



IFS ACADEMY

Training For The Future!!

Industry Domain Oriented
PG Diploma in Computer Aided Engineering (CAE)
(Duration: 4 Months)
(With 100% Placement Assistance)
Course Curriculum

Overview:

This course is designed for fresh B.E. / M.E. / M.Tech. Mechanical Engineers and CAD Working Professionals who wish to start their career in CAE as CAE Engineer / FE Analyst.

Course begins with review of Engineering Design Fundamentals followed by Finite Element Analysis theoretical concepts. ANSYS Design Modeler training will teach users how to create and modify their geometry in preparation for their analysis in ANSYS Workbench. You will learn and achieve an understanding of the main tools in SpaceClaim to create a geometry from scratch or to clean up and prepare an imported geometry for a CFD or a FEA modelling using ANSYS solutions. You will also learn how to create named selections and parameters and how to import them to ANSYS Workbench.

You will be trained on Finite Element Modeling using Hypermesh and in-depth Finite Element Analysis case studies using ANSYS Mechanical APDL & Workbench. You will also learn how to use ANSYS Text User Interface.

➤ ***Engineering Design, Solid Mechanics and Strength of Materials***

Free body diagrams and equilibrium; trusses and frames; virtual work; Stress and strain, stress-strain relationship and elastic constants, plane stress and plane strain, shear force and bending moment diagrams; bending and shear stresses; deflection of beams; torsion; Free and forced vibrations; Design for static and dynamic loading; failure theories; fatigue strength and the S-N diagram; Properties of engineering materials, yield criteria.

➤ **Introduction to Finite Element Analysis**

- What is Finite Element Analysis?
- History of FEA
- Analytical and numerical FEA
- Necessity of FEA
- Advantages and Limitation
- Discretization and other approximation in FEA
- Necessity of Meshing
- Convergence and mesh Refinement
- Human Error, Sources and detection, common mistake,
- Mesh Refinement, Mesh Convergence, Mesh Transition

➤ ***CAD Clean Up using ANSYS SpaceClaim (40 Hrs.)***

- Import Procedure
- Importing External CAD Files into SpaceClaim Direct Modeler (SCDM) using ANSYS Workbench
- Importing SpaceClaim Geometry into DesignModeler
- Starting SpaceClaim before ANSYS Workbench
- Starting ANSYS Workbench before SpaceClaim
- Importing SpaceClaim Geometry into Mechanical APDL
- Create a new geometry from scratch in ANSYS SpaceClaim
- Repair an imported geometry in ANSYS SpaceClaim
- Extract the fluid volume from solid bodies
- Create and edit an enclosure around a body
- Create spot welds for an FEA analysis in ANSYS Mechanical
- Using the Midsurface tool to simplify the thin models and mesh with shell elements
- Extracting then beams using Beam Extract tool
- Create Named Selections and parameters in ANSYS SpaceClaim
- Import the geometry to ANSYS Workbench with the Named Selections and the Parameter

➤ ***Finite Element Modeling using Hypermesh (60 Hrs.)***

- Review of Finite Element Theory
- Introduction to Hypermesh Overview
- Overall Meshing Process in Hypermesh
- Understanding Tetra & Hexa meshing
- Midsurface Extraction
- Understanding of Meshing quality criteria's
- FE Meshing of Complex Geometries using Shell & Solid Elements
- HM connections for ANSYS, Abaqus & LS Dyna
- Modeling of Automotive Plastic Trims and BIW Parts
- Automotive Exterior and Interior Trims
- Simulation Specific Model Building

➤ ***Finite Element Analysis using ANSYS Mechanical APDL & ANSYS Workbench***

ANSYS Mechanical APDL:

Before you start using ANSYS, Selection Logic, Solid Modeling, Meshing, Material Properties, Boundary Conditions, Solvers, Post-processing, Introduction to Non-Linear Analysis, Static Structural Analysis, Modal Analysis, Thermal Analysis, Tips & Tricks.

ANSYS Workbench:

Introduction to ANSYS Workbench, Mechanical Basics, General Preprocessing, Meshing in Mechanical, Static Structural Analysis, Modal Analysis, Thermal Analysis, Results and Postprocessing.

Types of Analysis:

- Static Structural Analysis
- Modal Analysis
- Harmonic Analysis
- Transient Dynamic Analysis
- Random Vibration Analysis
- Spectrum Analysis
- Introduction to Fatigue Analysis
- Non Linear and Contact Analysis

ANSYS Mechanical APDL using TUI (TEXT USER INTERFACE)

- Modeling using TUI
- Geometry clean-up using TUI
- Component Creation using TUI
- Element type selection
- Material property creation
- Assigning material properties
- Meshing control, sizing, map mesh, free mesh using TUI
- Creating weld using TUI
- Applying boundary condition using TUI
- Solver setting using TUI
- Post processing result using TUI
- Macro Basics
 - Creating a Macro
 - Macro with Arguments
 - Branching
 - Looping
 - General guidelines
 - Workshop

IFS Academy, Pune

T: +91-20-2543 0338, M: +91-98228 49628, +91-99224 40102, E: training@ifsacademy.org,

Visit Us At: www.ifsacademy.org