

Wound Composite Modeler for Abaqus



About this Course

Course objectives

Composite Overwrapped Pressure Vessels (COPVs) are increasingly being used in industry applications where their reduced weight over metallic vessel alternatives are an advantage for high pressure fluid storage. Simulating COPV performance with finite element analysis is a challenge because of the complex material structures and time consuming model setup requirements. This seminar provides an introduction to COPV modeling using the Wound Composite Modeler for Abaqus (WCM).

Course topics:

- Defining the liner (mandrel) geometry
- Generating composite layup for symmetric or unsymmetrical tanks
- Polar boss buildup controls
- Mesh partitioning and meshing of tank
- Material property generation and orientations
- Performing an autofrettage analyses
- Performing liner debonding analyses
- Automated tool for generating layer path plots
- Micro-mechanics module

Targeted audience

Simulation Analysts

Prerequisites

This course does not require previous experience with Abaqus and Abaqus/CAE, although it is recommended.

Day 1

- ▶ **Lecture 1** **Wound Composite Modeler for Abaqus**
 - Background of Composite Overwrapped Pressure Vessels (COPVs)
 - Tank Design, Netting analysis
 - Theory of composites for COPVs
 - Introduction to Wound Composite Modeler
 - Example finite element model
 - Workshop 1: Generating 2D Wound Composite Models

- ▶ **Lecture 2** **Wound Composite Modeler and 3D models**
 - Step-by-Step 3D model building procedure
 - Shell models
 - Brick vs. cylindrical elements
 - Workshop 2: Generating 3D Wound Composite Models

- ▶ **Lecture 3** **Wound Composite Modeler and Autofrettage Analysis**
 - What is autofrettage
 - Plasticity basics in Abaqus
 - Autofrettage/Debonding in WCM
 - Post-processing an autofrettage analysis
 - Workshop 3: Autofrettage Analysis
 - Workshop 4: COPV Design
 - Workshop 5: Buckling/Debonding Analysis

- ▶ **Lecture 4** **Wound Composite Modeler and Micro-mechanics**
 - Micro-mechanics basics
 - Theory and load cases

Day 2

- ▶ **Lecture 3** **Wound Composite Modeler and Autofrettage Analysis**
 - What is autofrettage
 - Plasticity basics in Abaqus
 - Autofrettage/Debonding in WCM
 - Post-processing an autofrettage analysis
 - Workshop 3: Autofrettage Analysis
 - Workshop 4: COPV Design
 - Workshop 5: Buckling/Debonding Analysis

- ▶ **Lecture 4** **Wound Composite Modeler and Micro-mechanics**
 - Micro-mechanics basics
 - Theory and load cases

Day 2

▶ **Hands-On** **Model Building of Customer's COPVs**

- Import and mesh 2D and 3D customer liner geometry
- Convert winding table layup from CAD winding software into WCM
- Build 2D and 3D models
- Post-process 2D and 3D models

Join the Community!

How can you maximize the robust technology of Abaqus FEA and Isight?

Connect with peers to share knowledge and get technical insights



 SIMULIA

Let the SIMULIA Learning Community be *Your* Portal to 21st Century Innovation

Discover new ways to explore how to leverage realistic simulation to drive product innovation. Join the thousands of Abaqus and Isight users who are already gaining valuable knowledge from the SIMULIA Learning Community.

For more information and registration, visit 3ds.com/simulia-learning.
Connect. Share. Spark Innovation.

 DASSAULT SYSTEMES | The 3DEXPERIENCE Company