

Autodesk Nastran In-CAD Essentials

The Autodesk Nastran In-CAD Essentials instructs students in the use of the Autodesk® Nastran® In-CAD software. The software is a finite element analysis (FEA) tool that is embedded directly in the Autodesk Inventor software as an Add-In. It is powered by the Autodesk Nastran solver and offers simulation capabilities specifically tailored for designers and analysts as a tool for predicting the physical behavior of parts or assemblies under various boundary conditions.

Objectives

Through a hands-on, practice-intensive curriculum, students acquire the knowledge required to work in the Autodesk Nastran In-CAD environment to setup and conduct FEA analyses on part and assembly models.

Duration

2 days

Prerequisites

This student guide assumes that a student has Finite Element Analysis (FEA) knowledge and can interpret results. The goal is to teach a user that is new to the Autodesk Nastran In-CAD software how to navigate the interface to analyze a model.

Topics Covered

- Getting Started
 - Autodesk Digital Prototyping
 - Introduction to FEA
 - Introduction to Autodesk Nastran In-CAD
 - Working in Autodesk Nastran In-CAD
- Working with the Default Analysis
 - Analysis & Subcases
 - Idealizations & Materials
 - Constraints & Loads
 - Connectors
- Working with the Mesh & Result Plots
 - Meshing Basics
 - Generating & Reviewing the Mesh
 - Customizing the Mesh
 - Loading Analysis Results
 - Visualizing Result Plots
 - Visualizing XY Plot Results
- Surface Contacts
- Working with Composites
- Nonlinear Static Analysis
 - Basics of a Nonlinear Analysis
 - Creating a Nonlinear Static Analysis
- Nonlinear Materials
- Nonlinear Transient Response Analysis
- Normal Modes Analysis
 - Basics of a Dynamic Analysis
 - Creating a Normal Modes Analysis
- Frequency Response Analysis
- Transient Response Analyzes