IFS Academy
Training For The Future!!

Advance 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 DesignModeler 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.

➢ *Basics of Engineering Design & 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 Design Modeler*

- Navigate within the graphical user interface
- Create, modify and/or cleanup their geometry in preparation for their analyses
- Generate 2-D sketches and convert them into 2-D or 3-D models
- Modify 2-D and 3-D geometry
- Import existing CAD geometry into ANSYS DesignModeler
- Create line bodies and their cross sections in preparation for FE beam analysis
- Repair:
- Automatically Finding Faults
- Generating the Repair Feature
- Viewing Faults/Results
- Repair Context Menu Options
- Repair Feature Types
- Create surface bodies in preparation for FE shell analysis
- Model assemblies
- Utilize parameters

➢ *CAD Clean Up using ANSYS SpaceClaim (70 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 (70 Hrs.)*

Introduction to FEA, Introduction to Hypermesh, Geometry Clean-up, Shell Meshing, 3D Solid Meshing, 1D Meshing and Connectors, Analysis Setup and Loading, Exporting and Solving, Introduction to HyperView.

Case Study & Tool Test: Student has to complete any one live project case study and submit report of the same followed by Tool Test.

Expert Lectures / Seminars on use of Hypermesh in CAE Industry.

> ### *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

Case Study & Tool Test: Student has to complete any one live project case study and submit report of the same followed by Tool Test.

Expert Lectures / Seminars on use of ANSYS in CAE Industry.

> ### *Soft Skills Training*
> Résumé writing, Aptitude Test, Technical Test, Group Discussion Techniques, Interview techniques, Communication & Presentation Skills, Personality Development.

Deliverables:

- Technical sessions from industrial and academic professionals on topics of basics of engineering design and materials.
- Seminars and guidance from industrial professionals on soft skills training.
- **Hard Copy of IFS Academy Certificate of Completion**
- **Exposure to live case studies and projects.**