special eigenvalue problem. A stip. So, the total stiffness matrix is the structural stiffness matrix plus the geometric structural stiffness matrix, and then you find out the mode and then frequency. You are sorry you find out that the eigenvalues and eigenvector values will give the buckling load vector and will give the buckling mode; you just plot it. Let me run it and show you that it is indeed running. Uh. I don't know whether it will run.



There is some warning coming, but do not worry about this warning. Let me explain to you what this warning means. Warning: the matrix is close to singular or badly scaled; the result may be inaccurate. This happens because sometimes, when you solve this problem, the geometry is unclear. 1.261 e to the power minus 28 what is ard anybody matrix is close to singular rn you don't know what is around all of you did numerical analysis right h you know have a condition number of a matrix what is condition number of a matrix huh what is condition number of a matrix huh This is very confusing; you don't know, do you? These are the eigenvalues, okay? So, the values you see are initially not assigned to any force; it was only one, and it is coming out as negative: -7.869 e to the power of 16. Of course, it is in Newton. So, that is the first fundamental buckling force, you know, the fundamental critical load for phi that will trigger buckling. The dimension of the stiffness matrix here is unique; it is 15 by 15 total. So, the number of eigenvalues will be equal to what? Whatever the dimension of the stiffness matrix is, right? So, squared stiffness matrix, right? So, this is the first mode because this is the lowest, and then it started increasing, and you can also find the respective eigenvector here. No, this is basically you see that diagonal in diagonal terms; all these values are there, right? What is the end vector? Okay. Let me see what is... Yeah, this one vector you can plot. This is the fundamental one. Huh? Okay. Uh, this is the fourth eigenvector. You can, because I have written it as comma 4, find out the vector. You see that some eigenvectors are very small. Why is that? So, -70 is -18? Because these are restrained degrees of freedom. It is almost zero. You see that -23, -20, -25, right? Because these are restrained. So, there is nothing in that, huh? So, you can find out the eigenvector and eigenvalue. This is the structure of the code, and you know you can all do this, okay? I'm removing it, so if you can recall. So, you can recall that in the finite element assignment, whatever I have shown you, I have given you this kind of frame. And this was subjected to some, you know, lateral distribution of these forces.

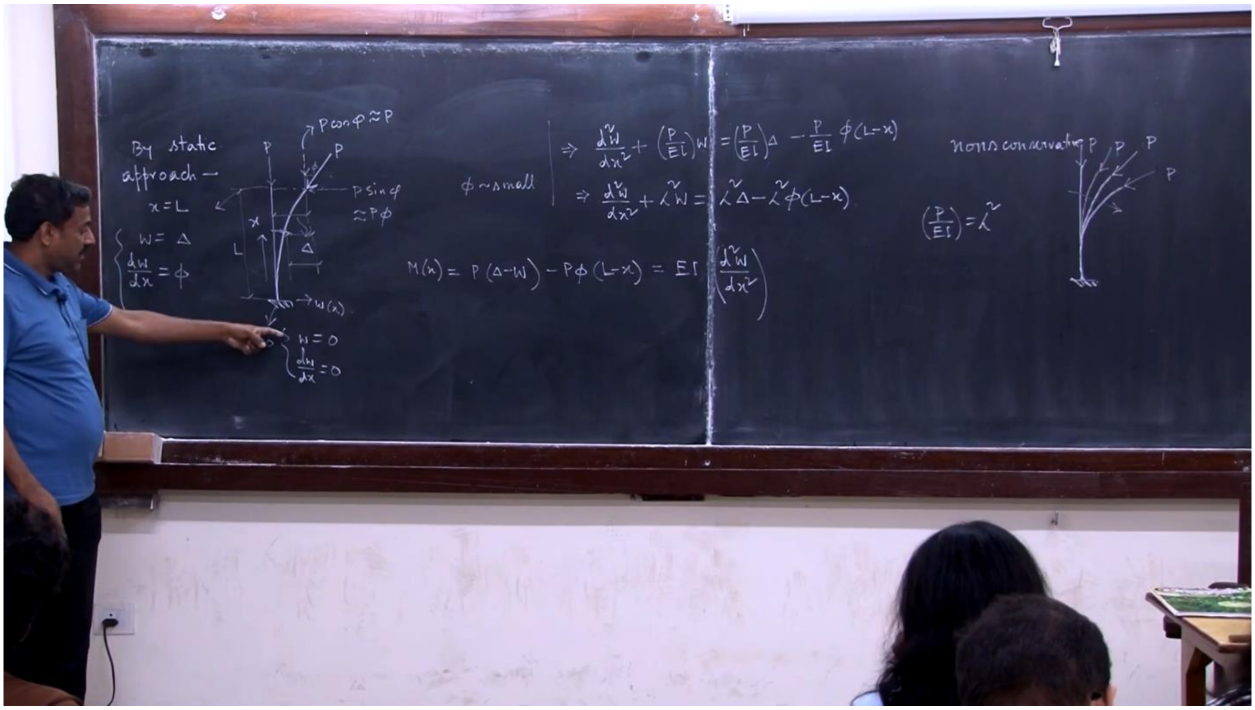


You know, that was also subjected to, you know, lateral distribution of forces, and that was also uniformly distributed forces. And I asked you to find out what this will be; you know what the W value for r is, so save the W value. So, once again, this is the frame and the two-dimensional frame; you can use the same computer code for solving this. You discretize it using elements you know, and you try to find out the nodal coordinates, nodal connectivity, element connectivity array, and fixity array. And all these things, and then you first solve the statically to find out the axial forces in the member, and then you construct the geometric stiffness matrix using the geometric eigenvalue, and then you solve for the. Uh, the eigenvalue problem that is forced to be solved for this, okay? So, this problem can be solved. Okay, at one point in time, let me remind you to please pay attention and try to concentrate on this. Because I can recall that in your experimental method course, you will see that there is an experiment. I don't know if it is being conducted this year or not, but you know about the buckling of frames. So, I can Recall that here. So these people were all confused that, how to find out the, theoretically how to find out the buckling load for a frame and they were all searching Timoshenko This young book is very simple, using either the matrix method or the finite element method. So, whatever you are learning, try to retain it, okay? Don't you know this? It shouldn't be volatile memory. Like in computers, there are two types of memory: permanent and volatile, right? It shouldn't be like volatile memory; you learn and then it evaporates from here. Okay. Anyway, okay.



So that's what I will cover a little bit, you know, just to sensitize you to computer programming and things. Now I will go to dynamic stability. So why dynamic stability? What was the problem with static stability? Until now, when we approach static stability, you know the static approach to stability; you know what we have approached. There are two formal methods: one is the most convenient, and then we love the energy method, right? The energy method and, of course, another way is the equilibrium approach; equilibrium directly solves the equilibrium approach. We have used both the equilibrium approach; of course, you have to write down the equilibrium equation in the deformed configuration or perturbation configuration. So, this approach worked well, you know, and then we solved a number of problems. We started with a toy problem. We demonstrated the different classes of behavior and different classes of systems. And then we have also used the same method for analyzing the buckling of columns, post-critical analysis in columns, buckling of plates, and post-critical analysis of plates, and now everything is right. Both the approach we have followed and the problem with the static are that, in all the cases we have considered, our loads were all conservative loads. Our loadings were all conservative. What is conservative loading? Conservative loading means there is no dissipation involved in the loading. Okay. So, when this load is working on the system, then of course the loading, like for a column or for a plate, you know the load is doing some work, right? The loading is occurring because the axial compression is acting, you know, out of plane; you know, axial deformation, right? And that is being stored as strain energy in the plate or in the column, right? So, as soon as the load is removed, that strain energy will also be gradually recovered. So, the energy stored there is not dissipated. There is no work done that you can see. So, that is conservative; that means whatever the system is, it is path independent, how the load is being applied and things like that, right? And there is no dissipation of energy. An example of a non-conservative force is friction. You see, if you pull a block on a rough surface, that friction force will act on it. The work done by that force will never be recovered. There will be some dissipation, but here all the forces were conservative. Now there are situations in which you know the forces may not be conservative. In that case, you will see that this approach, the energy equilibrium approach, will not work, and it basically took time for the scientists to realize that, you know. So, it is not that you know, in fact, all the static methods fail, and there was a problem that remained unsolved for a long time; it was later solved, and the way it was solved was by using dynamic methods, or let me show you that. An example of how it was failing: the static approach is failing, and we need to understand the dynamic approach; therefore, we have to invite the component that is causing non-conservatism in the system. Which is a component of non-conservative forces; if there is a component of non-conservative forces or there is a non-conservative force, then the treatment of stability should include the consideration of the dynamics problem. Otherwise, you cannot capture that influence. So, this fact was not accounted for in the past, and then it gave a false notion and a false result, and people didn't believe it, which caused a lot of confusion, but later the problem was solved. So, this problem is a column under follower force, under which is a follower force; follower force is a force that follows the deformation, okay. So, for here, what I will take, you know, I'll consider a column here. This column, as soon as it is subjected to a vertically downward force, you know, compression on the column, is conservative.

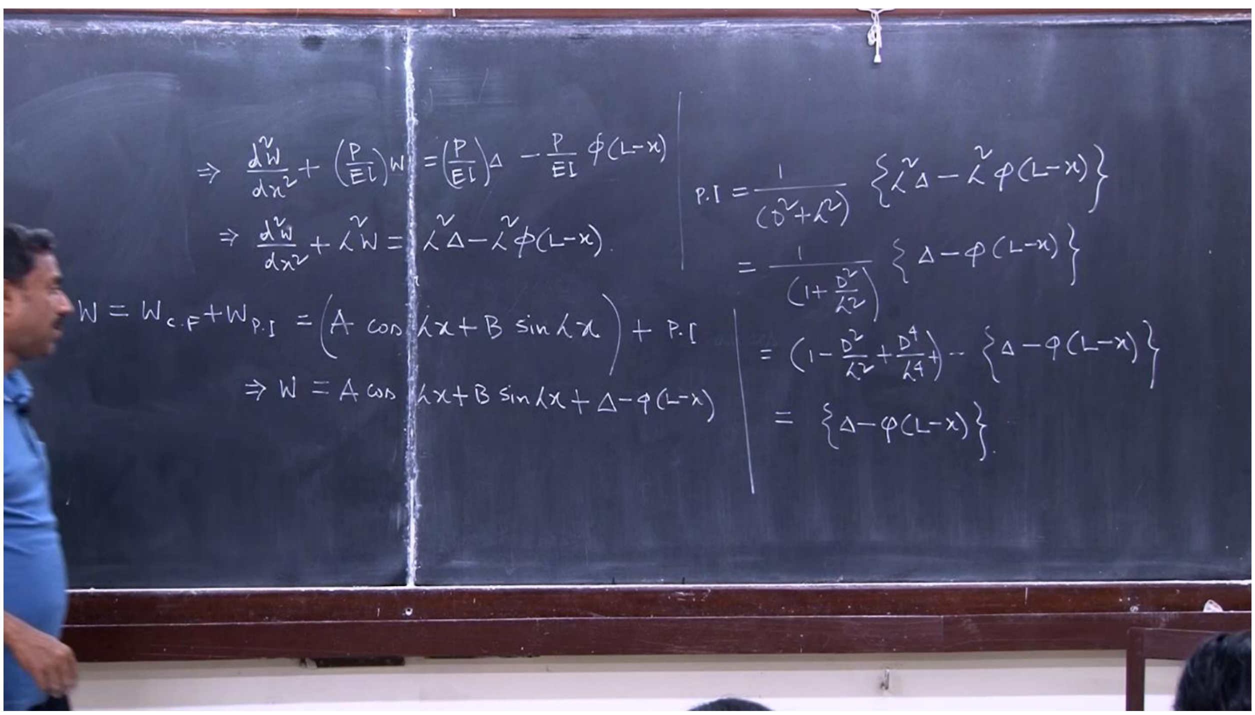
You know, even when it is deflecting, it was deflecting if it remains like this; these are all conservative. This is a conservative force, right? This is a conservative force, okay. Conservative force P acts vertically down; P remains vertical at all instances. However, now you consider the case, you know. However, huh? However, if P, you know, follows the tangent at the deformed tip, right? Then what will happen? So, if that, then you see that there will be a column. This column, and then you know. And then, here it is, P; that is fine, but as soon as it comes, then P follows the tangent. You see that P is like this. Then maybe here, when it comes, it follows the tangent P, okay? It is coming, and you see that the way it is changing; the P is changing its direction. Why? You know because this P is following the tangent of the deformed tangent to the column at the deformed tip. So, this is a non-conservative force. This is a case of a non-conservative force. Okay. This system becomes non-conservative why it is so? that I will explain you okay. So, people started to look into this. This problem they tried to solve using a static method, and all of them, you know, failed miserably. Actually, I will show you why it is so. So, now let us consider how people started solving it and why they were failing. Let me show it to you, and that was basically when people started using a dynamic approach. So, the static approach cannot solve this problem. Fine, you know. So, it will have two components. So, I'm assuming that this angle is Φ and this one is δ; this deflection is δ. So, P, this will be P you know well. Instead of treating this as five, I'll treat the angle as this; you know this to be ϕ, okay? This is ϕ, and I'm still assuming that ϕ is small, okay? I'm still assuming that ϕ is small. So, it is $p\sin\phi$; it will be nothing but $p\phi$, and this component is $p\cos\phi$. Can you see that? So, these are the component forces. Now you tell me one thing: I'm assuming that well X is defined like this, and my W is the deflection as a function of X. Now, this problem: what is the bending moment at any section X?



At any section, if I consider any section here or here, what is the bending moment? The bending moment will be contributed by these two components. One, of course, is P, you know; this is P into what? Will be that this distance, this distance is what? This is δ, and this is small δ. This is small δ minus w δ. Right? This is going to right this. So, this distance from here to here, this distance is nothing but $\delta - w$ because from here to here, it is w δ, right? So, this is trying to turn clockwise to the right, and this fellow is trying to make it anticlockwise. So, then minus P into Φ, and what is this section? This lever, what is this distance? This distance is nothing but from here to here; the length I'm assuming that column has is L. You know I'm assuming this column has a length of L, and from here to here is X at a distance X. This distance is$(L - X)$, right? If I hear, it is$(L - X)$ right. Then M is nothing but $EI\left(\frac{d^2w}{dx^2}\right)$ right; m is $EI\left(\frac{d^2w}{dx^2}\right)$. Another thing we are assuming is that this column has distributed elasticity, but it has a concentrated mass. So, the mass is concentrated here. I mean we do not need to consider mass here because we are essentially following a static approach. The column under follower force is what we are first trying to solve using a static approach. So here, what we are doing is solving this by static approaches. See what will happen. Okay. So, m is equal to this. Here, what we can get from here, you see that. Then, you will get $\frac{d^2w}{dx^2}$ and then $\frac{p}{EI}w$ is equal to $\frac{p}{EI}\Delta$ right minus $\frac{p}{EI}\phi(L-x)$. So, $\frac{p}{EI}$ is equal to λ^2; let us assume we have assumed it to be λ^2, right? All the previous examples, right? So, I can write this one as

$$\frac{d^2w}{dx^2} + \lambda^2\delta = \lambda^2\Delta - \lambda^2\phi(L - X)$$

right? So, that's the governing equation, and then we can have boundary conditions. Of course, you can understand the boundary conditions here: when $w = 0$, $\frac{d^2w}{dx^2}$ and $\frac{dw}{dx}$ equal to zero. You know the boundary conditions at $x = 0$. Here, $x = 0$, $w = 0$, and $\frac{dw}{dx}$ is equal to 0 because there is no slope. Then, the boundary conditions here at $x = l$ are $w = 0$ and $w = \Delta$. $\frac{dw}{dx}$ that is the slope. That is nothing but ϕ, right? These are the two boundary conditions, right? Two sets of boundary conditions: two sets of $x = l$, $x = 0$, and $x = l$, right? Now, the solution you know of, of course, is a complementary function plus a particular integral. So, the quadratic function, as you see, $\frac{d^2}{dw}$. You see that $d^2 + \lambda^2$. So, $d \pm i\lambda$.



So, c_1 to the power, Euler equation. So, you can write $A\cos\lambda x + b\sin\lambda x$; that is a complementary function, and then we can find the particular integral. So, how do we find the particular integral?

We'll see how to find the particular integral. The particular integral will be or the particular solution is $\left(\frac{1}{d^2+\lambda^2}\right)\{\lambda^2\Delta - \lambda^2\phi(l-x)\}$,.

You see that, okay? $\frac{1}{\left(1+\frac{d^2}{\lambda^2}\right)}\{\Delta - \phi(l-x\}$. I'm dividing by λ, okay? Dividing by λ^2 in the denominator, numerator, right? And then I will $\left(1-\frac{D^2}{\lambda^2}+\frac{d^4}{\lambda^4}\right)$, you know. So, ϕ minus capital δ minus ϕ means this. Okay. So, let us assume. So, w this means, w is $A\cos\lambda x + b\sin\lambda x$, particular integral is $\{\Delta - \phi(l-x)\}$ right okay So this is the solution that we obtained. Okay. So let us have the solution, and then we know the boundary conditions over there. So, in the next class, we'll try to see. Okay. Thank you very much for today's class.