

Introduction to Finite Element Analysis **Using MATLAB and Abaqus**

Target Audience

This course is intended for undergraduate and graduate engineering students, as well as practicing engineers, who seek a foundational understanding of the finite element method (FEM) along with its implementation using MATLAB and application using Abaqus. It is suitable for learners with prior exposure to basic mechanics, mathematics, and programming who want to develop both theoretical insight and practical simulation skills. The course is particularly relevant for those preparing for roles or evaluations in computational mechanics, structural analysis, and CAE, where a balanced understanding of theory and software application is required.

Course Outcomes

- Understand the fundamentals of the finite element method
 - Apply matrix-based analysis to trusses, beams, and frames
 - Analyze stress and strain relationships in materials
 - Formulate problems using weighted residual methods
 - Develop finite element approximations for engineering problems
 - Apply numerical integration techniques in FEM
 - Implement basic finite element codes using MATLAB
 - Perform simulations using Abaqus (interactive and keyword approaches)
 - Interpret and validate simulation results
 - Prepare for further study or application of FEM in engineering practice
-

Course Objectives

- Provide a structured introduction to finite element theory
 - Develop understanding of mathematical and numerical foundations
 - Enable implementation of FEM using MATLAB
 - Introduce practical simulation workflows using Abaqus
 - Reinforce learning through examples and problem-solving
 - Build competency in interpreting and validating engineering results
 - Support preparation for academic and industry-level FEM applications
-

Course Outline

The course comprises 56-hours of theory and labs and is divided into 11 different chapters. Each chapter will be followed by hands-on lab exercises to reinforce learning and gauge understanding of the topics covered.

Table of Contents

Chapter 1 Introduction

- 1.1 Prologue
 - 1.2 Finite Element Analysis and the User
 - 1.3 Aim of the Book
 - 1.4 Book Organization
-

Chapter 2 Bar Element

- 2.1 Introduction
- 2.2 One-Dimensional Truss Element
 - 2.2.1 Formulation of the Stiffness Matrix: The Direct Approach
 - 2.2.2 Two-Dimensional Truss Element
- 2.3 Global Stiffness Matrix Assembly
 - 2.3.1 Discretization
 - 2.3.2 Elements' Stiffness Matrices in Local Coordinates
 - 2.3.3 Elements' Stiffness Matrices in Global Coordinates
 - 2.3.3.1 Element 1
 - 2.3.3.2 Element 2
 - 2.3.3.3 Element 3
 - 2.3.4 Global Matrix Assembly
 - 2.3.4.1 Only Element 1 Is Present
 - 2.3.4.2 Only Element 2 Is Present
 - 2.3.4.3 Only Element 3 Is Present
 - 2.3.5 Global Force Vector Assembly
- 2.4 Boundary Conditions
 - 2.4.1 General Case
- 2.5 Solution of the System of Equations
- 2.6 Support Reactions
- 2.7 Members' Forces
- 2.8 Computer Code: truss.m

- 2.8.1 Data Preparation
 - 2.8.1.1 Nodes Coordinates
 - 2.8.1.2 Element Connectivity
 - 2.8.1.3 Material and Geometrical Properties
 - 2.8.1.4 Boundary Conditions
 - 2.8.1.5 Loading
- 2.8.2 Element Matrices
 - 2.8.2.1 Stiffness Matrix in Local Coordinates
 - 2.8.2.2 Transformation Matrix
 - 2.8.2.3 Stiffness Matrix in Global Coordinates
 - 2.8.2.4 “Steering” Vector
- 2.8.3 Assembly of the Global Stiffness Matrix
- 2.8.4 Assembly of the Global Force Vector
- 2.8.5 Solution of the Global System of Equations
- 2.8.6 Nodal Displacements
- 2.8.7 Element Forces
- 2.8.8 Program Scripts
- 2.9 Problems
 - 2.9.1 Problem 2.1
 - 2.9.2 Problem 2.2
- 2.10 Analysis of a Simple Truss with Abaqus
 - 2.10.1 Overview of Abaqus
 - 2.10.2 Analysis of a Truss with Abaqus Interactive Edition
 - 2.10.2.1 Modeling
 - 2.10.2.2 Analysis
 - 2.10.3 Analysis of a Truss with Abaqus Keyword Edition

Chapter 3 Beam Element

- 3.1 Introduction
- 3.2 Stiffness Matrix
- 3.3 Uniformly Distributed Loading
- 3.4 Internal Hinge
- 3.5 Computer Code: beam.m
 - 3.5.1 Data Preparation
 - 3.5.1.1 Nodes Coordinates

- 3.5.1.2 Element Connectivity
 - 3.5.1.3 Material and Geometrical Properties
 - 3.5.1.4 Boundary Conditions
 - 3.5.1.5 Internal Hinges
 - 3.5.1.6 Loading
 - 3.5.1.7 Stiffness Matrix
 - 3.5.2 Assembly and Solution of the Global System of Equations
 - 3.5.3 Nodal Displacements
 - 3.5.4 Element Forces
 - 3.6 Problems
 - 3.6.1 Problem 3.1
 - 3.6.2 Problem 3.2
 - 3.6.3 Problem 3.3
 - 3.7 Analysis of a Simple Beam with Abaqus
 - 3.7.1 Interactive Edition
 - 3.7.2 Analysis of a Beam with Abaqus Keyword Edition
-

Chapter 4 Rigid Jointed Frames

- 4.1 Introduction
- 4.2 Stiffness Matrix of a Beam–Column Element
- 4.3 Stiffness Matrix of a Beam–Column Element in the Presence of Hinged End
- 4.4 Global and Local Coordinate Systems
- 4.5 Global Stiffness Matrix Assembly and Solution for Unknown Displacements
- 4.6 Computer Code: frame.m
 - 4.6.1 Data Preparation
 - 4.6.1.1 Nodes Coordinates
 - 4.6.1.2 Element Connectivity
 - 4.6.1.3 Material and Geometrical Properties
 - 4.6.1.4 Boundary Conditions
 - 4.6.1.5 Internal Hinges
 - 4.6.1.6 Loading
 - 4.6.2 Element Matrices
 - 4.6.2.1 Stiffness Matrix in Local Coordinates
 - 4.6.2.2 Transformation Matrix
 - 4.6.2.3 Stiffness Matrix in Global Coordinates

- 4.6.2.4 “Steering” Vector
 - 4.6.2.5 Element Loads
 - 4.6.3 Assembly of the Global Stiffness Matrix
 - 4.6.4 Solution of the Global System of Equations
 - 4.6.5 Nodal Displacements
 - 4.6.6 Element Forces
 - 4.7 Analysis of a Simple Frame with Abaqus
 - 4.7.1 Interactive Edition
 - 4.7.2 Keyword Edition
-

Chapter 5 Stress and Strain Analysis

- 5.1 Introduction
 - 5.2 Stress Tensor
 - 5.3 Deformation and Strain
 - 5.4 Stress–Strain Constitutive Relations
 - 5.5 Solved Problems
-

Chapter 6 Weighted Residual Methods

- 6.1 Introduction
 - 6.2 General Formulation
 - 6.3 Galerkin Method
 - 6.4 Weak Form
 - 6.5 Integrating by Part over Two and Three Dimensions (Green Theorem)
 - 6.6 Rayleigh Ritz Method
-

Chapter 7 Finite Element Approximation

- 7.1 Introduction
 - 7.2 General and Nodal Approximations
 - 7.3 Finite Element Approximation
 - 7.4 Basic Principles for the Construction of Trial Functions
 - 7.5 Two-Dimensional Finite Element Approximation
 - 7.6 Shape Functions of Some Classical Elements for C0 Problems
-

Chapter 8 Numerical Integration

- 8.1 Introduction

- 8.2 Gauss Quadrature
 - 8.3 Integration over a Reference Element
 - 8.4 Integration over a Triangular Element
 - 8.5 Solved Problems
-

Chapter 9 Plane Problems

- 9.1 Introduction
 - 9.2 Finite Element Formulation for Plane Problems
 - 9.3 Spatial Discretization
 - 9.4 Constant Strain Triangle
 - 9.5 Linear Strain Triangle
 - 9.6 The Bilinear Quadrilateral
 - 9.7 The 8-Node Quadrilateral
 - 9.8 Solved Problem with MATLAB
-

Chapter 10 Axisymmetric Problems

- 10.1 Definition
 - 10.2 Strain–Displacement Relationship
 - 10.3 Stress–Strain Relations
 - 10.4 Finite Element Formulation
 - 10.5 Programming
 - 10.6 Analysis with Abaqus
-

Chapter 11 Thin and Thick Plates

- 11.1 Introduction
- 11.2 Thin Plates
- 11.3 Thick Plate Theory or Mindlin Plate Theory
- 11.4 Linear Elastic Finite Element Analysis of Plates
- 11.5 Boundary Conditions
- 11.6 Computer Program for Thick Plates Using the 8-Node Quadrilateral
- 11.7 Analysis with Abaqus