

Finite Element Method

Profile and Objective

As a well-established universal simulation and optimization tool, the finite element method (FEM) enables the optimization of single components such as turbine blades or clutch couplings, up to the simulation of entire vehicles. The simulation of complex accident scenarios helps the designer to identify weak points without having to produce expensive prototypes.

+ Software

Abaqus Expanded Teaching Edition:

- Abaqus/C
- Abaqus/S
- Abaqus/E

Limitation: compared to the Research Edition:

- Maximum model size: 250000 nodes
- No user subroutines

Number of licences: 20

+ Lectures and Seminars

FEM I: Statics and dynamics

- Geometry and meshing
- Loads and boundary

condition:

- Procedure
- Linear elasticity and damping
- Elements
- Contact and interaction
- Post processing

Corresponding lecture:
Introduction to Abaqus FEA

FEM II: Material models

- Hyperelasticity
 - Viscoelasticity
 - Plasticity
 - Damage and failure
 - Anisotropic
-