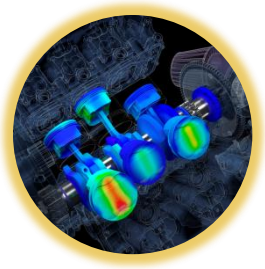
Autodesk® Simulation Multiphysics

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