Georgia Institute of Technology
School of Civil and Environmental Engineering

Finite Element Methods

CEE6504-Spring 2020

Project Report

Group-01

Group Members:
Zichen Bai (GT ID: 903148582)
Yanzi Wu (GT ID: 903524967)

Date:

2020/4/26

Total potential energy approach

The total potential energy $\mathbf{\Pi}$ of an elastic body is defined as the sum of total strain energy (U) and the work potential (WP):

$$\mathbf{\Pi} = \text{Strain energy} + \text{Work potential} \qquad \text{Eq. 1}$$

For linear elastic materials, the strain energy per unit volume in the body is $\frac{1}{2}\boldsymbol{\sigma}^{\boldsymbol{T}}\boldsymbol{\epsilon}$. The total strain energy U is given by:

$$U = \frac{1}{2}\int_V \boldsymbol{\sigma}^{\boldsymbol{T}}\boldsymbol{\epsilon}\, d\mathbf{V} \qquad \text{Eq. 2}$$

The work potential (WP) is given by:

$$WP = -\int_V \mathbf{u}^{\boldsymbol{T}}\mathbf{f}\, d\mathbf{V} - \int_S \mathbf{u}^{\boldsymbol{T}}\mathbf{T}\, d\mathbf{S} - \sum_{\boldsymbol{i}} \boldsymbol{u}_{\boldsymbol{i}}^{\boldsymbol{T}}\, P_i \qquad \text{Eq. 3}$$

For conservative systems, of all the kinematically admissible displacement fields, those corresponding to equilibrium extremize the total potential energy. If the extremum condition is a minimum, the equilibrium state is stable.

Shape functions

The shape functions are a group of piecewise basis functions, which can be used to express the displacement at each node of each element. The piecewise basis functions can be defined as $\boldsymbol{\Phi}_{\boldsymbol{i}}(\boldsymbol{x})$ which satisfy the following relations:

$$\boldsymbol{\Phi}_{\boldsymbol{i}}(\boldsymbol{x}) = \begin{matrix} \mathbf{1} & \boldsymbol{at\ node\ i} \\ \mathbf{0} & \boldsymbol{if\ node\ j\ is\ not\ equal\ to\ i} \end{matrix} \qquad \text{Eq. 4}$$

$$\sum_{\boldsymbol{i}} \boldsymbol{\Phi}_{\boldsymbol{i}}(\boldsymbol{x}) = \mathbf{1} \qquad \text{Eq. 5}$$

The displacement field can be expressed by the shape functions, where $\boldsymbol{Q}_{\boldsymbol{i}}(\boldsymbol{x})$ is the nodal displacement.

$$\boldsymbol{u} = \sum_{\boldsymbol{i=1}}^{\boldsymbol{n}} \boldsymbol{Q}_{\boldsymbol{i}}(\boldsymbol{x})\, \boldsymbol{\Phi}_{\boldsymbol{i}}(\boldsymbol{x}) \qquad \text{Eq. 6}$$

Stiffness matrix

The stiffness matrix is applied at a set of nodal coordinates of the structure, which relates the nodal concentrated forces to the nodal displacements at the same set of coordinates. The relationship of stiffness matrix, nodal concentrated forces and nodal displacements can be described as the equation:

$$[K]\{q\} = \{f\} \qquad \text{Eq. 7}$$

K matrix can be obtained by applying the strain energy equation (Eq. 2), where $\boldsymbol{\sigma} = \mathbf{DBq}$ **and** $\boldsymbol{\epsilon} = \mathbf{Bq}$**.** D is the strain and stress matrix corresponding to the material properties. Taking 3-node element for example, the B matrix can be expressed as the form below:

$$\boldsymbol{B} = \begin{matrix} \frac{\partial N1}{\partial x} & \mathbf{0} & \frac{\partial N2}{\partial x} & \mathbf{0} & \frac{\partial N3}{\partial x} & \mathbf{0} \\ \mathbf{0} & \frac{\partial N1}{\partial y} & \mathbf{0} & \frac{\partial N2}{\partial y} & \mathbf{0} & \frac{\partial N3}{\partial y} \\ \frac{\partial N1}{\partial y} & \frac{\partial N1}{\partial x} & \frac{\partial N2}{\partial y} & \frac{\partial N2}{\partial x} & \frac{\partial N3}{\partial y} & \frac{\partial N3}{\partial x} \end{matrix} \qquad \text{Eq. 8}$$

$\mathbf{N_i}$ is the shape function of the corresponding node. Thus, the strain energy equation and stiffness matrix can be written as:

$$U_e = \frac{1}{2} q^T \int_e [\mathbf{B}^{\boldsymbol{T}}\mathbf{DBA}\, d\mathbf{x}]\mathbf{q} = \frac{1}{2} q^T kq \qquad \text{Eq. 9}$$

$$k_e = \int_e \mathbf{B}^{\boldsymbol{T}}\mathbf{DBA}\, d\mathbf{x} \qquad \text{Eq. 10}$$

The Gauss-Legendre quadrature approach can be applied to calculate the integration of k matrix. According to this method, the stiffness equation can be converted as the form below:

$$k_e = \sum_{i=1}^{n\ points} B^T * D * B * \det(J) * t * w_i \qquad \text{Eq. 11}$$

Where J is the Jacobian matrix, t is the thickness of the element, n represents the number of integration points and w_i is the integration weight corresponding to each integration point. Once the stiffness matrix of each element has been obtained, they can be assembled to find the total stiffness matrix of the structure. The assembling process is based on the connection pattern of the elements. The entries of stiffness matrix corresponding to the shared nodes will be added together.

Boundary conditions

The boundary condition is a limitation of nodal displacement. When the boundary condition has been imposed, the rows and columns associated with the boundary nodes and the degree of freedom should be cancelled.

Nodal displacements and reaction forces

The traction force on the edge of each element can be converted to nodal forces by the following equation:

$$f = \int_{-1}^{1}\int_{-1}^{1} \begin{matrix} T_x \\ T_y \end{matrix} * N^T * \det(J) * d\xi_1 d\xi_2 \qquad \text{Eq. 12}$$

By the Gauss-Legendre quadrature approach, this equation can be solved numerically as well. After the nodal forced and traction forces have been obtained and the boundary condition has been imposed, the nodal displacement can be solved by equation [K]{q} = {f}. Next, the reaction forces can be obtained by substituting the nodal displacement back to the same equation.

Elements stresses and strains

The element strain can be calculated by multiplying B matrix by the nodal displacement vector ($\boldsymbol{\epsilon} = \mathbf{Bq}$). The strain of each integration point can be obtained by substituting the integration point into the B matrix. The element stress can be calculated by multiplying the strain vector by the strain and stress matrix D ($\boldsymbol{\sigma} = \mathbf{DBq}$).

MATLAB code and development

In this project, six types of element were defined to solve the actual problems. Each element is shown in the figure below:

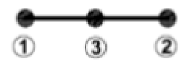
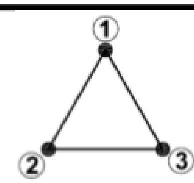
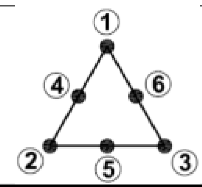
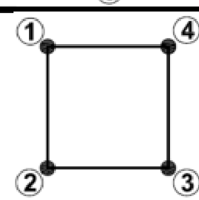
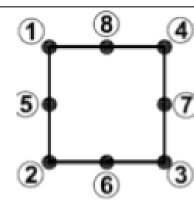
Element	Order	No of Nodes per Each Element (nElemNode)	
Bar	1	2	
	2	3	
Triangular	1	3	
	2	6	
Quadrilateral	1	4	
	2	8	

Figure 1: Types of element

The first step was to obtain the stiffness matrix of each element by applying the Gauss-Legendre quadrature approach. The shape function and its derivative of each element and the integration points and weights were obtained in the function file shapeFunctions.m and integrationPoints.m. Next, the stiffness matrix was obtained in the file elemStiff.m. In the section (B) of the file fem2D.m, the global stiffness matrix was formed by assembling each element stiffness matrix. The next step was to convert the traction forces to nodal forces and apply all nodal forces. The traction forces were converted in function file elemEquivalentLoad.m and the nodal forces were applied in section (C) of file fem2D.m. Next, the boundary conditions were imposed in section (D) of fem2D.m, and the stresses and strains were calculated in section (E). The section (F) was to store all the results and they should be outputted by the file printOutput.m. The results summary included the nodal displacements, the reaction forces, and the strains and stresses of each element at each integration point.