

Learning Module

Metal Forming with Abaqus

Metal forming processes are highly nonlinear because they involve geometric, material and contact nonlinearities.

Objectives

In this course you will learn practical modeling skills and techniques for:

- Stamping.
- Hydroforming.
- Punch stretching.
- Forging.
- Rolling.
- Drawing.
- Superplastic forming.

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

24 hours

Discipline

Advanced Abaqus

Contents

Overview - Metal Forming with Abaqus

- 1 - Introduction
- 2 - Solution Procedures with Abaqus
- 3 - Contact
- 4 - Elements
- 5 - Materials
- 6 - Adaptive Meshing
- 7 - Quasi-Static Analyses
- 8 - Transferring Results between Abaqus Analyses
- 9 - Model Change
- 10 - Thermal Effects
- 11 - Rolling Analysis
- 12 - Multi-Pass Rolling Analysis
- 13 - Drawbead Modeling
- 14 - Hydroforming
- 15 - Superplastic Forming Analysis with Abaqus
- 16 - One-step Inverse Analysis