Finite Element Method and its Applications to Solid Mechanics

Course Contents:

The class presents the basis of the finite element method. The idea of the method is explained by studying the solution of a complicated problem by replacing it by a simpler one. In the finite element method, the solution region is considered as built up of many small, interconnected subregions called finite elements. In each piece or element, a convenient approximate solution is assumed and the conditions of overall equilibrium of the structure are derived. During the course the method is applied in the field of structural mechanics, different types of engineering problems, such as heat conduction, fluid dynamics, seepage flow, and electric and magnetic fields.

Subjects covered by the lectures are:

Section 1. The Finite Element Method

Topic 1.1. Overview of the Finite Element Method

Basic concept. General applicability of the method. Engineering applications of the finite element method. General description of the finite element method. One-dimensional problems with linear interpolation model. One-dimensional problems with cubic interpolation model. Derivation of finite element equations using a direct approach. Commercial finite element program packages. Solutions using finite element software.

Section 2. Basic Procedure

Topic 2.1. Discretization of the Domain

Basic element shapes. Discretization process. Node numbering scheme. Automatic mesh generation.

Topic 2.2. Interpolation Models

Polynomial form of interpolation functions. Simplex, complex, and multiplex elements. Interpolation polynomial in terms of nodal degrees of freedom. Selection of the order of the interpolation polynomial. Convergence requirements. Linear interpolation polynomials in terms of global coordinates. Interpolation polynomials for vector quantities. Linear interpolation polynomials in terms of local coordinates. Integration of functions of natural coordinates.

Topic 2.3. Higher Order and Isoparametric Elements

Higher order one-dimensional elements. Higher order elements in terms of natural coordinates. Higher order elements in terms of classical interpolation polynomials. One-dimensional elements using classical interpolation polynomials. Two-dimensional (rectangular) elements using classical interpolation polynomials. Continuity conditions. Comparative study of elements. Isoparametric elements. Numerical integration.

Topic 2.4. Numerical Solution of Finite Element Equations

Solution of equilibrium problems. Solution of eigenvalue problems. Solution of propagation problems. Parallel processing in finite element analysis.

Section 3. Application to Solid Mechanics Problems

Topic 3.1. General Problems in Solid Mechanics and Non-linearity

Small deformation solid mechanics problems. Variational forms for non-linear elasticity. Viscoelasticity – history dependence of deformation. Classical time-independent plasticity theory. Computation of stress increments. Isotropic plasticity models. Generalized plasticity. Basic formulation of creep problems. Viscoplasticity – a generalization. Some special problems of brittle materials. Non-uniqueness and localization in elasto-plastic deformations. Non-linear quasi-harmonic field problems.

Topic 3.2. Material Constitution for Finite Deformation

Isotropic elasticity (hyperelastic, porous elastic metals). Isotropic viscoelasticity. Plasticity models. Creep models. Power-law creep. Incremental formulations. Rate constitutive models. Numerical examples.

Topic 3.3. Treatment of Constraints – Contact and Tied Interfaces

Node–node contact: Hertzian contact. Tied interfaces. Node–surface contact. Surface–surface contact.

Topic 3.4. Basic Equations and Solution Procedure

Basic equations of solid mechanics. Formulations of solid and structural mechanics. Formulation of finite element equations (static analysis). Nature of finite element solutions.

Topic 3.5. Analysis of Trusses, Beams, and Frames

Space truss element. Beam element. Space frame element. Characteristics of stiffness matrices.

Topic 3.6. Analysis of Plates

Triangular membrane element. Quadratic triangle element. Rectangular plate element (in-plane forces). Bending behavior of plates. Finite element analysis of plates in bending. Triangular plate bending element. Analysis of three-dimensional structures using plate elements.

Topic 3.7. Analysis of Three-Dimensional Problems

Tetrahedron element. Hexahedron element. Analysis of solids of revolution.

Topic 3.8. Dynamic Analysis

Dynamic equations of motion. Consistent and lumped mass matrices. Consistent mass matrices in a global coordinate system. Free vibration analysis. Dynamic response using finite element method. Nonconservative stability and flutter problems. Substructures method.

Section 4. Application to Heat Transfer Problems

Topic 4.1. Formulation and Solution Procedure

Basic equations of heat transfer. Statement of the problem. One-dimensional problems. Two-dimensional problems. Three-dimensional problems.

Section 5. Application to Fluid Mechanics Problems

Topic 5.1. Basic Equations of Fluid Mechanics

Basic characteristics of fluids. Methods of describing the motion of a fluid. Continuity equation. Solution procedure. Inviscid and incompressible flows. Viscous and non-Newtonian flows.

Section 6. Solution and Applications of Quasi-Harmonic Equations

Topic 6.1. Solution of Quasi-Harmonic Equations

Finite element equations for steady-state problems. Solution of Poisson's equation. Transient field problems.

Section 7. ABAQUS, ANSYS Software, FEMAP and MATLAB Programs for Finite Element Analysis

Topic 7.1. Finite Element Analysis Using ABAQUS

Starting Abaqus/CAE. Creating part. Creating material. Defining the assembly. Configuring analysis. Applying boundary conditions and loads to the model. Meshing the model. Partitioning the model. Assigning the Abaqus element type. Seeding the part instances. Creating an analysis job. Checking the model. Running the analysis. Postprocessing with Abaqus/CAE. Generating solution contours. Generating report of Field Outputs. Main principles of operation with documentation for Abaqus.

Topic 7.2. Finite Element Analysis Using ANSYS

GUI Layout in ANSYS. Terminology. Finite element Discretization. System of units. Stages in solution.

Topic 7.3. Finite Element Analysis Using MSC Visual NASTRAN

Software package structure. User's interface. Means of model geometry creation. Modeling of constructions with finite elements. Means of the automated mesh creation. Loads and constraints. Display of model and results. Linear static construction analysis. Nonlinear static construction analysis. The stability analysis. Dynamic construction analysis. Construction optimization. Error analysis.

Topic 7.4. MATLAB Programs for Finite Element Analysis

Solution of linear system of equations using Choleski method. Incorporation of boundary conditions. Analysis of space trusses. Analysis of plates subjected to in-plane loads using CST elements. Analysis of three-dimensional structures using CST elements. Temperature distribution in one-dimensional fins. Temperature distribution in one-dimensional fins including radiation heat transfer. Two-dimensional heat transfer analysis. Confined fluid flow around a cylinder using potential function approach. Torsion analysis of shafts.

Learning Outcomes of the Course:

Through a deep understanding of the theory and the realization of a project, the student will be able to apply numerical tools to use the finite element method and study a wide class of mechanical problems. In particular:

- He will have a deep understanding of the finite element method and will be able to apply it, finish and explain the results.
- He will be able to apply the FEM method realization in different software.
- He will be able to apply the method to a wide class of problems.
- He will be able to analyse and to evaluate (justify and criticise) this method.
- He will be able to analyse new problems.

Planned Learning Activities:

Topic 1.1. Overview of the Finite Element Method

Problems 1 – 5 Obtain an approximate displacement equation for the simply supported beam of length H and section property EI shown in Figure 1. Assume that the trial displacement equation is $y(x)=A \sin \pi x/H$. Compare the deflection at the center with the theoretical value $y=-5WH^4/384EI$. The governing differential equation is

$$EI\frac{d^2y}{dx^2} - \frac{Wx(H-x)}{2} = 0$$

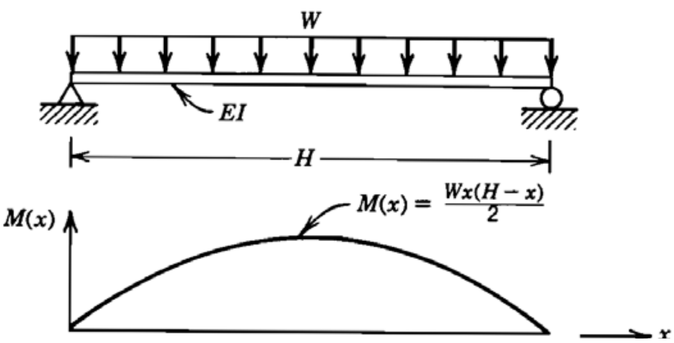

Figure 1.1

1. Evaluate A by minimizing the integral

$$\Pi = \int_0^H \left[\frac{EI}{2}\left(\frac{dy}{dx}\right)^2 + \left(\frac{Wx(H-x)}{2}\right)y\right]dx$$

2. Evaluate A by requiring that the residual vanish at (a) x=H/3, and (b) x=H/2.
3. Evaluate A using the subdomain method.
4. Evaluate A using Galerkin's method.
5. Evaluate A using the least squares method.

Topic 2.1. Discretization of the Domain

Problems:

1. The nodal coordinates X_i and X_j and the nodal values of Φ_i and Φ_j for several linear elements are given in Figure 2. Evaluate φ at the given value of x. The x values are in centimeters, and Φ_i and Φ_j are in degrees Celsius.

	x	X_i	X_j	Φ_i	Φ_j
(a)	0.8	0.0	1.5	60	43
(b)	3.6	3.0	4.5	27	33
(c)	7.1	6.5	7.5	63	51
(d)	1.8	0.5	3.0	0	−15
(e)	2.2	1.0	3.0	60	67

Figure 2

2. Evaluate $\frac{d\varphi}{dx}$ for the corresponding element in problem 1.
3. The shape functions for the quadratic element shown in Figure 3 are

$$N_i - \frac{2}{L^2}(x - X_j)(x - X_k)$$

$$N_j = \frac{4}{L^2}(x \quad X_i)(x \quad X_k)$$

$$N_k = \frac{2}{L^2}(x - X_i)(x - X_j)$$

(a) Show that these shape functions equal one at their own node and are zero at the other two nodes. Also show that the shape functions sum to one.
(b) Show that the derivatives of N_i, N_j and N_k with respect to x sum to zero.

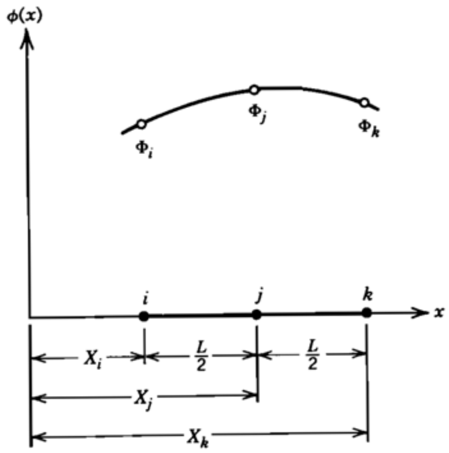

Figure 3

Topic 2.3. Higher Order and Isoparametric Elements

Problems:

1. Evaluate one of the following integrals related to the force vector for the rectangular element.

a. $\int_A QN_i dA$ b. $\int_A QN_j dA$ c. $\int_A QN_m dA$

2. Write the transformation equations that apply between the xy- and st- coordinate systems of the rectangular element. Use these equations and the chain rule equations

$$\frac{\partial N_\beta}{\partial s} = \frac{\partial N_\beta}{\partial x}\frac{\partial x}{\partial s} + \frac{\partial N_\beta}{\partial y}\frac{\partial y}{\partial s}$$

$$\frac{\partial N_\beta}{\partial t} = \frac{\partial N_\beta}{\partial x}\frac{\partial x}{\partial t} + \frac{\partial N_\beta}{\partial y}\frac{\partial y}{\partial t}$$

Topic 3.5. Analysis of Trusses, Beams, and Frames

Problems:

1. Obtain the final system of finite element equations for the nodal deflections of the stepped beam shown in Figure 4. Solve the equationsand calculate the deflection at x=3H/16.

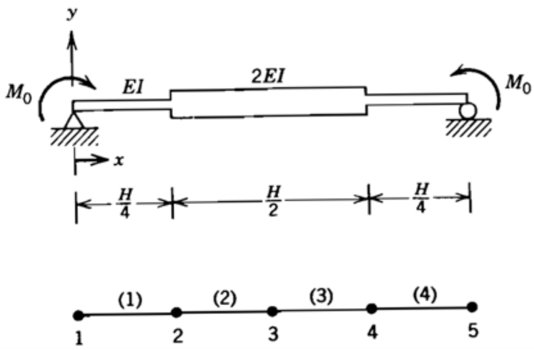

Figure 4

2. Develop and solve the system of finite element equations for an approximate solution to the differential equation $d^2\varphi/dx^2+Q=0$ using the value for Q and the boundary conditions given in the following table. Divide the interval [0,2] into four elements, each with a length of 0.5 cm. The nodes and elements are numbered as shown in Figure 5.

	Q	$\phi(0)$	$\phi(2)$
(a)	4	0	3.0
(b)	6	1	2.0
(c)	−3	2	0.5
(d)	2	−1	1.5
(e)	−5	3	−2.0

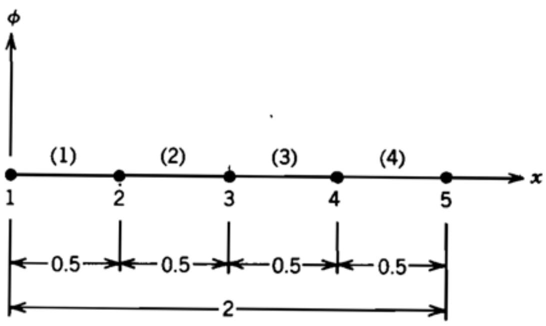

Figure 5

3. Each joint of the structural system is a pinned joint. The node numbers and element numbers are given on the Figure 6. The cross-sectional area of each member in cm^2 is underlined. Each member is made from steel, E=20×10^6 N/cm^2. All lengths are given in centimetersю Calculate the unknown nodal displacements and the axial forces produce a system that is equilibrium. There is no temperature change.

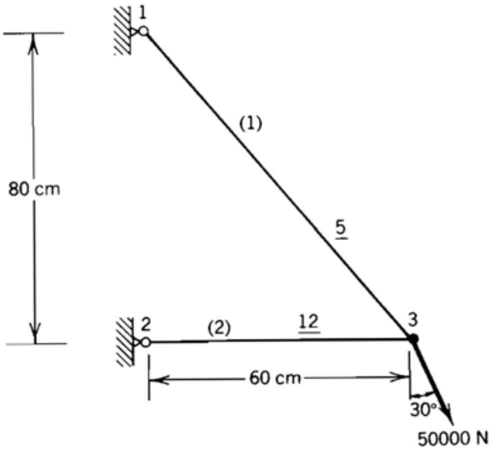

Figure 6

4. Calculate the nodal displacements and the internal member forces for the beam loadings shown in Figure 7. Construct the shear force and bending moment diagram. Check the accuracy of your solution by considering the equilibrium. Use the element and nodes shown. Use E=20×10^6 N/cm^2 and l=8000cm^2.

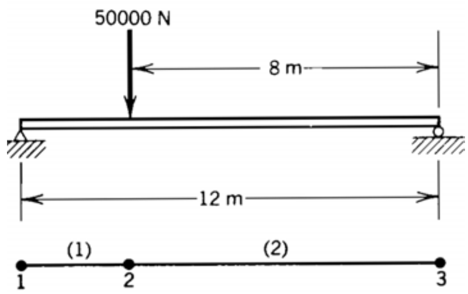

Figure 7

5. Calculate the nodal forces (internal) for member two of the configuration shown in Figure 8.

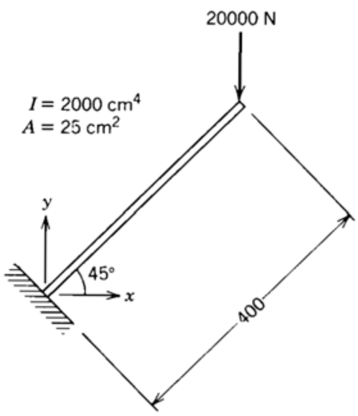

Figure 8

Topic 3.6. Analysis of Plates

Problems:

1. Verify that N_i for the triangular element is equal to one at node I and equal to zero at nodes j and k.
2. Verify that the shape functions for the triangular element sum to one, that is, $N_i+N_j+N_k=1$. Comment on the behavior of the following summations:
   (i) $a_i+a_j+a_k$.
   (ii) $b_i+b_j+b_k$.
   (iii) $c_i+c_j+c_k$.
3. Evaluate $\{f_p^{\{e\}}\}$ for the surface stress p_z shown acting on the rectangular element in Figure 9.

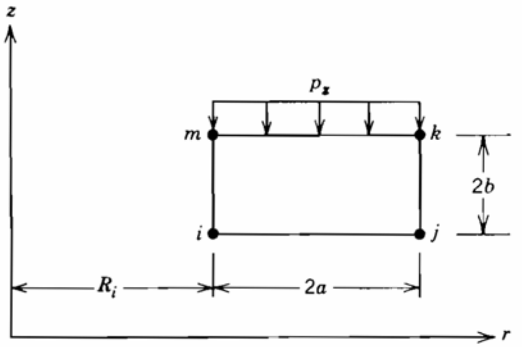

Figure 9

Topic 7.1. Finite Element Analysis Using ABAQUS

Problems:

1. Truss example.
   a. Analysis of an overhead hoist.
   b. Static calculation of a console beam.
2. Dynamic problem of the free oscillations of the fixed beam.
3. The analysis of frequency response. The round plate rigidly anchored on a contour.
4. 2D example: a rectangular plate with a hole in 2D plane stress.
5. 3D example: analysis of 3D elastic solid.
   a. Modeling of a static loading of rotary loops.
   b. A fixed beam bending. Various types of elements usage. Change of grid settings.
6. The contact problem.
   a. A solid full-sphere falling on the free end of the fixed beam. Various initial conditions.
   b. Interaction of the fixed beam and the elastic cross-loaded cylinder lying on it. Record of results in the video clip.
7. Static linear problem.
   a. Heating and cooling of a fixed beam. Research of originating temperature stresses.
   b. Electrostatic interaction of a fixed beam with charged bodies of the various geometrical form.
8. Cyclic loading of a rectangular plate with a central circular hole.
9. Import/export of geometry and models.
10. Creation of scripts in Abaqus.

Topic 7.3. Finite Element Analysis Using MSC Visual NASTRAN

Problem: The analysis of a rectangular caisson.

Classical problem about loading a rectangular caisson with a perpendicular force, applied to the tip rib, with no torsion. The caisson is fixed in accordance with symmetry conditions (see Figure 10). Caisson's parameters:

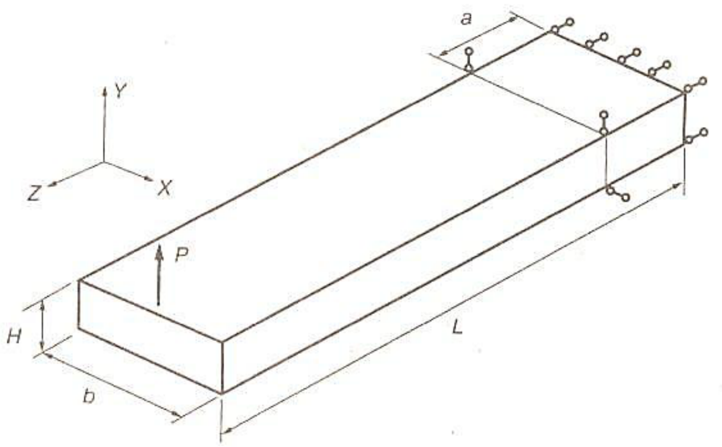

Figure 10

Length L=6050 mm; width b=1210 mm; height H=300 mm; length of centre wing a=550 mm; thickness of a skin δ_{skin}=2.0 mm; thickness of a spar wall $\delta_{spar\ wall}$=2.5 mm; the total area of stringers on the upper surface $\Sigma\Phi_{str}$=2000 mm^2; the total area of stringers on the inner surface $\Sigma\Phi_{str}$=2000 mm^2; stringer's eccentricity h_{str}=50 mm; the area of a belt of each spar F_{belt}=220 mm^2 ; belt's eccentricity are null; material parameters: E=72000 MPa; v=0.3.

Recommended or Required Readings:

1. Rao S. S. The Finite Element Method In Engineering. Amsterdam, Boston, Heidelberg, London, New York, Oxford, Paris, San Diego, San Francisco, Singapore, Sydney, Tokyo: Elsevier, 2011. 727p.
2. Zienkiewicz O.C., Taylor R.L. The Finite Element Method for Solid and Structural Mechanics. Amsterdam, Boston, Heidelberg, London, New York, Oxford, Paris, San Diego, San Francisco, Singapore, Sydney, Tokyo: Elsevier, 2005. 648p.
3. Professor Suvranu De. Abaqus Handout. Rensselaer Polytechnic Institute. Department of Mechanical, Aerospace and Nuclear Engineering. 61 p.
4. Nushtaev D. V. Abaqus. The manual for beginners. The step by step instruction. TESIS, Moscow 2010, 78 p.
5. Abaqus. Complex application in engineering tasks. TESIS, Moscow, 2008. 99 p.
6. SIMULIA Abaqus/CAE User`s Manual
7. SIMULIA Abaqus Example Problems Manual
8. Segerlind L. J. Applied FiniteElement Analysis. Inc. New York/London/Sydney/Toronto: John Wiley and Sons, 1976. 393p.
9. Rychkov S.P. Construction modeling in MSC visual NASTRAN for Windows. Moscow: NT Press. 2004. 552p.

10. Kuna M. Finite Elements in Fracture Mechanics. Theory – Numerics – Applications. Springer Dordrecht Heidelberg New York London. V. 201
11. Chen H.F., Ponter A.R.S. Shakedown and limit analyses for 3-D structures using the linear matching method. International Journal of Pressure Vessels and Piping, 2001, v. 78: 443-451
12. Chen H.F., Engelhardt M.J., Ponter A.R.S. Linear matching method for creep rupture assessment. International Journal of Pressure Vessels and Piping,2003, v.80:213-220
13. Chen H.F., Ponter A.R.S. Integrity assessment of a 3D tubeplate using the linear matching method. Part 1. Shakedown, reverse plasticity and ratcheting. International Journal of Pressure Vessels and Piping, 2005, v. 82: 85-94
14. Chen H.F., Ponter A.R.S., Ainsworth R.A. The linear matching method applied to the high temperature life integrity of structures. Part 2. Assessments beyond shakedown involving changing residual stress fields. International Journal of Pressure Vessels and Piping, 2006, v. 83: 136-147
15. Chen H.F., Ponter A.R.S. On the behaviour of a particulate metal matrix composite subjected to cyclic temperature and constant stress. Computational Materials Science, 2005, v. 34:425-441
16. Benoit A., Maitournam M.H., Remy L., Oger F. Cyclic behaviour of structures under thermomechanical loadings: Application to exhaust manifolds. International Journal of Fatigue, 2012, v. 38:65-74
17. Moes N., Dolbow J.,Belytschko T. A finite element method for crack growth without remeshing. International Journal for Numerical Methods in Engineering, 1999, v. 46:131-150
18. Hansbo A., Hansbo P. A finite element method for the simulation of strong and weak discontinuities in solid mechanics. Computer Method in Applied Mechanics and Engineering, 2004, v. 193: 3524-3540
19. Song J., Areias P., Belytschko T. A method for dynamic crack and shear band propagation with phantom nodes. International Journal for Numerical Methods in Engineering, 2006, v. 67: 868-893
20. Giner E., Sukumar N., Taracon J.E., Fuenmayor F.J. An Abaqus implementation of the extended finite element method. Engineering Fracture Mechanics, 2009, v. 76: 347-368
21. Extended Finite Element Method (XFEM), 2009, Abaqus 6.9 Update Seminar, Dassault Systemes
22. Abaqus 6.9 – Extended Functionality (EF) Overview, Abaqus 6.9 – EF Update Webinar, Dassault Systemes
23. J. Ciambella, M. Destrade, R. W. Ogden, On the ABAQUS FEA model of finite viscoelasticity. Rubber Chem. Technol., 2009, 82(2), 184–193