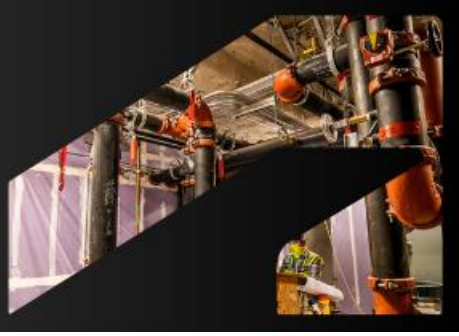
INVENTOR NASTRAN
FEA Simulation



Course Duration: 16 Hours

Overview:

This training will cover the use of Inventor Nastran for Finite Element Analysis (FEA) application. This course offers frequently used simulation capabilities that span across multiple analysis types, such as linear and nonlinear stress, dynamics, and heat transfer.

Learning Objectives:

After completing this session, you will be able to:

- Understand FEA Workflow and Elements
- Perform FEA Simulations
- Extract FEA simulation Results
- Create Simulation Reports

Prerequisites:

It is recommended to have a proper knowledge in Materials Engineering, Mechanics of deformable bodies and Physics.

Acquisition:

Trainees will get an industry recognized Certificate of Completion.

Notes:

The Course topics and duration may be modified by the instructor based upon the knowledge and skill level of the trainees.

Topics Covered:

Touring with Inventor Nastran workspace

Understanding FEA Workflow and Elements

- Material to the model
- Load and Constraint
- Generating mesh
- Starting an analysis
- Linear Static Analysis Basic
- Refining mesh
- Linear Static Analysis Exercises
- Result Visualization

FEA Studies

- Linear Static
- Normal Mode
- Linear Buckling
- Nonlinear Static
- Nonlinear Buckling
- Transient and Frequency Response
- Fatigue
- Heat Transfer

For inquiries, please call or email:

8899-7853 loc 2158/2154
09399734872
inquiry@mscorp.com.ph

maximum SOLUTIONS CORPORATION

AUTODESK
Solutions Provider
Gold Partner
Authorized Training Center

Maximum Engineering Solutions Hub