

Applied continuum mechanics with Abaqus

Ph.D. Eng. Marcin Nowak

Department of Mechanics of Materials

Course Description

During the lectures, we will explore the practical application of acquired knowledge in the field of continuum mechanics through numerical simulations. The primary objective of this course is to introduce students to the Abaqus program, utilizing a series of computational examples as learning tools. Abaqus is a powerful finite element analysis software program used for simulating various types of engineering and scientific problems. It is widely used in industry and academia for solving complex simulations. Abaqus includes a range of capabilities for modelling and analysing structures, fluids, thermal effects, and electromagnetic fields, among other physics.

Requirements

- Basic knowledge of Mechanical/Civil engineering. Understanding the concepts of stress and strain, the continuum hypothesis and fundamental equations of continuum mechanics.
- Basic knowledge of Finite Element Method.
- Abaqus Learning Edition program (available free of charge to anyone)

Content

- Introduction: syntax and conventions, model definition, job execution, documentation, a quick review of the finite element method
- Material models: linear and nonlinear elastic models, plasticity models, viscoelasticity models, progressive damage and failure, heat transfer properties
- Linear dynamics: modal-based solutions, extracting real eigenvalues, damping, base motion excitation
- Nonlinearity: sources of nonlinearity, the solution of nonlinear problems
- Contact: features of general contact, defining contact, limitations of general contact, contact pairs, contact logic and diagnostics tools
- Rigid body definition, analytical rigid surface, rigid elements, rigid body constraints
- Finite elements: continuum elements, structural elements, inertial, rigid, and capacitance elements
- Running simulation: use of Grafen computing cluster in IPPT PAN
- Analysis techniques: Eulerian analysis, particle methods, co-simulation, design sensitivity analysis, parametric studies
- Viewing the output from analysis: Reading the output database, customizing a model plot, displaying the deformed model shape, animating a contour plot
- Python scripting: Python language basics, automation of the model creation process
- Extension of the Abaqus: Abaqus User Subroutines, creating your own mechanical constitutive behaviour of a material

The total number of hours: lectures 30 hours, laboratory exercises: 0 hours, self-teaching: 15 hours, tutoring and consultations: 15 hours.