Autodesk Authorized Training Center

AEC SYSTEMS
EXPERIENCE. KNOWLEDGE. SOLUTIONS.

Course Title: Inventor – Simulation

Course Code: INV-2SIM

Duration: 2 Days

Courseware Description

Learn the fundamental principles and recommended workflows for analyzing designs and creating dynamic simulations of mechanisms using Autodesk® Inventor® Professional. Users learn how to validate digital prototypes by simulating the operation of mechanisms and motorized assemblies. They also learn how to analyze parts and assemblies, perform parametric design studies, and use modal analysis. Hands-on exercises representing real-world, industry-specific design scenarios are included.

Objectives

To introduce users to the user interface, tools, and recommended workflows in the Autodesk Inventor Professional10 Dynamic Simulation and Stress Analysis environments. After completing this class, users will be able to:

- Validate digital prototypes by creating dynamic simulations of mechanisms using joints and environmental constraints.
- Eliminate redundancies in a design.
- Interpret Dynamic Simulation results.
- Analyze parts and assemblies and perform parametric design studies.
- Practice solving real-world design problems.

Who Should Attend

Experienced Autodesk Inventor users

Prerequisites

- Completion of INV-1 course or equivalent knowledge of Inventor parametric part and assembly design.
- Drafting, design, or mechanical engineering principles.
- Microsoft® Windows® 7, Windows® Vista or Microsoft® Windows® XP.

Course Outline

Day 1: Stress Analysis

- Introduction and overview of Finite Element theory
- Examining the capabilities of the Inventor stress analysis environment
- Creating F.E. simulations and working with simulation settings
- Creating and manipulating constraints
- Creating and manipulating loads
- Meshing, working with general mesh settings and creating local refinements
- Running a stress analysis simulation
- Interpreting and formatting results
- Animating and probing results
- Creating simulation reports
- Running an assembly analysis
- Creating and manipulating contacts
- Performing parametric studies
- Running a modal analysis

Day 2: Dynamic Simulation

- Introduction and overview of Dynamic Simulation
- The Simulation Plater, the Output Grapher and construction mode
- Converting, creating and manipulating joints
- Adding friction, forces, damping and imposed motion
- Running simulations and analysing results
- Animating and recording simulations
- Exporting, importing and comparing results from multiple simulations
- Sharing dynamic simulation results with FEA

Frame Analysis

- This section of the course is yet to be developed. This functionality is new in the 2011 version of the software... Should be ready by next month.