Learning Module

Crashworthiness Analysis with Abaqus

This course is the ideal way to obtain a working knowledge of how to use Abaqus for crashworthiness analysis.

Objectives

Upon Completion Of This Course You Will Be Able To:
- Abaqus fundamentals and input syntax.
- General "automatic" contact modeling.
- Element selection for crash simulation.
- Constraints and connections modeling.
- Material models used in crash simulation.
- Multiple mechanism damage and failure modeling.

Knowledge Prerequisites

No previous knowledge of Abaqus is required, but knowledge of finite elements and engineering mechanics is necessary.

Language(s) for selected release

English

Brands

Simulia

Available Releases

SIMULIA 2021, SIMULIA 2020, SIMULIA 2019, SIMULIA 2018, SIMULIA 2017, SIMULIA 2016, SIMULIA V6.12

Duration

24 hours

Discipline

Advanced Abaqus

Contents

Overview - Crashworthiness Analysis with Abaqus
1 - Introduction and Motivation
2 - Setting up an Abaqus analysis
3 - Explicit Dynamics in Abaqus
4 - Contact Modeling
5 - Element Technology
6 - Constraints and Connections
7 - Material Modeling
8 - Advanced Analysis Techniques
9 - Output
10 - Co-simulation
Appendices

Общество с ограниченной ответственностью “Системы Инженерного Анализа”
603107, Россия, г. Н. Новгород, ул. Ларина, д. 7; ИНН 5261108537 / КПП 526101001
www.caecis.com, info@caecis.com, тел: +7 831 283 00 15