

Learning Module

Introduction to Abaqus Scripting

This seminar covers basic usage of the Abaqus Scripting Interface and Python's syntax. It includes numerous hands-on exercises for the student to learn to automate tasks that are common to most analysts.

Objectives

Upon Completion Of This Course You Will Be Able To:

- Help students to develop a high level understanding of the Abaqus scripting capabilities.
- Organize and present the technical details of Python and the Abaqus Scripting Interface.
- Expose the strengths and weaknesses of Abaqus scripting.
- Encourage the student to use scripting in new ways.

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 Scripting

1 - Overview of Scripting in Abaqus

2 - Running Python

3 - Python Core Language Part 1

4 - Python Core Language Part 2

5 - Abaqus Scripting Interface Basics

6 - Abaqus Object Model

7 - Postprocessing with Abaqus Scripting

8 - Miscellaneous Topics

A1 - Optional Topics