

Learning Module

Heat Transfer and Thermal-Stress Analysis with Abaqus

The success of many structural designs requires a thorough understanding of both the thermal and mechanical response of the design. Temperature-dependent material properties, thermally-induced deformation, and temperature variations all may be important design considerations.

Objectives

Upon Completion Of This Course You Will Be Able To:

- Perform steady-state and transient heat transfer simulations.
- Solve cavity radiation problems.
- Model latent heat effects.
- Perform adiabatic, sequential, and fully coupled thermal-stress analyses.
- Model contact in heat transfer problems.

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 - Heat Transfer and Thermal-Stress Analysis with Abaqus

- 1 - Heat Transfer and Stress Analysis Overview
- 2 - Heat Transfer Basics
- 3 - Geometry, Material Properties, and Elements
- 4 - Analysis Procedures and Convergence
- 5 - Boundary Conditions and Loads
- 6 - Thermal Interfaces
- 7 - Thermal Output and Postprocessing
- 8 - Thermal-Stress Analysis
- 9 - Sequentially-Coupled Thermal-Stress Analysis
- 10 - Fully-Coupled Thermal-Stress Analysis
- 11 - Adiabatic Analysis
- A1 - Heat Transfer Theory
- A2 - Heat Transfer Theory
- A3 - Cavity Radiation
- A4 - Thermal Fatigue