Introduction to Finite Element Analysis

Coordinators: Dr. Indrasen Singh; Dr. D. Subba Reddy; Dr. K. E. Prasad; Dr. Vinod Kumar and Dr. S. Chaudhary

Duration of the course: ***March 22 – 27, 2018 (6 days)***

Objectives:

This course is intended to provide graduate students, engineers, researchers, and new faculty members working in material science engineering, mechanical, civil and automotive industries as well as numerical analysts and materials scientists with the theory and applications of linear finite element analysis of problems from solid and structural mechanics.

Tentative Speakers:

1. Prof. A. R. Kumar (IIT Madras)
2. Prof. A. N. Reddy (IIT Guwahati)
3. Prof. I. Singh (IIT Indore)
4. Prof. D. Subba Reddy (IIT Indore)
5. Prof. K. E. Prasad (IIT Indore)
6. Prof. Sandeep Chaudhary (IIT Indore)

Course guidelines and time allotted are as follows:

Day 1- Introduction to Finite Element Methods (4 Lecture hours + 2 Tutorial hours)

Day 2- Finite Element formulation of 1D structural problem (4 Lecture hours + 2 Tutorial hours)

Day 3- Finite Element formulation of 2D structural problems (4 Lecture hours + 2 Tutorial hours)

Day 4- Numerical Integration (4 Lecture hours + 2 Tutorial hours)

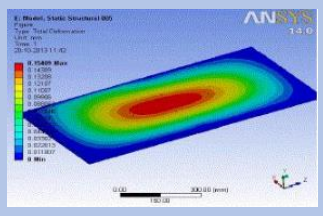
Day 5- Solving 2D structural problems and other exercise problems using ANSYS / ABAQUS, (2 Lecture hours + 4 Tutorial hours)

Day 6- Solving 2D structural using ANSYS / ABAQUS (2 Lecture hours + 4 Tutorial hours)

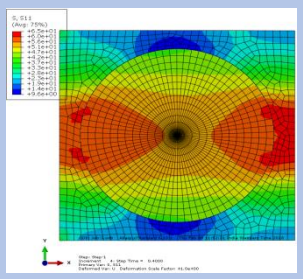
Registration Fee: *Free for TEQIP sponsored Institutes and Rs. 5,000 for others.*

Day 1- Introduction to Finite Element Methods

- Introduction to the necessity of numerical treatment of mechanical and structural problems will be discussed. At structural mechanics level, concept of principle of virtual work will be introduced with an example problem. Moreover, different branches of continuum mechanics will be briefly presented.
- Generalized deformation-strain (3D) relations will be introduced (Green-Lagrangian strain tensor).
- Cauchy's stress tensor as well as generalized balance of momentum equation for a linear elasto-dynamics will be introduced.
- Constitutive equations along with the plane stress and strain cases will be introduced.
- Generalized form of Principle of Virtual Work (PVW) for linear elasto dynamics will be derived
- Finally, example problems of plane stress and strain will be solved using commercial software



Plane stress FEA of thin plate



Plane strain analysis

Day 2- Finite Element formulation of 1D structural problem

- Formulation of the stiffness matrix and force vector from element level matrix for bar element.
- Assembly of the element stiffness matrix and force vector into the global stiffness matrices and force vector.
- Treatment of the boundary conditions: the elimination approach.
- Example problems: finding stress and strain distribution in a 1D bar problem.
- Discussion of the 2D and 3D Truss structural members.
- Transformation of element stiffness matrix and force vectors from local co-ordinates system to the global coordinate system.
- Solving 2D truss problems.
- Brief Introduction to Beam elements and frames.

Day 3- Finite Element formulation of 2D structural problems

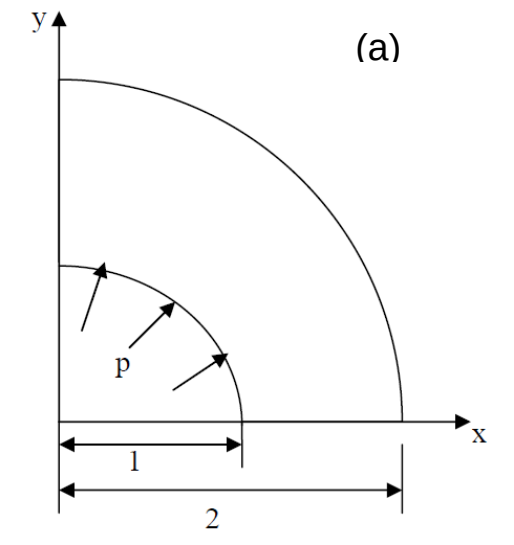
- Derivation of the stiffness matrix and force vectors from element level matrix for a four-node quadrilateral (bi-linear) element corresponding to plane stress and strain formulations.
- Assembly of the element stiffness matrix and force vector into the global stiffness matrices and force vector.
- Imposition of the boundary conditions (Discuss only elimination method).
- Example problems.

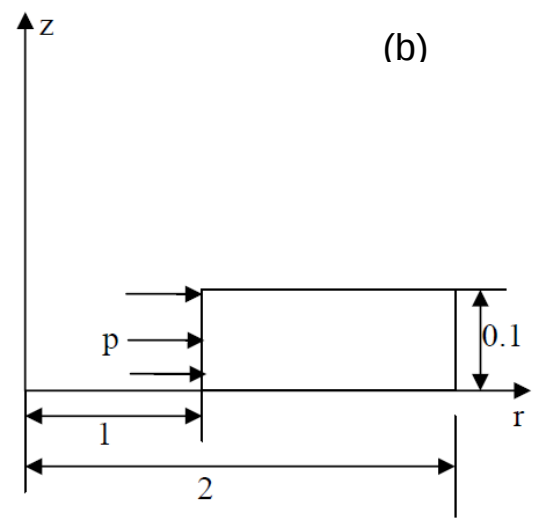
Day 4- Numerical Integration

- Isoparametric mapping.
- Shape functions in terms of natural coordinates, Derivatives of shape functions, Jacobian of mapping.
- Element stiffness matrix and force vectors in terms of natural coordinates.
- 1D and 2D Gauss quadrature methods.
- Reduced Integration and full integrations.
- Convergence of FEM.
- Example problems.

Day 5- Solving 2D structural problems using ANSYS / ABAQUS.

- Simulation of plane stress / strain simple shear and tension test using one element in Abaqus/ANSYS.
- Solve the problem of a thick cylinder subjected to internal pressure, p, using plane strain formulations with 4-node quadrilateral elements. Noting symmetry under plane strain conditions, model one quarter of the cylinder as shown in Fig. (1a). Prescribe normal displacement and shear traction as zero on the x = 0 line and y = 0 line. Assume Young's Modulus E = 80.0 GPa, Poisson's Ratio ν = 0.3, and take p=1. Compare FEM results with analytical solution (see Timoshenko and Goodier, Theory of Elasticity, pg.68-71)
- Repeat the above problem using axis symmetric formulations (refer Fig. (b)).





Day 6- Solving 2D structural using ANSYS / ABAQUS.

- Using ANSYS/Abaqus find stress concertation at the hole in a rectangular plate subjected to tension (refer Fig. 2). Assume Young's Modulus E = 80.0 GPa, Poisson's Ratio ν = 0.3. Solve this problem using plane stress and strain formulations and compare the results. Also, obtain the following:
    a. The variation of σ_{xx} stress along y-axis above the hole.
    b. The variation of σ_{yy} stress along x-axis. Compare with analytical solution (see Timoshenko and Goodier, Theory of Elasticity.).