FINITE ELEMENT METHOD: AN INTRODUCTION

Uday S. Dixit

Department of Mechanical Engineering, Indian Institute of Technology Guwahati-781 039, India

1. Introduction

Finite element method (FEM) is a numerical method for solving a differential or integral equation. It has been applied to a number of physical problems, where the governing differential equations are available. The method essentially consists of assuming the piecewise continuous function for the solution and obtaining the parameters of the functions in a manner that reduces the error in the solution. In this article, a brief introduction to finite element method is provided. The method is illustrated with the help of the plane stress and plane strain formulation.

2. FEM formulation for a linear differential equation

A linear differential equation can be of the following form:

$$L\boldsymbol{u} + \boldsymbol{q} = \boldsymbol{0}, \tag{1}$$

where $\boldsymbol{u}$ is the vector of primary variables of the problem, which are functions of the coordinates, L is the differential operator and $\boldsymbol{q}$ is the vector of known functions. This differential equation will be subjected to boundary conditions, which are usually of two types- (i) the essential boundary conditions (ii) the natural boundary conditions. The essential boundary conditions are the set of boundary conditions that are sufficient for solving the differential equations completely. The natural boundary conditions are the boundary conditions involving higher order derivative terms and are not sufficient for solving the differential equation completely, requiring atleast one essential boundary condition. For example, consider the differential equation:

$$\frac{\mathrm{d}}{\mathrm{d}x}\left(EA\frac{\mathrm{d}u}{\mathrm{d}x}\right) + q = 0. \tag{2}$$

This problem can be solved completely under one of the following two conditions:
(i) u is prescribed at both ends.
(ii) u is prescribed at one end and $\mathrm{d}u/\mathrm{d}x$ is prescribed at the same or other end.
However, the problem cannot be solved if only $\mathrm{d}u/\mathrm{d}x$ is prescribed at both ends. Thus, we surely require one boundary condition prescribing u. Therefore, for this problem $u = u^*$ is an essential boundary condition and $\mathrm{d}u/\mathrm{d}x = (\mathrm{d}u/\mathrm{d}x)^*$ is a natural boundary condition, where * indicates the prescribed value. Now consider the differential equation

$$\frac{\mathrm{d}^2}{\mathrm{d}x^2}\left(EI\frac{\mathrm{d}^2w}{\mathrm{d}x^2}\right) - q = 0, \tag{3}$$

This differential equation can be solved completely by specifying w and $\mathrm{d}w/\mathrm{d}x$ at both ends. One can also specify $\mathrm{d}^2w/\mathrm{d}x^2$ and/or $\mathrm{d}^3w/\mathrm{d}x^3$ as boundary conditions, however out of total four boundary conditions, two must be of one of the following forms:
(i) w prescribed at both ends.
(ii) w prescribed at one end and $\mathrm{d}w/\mathrm{d}x$ prescribed at the other end.
Thus, the prescribed values of w and $\mathrm{d}w/\mathrm{d}x$ form the part of essential boundary conditions and prescribed values of $\mathrm{d}^2w/\mathrm{d}x^2$ and $\mathrm{d}^3w/\mathrm{d}x^3$ form the part of natural boundary conditions.

Two popular FEM formulations are Galerkin formulation and Ritz formulation. In Galerkin formulation, the primary variable is approximated by a continuous function inside the element. When the approximate primary variable $\boldsymbol{u}^e$ is substituted in Eq. (1), we shall get residue depending on the approximating function, i.e.,

$$L\boldsymbol{u}^e + \boldsymbol{q} = \boldsymbol{R}. \tag{4}$$

Ideally, the residue should be zero everywhere. In that case, approximation becomes equal to true value. As it is very difficult to make the residue 0 at all points, we make the weighted residual equal to zero, i.e.,

$$\int_D wR \, dA = 0 \; , \tag{5}$$

where w is the weight function. In order to weaken the requirement on the differentiability of the approximating function, we integrate Eq. (5) by parts to redistribute the order of derivative in w and R. In Galerkin method, the weight function is chosen of the same form as the approximating function. The approximating function is some algebraic function. It is common to replace the unknown coefficients of the function by unknown nodal degrees of freedom. Thus, typically,

$$u^e = [N]\{u^{ne}\} \; . \tag{6}$$

where $[N]$ is the matrix of shape functions and $\{u^{ne}\}$ is the nodal degrees of freedom.

In Ritz formulation, the differential equation Eq. (1) is converted into an integral form using calculus of variation. (Sometimes the integral form itself may be easily derivable from the physics of the problem.) The approximation (Eq. (6)) is substituted in the integral form and the form is extremized by partially differentiating with respect to $\{u^{ne}\}$.

After obtaining the elemental equations, the assembly is performed. A simple way of assembly is to write equations for each element in global form and then add each similar equations of all the elements, i.e., we add the equation number 1 from each element to obtain the first global equation, all equation number 2 are added together to give second equation, and so on. The boundary conditions are applied to assembled equation and then are solved by a suitable solver. Then, post-processing is carried out to obtain the derivatives.

3. Formulation for plane stress and plane strain

Consider a linear elastic solid of domain Ω and having uniform thickness bounded by two parallel planes on any closed boundary Γ as shown in Fig. 1. The meaning of boundary conditions is explained in Figure 2. If the thickness in z direction is small compared with the size Ω of the domain, the problem may be approximated as a plane stress problem. The following assumptions are made. The body forces, if any exist, cannot vary in the thickness direction and cannot have components in the z direction; the applied boundary forces must be uniformly distributed across the thickness (*i.e.* constant in the z direction); and no loads can be applied on the parallel planes bounding to the bottom surfaces. The assumption that the forces are zero on the parallel planes implies that for plane stress problems the stresses in the z direction are negligibly small *i.e.*,

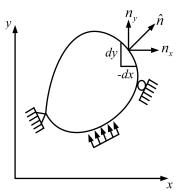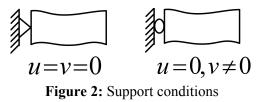$$\sigma_{xz}=0 \quad \sigma_{yz}=0 \quad \sigma_z=0 \tag{7}$$

Figure 1: A solid of domain Ω

$$u=v=0 \qquad u=0, v\neq 0$$

Figure 2: Support conditions

Plane strain is defined as a deformation state in which there is no deformation in z-direction and deformations in other directions are functions of *x* and *y* but not of *z*. Thus, stain components $\varepsilon_z = \varepsilon_{yz} = \varepsilon_{zx} = 0$. In plane strain problems non-zero stress components are $\sigma_x, \sigma_y, \sigma_{xy}$ and σ_z. However, σ_z is not an independent component and can be obtained if σ_x and σ_y are known. This makes the FEM formulation for plane stress and plane strain problems similar. Only difference is in the constitutive matrices for both problems. In this section FEM formulation for plane stress and plane strain problems will be discussed.

The governing equations for the plane elasticity problems are given by

$$\frac{\partial \sigma_x}{\partial x} + \frac{\partial \sigma_{xy}}{\partial y} + f_x = \rho \frac{\partial^2 u}{\partial t^2} \tag{8}$$

$$\frac{\partial \sigma_{xy}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + f_y = \rho \frac{\partial^2 v}{\partial t^2} \tag{9}$$

where f_x and f_y denote the body forces per unit volume along the *x* and *y* directions, respectively and ρ is the density of the material. σ_x, σ_y are the normal stresses and *u*, *v* are the displacements in *x* and *y* directions respectively, σ_{xy} is the shear stress on the *xz* and *yz* planes. Strain-displacement relations are given by

$$\varepsilon_x = \frac{\partial u}{\partial x} \quad , \quad \varepsilon_y = \frac{\partial v}{\partial y} \quad , \quad 2\varepsilon_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \tag{10}$$

For plane stress problems, stress and strain are related by the constitutive matrix **D**, in the following manner:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_{xy} \end{Bmatrix} = \begin{bmatrix} d_{11} & d_{12} & 0 \\ d_{21} & d_{22} & 0 \\ 0 & 0 & d_{33} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ 2\varepsilon_{xy} \end{Bmatrix} \tag{11}$$

where d_{ij} ($d_{ij} = d_{ji}$) are the elasticity (material) constants for an orthotropic material with the material principal directions coinciding with the co-ordinate axes (*x,y*) used to describe the problem. For an isotropic material in plane stress d_{ij} are given by

$$d_{11} = d_{22} = \frac{E}{1-v^2} \quad , \quad d_{12} = d_{21} = \frac{Ev}{1-v^2}, \quad d_{33} = \frac{E}{2(1+v)}, \tag{12}$$

where *E* is Young's modulus of the material and *v* is Poisson's ratio. For plane strain problems:

$$d_{11} = d_{22} = \frac{E(1-v)}{(1+v)(1-2v)}, \quad d_{12} = d_{21} = \frac{Ev}{(1+v)(1-2v)}, \quad d_{33} = \frac{E}{2(1+v)}. \tag{13}$$

For the given problem, essential or geometric boundary conditions are

$$u = \bar{u} \quad , \quad v = \bar{v} \quad \text{on } \Gamma_u \tag{14}$$

and natural boundary conditions are

3

$$t_x = \sigma_x n_x + \sigma_{xy} n_y = \bar{t}_x \quad \text{on } \Gamma_t \tag{15}$$

$$t_y = \sigma_{xy} n_x + \sigma_y n_y = \bar{t}_y \quad \text{on } \Gamma_t \tag{16}$$

where n_x, n_y are the components of the unit normal vector **n** on the boundary Γ. Γ_u and Γ_t are portions of the boundary Γ ($\Gamma = \Gamma_u \cup \Gamma_t$). $\bar{t}_x$, $\bar{t}_y$ are specified boundary stresses or tractions, and $\bar{u}, \bar{v}$ are specified displacements. Only one element of each pair, (u, t_x) and (v, t_y) may be specified at a boundary point.

In the Ritz FEM method, the variables whose values are to be determined are approximated by piecewise continuous polynomials. The coefficients of these polynomials are obtained by minimizing the total potential energy of the system. In FEM, usually, these coefficients are expressed in terms of unknown values of primary variables. Thus, if an element has got 4 nodes, the displacement field u can be approximated as

$$u = \sum_{i=1}^{4} N_i u_i \tag{17}$$

where u_i are the nodal displacements in x-direction and N_i are the shape functions, which are functions of coordinates.

For plane elastic body, the total potential energy of an element is given by (using index notations),

$$\Pi^e = \int_{V^e} \frac{1}{2} \sigma_{ij} \varepsilon_{ij} \mathrm{d}V - \int_{V^e} f_i u_i \mathrm{d}V - \int_{\Gamma^e} t_i^* u_i \mathrm{d}S \tag{18}$$

where, V^e denotes the volume of element e, Γ^e is the boundary of domain Ω^e, σ_{ij} and ε_{ij} are the components of stress and strain tensors, respectively and f_i and $\bar{t}_i$ are the components of body force and boundary stress vectors, respectively. Note that

$$\sigma_{11} = \sigma_x, \ \sigma_{12} = \sigma_{xy}, \sigma_{22} = \sigma_y \tag{19}$$

$$f_1 = f_x, \ f_2 = f_y, t_1 = t_x, t_2 = t_y \tag{20}$$

The first term in equation (18) corresponds to strain energy stored in the element, the second represents the work potential of the body force, and the third represent the work potential of surface forces. For plane stress problems with thickness h^e, it is assumed that all quantities are independent of the thickness co-ordinates z. Hence,

$$\Pi^e = h^e \int_{\Omega^e} \frac{1}{2} (\sigma_x \varepsilon_x + \sigma_y \varepsilon_y + 2\sigma_{xy} \varepsilon_{xy}) \mathrm{d}x\mathrm{d}y$$

$$-h^e \int_{\Omega^e} (f_x u + f_y v)\mathrm{d}x\mathrm{d}y - h^e \int_{\Gamma^e} (t_x u + t_y v)\mathrm{d}s \tag{21}$$

where f_x and f_y are the body forces per unit area, and t_x and t_y are boundary forces per unit length. Equation (21) can be rewritten as

$$\Pi^e = h^e \int_{\Omega^e} \frac{1}{2} \left(\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ 2\varepsilon_{xy} \end{Bmatrix}^T \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_{xy} \end{Bmatrix} \right) \mathrm{d}x\mathrm{d}y - h^e \int_{\Omega^e} \begin{Bmatrix} u \\ v \end{Bmatrix}^T \begin{Bmatrix} f_x \\ f_y \end{Bmatrix} \mathrm{d}x\mathrm{d}y - h^e \oint_{\Gamma^e} \begin{Bmatrix} u \\ v \end{Bmatrix}^T \begin{Bmatrix} t_x \\ t_y \end{Bmatrix} \mathrm{d}s \tag{22}$$

The finite element model of the plane elasticity equations is developed using the matrix form in (22). The displacements u and v are approximated by the Lagrange family of interpolation functions (shape functions). Let u and v are approximated over Ω^e by the finite element interpolations

$$u \approx \sum_{i=1}^{n} u_i^e N_i^e(x, y) \ , v \approx \sum_{i=1}^{n} v_i^e N_i^e(x, y) \tag{23}$$

where n is the number of nodes representing the element e, N_i^e are the displacement shape functions, u_i^e and v_i^e are the nodal displacements in x- and y- directions respectively. The displacements and strains over element e are given by

$$\begin{Bmatrix} u^e \\ v^e \end{Bmatrix} = \begin{Bmatrix} \sum_{i=1}^{n} u_i^e N_i^e \\ \sum_{i=1}^{n} v_i^e N_i^e \end{Bmatrix} = \begin{bmatrix} N_1 & N_2 ... N_n & 0 & 0 .. & 0 \\ 0 & 0 .....0 & N_1 & N_2 ... N_n \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ . \\ u_n \\ v_1 \\ v_2 \\ . \\ v_n \end{Bmatrix}$$

$$= \begin{bmatrix} N_1 & 0 & N_2 & 0 & ... & N_n & 0 \\ 0 & N_1 & .0 & N_2 & .. & 0 & .N_n \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ .u_2 \\ v_2 \\ .. \\ .. \\ u_n \\ v_n \end{Bmatrix} \equiv \begin{bmatrix} N^e \end{bmatrix} \{ \Delta^e \} \tag{24}$$

and

$$\{ \varepsilon^e \} = [B^e] \{ \Delta^e \}, \{ \sigma^e \} = [D^e][B^e] \{ \Delta^e \} \tag{25}$$

where $[B^e] = [T^e][N^e]$ is called Gradient matrix and $[T^e]$ is the matrix of differential operators. Substituting these expressions for the displacements and strains into (22)

$$\Pi^e = h^e \int_{\Omega^e} \{ \Delta^e \}^T ([B^e]^T [D^e][B^e] \{ \Delta^e \}) \mathrm{d}x\mathrm{d}y - h^e \int_{\Omega^e} \{ \Delta^e \}^T [N^e]^T \begin{Bmatrix} f_x \\ f_y \end{Bmatrix} \mathrm{d}x\mathrm{d}y$$

$$-h^e \int_{\Gamma^e} \{ \Delta^e \}^T [N^e]^T \begin{Bmatrix} t_x \\ t_y \end{Bmatrix} \mathrm{d}s = \{ \Delta^e \}^T ([k^e]^T \{ \Delta^e \} - \{ f^e \} - \{ Q^e \}) \tag{26}$$

Minimizing this, *i.e.*, differentiating the above expression with respect to $\{ \Delta^e \}$, we get

$$[k^e] \ \{ \Delta^e \} = \{ f^e \} + \{ Q^e \} \tag{27}$$

where

$$[k^e] = h^e \int_{\Omega^e} [B^e]^T [D^e][B^e] dxdy ,$$

$$\{f^e\} = h^e \int_{\Omega^e} [N^e]^T \begin{Bmatrix} f_x \\ f_y \end{Bmatrix} dxdy , \quad \{Q^e\} = h^e \int_{\Gamma^e} [N^e]^T \begin{Bmatrix} t_x \\ t_y \end{Bmatrix} ds \qquad (28)$$

The element stiffness matrix $[k^e]$ is of order $2n$ x $2n$ and the elemental load vector

$$[F^e] = \{f^e\} + \{Q^e\} \qquad (29)$$

is of order $2n$ x 1 where n is the number of nodes of the element.

Shape functions or interpolation functions N_i are used in the finite element analysis to interpolate the nodal displacements of any element to any point within each element. The interpolation functions for the four nodded quadrilateral elements shown in Fig. 3 are

$$N_1 = \frac{(1-\xi)(1-\eta)}{4} , \quad N_2 = \frac{(1+\xi)(1-\eta)}{4} , N_3 = \frac{(1+\xi)(1+\eta)}{4} , N_4 = \frac{(1-\xi)(1+\eta)}{4} \qquad (30)$$

where ξ and η are the natural co-ordinates for the physical co-ordinates x and y, respectively. In natural coordinate system, the coordinates of four nodes are (-1,-1), (1,-1), (1,1) and (-1,1).

One of the earliest finite elements is a three nodded triangular element shown in Fig. 4. An arbitrarily located point P divides a triangle 1-2-3 into three sub-areas A_1, A_2, and A_3. Then, the natural coordinates of the point P are defined as ratios of areas:

$$\xi_1 = \frac{A_1}{A} \qquad \xi_2 = \frac{A_2}{A} \qquad \xi_3 = \frac{A_3}{A} \qquad (31)$$

where A is the area of triangle 1-2-3. Since $A = A_1 + A_2 + A_3$, the ξ_i are not independent. They satisfy the constraint equation

$$\xi_1 + \xi_2 + \xi_3 = 1 \qquad (32)$$

For this triangle, shape functions in terms of natural coordinates are given as

$$N_1 = \xi_1 \qquad N_2 = \xi_2 \qquad N_3 = \xi_3 \qquad (33)$$

It can be shown that displacement field obtained using these shape functions provides constant strain inside the triangular element. This element is, therefore, called constant strain triangle (CST).
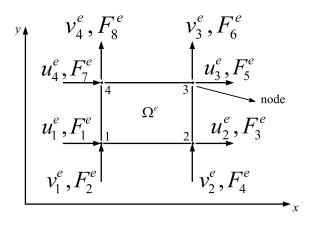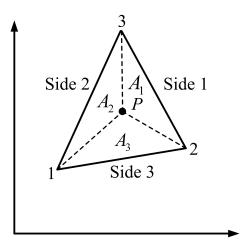

Figure 3 A quadrilateral element

Figure 4: A triangular element

The evaluation of the element matrices in equation (8) is done by using numerical integration techniques. For all area and line integrals Gauss-Quadrature rule is used. All physical domain integration is transformed to the (ξ, η) plane as shown in Fig. 5.
As a result,

$$\int_{\Omega} f(x, y)dxdy = \int_{-1}^{1}\int_{-1}^{1} f(\xi, \eta)|J|d\xi d\eta \tag{34}$$

where $|J|$ is the determinant of the Jacobian matrix of the transformation and is given by

$$|J| = \begin{vmatrix} J_1 & J_2 \\ J_3 & J_4 \end{vmatrix} = \begin{vmatrix} \sum_{i=1}^{n} N_{i,\xi}^{e}x_i & \sum_{i=1}^{n} N_{i,\xi}^{e}y_i \\ \sum_{i=1}^{n} N_{i,\eta}^{e}x_i & \sum_{i=1}^{n} N_{i,\eta}^{e}y_i \end{vmatrix} \tag{35}$$
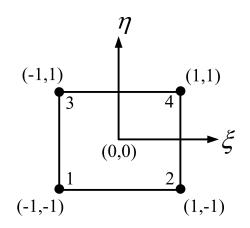

Figure 5: Natural co-ordinate system.

where n is the number of nodes of the element, $N_i^e(\xi, \eta)$ is the shape function corresponding to node i, (x_i, y_i) is the physical co-ordinates of nodes i. In writing equation (26), the isoparametric formulation has been used, *i.e.*, the interpolation functions used for the geometry variables (x, y) and field variables (u, v) are same. Also, the spatial derivatives are transformed to the (ξ, η) plane using

$$\begin{Bmatrix} N_{i,x} \\ N_{i,y} \end{Bmatrix} = [J^{-1}] \begin{Bmatrix} N_{i,\xi} \\ N_{i,\eta} \end{Bmatrix} \tag{36}$$

where

$$[J^{-1}] = \begin{bmatrix} \dfrac{J_4}{|J|} & \dfrac{-J_2}{|J|} \\ \dfrac{-J_3}{|J|} & \dfrac{J_1}{|J|} \end{bmatrix}$$

Finally, the Gauss-Quadrature scheme gives

$$\int_{-1}^{1}\int_{-1}^{1} f(\xi,\eta)d\xi d\eta = \sum_{r=1}^{n1}\sum_{s=1}^{n2} w_r w_s f(\xi_r,\eta_s) \tag{37}$$

where n_1=number of Gauss points in ξ direction,

n_2=number of Gauss points in η direction,

w_r, w_s=weights of corresponding Gauss points.

If the displacement field within each element is assumed to be bilinear then, 2×2 Gauss quadrature exactly integrates all terms of the elemental stiffness matrix.

Now, considering the evaluation of boundary integral of the type

$$Q_i^e = \int_{\Gamma^e} q_n^e N_i(s)\mathrm{d}s \tag{38}$$

where q_n^e is a known function (here boundary stress) of the distance s along the boundary Γ^e. It is not necessary to compute such integrals when a portion of Γ does not coincide with the boundary Γ of the total Ω. This is because for any interior boundary the stresses from adjacent elements cancel each other. The 2-D line integral in (x, y) plane is transformed to 1-D line integral in the natural co-ordinate plane by using the fact that along any element side one of the natural co-ordinates is constant. Thus,

$$\int_{0}^{s} f(x,y)\mathrm{d}s = \int_{-1}^{1} f(\xi)J_b d\zeta \tag{39}$$

where the boundary Jacobian $J_b = \sqrt{\left(\dfrac{\mathrm{d}x}{\mathrm{d}\zeta}\right)^2 + \left(\dfrac{\mathrm{d}y}{\mathrm{d}\zeta}\right)^2}$.

Once the element matrices are obtained, they are assembled to form the set of linear simultaneous equations, the solution of which yields the displacement field. The assembly is based on the principle of maintaining the continuity of the primary variable, in this case displacement and the equilibrium of the secondary variables, here forces and tractions.

The two types of boundary conditions are used:

1. Essential or geometric boundary conditions which are imposed on the primary variable like displacements, and

2. Natural or force boundary conditions which are imposed on the secondary variable like forces and tractions.

The force boundary conditions are imposed during the evaluation of the element matrices itself while the prescribed displacement boundary conditions are imposed after the assembly of the element matrices. Then the global system of linear equations are solved by any numerical technique to get the displacements at global nodes.

In finite element calculations, one often has a need for accurate estimates of the derivatives of the primary variable. For example, in plane stress or plane strain analysis, the primary unknowns to be computed are the displacement components of the nodes. However in many cases the strains and stresses are the prime importance, which are computed from the derivatives of the displacements. As the finite

element solutions is only an interpolate solutions, it was exact at the nodes and approximate elsewhere. Such accuracy is rare but, in general, one finds that the computed values of the primary variables are most accurate in the nodes points. Thus, for the sake of simplicity it is assumed that the element's nodal values are exact. It was observed that derivatives estimates are least accurate at the nodes, generally and most accurate at the Gauss points. These points are also called as *Barlow points* or *optimal points*. Thus, the center of the linear element is taken as the optimal position for sampling the first derivative.

As an example of finding out the stress in whole domain, consider the problem of a plate with the hole loaded by uniformly distributed tensile load (Fig. 6). This problem was solved on ANSYS, commercial FEM software. Because of the symmetry of the problem, only a quarter plate needs to be analyzed. On the line of symmetries, tractions and normal displacement components will be zero. Figure 7 shows the finite element mesh and boundary conditions. Here, quadrilateral elements have been used. Figure 8 shows the contours of longitudinal stresses. Applied tensile stress was 100 MPa. Note that in the domain far away from the hole stresses are close to this value. There is a region of high stress concentration near the highest vertical point of the circle, the stresses there being around 336 MPa. One can obtain the contours of other stress components and equivalent stresses also.
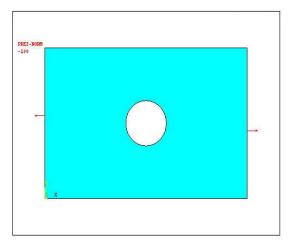

Figure 6: A plate with a hole loaded by uniformly distributed tensile load on both sides
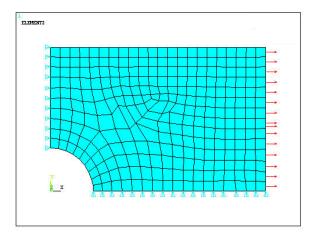

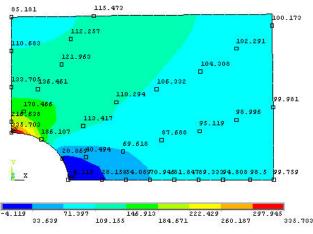**Figure 7:** Finite element mesh for solving quarter plate problem

Figure 8: Contours of longitudinal stresses

4. Conclusions

In this article a brief introduction to finite element method and finite element formulation of plane stress and plane strain problems has been described. Similar procedure is employed for the FEM formulation of the other problems. Following are the steps of the finite element formulation: (i) pre-processing that includes mesh generation (ii) obtaining the assembled system of equations, for which the elemental matrices and vectors need to be evaluated (iii) applying the boundary conditions (iv) solving the linear system of equations and (v) post-processing. The time-dependent problems are solved by treating the derivative with time by the finite difference approximation. The derivatives with respect to spatial variables are tackled in the usual way of FEM.

For further details readers can refer the lecture notes of the author on "Finite Element Methods in Engineering". These notes are available on http://www.iitg.ernet.in/scifac/qip/public_html/cd_cell/cdc_06_07.htm.