Inventor Simulation and Analysis

SolidCAD
A Cansel Company

AUTODESK.
ACADEMIC AUTHORIZED
TRAINING CENTER

Course Description:

"Inventor Simulation and Analysis" program introduces designers to the built-in toolset for performing motion simulation and linear, structural finite element analysis (FEA) to help users understand how their designs will perform. Stress, strain, deflection, modal (natural shapes and frequencies) and dynamic (rigid-body) simulations are covered.

Upon completion, users will have the ability to confidently validate their digital prototypes and make early decisions to optimize their designs.

Learning Objectives:

Stress and Frame Analysis overview
- Frame vs. Stress Analysis environments
- Global and local mesh control, mid-plane meshing
- Element types (beam, tetrahedral, shell)
- Boundary conditions (loads, constraints)
- Analyzing assemblies (contact options)
- Automatic convergence
- Parametric studies and design optimization
- Modal Analysis overview
- Generative Design *(Shape Generator)*
- Results interpretation for linear static stress and modal analyses

Dynamic Simulation overview
- Creating joints
- Defining constant and variable loads
- Repairing redundancies
- Understanding results
- Sharing dynamic simulation results with Stress Analysis

Supplemental Learning:

- Autodesk Nastran In-CAD Essentials
- Autodesk Fusion 360 Ultimate – Simulation
- Autodesk Inventor Productivity
- Autodesk Inventor Sheet Metal
- Autodesk Inventor Tube & Pipe
- Autodesk Inventor for Cable & Wire Harness