Finite Element Analysis of Composite Materials Using Abaqus

Target Audience

This course is designed for advanced undergraduate and graduate students in mechanical, aerospace, civil, and related engineering disciplines, as well as practicing engineers seeking specialization in composite materials and finite element analysis. It is particularly suitable for learners with prior knowledge of mechanics of materials and basic finite element concepts who aim to develop deeper expertise in composite behavior, damage mechanics, and advanced simulation using Abaqus. The course is aligned with industry and research-oriented applications, helping learners prepare for advanced CAE roles, simulation-based design tasks, and technical evaluations involving composite structures.

Course Outcomes

- Understand the mechanics of orthotropic and anisotropic composite materials
- Apply finite element procedures to analyze composite structures using Abaqus
- Analyze laminate behavior including elasticity, strength, and failure criteria
- Evaluate buckling and free-edge stress effects in composite laminates
- Apply computational micromechanics techniques for material modeling
- Model viscoelastic behavior of composite materials
- Analyze continuum and discrete damage mechanisms in composites
- Simulate delamination using cohesive zone and fracture mechanics approaches
- Apply fatigue analysis methods for composite materials
- Develop and implement user-defined material models and scripts in Abaqus
- Troubleshoot and validate advanced composite simulations

Course Objectives

- Provide a structured understanding of composite material mechanics and FEM integration
- Develop proficiency in applying Abaqus for advanced composite analysis
- Enable learners to model complex behaviors such as damage, fatigue, and delamination
- Strengthen analytical and computational skills through detailed examples
- Reinforce concepts through problem-solving and simulation-based exercises
- Prepare learners for advanced simulation roles and research applications

- Build capability to extend Abaqus functionality using scripting and subroutines

Course Outline

The course comprises 48-hours of theory and labs and is divided into 12 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 Mechanics of Orthotropic Materials

- Lamina Coordinate System
- Displacements
- Strain
- Stress
- Contracted Notation
    - Alternate Contracted Notation
- Equilibrium and Virtual Work
- Boundary Conditions
    - Traction Boundary Conditions
    - Free Surface Boundary Conditions
- Continuity Conditions
    - Traction Continuity
    - Displacement Continuity
- Compatibility
- Coordinate Transformations
    - Stress Transformation
    - Strain Transformation
- Transformation of Constitutive Equations
- 3D Constitutive Equations
    - Anisotropic Material
    - Monoclinic Material
    - Orthotropic Material
    - Transversely Isotropic Material
    - Isotropic Material
- Engineering Constants

  - Restrictions on Engineering Constants
- From 3D to Plane Stress Equations
- Apparent Laminate Properties

Chapter 2 Introduction to Finite Element Analysis

- Basic FEM Procedure
  - Discretization
  - Element Equations
  - Approximation over an Element
  - Interpolation Functions
  - Element Equations for a Specific Problem
  - Assembly of Element Equations
  - Boundary Conditions
  - Solution of the Equations
  - Solution Inside the Elements
  - Derived Results
- General Finite Element Procedure
- Solid Modeling, Analysis, and Visualization
  - Interacting with Abaqus
  - Model Geometry
  - Manual Meshing
  - Solid Modeling
  - The CAE Window
  - Material and Section Properties
  - Assembly
  - Solution Steps
  - Loads
  - Boundary Conditions
  - Meshing and Element Type
  - Solution Phase
  - Post-processing and Visualization
- Interactions and Constraints
  - Tie Constraint
  - Rigid Body Constraint

- Display Body
- Coupling
- Adjust Points
- MPC Constraint
- Shell-to-solid Coupling
- Embedded Region
- Multi-point Constraint Equations

Chapter 3 Elasticity and Strength of Laminates

- Kinematic of Shells
  - First-Order Shear Deformation Theory
  - Kirchhoff Theory
  - Simply Supported Boundary Conditions
- Finite Element Analysis of Laminates
  - Element Types and Naming Convention
  - Thin (Kirchhoff) Shell Elements
  - Thick Shell Elements
  - General-purpose (FSDT) Shell Elements
  - Continuum Shell Elements
  - Sandwich Shells
  - Nodes and Curvature
  - Drilling Rotation
  - A, B, D, H Input Data for Laminate FEA
  - Equivalent Orthotropic Input for Laminate FEA
  - LSS for Multi-directional Laminate FEA
  - FEA of Ply Drop-Off Laminates
  - FEA of Sandwich Shells
  - Element Coordinate System
- Failure Criteria
  - 2D Failure Criteria
  - 3D Failure Criteria
- Predefined Fields

Chapter 4 Buckling

- Eigenvalue Buckling Analysis
    - Imperfection Sensitivity
    - Asymmetric Bifurcation
    - Post-critical Path
- Continuation Methods

Chapter 5 Free Edge Stresses

- Poisson’s Mismatch
    - Interlaminar Force
    - Interlaminar Moment
- Coefficient of Mutual Influence
    - Interlaminar Stress due to Mutual Influence

Chapter 6 Computational Micromechanics

- Analytical Homogenization
    - Reuss Model
    - Voigt Model
    - Periodic Microstructure Micromechanics
    - Transversely Isotropic Averaging
- Numerical Homogenization
- Local-Global Analysis
- Laminated RVE

Chapter 7 Viscoelasticity

- Viscoelastic Models
    - Maxwell Model
    - Kelvin Model
    - Standard Linear Solid
    - Maxwell-Kelvin Model
    - Power Law
    - Prony Series
    - Standard Nonlinear Solid
    - Nonlinear Power Law
- Boltzmann Superposition

    - Linear Viscoelastic Material
    - Un-aging Viscoelastic Material
- Correspondence Principle
- Frequency Domain
- Spectrum Representation
- Micromechanics of Viscoelastic Composites
    - One-Dimensional Case
    - Three-Dimensional Case
- Macromechanics of Viscoelastic Composites
    - Balanced Symmetric Laminates
    - General Laminates
- FEA of Viscoelastic Composites

Chapter 8 Continuum Damage Mechanics

- One-Dimensional Damage Mechanics
    - Damage Variable
    - Damage Threshold and Activation Function
    - Kinetic Equation
    - Statistical Interpretation of the Kinetic Equation
    - One-Dimensional Random-Strength Model
    - Fiber Direction, Tension Damage
    - Fiber Direction, Compression Damage
- Multidimensional Damage and Effective Spaces
- Thermodynamics Formulation
    - First Law
    - Second Law
- Kinetic Law in Three-Dimensional Space
    - Return-Mapping Algorithm
- Damage and Plasticity

Chapter 9 Discrete Damage Mechanics

- Overview
- Approximations
- Lamina Constitutive Equation

- Displacement Field
  - Boundary Conditions for $\Delta T = 0$
  - Boundary Conditions for $\Delta T \neq 0$
- Degraded Laminate Stiffness and CTE
- Degraded Lamina Stiffness
- Fracture Energy
- Solution Algorithm
  - Lamina Iterations
  - Laminate Iterations

Chapter 10 Delaminations

- Cohesive Zone Model
  - Single Mode Cohesive Model
  - Mixed-mode Cohesive Model
- Virtual Crack Closure Technique
- Determination of CZM Parameters
  - Elastic Loading
  - Softening Behavior
  - Linear CZM
  - Tabular CZM
  - Mixed-mode Parameter
- Modeling Considerations
  - Damage Stabilization Cohesive
  - Damping
  - Symmetry
  - ENCASTRE
  - Mesh Refinement
  - Abaqus CAE and Input File
- Script for Example
  - DCBCZMparams.py

Chapter 11 Fatigue

- Temperature Dependent Properties
- The Quasi-Static Problem

- The First Cycle of a Fatigue Test
- Fatigue Damage Criterion
- Master Paris Law
- Thermal and Fatigue Damage Prediction
    - Implementation
- Thermomechanical Equivalence
    - Damage Equivalence
    - Defect Nucleation Equivalence
- Software Implementation
- Uniaxial Mechanical Test

Chapter 12 Abaqus Programmable Features

- User Materials in Abaqus Standard
- UMAT for Linear Elastic Shells
    - User Subroutine Interface
    - State Variables and Constants
    - User's Code
    - UMATS8R5.FOR
- UMAT for Orthotropic Viscoelasticity
    - User Subroutine Interface
    - State Variables and Constants
    - User's Code
    - UMAT3DVISCO.FOR
- Constraint Equations and Periodicity
    - Ex 6.4.py
    - PBC 2d.py
    - Mesoscale Effective Stress from FEA-RVE Solution
- UMAT 1D
    - User Subroutine Interface
    - State Variables and Constants
    - User's Code
    - UMAT1D83.FOR
- UMAT Plane Stress
    - User Subroutine Interface

- State Variables and Constants
  - User's Code
  - UMATPS85.FOR
- UGENS. User General Section
  - Alternative Solution for Example 9.1
  - Alternative Solution for Example 11.1
- Execute Abaqus from MATLAB