AUTODESK NASTRAN IN-CAD ESSENTIAL TRAINING

Objectives

The main objective of this course is to guide new users in the use of the Autodesk® Nastran In-CAD. After completing this course users will be able to:

- ✓ Activate and navigate the Autodesk Nastran In-CAD environment to conduct FEA analyses.
- ✓ Create, edit, and assign idealizations and materials (linear, nonlinear, and composites).
- ✓ Manage the creation, setup, and modification of analyses and subcases that are used to analyze both static and dynamic models. Specific analyses types covered in this student guide include: Linear Static, Nonlinear Static, Nonlinear Transient Response, Normal Modes, Direct Frequency Response, Modal Frequency Response, Direct Transient Response, Modal Transient Response
- ✓ Create constraints with the required degrees of freedom and assign them to entities.
- ✓ Create loads that accurately represent the magnitude and location of the loads the model will experience in the working environment.
- ✓ Create Connector elements to simulate how a physical connector such as a rod, cable, spring, rigid body, or bolt will affect the model.
- ✓ Create Surface Contact elements to define contact between interacting components.
- ✓ Assign global and local mesh settings.
- ✓ Run an Autodesk Nastran In-CAD analysis.
- ✓ Review and create plots for analyzing the results.

Pre-requisites

This guide is designed for new users of Inventor. It is recommended that you have a working knowledge of:

- Microsoft® Windows® 7, Microsoft® Windows® 8 or Microsoft® Windows® 10
- Finite Element Analysis (FEA) with the ability to interpret the results.

TRAINING PROGRAMME DAY 1

Chapter 1: Getting Started

- Autodesk Digital Prototyping
- Introduction to FEA
- Introduction to Autodesk Nastran In-CAD
- Working in Autodesk Nastran In-CAD

Chapter 2: Working with the Default Analysis

- Analysis & Subcases
- Idealizations and Materials
- Constraints and Loads
- Connectors

Chapter 3: Working with Mesh & Result Plots

- Meshing Basics
- Generating & Reviewing the Mesh
- Customizing the Mesh
- Loading Analysis Results
- Visualizing Result Plots
- Visualizing XY Plot Results

Chapter 4: Surface Contacts

- Surface Contacts

Chapter 5: Working with Composites

- Working with Composites

TRAINING PROGRAMME DAY 2

Chapter 6: Nonlinear Static Analysis

- Basics of a Nonlinear analysis
- Creating a Nonlinear Static Analysis

Chapter 7: Nonlinear Materials

- Working with Nonlinear materials

Chapter 8: Nonlinear Transient Response Analysis

- Creating a Nonlinear Transient Response Analysis

Chapter 9: Normal Modes Analysis

- Basics of a Dynamic Analysis
- Creating a Normal Modes Analysis

Chapter 10: Frequency Response Analysis

- Creating Frequency Response Analysis

Chapter 11: Transient Response Analysis

- Creating Direct & Modal Transient Response Analysis