



# IFS ACADEMY

Training For The Future!!

## ***Computational Fluid Dynamics using*** **ANSYS ICEM CFD & FLUENT / CFX with Introduction to CAD** **(3D Modeling)**

***Course Curriculum (Duration: 100 Hrs.)***

### **Section 1: Introduction to CAD**

Different CAD Softwares, Graphical User Interface, Creating Sketcher Geometry and using Sketcher Tools, Selecting and Editing Geometry, Part Modeling, Feature based Modeling, Assembly with constraints. Import and Export File options.

### **Section 2: Before you start using ANSYS FLUENT**

#### **a. Review of viscous flow theory:**

Review of fundamental concepts – continuum, control volume, Eulerian and Lagrangian methods of description of fluid flow; Reynolds transport equation – integral and differential forms of continuity, momentum, and energy equations.

#### **b. Navier-Stokes equations**

Navier-Stokes equations and boundary conditions; Nondimensionalization of equations and order of magnitude analysis, dimensionless parameters and their significance. Exact solution of incompressible Navier-Stokes equations – Couette flow, flow between rotating cylinders, fully developed flow through ducts.

#### **c. Introduction to Turbulence**

Introduction to turbulent flow, stability of laminar flow, mean motion and fluctuation, time averaged turbulent flow equations, Reynolds stresses, boundary layer equations, boundary conditions, eddy viscosity, mixing length hypothesis.

#### **d. Computational Fluid Dynamics**

##### **Module 1**

Experimental, theoretical and numerical methods of predictions; physical and mathematical classifications partial differential equations; computational economy; numerical stability; validation of numerical results; round-off-error and accuracy of numerical results; iterative convergence, condition for convergence, rate of convergence; under – and over – relaxations, termination of iteration; tridiagonal matrix algorithm; discretization – converting derivatives to their finite difference forms – Taylor's series approach, polynomial fitting approach; discretization error.

##### **Module II**

Steady one-dimensional conduction in Cartesian and cylindrical coordinates; handling of boundary conditions; two – dimensional steady state conduction problems in Cartesian and cylindrical co-ordinates– point-by-point and line-by-line method of solution, dealing with Dirichlet, Neumann, and Robins type boundary conditions; formation of discretized equations for regular and irregular boundaries and interfaces; grid generation methods; adaptive grids.

### **Module III**

One-, two, and three-dimensional transient heat conduction problems in Cartesian and cylindrical co-ordinates – explicit, implicit, Crank-Nicholson and ADI schemes; stability criterion of these schemes; conservation form and conservative property of partial differential and finite difference equations; consistency, stability and convergence for marching problems.

### **Module IV**

Finite volume method for diffusion and convection–diffusion problems – steady one-dimensional convection and diffusion; upwind, hybrid and power-law schemes, discretization of equation for two-dimension, computation of the flow field using stream function–vorticity formulation; SIMPLE, SIMPLER, SIMPLEC and QUICK schemes, solution algorithms for pressure–velocity coupling in steady flows; numerical marching techniques.

## **Section 3: ICEM-CFD and FLUENT**

- **ANSYS ICEM-CFD:**

1. **Introduction**

- a. ANSYS ICEM CFD Overview
- b. GUI & Layout
- c. ICEM CFD File Management
- d. Menu Introduction
- e. Work Flow and Meshing Process
- f. Workbench ICEM Link

2. **Geometry Basics**

- a. Geometry Handling
- b. Importing / editing geometry and mesh
- c. Geometry creation, repair and simplification
- d. Faceted Geometry Handling
- e. Workshop

3. **Shell Meshing**

- a. Shell meshing
- b. Local & Global Mesh Settings
- c. Mesh Methods, Types and Computation
- d. Workshops

4. **Auto-Volume Meshing**

- a. Introduction to Volume meshing
- b. General Meshing Procedure
- c. Mesh Types
- d. Mesh Methods & Settings
- e. Mesh Density Option
- f. Periodicity

5. **Prism Meshing**

- a. General Procedure
- b. Prism Global Parameters
- c. Local Prism Settings
- d. Quality Control Option
- e. Structured & Unstructured Mesh

- f. Workshops

**6. Hexa Meshing**

- a. Blocking
- b. Nomenclature
- c. Blocking Process
- d. O-Grid
- e. Edit O-Grid
- f. Workshops

**7. Solver Output**

- a. Mesh quality checks and improvement
- b. Mesh editing tricks and best practices
- c. Mesh export and solver selection
- d. Output Options
- e. File and directory structure
- f. Accessing ICEM CFD from Workbench

• **ANSYS FLUENT:**

- a. Introduction to FLUENT & ANSYS Products
- b. Basic fluid flow and CFD (Theory topics are as mentioned above.)
- c. Reading the grid (mesh) and editing of grid (mesh)
- d. Materials
- e. Boundary condition setup
  - Cell zones – fluid / solid
  - Porous media
  - General guidelines
  - Different boundary conditions
- f. Solver theory
- g. Solver settings
  - Setting solver parameters
  - Convergence
  - Accuracy
  - Unsteady flow modeling
  - Available solvers
- h. Fluid Flow Modeling (Pressure & Velocity)
- i. Turbulence Modeling
- j. Aerodynamics Modeling
- k. Porous Media
- l. Turbo Machinery
- m. Multi-Phase
- n. Heat transfer Modeling
- o. Transient Flow Modeling
- p. Theory of Discretization
- q. Convergence settings and monitoring
- r. Post processing

**Section 4:**

- Hand on Workshops, Exercises and Live Case Studies
- Live Case Study CAD Models with Problem Description shall be given to the students.

\*\*\*\*\*

IFS Academy, Pune

Phone: +91-20-6400 7296, Email: [training@ifsacademy.org](mailto:training@ifsacademy.org),

Visit Us At: [www.ifsacademy.org](http://www.ifsacademy.org)