

Learning Module

Substructures and Submodeling with Abaqus

The size and complexity of designs that are analyzed and tested with Abaqus continues to grow. Substructures and submodeling are two effective techniques that allow the analyst to study problems that are too large to simulate with a conventional modeling approach.

Objectives

Upon completion of this course you will be able to:

- Understand the difference between substructuring and submodeling.
- Build, translate, rotate and reflect substructures.
- Build preloads into substructures.
- Design meshes for submodel analysis.
- Perform solid-to-solid, shell-to-shell, and shell-to solid submodeling.

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

16 hours

Discipline

Advanced Abaqus

Contents

Overview - Substructures and Submodeling with Abaqus

- 1 - Introduction to Substructures
- 2 - Using Static Substructuring in Abaqus
- 3 - Linear Perturbations about a Preloaded State
- 4 - Dynamic Substructuring
- 5 - Substructure Output
- 6 - Substructuring Examples
- 7 - Using Substructures with Abaqus Explicit
- 8 - Introduction to Submodeling
- 9 - Submodeling in Abaqus
- 10 - Abaqus Usage and Examples (Part 1)
- 11 - Abaqus Usage and Examples (Part 2)
- 12 - Submodeling Practices
- 13 - Limitations of Submodeling
- A1 - Theory of Substructures