Learning Module

Modeling Stents Using Abaqus

This course focuses on the use of Abaqus for modeling and analyzing stents. However, its content can also be useful when modeling other types of medical devices. The course is targeted at engineers responsible for the design of medical devices who are looking to accelerate their understanding of the highly complex mechanical behavior associated with performance of such devices.

Objectives

Upon Completion Of This Course You Will Be Able To:
- Create geometry for modeling stents and tools.
- Choose the proper element type.
- Choose material models: elastic-plastic (Stainless Steel), superelastic-plastic (Nitinol), hyperelastic (vessels).
- Perform stent analyses: Static, Implicit and Explicit Dynamics.
- Define contact and constraints.
- Postprocess stent analyses.
- Perform fatigue evaluation.

Knowledge Prerequisites

This course is recommended for engineers with experience using Abaqus.

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

Language(s) for selected release

English

Contents