

Learning Module

Introduction to Abaqus/CAE

Abaqus/CAE provides a complete interactive environment for creating Abaqus models, submitting and monitoring analysis jobs and viewing and manipulating simulation results.

Objectives

Upon Completion Of This Course You Will Be Able To:

- Use Abaqus/CAE to create complete finite element models.
- Use Abaqus/CAE to submit and monitor analysis jobs.
- Use Abaqus/CAE to view and evaluate simulation results.

Knowledge Prerequisites

None

Language(s) for selected release

English

Brands

Simulia

Available Releases

SIMULIA 2021, SIMULIA 2020, SIMULIA 2019, SIMULIA 2018, SIMULIA 2017, SIMULIA 2016, SIMULIA V6.14, SIMULIA V6.13, SIMULIA V6.12

Duration

16 hours

Discipline

Advanced Abaqus

Contents

Overview - Introduction to Abaqus-CAE

1 - Introducing Abaqus-CAE

2 - Working with Geometry in Abaqus-CAE

3 - Working with Models Created Outside Abaqus

4 - Material Properties and Assemblies

5 - Steps, Interactions, and Loads

6 - Meshing

7 - Job Management and Results Visualization