

Learning Module

Electromagnetic Analysis with Abaqus

Abaqus provides computational electromagnetic capabilities for the simulation of problems involving steady-state electrical conduction, piezoelectric phenomena and low-frequency eddy currents. In this course, you will learn how to analyze low frequency eddy current problems in Abaqus/Standard.

Objectives

Upon Completion Of This Course You Will Be Able To:

- Set up and create electromagnetic models with Abaqus.
- Perform low frequency eddy current analyses with Abaqus.
- Perform transient eddy current analyses with Abaqus.
- Perform magnetostatic analyses with Abaqus.

Knowledge Prerequisites

This course is recommended for engineers with experience using Abaqus.

Language(s) for selected release

English

Contents

- Overview - Electromagnetic Analysis with Abaqus
- 1 - Introduction to Computational Electromagnetics
- 2 - Geometry, Material Properties, Elements and Meshing
- 3 - Loads and Boundary Conditions
- 4 - Output and Transfer of Results

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

8 hours

Discipline

Advanced Abaqus