



IFS ACADEMY

Training For The Future!!

Advanced Diploma in CAE

(Duration: 3 Months)

(With 100% Placement Assistance)

Course Curriculum

➤ ***Basics of Engineering Design***

Applied Mechanics, Strength of Materials, Machine Design, Theory of Machines, Physical Metallurgy, Thermal Engineering, Fluid Dynamics, Mechanical Vibrations.

➤ ***Basics of Finite Element Analysis***

Introduction, History, Methods to solve Finite Element Analysis equations, Stiffness Matrix derivation, 1D, 2D and 3D problems, Types of Analysis, Softwares used in the market.

➤ ***Finite Element Modeling using Hypermesh***

Introduction to Finite Element Modeling, Quality Parameters, GUI, Geometry creation and clean up, Mid Surface extraction, Element Offset, Plane, Cone Torus, Quality Index and Optimising quality. Organise element, Split Element, Replace. Tool page. Mask element, Translate-Reflect-Scale-Count-Number-Renumber. Check Element, Edges, faces. Features, Normal, 1D Bar, Rigid, Spot weld, Rbe3, Spring, FE joints. Drag, Spin, Line Drag, Element offset, Loading & BC's. Exporting FE data to various solvers.

➤ ***Finite Element Analysis using ANSYS***

Introduction to FEA, Elements Study, About ANSYS Inc., ANSYS Products, Selection Logic, Solid Modeling, Meshing, Material Properties, Loading and Boundary Conditions, Solvers, Post Processing, Static Structural, Thermal and Modal Analysis, Introduction to Non Linear Analysis, Tips & Tricks, Project Work.

➤ ***Finite Element Analysis using ANSYS Workbench***

Introduction to ANSYS Workbench, GUI, Procedure, Static Structural, Thermal and Modal Analysis, Introduction to Non Linear Analysis and Contact, Project Work.

➤ **Computational Fluid Dynamics using ANSYS ICEM CFD & Fluent**

Introduction to fluid mechanics & CFD. Types of CFD analysis. Softwares available in the market. Project Work.

➤ **ICEM CFD:**

Introduction to ANSYS ICEM CFD, Importing / editing geometry and mesh, Geometry cleanup and wet volume creation, Shell meshing, Volume meshing, Prism meshing, Unstructured and block structured mesh creation, Multi-block Structured mesh and O-grid generation, Mesh quality checks and improvement, Mesh editing tricks and best practices, Mesh export and solver selection, File and directory structure, Accessing ICEM CFD from Workbench.

➤ **Fluent:**

Introduction to FLUENT & ANSYS Products, Basic fluid flow and CFD, Reading the grid (mesh) and editing of grid (mesh), Materials, Boundary condition setup, Cell zones – fluid / solid, Porous media, General guidelines, Different boundary conditions, Solver theory, Solver settings, Setting solver parameters, Convergence, Accuracy, Unsteady flow modeling, Available solvers, Turbulence modeling, Heat transfer modeling, Complex flow modeling, Convergence settings and monitoring, Advanced physics, Post processing

➤ **Soft Skills Training**

Résumé writing, Aptitude Test, Technical Test, Group Discussion Techniques, Interview techniques, Communication & Presentation Skills, Personality Development.

Duration: 3 Months

Batches: Morning / Afternoon

Contact:

IFS Academy, Pune

Phone: +91-20-6400 7296

Mob. No.: +91-99224 40102, +91-98228 49628

Email: training@infinitesolutionsindia.com

Visit Us At: <http://www.ifsacademy.org>