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

Basic concept. General applicability of the method. Engineering applications of the finite element method. General description of the 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 2.1. General Problems in Solid Mechanics. Nonlinear Problems in Solid Mechanics

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 2.2. Material Constitution for Finite Deformation

Isotropic elasticity. Isotropic viscoelasticity. Plasticity models. Incremental formulations. Rate constitutive models. Numerical examples.

Topic 2.3. Treatment of Constraints – Contact and Tied Interfaces

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

Topic 2.4. 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.

Section 3. ABAQUS Software

Topic 3.1. Starting Abaqus/CAE

Components of the main window. Title bar. Menu bar. Toolbars. Context bar. Model Tree. Results Tree. Toolbox area. Canvas and drawing area. Viewport. Prompt area. Message area. Command line interface.

Topic 3.2. TRUSS EXAMPLE: Analysis of an Overhead Hoist

Creating part. Creating material. Defining and assigning section properties: defining a truss section, assigning the section to the frame. Defining the assembly. Configuring analysis. Creating a static linear perturbation analysis step. Requesting data output. Examining output requests. Applying boundary conditions and loads to the model: applying boundary conditions to the frame, applying a load to the frame, applying a concentrated force to the frame. Meshing the model. Assigning the ABAQUS element type. Creating the mesh. Creating an analysis job. Checking the model (running a data check analysis, monitoring the status of a job). Running the analysis.

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

Creating a part. Creating a material. Defining and assigning section properties. Defining the assembly. Configuring your analysis. Applying boundary conditions and loads to the model. Meshing: structured meshing, swept meshing, free meshing. Remeshing and changing element types. Creating an analysis job. Checking the model (log tab, errors and warnings). Running the analysis.

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

Importing parts. Importing an assembly. Creating the cube. Adding the flange to the base feature. Creating a material. Defining a section. Assigning the section. Assembling the model by creating an instance of the hinge. Defining analysis steps. Selecting a degree of freedom to monitor. Constraining the hinge. Applying the pressure and the concentrated load to the hinge. Meshing the assembly. Partitioning the model. Assigning the ABAQUS element type. Seeding the part instances. Meshing the assembly. Creating and submitting a job. Viewing the results of the analysis.

Topic 3.6.2. 3D Body with a Spherical Cavity

Creating a part. Creating a material. Defining and assigning section properties. Defining the assembly. Configuring your analysis. Applying boundary conditions and loads to the model. Creating an analysis job. Running the analysis. Coefficient of stress concentration. Comparison of analytical and numerical solutions.

Topic 3.6.3. 3D Body with a Spherical Inclusion

Creating a part. Creating a material. Defining and assigning section properties. Defining the assembly. Configuring your analysis. Applying boundary conditions and loads to the model. Creating an analysis job. Running the analysis.

Topic 3.8. Main Principles of Operation with Documentation for ABAQUS

Abaqus/CAE User's Manual. Abaqus Analysis User's Manual. Abaqus Example Problems Manual. Getting Started with Abaqus: Interactive Edition. Getting Started with Abaqus: Keywords Edition.

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. Extended Finite Element Method (XFEM). Application to Fracture Mechanics

Topic 3.11.1 Basic XFEM Concepts

Topic 3.11.2 Modeling Approaches

Stationary cracks. Contour integral calculation. Propagation cracks. Cohesive segments approach. Linear elastic fracture mechanics approach.

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

Global/local modeling approach. Co-Simulation.

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 diametricly squeezed disk in 2D plane stress.
    c. A diametricly 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.
    b. A fixed beam bending. Various types of elements usage. Change of grid settings.

    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.

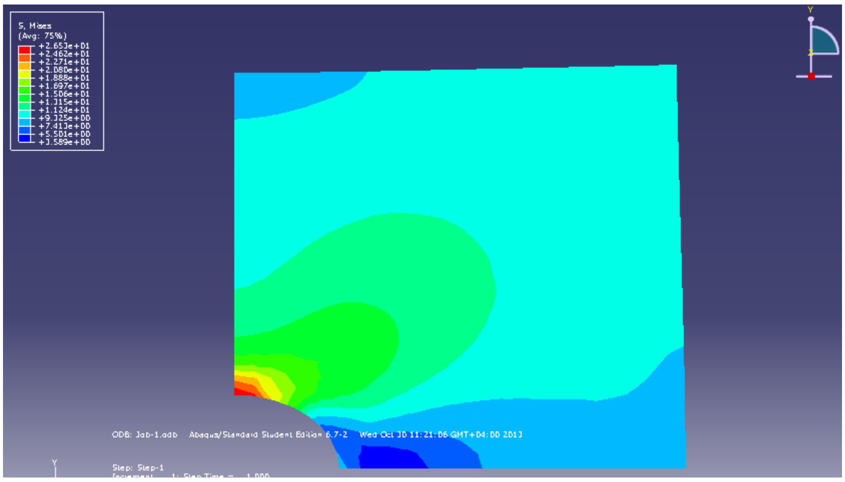

Figure 1

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

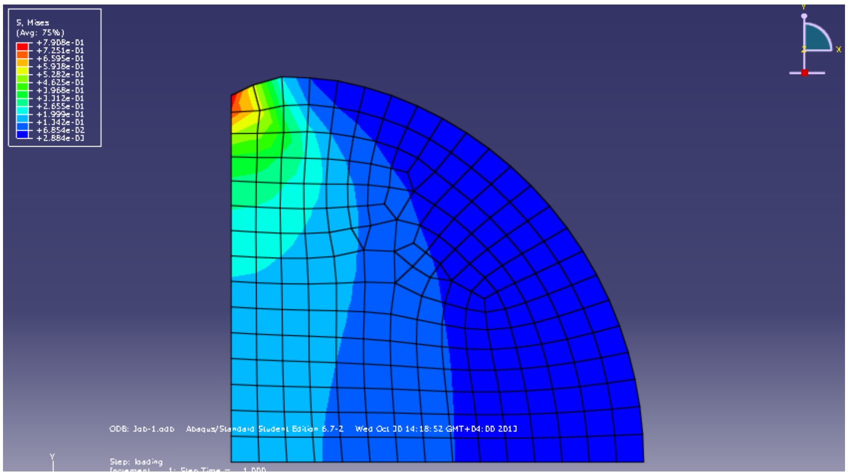

Figure 2

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. Kuna M. Finite Elements in Fracture Mechanics. Theory – Numerics – Applications. Springer Dordrecht Heidelberg New York London. V. 201. 2010.
10. W.T. Koiter. General Theorems for Elastic-Plastic Solids. 1960. North-Holland publishing company. Amsterdam.79p.
11. 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
12. 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
13. 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
14. 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
15. Extended Finite Element Method (XFEM), 2009, Abaqus 6.9 Update Seminar, Dassault Systemes
16. Abaqus 6.9 – Extended Functionality (EF) Overview, Abaqus 6.9 – EF Update Webinar, Dassault Systemes
17. J. Ciambella, M. Destrade, R. W. Ogden, On the ABAQUS FEA model of finite viscoelasticity. Rubber Chem. Technol., 2009, 82(2): 184–193
18. 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
19. 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
20. 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

21. 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
22. 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
23. Babuska I., Whiteman J., Strouboulis T. Finite Elements. An Introduction to the Method and Error Estimation. Oxford: Oxford University Press, 2010. 336 p.
24. Davies A.J. The Finite Element Method. An Introduction with Partial Differential Equations. Oxford: Oxford University Press, 2011. 312 p.

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