AUTODESK INVENTOR : STRESS ANALYSIS TRAINING

Objectives:

The objective of the training course is to teach trainee on the fundamental principles and recommended workflows for analyzing designs using Autodesk Inventor Professional. Trainee also learn how to analyze parts and assemblies, perform parametric design studies, and use modal analysis in the FEA and Frame Analysis Environment. After completing this course trainee will be able to:

- ✓ Understanding the Finite Element Analysis (FEA) fundamentals
- ✓ Analyze stress and modal analysis for parts and assemblies
- ✓ Analyze frame analysis
- ✓ Generate and Interpret the FEA results

Pre-requisites

- Knowledge of Autodesk Inventor user interface and working environments.
- Knowledge of part modeling, assembly modeling, and drawing view creation and annotation, is recommended.

Training Programme Day 1

Chapter	Topic	Duration	Time
Chapter 1	**Introduction to Engineering Analysis** • Stress Analysis Overview • Frame Analysis Overview	**1-hour**	**09.00 AM-10.00 AM**
Chapter 2	**Part Modal and Stress Analysis** • Creating Load and Constraint • Assign Material • Preview Mesh • Running the Analysis • Reviewing and Interpreting Analysis Results • Animating and Reporting Analysis Results	**3-hour**	**10.00 AM-01.00 PM**

Chapter 3	**Assembly Stress Analysis** • Stress Analysis Environment • Excluding Components • Assign Materials • Add Constraints and Load • Stress Analysis Setting • Contact Conditions • Generate Meshes • Run the Simulation • Reviewing and Interpreting Analysis Results	**3-hour**	**02.00 PM-05.00 PM**

Training Programme Day 2

Chapter	Topic	Duration	Time
Chapter 4	**Contacts and Mesh Refinement** • Specify and Preview Meshes • Specify Local Mesh Controls • View and Compare the Results	**2-hour**	**09.00 AM-11.00 AM**
Chapter 5	**Assembly Modal Analysis** • Create a simulation study • Assign Materials and Add Constraints • Create Manual Contact • Specify Mesh Options • Preview Mesh and Run the Simulation • Reviewing and Interpreting Analysis Results	**2-hour**	**11.00 AM-01.00 PM**
Chapter 6	**FEA Assembly Optimization** • Modify Mesh • Create Parametric Geometry • Optimization Criteria • Reviewing and Interpreting Analysis Results • Reviewing and animating 3D plots • Reviewing XY Plots	**2-hour**	**02.00 PM-04.00 PM**
Chapter 7	**Stress Analysis Contacts** • Contact Types • Bonded Contact • Separation Contact • Sliding and No Separation Contact • Separation and No Sliding Contact • Shrink Fit and No Sliding Contact • Spring Contact	**1-hour**	**04.00 PM-05.00 PM**

Training Programme Day 3

Chapter	Topic	Duration	Time
Chapter 8	**Frame Analysis** • Frame Analysis Environment • Frame Analysis Settings • Assign Materials • Change Beam Properties • Change Direction of Gravity • Add Constraints • Add Constraints to the Next Beam • Add Loads • Run the Simulation • View and Interpret Results	**2-hour**	**09.00 AM-11.00 AM**
Chapter 9	**Frame Analysis Results** • Get Started • Frame Analysis Environment • View and Interpret the Results • Display Maximum and Minimum Values • View Beam Detail • Display and Edit Diagrams • Adjust Displacement Display • Animate the Results • Generate Report	**2-hour**	**11.00 AM-01.00 PM**
Chapter 10	**Frame Analysis Connections** • Change Direction of Gravity • Add Custom Nodes • Add Custom Nodes • Change Color of Custom Nodes • Assign Rigid Links • Assign a Release • View the Updated Results	**1-hour**	**02.00 PM-03.00 PM**
Chapter 11	**Modal Type of Frame Analysis** • Frame Analysis Environment • Create a Simulation Study • Run the Simulation • View the Results • Animate the Results	**2-hour**	**03.00 PM-05.00 PM**