

Learning Module

Abaqus/Explicit: Advanced Topics

The course emphasizes practical skills and techniques that are needed for analyses with Abaqus/Explicit.

Objectives

Upon completion of this course you will be able to:

- Use the explicit dynamics method effectively, including the application of general contact, mass scaling, and adaptive remeshing.
- Use Abaqus/Explicit and Abaqus/Standard together to solve difficult problems, including results transfer and co-simulation.
- Model high-strain-rate deformation and failure.
- Filter output.

Knowledge Prerequisites

This course is recommended for engineers with experience using Abaqus

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

24 hours

Discipline

Advanced Abaqus

Contents

Overview - Abaqus (Explicit) Advanced Topics

1 - Overview of Abaqus (Explicit)

2 - Elements

3 - Contact Modeling

4 - Quasi-Static Analyses

5 - Constraints and Connections

6 - Impact and Postbuckling Analyses

7 - Material Damage and Failure

8 - Importing and Transferring Results

9 - Managing Large Models

10 - Output Filtering

A1 - Explicit Dynamics Algorithm

A2 - Contact Pairs

A3 - Co-simulation