BITS Pilani K. K. Birla Goa Campus

Design Project : Numerical Solution of PDEs by FEM

ME F376

Simulation of a thin elastic plate by Finite Element Method

Authors:
Aabir Abubaker
ID:2013A4B5280G
Parth Thakkar
ID:2013A4TS022G
Kushal Joshi
ID:2013A4TS136G
Abhishek Suresan
ID:2013A4PS256G

1 Introduction

Finite Element Method (FEM) is a numerical technique to solve a set of Partial Differential Equations (PDEs). Most of the complex engineering problems are governed by a set of partial differential equations which are difficult or impossible to solve analytically. In such cases, numerical technique like FEM is useful as it gives an approximate solution to any complex set of partial differential equations.

The basic idea behind Finite Element Method is to convert partial differential equations into a system of linear algebraic equations which are easy to solve numerically.The solution is assumed to be belonging to a mathematical space in which its values at the boundaries are already known (Dirichlet boundary conditions).

In order to solve the PDE numerically,the governing partial differential equation is converted to its weak form. This is done by multiplying the PDE by a weight function which belongs to another mathematical space. We are free to choose the weight function and its values at the domain boundaries. This is called the weak form of partial differential equation because integration reduces the order of the derivatives and thus relaxes differentiability condition by an order. The strong form and the weak form are both equivalent and the solution of the weak form is inturn the solution of the strong form. The weak form thus generated is called infinite dimensional weak form.

The infinite dimensional weak form is then converted to a finite dimensional weak form. The finite dimensional solution and the weight functions are considered to be a part of a finite dimensional subspaces. In order to obtain the finite dimensional solution and weighting functions, the domain is divided in to disjoint sub-domains or elements. Since the entire domain can be constructed by summing the elements, the problem reduces to representing the finite dimensional solution and weighting functions over the elements. In order to represent finite dimensional solutions and weight functions, we select suitable basis functions and the solution and weight functions are represented as a linear combinations of these basis functions, with coefficients equal to the degrees of freedon of the nodes. The number of basis functions is equal to the number of nodes in an element. In order to simplify the problem, cartesian domain is converted into a bi-unit domain in where nodal positions vary from -1 to 1 in all dimensions. Basis functions are represented in this bi-unit domain.

These basis functions are now plugged into the weak form of PDE. The integration domain is now converted in terms of bi-unit domain. This gives rise to element level integrals which can be added for all elements to give the solution for the entire domain. Integrations are performed numerically using gaussian quadrature techniques and summation over all elements gives rise to a system of linear equations which are represented in matrix form as

$$F = Kd \tag{1}$$

where, F = force matrix
K = stiffness matrix
d = degrees of freedom of nodes

Once the degrees of freedom are calculated, the solution can be calculated since the solution was assumed to be a linear combination of basis functions and degrees of freedom of nodes.

In the following chapters, mathematical formulation of an elasticity problem is presented using FEM technique along with the results.

2 Description and Formulation of the problem

The physical problem that we are simulating is that of a thin elastic plate which is perpendicular to the horizontal. The two vertical sides are fixed and the two horizontal sides are loaded. The elastic force equation is solved subject to boundary conditions as follows: the two vertical sides have Dirichlet Boundary conditions and the two horizontal sides have traction boundary conditions.

Consider a rectangular plate with length l and breadth b. Let the plate be fixed on the two sides with length b. Let the Young's Modulus of the material of the plate be E and the Poisson's ratio be ν. We are assuming that the problem is two-dimensional. Let the body force acting on the plate be $\vec{f}$ N/m^3. Let the plate be loaded on the sides with length l by traction forces with boundary values of traction $\vec{t}$. Let the domain of analysis, i.e. the solid area of the plate be Ω and the boundary of the plate be $\partial\Omega = \partial\Omega^D + \partial\Omega^t$, where $\partial\Omega^D$ is the Dirichlet boundary and $\partial\Omega^t$ is the traction boundary. In the subsequent equations, the Einstein convention for summation is used unless specified otherwise. Thus in mathematical terms, the equations to be solved are as follows:

$$\sigma_{ij,j} + f_i = 0 \tag{2}$$

$$\sigma_{ij} = C_{ijkl}\epsilon_{kl} \tag{3}$$

$$\epsilon_{kl} = \frac{1}{2}(u_{k,l} + u_{l,k}) \tag{4}$$

$$\mathbf{u} = \mathbf{0} \quad \text{on } \partial\Omega^D \tag{5}$$

$$\sigma_{ij}n_j = t_i \quad \text{on } \partial\Omega^t \tag{6}$$

where, $\mathbf{u}$ is the displacement field with ith component u_i, σ_{ij} is the stress tensor, ϵ_{ij} is the strain tensor, C_{ijkl} is the fourth order stiffness tensor and $\mathbf{n}$ is the normalised normal vector to the boundary under consideration with jth component n_j.

The above problem is expressed as a differential equation and it's boundary conditions. This form of expressing the problem is called the Strong Form. In FEM, the problem is converted into an Integral Equation, which is called the Weak Form. Consider a function $\mathbf{w}\epsilon[H_0^1(\Omega)]^2$, which is used as a test function. Taking the dot product of $\mathbf{w}$ with equation (1), integrating it over Ω and using Green's formula, we get the following expression after some simplification:

$$\int_\Omega w_{i,j}\sigma_{ij}dV = \int_\Omega w_i f_i dV + \int_{\partial\Omega} w_i \sigma_{ij} n_j dS$$

Since $\mathbf{w} = \mathbf{0}$ on $\partial\Omega^D$, using equation (5) we get

$$\int_\Omega w_{i,j}\sigma_{ij}dV = \int_\Omega w_i f_i dV + \int_{\partial\Omega^t} w_i t_i dS$$

Using the equation (3) and the properties of symmetry of the stiffness and strain tensors, we get

$$\int_\Omega w_{i,j}C_{ijkl}u_{k,l}dV = \int_\Omega w_i f_i dV + \int_{\partial\Omega^t} w_i t_i dS$$

The mesh used is a rectangular mesh which is uniform in the x and y directions. Let the domain of element e be Ω_e. Thus, on discretising the domain, we get

$$\sum_e \int_{\Omega_e} w_{i,j}C_{ijkl}u_{k,l}dV = \sum_e \int_{\Omega_e} w_i f_i dV + \int_{\partial\Omega^t} w_i t_i dS \tag{7}$$

Now, let us consider an element e. Each element is a rectangle with four nodes each. They are given local node numbers 1, 2, 3 and 4. FEM assumes the solution to be a linear combination of basis functions. So, or a particular element, the solution $\mathbf{u_e}$ is taken to be a linear combination of the four basis functions N^A, where A goes from 1 to 4. Thus,

$$u_{ie} = \sum_{A=1}^{4} N^A d_{ie}^A$$

where, d_{ie}^A is the ith component of the displacement of the node A. The test function is taken to be of a similar form, i.e.

$$w_{ie} = \sum_{A=1}^{4} N^A c_{ie}^A$$

where, c_{ie}^A is the ith component of the test function at the node A.

The basis function is taken to be a bilinear basis function in the domain. Important properties of the basis function are :

1) The basis function N^A takes the value 1 at node A and the value 0 at other nodes in the element. It is linear along x and linear along y.

2) The sum of all the basis functions of an element is equal to 1.

Consider one element e from the summation on the LHS. On substitution of u and w in terms of the basis functions, it becomes

$$\sum_{A,B} c_{ie}^A \left(\int_{\Omega_e} N_{,j}^A C_{ijkl} N_{,l}^B dV \right) d_{ke}^B = \sum_{A,B} c_{ie}^A K_{eik}^{AB} d_{ke}^B$$

We used Two-point Gaussian integration in order to evaluate the above double integral to get K_{eik}^{AB}. Thus,

$$K_e^{AB} = \begin{bmatrix} K_{e11}^{AB} & K_{e12}^{AB} \\ K_{e21}^{AB} & K_{e22}^{AB} \end{bmatrix}$$

The element level stiffness matrix K_e is given by:

$$\boldsymbol{K_e} = [K_e^{AB}]_{A=1,B=1}^{A=4,B=4} \tag{8}$$

Now, let us consider the element e in one of the summed terms in the first term on the RHS of equation (7). Substituting the value of w_i into the term, we get:

$$\sum_A c_{ie}^A \int_{\Omega_e} N^A f_i dV = \sum_A c_{ie}^A F_{ie}^A$$

Now, consider the second term in the weak form. We substitute the value of w_i into the term, we get,

$$\sum_{A\epsilon An} c_{ie}^A \int_{\Omega_e^t} N^A t_i dS = \sum_{A\epsilon An} c_{ie}^A F_{ie}^{tA}$$

The complete finite dimensional weak form can be written as

$$\sum_e \sum_{A,B} c_{ie}^A K_{eik}^{AB} d_{ke}^B = \sum_e \sum_A c_{ie}^A F_{ie}^A + \sum_{e\epsilon En} \sum_A c_{ie}^A F_{ie}^{tA} \tag{9}$$

We write the above equation in matrix vector form

$$\sum_{e} \boldsymbol{c_e^A} \boldsymbol{K_e^{AB}} \boldsymbol{d_e^B} = \sum_{e} \boldsymbol{c_e^A} \boldsymbol{F_e^A} + \sum_{e \epsilon En} \sum_{A} c_{ie}^{A} F_{ie}^{tA} \tag{10}$$

Now, the element level stiffness matrix and the element level force matrix are to be assembled in a single matrices. To do this, the neighbouring elements of each node are checked.

Assembly of the stiffness and force matrices:

The global level equation is formulated by clubbing up all the equations of the nodes which do not have Dirichlet boundary conditions. However, the equations contain the values of the degrees of freedom at all the nodes, even those having the dirichlet boundary conditions. Let N_{nodes} be the total number of nodes in the mesh and N_D be the number of nodes at which Dirichlet boundary conditions are applied. Thus, the global level equations are formulated as follows:

$$\boldsymbol{c^T} \bar{\boldsymbol{K}} \bar{\boldsymbol{d}} = \boldsymbol{c^T} \boldsymbol{F_b} + \boldsymbol{c^T} F^t \tag{11}$$

where, **c** contains the values of test functions at all the nodes except the ones at the Dirichlet boundaries. It is a matrix of $N_{nodes} - N_D$ 2-dimensional vectors. **d** contains the values of the displacements at all the nodes including those on which Dirichlet boundary conditions are applied. It is a matrix of N_{nodes} 2-dimensional vectors. The $\bar{K}$ has $(N_{nodes} - N_D) \times N_{nodes}$ 2×2 matrices. It is obtained by adding the contribtions of each of the relevant components of the element level stiffness matrices of the relevant elements. The F_b matrix is obtained by a similar assembly of the element level body force matrices of the elements. The F^t matrix is obtained by assembling the element level traction force matrices on the traction boundaries and the other values in this matrix are zeros. These two matrices have $N_{nodes} - N_D$ 2-dimensional vectors.

A point to note is that some values in the $\bar{d}$ matrix are known - the ones on the Dirichlet boundaries. So, those values are multiplied with the relevant columns in the $\bar{K}$ matrix, and those columns are shifted to the other side of the equation. The total number of these columns is $2 \times (N_{nodes} - N_D)$. Thus, the equation becomes

$$\boldsymbol{c^T}(\boldsymbol{K}\boldsymbol{d} + \boldsymbol{D}) = \boldsymbol{c^T} \boldsymbol{F_b} + \boldsymbol{c^T} F^t \tag{12}$$

where, **K** is the Global Stiffness Matrix. It is a square matrix with $(N_{nodes} - N_D) \times (N_{nodes} - N_D)$ 2×2 matrices. **d** is the matrix of the degrees of freedom for which we want to solve. It is a column matrix of $N_{nodes} - N_D$ 2-dimensional vectors. **D** is the column matrix which is the total sum of the values of degrees of freedom at the Dirichlet nodes multiplied with the relevant columns of $\bar{K}$. Thus, we finally get

$$\boldsymbol{c^T}(\boldsymbol{K}\boldsymbol{d} + \boldsymbol{D} - \boldsymbol{F_b} - F^t) = 0 \boldsymbol{c^T}(\boldsymbol{K}\boldsymbol{d} - \boldsymbol{F}) = 0 \tag{13}$$

where **F** is the Global Force Matrix. This equation holds for any value of **c**. Thus, we have

$$\boldsymbol{K}\boldsymbol{d} = \boldsymbol{F} \tag{14}$$

Thus, we obtain the above system of linear equations. We have solved this system of equations using the Successive Over-Relaxation method, using relaxation factor = 1.6 and taking initial guess as zeros.

References

[1] Krishna Garikipati, *The Finite Element Method for Problems in Physics,* Coursera Lecture Series

[2] Claes Johnson, *Numerical Solution of Partial Differential Equations by Finite Element Method*, Dover Publications, 2009

[3] `https://github.com/stangmechanic/NE155_Homework_3/blob/master/SOR.py`