Introduction to Autodesk Inventor Nastran

COURSE DESCRIPTION

Autodesk Inventor Nastran is a powerful Finite Element Analysis (FEA) package built into Inventor. This course teaches the capabilities of Nastran including complex stress, impact, fatigue, vibrations and thermal analysis.

COURSE MODULES

- Introduction to Finite Element Analysis
- Idealisations
- Contacts
- Constraints
- Leads
- Meshes
- Reviewing Results
- Sections
- Animations

STUDY TYPES

- Linear and Nonlinear Static Analysis
- Normal Modes Analysis
- Linear and Nonlinear Buckling Analysis
- Prestress Static Analysis
- Prestress Normal Modes
- Direct and Modal Transient Response
- Impact Analysis
- Nonlinear Transient Response
- Direct, Modal and Random Frequency Response
- Shock/Response Spectrum

OTHER INFORMATION

Prerequisites

Prior knowledge of Inventor Solid Modelling is required. FEA experience is beneficial.

Course Duration

2 Days

Next Steps

A bespoke course

man+machine