Autodesk Inventor Stress and Frame Analysis

You will learn how to drive the simulation capabilities of Autodesk Inventor Professional to perform FEA stress and frame analysis on models and digital prototypes. This will enable you to validate designs, eliminate redundancies and solve real-world problems before a physical prototype is produced.

Course Modules

- *Simulation Overview*
- *The Stress Analysis Environment*
- *Introduction to Static Analysis*
- *Introduction to Model Analysis*
- *Introduction to Frame Analysis*
- *Stress Analysis Workflows in Inventor Professional*
- *User Interface*
- *Simulation Preparation*
- *Simulation Pre-Solve*
- *Meshing*
- *Mesh Refinement*
- *Convergence*
- *Simulation Solving*
- *Displaying Results*
- *Viewing Different Results*
- *Animating*
- *Probing*
- *Convergence Plotting*
- *Exporting Reports*

Prerequisites

The class assumes prior knowledge of Inventor basics as taught in Inventor Introduction to Solid Modelling.

Course Duration

2 days

Next Steps

Inventor Dynamic Simulation