Automotive Powertrain Assembly Analysis with Abaqus

V6
R6.11

About this Course

Course objectives

Upon completion of this course you will be able to:

- Simulate engine assembly and operation conditions including the effects of bolt loading, frictional contact, nonlinear gaskets, rubber components, etc.

Targeted audience

Simulation Analysts

Prerequisites

This course is recommended for engineers with experience using Abaqus/Standard.

1 days

Day 1

- Lecture 1 — Introduction and Motivation
- Lecture 2 — Contact
- Lecture 3 — Gaskets and Bolt Loading
- Lecture 4 — Thermal Stress Analysis
- Lecture 5 — Dynamics—NVH Effects
- Lecture 6 — Manufacturing Process

Appendices

- Appendix 1 Maximizing Success with Contact in Abaqus/Standard
- Appendix 2 Large Model Management
- Appendix 3 Materials for Powertrain

Legal Notices

Revision Status

Lecture 1	8/12	Re-issued for 6.11
Lecture 2	8/12	Re-issued for 6.11
Lecture 3	8/12	Re-issued for 6.11
Lecture 4	8/12	Re-issued for 6.11
Lecture 5	8/12	Re-issued for 6.11
Lecture 6	8/12	Re-issued for 6.11
Appendix 1	8/12	Re-issued for 6.11
Appendix 2	8/12	Re-issued for 6.11
Appendix 3	8/12	Re-issued for 6.11

Lesson 1: Introduction and Motivation

Lesson content:

- Background
- Powertrain Applications (not comprehensive)
  - Exhaust Manifold
  - Crank Bore Distortion
  - Four Pinion Differential Carrier
  - Cylinder Head and Block Thermal Structural Analysis
  - Valve Body Sealing
  - Sealing Systems
  - Water Pump Sealing—Paper/Silicone Gasket
  - Valve Cover Gasket—Elastomeric Seals
  - Hyperelastic Material Calibration
  - Sealing Analysis with Elastomeric Gaskets
  - Dynamic Park System Simulation
  - Composite Intake Manifold Analysis
  - Roller Rocker Arm Pedestal
  - Natural frequencies of engine-transmission assembly
- Summary of Relevant Abaqus Features for Powertrain

75 minutes

Lesson 2: Contact

Lesson content:

- Contact Analysis in Powertrain
- How to Approach Contact Analyses
- Application: Coolant Manifold Cover Assembly
  - *TIE
  - Adjusting surfaces
- Application: 3-D Rubber Seal
  - *CONTROLS
  - Contact output
- Application: Press Fit Analysis of Valve Seats
  - Submodeling
  - Interference fit problems
- Bolted Joints with Threads

90 minutes

Lesson 3: Gaskets and Bolt Loading

Lesson content:

- Introduction
- Gasket Element Formulations
- Gasket Element Library
- Defining Gasket Element Geometry
- Gasket Element Behavior
- Using Gasket Elements in a Model
- Application: Coolant Manifold Cover Gasket
- Application: Transmission Pan Gasket
- Application: Engine Bore Distortion
- Gasket Element Output Variables
- Practical Tips for Gasket Usage

2 ½ hours

Lesson 4: Thermal Stress Analysis

Lesson content:

- Thermal-Stress Procedures in Abaqus
- Sequentially Coupled Thermal-Stress Analysis
- Temperature Application
- Thermal Stress Example: Exhaust Manifold
- Using CFD Results with Abaqus

1 hour

Lesson 5: Dynamics—NVH

Lesson content:

- Introduction
- Natural Frequency Extraction
- Steady-State Analysis
- Mode-Based Steady-State Analysis
- Direct Steady-State Analysis
- Frequency Domain Analysis with the Subspace Method
- Dynamic Gaskets
- Structural Acoustics

90 minutes

Lesson 6: Manufacturing Processing Effects

Lesson content:

- Introduction
- Example: Machining of a Coolant Manifold Assembly
  - Transferring results between Abaqus/Standard analyses
- Example: Oil Pan Vibration
  - Manufacturing process effects on steady-state vibration and fatigue life

45 minutes

Appendix 1: Maximizing Success with Contact in Abaqus/Standard

Lesson content:

- Understanding Abaqus Solution Algorithms
- Overview of Contact in Abaqus/Standard
- Contact Discretization
- Relative Sliding Between Bodies
- The Contact Algorithm in Abaqus
- Understanding the Message File
- Contact Diagnostics (Visual)
- Systematic Modeling Practices
- Troubleshooting Contact Analyses

2 ¼ hours

Appendix 2: Large Model Management

Lesson content:

- What is a Large Model?
- Managing Computer Resources for Large Models
- Analysis Techniques to Manage Large Models
  - Restart
  - Output control
  - Parts and assemblies
  - Submodeling
  - Substructuring

30 minutes

Appendix 3: Materials for Powertrain

Lesson content:

- Introduction
- Linear Elasticity (Hooke's Law)
- Abaqus Rubber Material Models
- Example: Curve Fitting Rubber Test Data
- Solid Metal Plasticity
- Abaqus Pressure-Dependent Plasticity Models
- Example: Application of a Crankshaft Seal
- Gray Cast Iron Plasticity

3 ½ hours