

## Autodesk Nastran In-CAD Essentials Training

**Course Length: 2 days**

The Autodesk Nastran In-CAD Essentials training course instructs students in the use of the Autodesk®

Nastran® In-CAD software. This training course was written using the 2019.1.0.200 build 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 behaviour of parts or assemblies under various boundary conditions. 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 analysis on part and assembly models.

### Topics Covered

- Activate and navigate the Autodesk Nastran In-CAD environment to conduct FEA analysis on part and assembly models.
- Create, edit, and assign idealizations and materials (linear and nonlinear) for use in an analysis (including composites).
- Manage the creation, setup, and modification of analyses and subcases that are used to analyse both static and dynamic models. Specific analyses types that are covered in this learning guide include:
  - Linear Static
  - Nonlinear Static
  - Nonlinear Transient Response
  - Normal Modes
  - Direct Frequency Response
  - Modal Frequency Response
  - Direct Transient Response
  - Modal Transient Response
  - Random Response
  - Shock/Response Spectrum
- Create constraints with the required degrees of freedom and assign them to entities in the model.
- 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 in an assembly.
- Assign global and local mesh settings.
- Run an Autodesk Nastran In-CAD analysis.

- Review and create result plots for analysing the results of an Autodesk Nastran In-CAD analysis.

## **Prerequisites**

This training course assumes that a student has Finite Element Analysis (FEA) knowledge, can interpret results, and in general, knows how a model should be setup for an analysis. The main goal of this training course is to teach a user that is new to the Autodesk® Nastran® In-CAD software how to navigate the interface to successfully analyse a model.

This learning guide was written using the 2019.1.0.200 build of the Autodesk® Nastran® In-CAD software. The software user-interface and workflow may vary if older or newer versions of the software are being used.

## Training Guide Contents

### Chapter 1: Getting Started

- Lesson: Autodesk Digital Prototyping
- Lesson: Introduction to FEA
- Lesson: Introduction to Autodesk Nastran In-CAD
- Lesson: Working in Autodesk Nastran In-CAD
- Exercise: Cantilever Beam Exercise

### Chapter 2: Working with the Default Analysis

- Lesson: Analysis & Subcases
- Lesson: Idealizations & Materials
- Lesson: Constraints & Loads
- Lesson: Connectors
- Exercise: Cast Lever Boundary Conditions I
- Exercise: Cast Lever Boundary Conditions II

### Chapter 3: Working with the Mesh and Result Plots

- Lesson: Meshing Basics
- Lesson: Generating & Reviewing the Mesh
- Lesson: Customizing the Mesh
- Lesson: Loading Analysis Results
- Lesson: Visualizing Result Plots
- Lesson: Visualizing XY Plot Results
- Exercise: Refining the Mesh
- Exercise: Working with Line Elements

### Chapter 4: Surface Contacts

- Lesson: Surface Contacts
- Exercise: Contacts & Symmetry in an Assembly Model

### Chapter 5: Working with Composites

- Lesson: Working with Composites
- Exercise: Using Composite Materials in a Bike Frame

### Chapter 6: Nonlinear Static Analysis

- Lesson: Basics of a Nonlinear Analysis
- Lesson: Creating a Nonlinear Static Analysis
- Exercise: Flat Walled Tank

### Chapter 7: Nonlinear Materials

- Lesson: Working with Nonlinear Materials
- Exercise: Flexural Test Fixture

## **Chapter 8: Nonlinear Transient Response Analysis**

- Lesson: Creating a Nonlinear Transient Response Analysis
- Exercise: Ball Impact

## **Chapter 9: Normal Modes Analysis**

- Lesson: Basics of a Dynamic Analysis
- Lesson: Creating a Normal Modes Analysis
- Exercise: Muffler I - Determining Natural Frequencies
- Exercise: Muffler II - Modal Avoidance

## **Chapter 10: Frequency Response Analysis**

- Lesson: Creating a Frequency Response Analysis
- Exercise: Muffler III - Frequency Response

## **Chapter 11: Transient Response Analysis**

- Lesson: Creating Direct & Modal Transient Response Analysis
- Exercise: Wing

## **Chapter 12: Random Response Analysis**

- Lesson: Creating a Random Response Analysis
- Exercise: Muffler IV - Random Response

## **Chapter 13: Shock/Response Spectrum Analysis**

- Lesson: Creating a Shock/Response Spectrum Analysis
- Exercise: Multi-Story Building

## **Appendix A: Dynamic Analysis Theory**