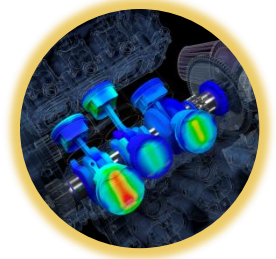


Autodesk® Simulation Multiphysics



Course Overview

Duration:
2 days

Who Should Attend?

Autodesk Simulation Multiphysics users that need to study multiple physical factors acting simultaneously, including fluid flow.

Autodesk Simulation Mechanical Part I is a prerequisite for this course.

To register for upcoming classes Email:
NA.MFG.simulation.training@autodesk.com

Phone: +1.412.967.2779

Course Description

This course will introduce you to performing computational fluid dynamics (CFD) with Autodesk® Simulation Multiphysics. Coupling of multiple physics phenomenon such as fluid flow, heat transfer, and mechanical stress/strain are also covered. This course assumes you have a working knowledge of the Autodesk Simulation user interface and basic FEA theory. It is recommended that you attend the Autodesk Simulation Mechanical – Part I course prior to attending this class.

Course Outline - Autodesk Simulation Multiphysics

Engineering Theory Review

- FEA Methods
- Fluid Flow Equations

Fluid Flow Analysis

- Fluid Elements
- Meshing
- Loading
- Convergence Controls
- Turbulence

Results Evaluation

- Result Types
- View Control
- Particle Paths
- Streamlines

Additional Loading Options

- Fans
- Rotating Reference Frames

Open Channel Flow

- Free Surface
- Initial Fluid Volume
- Results Evaluation

Multiphysics

- Forced Convection
- Natural Convection
- Fluid Structure Interaction
- Thermal Stress
- Joule Heating

Autodesk®

Autodesk is a registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document. © 2012 Autodesk, Inc. All rights reserved.