Application of a Suite of Engineering Simulation Programme ABAQUS to Solid Mechanics Problems

Course Contents:

The class presents the basis of solving mechanical problems with the help of ABAQUS software. ABAQUS contains an extensive library of elements that can model virtually any geometry. Geometry from a many different CAD software packages may be imported. Using ABAQUS, the students should be able to use various different material models to simulate the behavior of most typical engineering materials including metals, rubber, polymers, composites, reinforced concrete, crushable and resilient foams, and geotechnical materials such as soils and rock. ABAQUS offers a wide range of capabilities for simulation of linear and nonlinear applications. Problems with multiple components are modeled by associating the geometry defining each component with the appropriate material models and specifying component interactions. In a nonlinear analysis ABAQUS automatically chooses appropriate load increments and convergence tolerances and continually adjusts them during the analysis to ensure that an accurate solution is obtained efficiently. Static analysis can be performed as well as dynamic. The course is intended to serve as a quick introduction to the software for the students.

Subjects covered by the lectures are:

Section 1. Finite Element Method

Topic 1.1. Overview of Finite Element Method


Topic 2.1. Discretization of the Domain

Basic element shapes. Discretization process. Automatic mesh generation.

Section 2. Application to Solid Mechanics Problems


Topic 2.2. Material Constitution for Finite Deformation

Topic 2.3. Treatment of Constraints – Contact and Tied Interfaces


Topic 2.4. Basic Equations and Solution Procedure


Section 3. ABAQUS Software

Topic 3.1. Starting Abaqus/CAE


Topic 3.2. TRUSS EXAMPLE: Analysis of an Overhead Hoist


Topic 3.3. TRUSS EXAMPLE: Postprocessing with Abaqus/CAE

Displaying node numbers. Displaying element numbers. Changing the deformation scale factor. Overlaying the undeformed model shape on the deformed model shape. Tabular data reports. Generating field data reports.

Topic 3.4. 2D EXAMPLE: A Rectangular Plate with a Hole in 2D Plane Stress


Topic 3.5. 2D EXAMPLE: Postprocessing with Abaqus/CAE

Generating solution contours. Generating report of Field Outputs.
Topic 3.6. 3D EXAMPLE

Topic 3.6.1. Analysis of 3D Elastic Solid


Topic 3.6.2. 3D Body with a Spherical Cavity


Topic 3.6.3. 3D Body with a Spherical Inclusion


Topic 3.8. Main Principles of Operation with Documentation for ABAQUS


Topic 3.9. Creation of Scripts in ABAQUS

The simple scenario in ABAQUS. Automation of repeated actions. Parametric analysis. Creating and changing of modeling databases and the models created in ABAQUS. Access to databases of results (an output file of results).

Topic 3.10. System of Units


Topic 3.11.1 Basic XFEM Concepts

Topic 3.11.2 Modeling Approaches


Topic 3.11.3 XFEM Simultaneously Used with Other Fracture and Failure Techniques.

Bulk material failure and interfacial delamination.
Topic 3.11.4 Analysis Procedure

Static. Implicit dynamic. Low cycle fatigue.

Topic 3.11.5 XFEM Used with Other Analysis Techniques


Topic 3.11.6 Elements, Outputs and Others.

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) these method.
- He will be able to analyse new problems.
- He will be able to apply Simulia ABAQUS for complex problems of solid mechanics.
- He will be able to apply Simulia ABAQUS for complex problems of fracture mechanics.
- He will be able to use proper modeling techniques for capturing crack-tip singularities in fracture mechanics problems.
- He will be able to use Abaqus/CAE to create meshes appropriate for fracture studies.
- He will be able to calculate stress intensity factors and contour integrals around a crack tip.
- He will be able to simulate material damage and failure.
- He will be able to simulate crack growth using cohesive behavior, VCCT, and XFEM.
- He will be able to simulate low-cycle fatigue crack growth.

Planned Learning Activities:

Topic 7.1. Finite Element Analysis Using ABAQUS

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. A rectangular plate with a hole in 2D plane stress.
   b. A diametrically squeezed disk in 2D plane stress.
   c. A diametrically squeezed disk with a cut in 2D plane stress.
5. 3D example: analysis of 3D elastic solid.
   a. Modeling of a static loading of rotary loops.
c. 3D body with a spherical cavity.
d. 3D body with a spherical inclusion.

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.
   c. Space with a spherical cavity.
   d. Space with a spherical inclusion.
   e. Space with an ellipsoid cavity.
   f. Space with an ellipsoid inclusion.

7. Cyclic loading of a rectangular plate with a central circular hole.

8. Import/export of geometry and models.

9. Creation of scripts in Abaqus.

10. Prove that distribution of Mises stress for a rectangular plate with a central hole will be as shown in the Figure 1.

![Figure 1](image1.png)

11. Prove that distribution of Mises stress for a diametricly squeezed disk will be as shown in the Figure 2.

![Figure 2](image2.png)

**Recommended or Required Readings:**
6. SIMULIA Abaqus/CAE User’s Manual
7. SIMULIA Abaqus Example Problems Manual
15. Extended Finite Element Method (XFEM), 2009, Abaqus 6.9 Update Seminar, Dassault Systemes

WORKSHOP

Workshop 1 Crack in a Three-point Bend Specimen
Workshop 2 Crack in a Helicopter Airframe Component
Workshop 3 Crack Growth in a Three-point Bend Specimen using Cohesive Connections (Part 1)
Workshop 4 Crack Growth in a Helicopter Airframe Component using Cohesive Elements
Workshop 3 Crack Growth in a Three-point Bend Specimen using Cohesive Connections (Part 2)
Workshop 5 Crack Growth in a Three-point Bend Specimen using VCCT
Workshop 6 Crack Growth in a Three-point Bend Specimen using XFEM
Workshop 7 Modeling Crack Propagation in a Pressure Vessel with Abaqus using XFEM