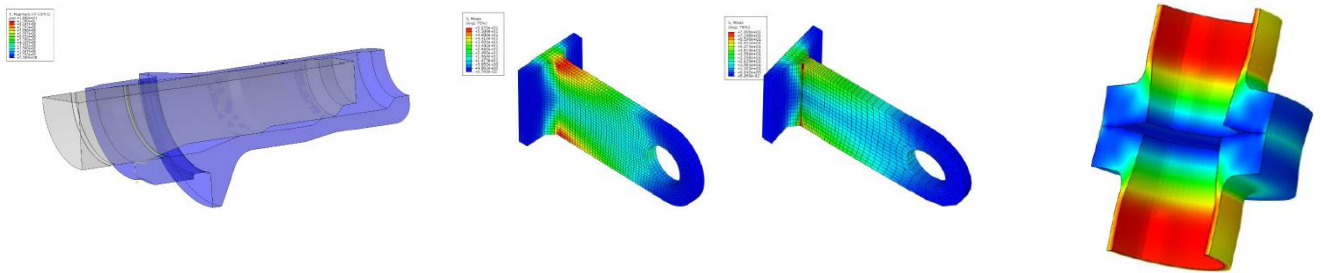


Introduction to Abaqus - Part 1 (3 Days)

Introduction:

Introduction to Abaqus is an intensive hands-on training that will familiarize the participants with the contents of the world's most powerful FEA package. During these 3 days we will go from linear elastic beams to complex assemblies containing pre-tensions and non-linear materials. Practical cases will introduce useful features and techniques in a natural and progressive manner to help the participants to create efficient models in a day-to-day setting. This compressed yet comprehensive course is highly recommended for all new Abaqus users.



Day 1 - Introduction to Abaqus

Morning (08:00-09:00):

- Installation and setup of Abaqus 2017 on participants computers

Morning (09:00-12:00):

- Lesson: Introduction to the Finite Element Method
- Lesson: Introduction to Abaqus/CAE
- 2 Workshops

Afternoon (12:45-16:00):

- 4 Workshops and Discussions

Day 2 - Meshing and Geometry

Morning (9:00-12:00):

- Lesson: Working with Geometry in Abaqus/CAE
- 2 Workshops

Afternoon (12:45-16:00):

- Lesson: Elements and Meshing in Abaqus/CAE
- 3 Workshops

Day 3 - Introduction to Contact

Morning (9:00-12:00):

- Lesson: Introduction to Contact
- 2 Workshops

Afternoon (12:45-16:00):

- Lesson: Contact Methods
- 3 Workshops
- Lesson: Troubleshooting