



# IFS ACADEMY

Training For The Future!!

## ***Advanced Computational Fluid Dynamics using ANSYS Workbench & FLUENT Course Curriculum (Duration: 80 Hrs.)***

### **Section 1: 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 2: ANSYS WORKBENCH**

### **1. Introduction to ANSYS Workbench**

- a. ANSYS Workbench
- b. Workbench Overview
- c. Basic Workflow
- d. Alternate Workflow
- e. Cell States
- f. Sharing Data between Different Solvers
- g. File Location on Disk
- h. Use of Archive / Restore
- i. Working with Parameters / Refresh and Update

### **2. DesignModeler**

- a. What is DesignModeler?
- b. Launching DesignModeler
- c. DesignModeler Interface
- d. Tree Outline
- e. Details View
- f. Status / Info Bar
- g. Toolbars: Selection Tool
- h. Toolbars: View Controls

### **3. Planes and Sketches and Modeling**

- a. Concept of Planes and Sketches
- b. New Plane and Sketch Creation
- c. Sketching Interface
- d. Draw Toolbox Examples
- e. Modify Toolbox
- f. Dimensions Toolbox
- g. Constraints Toolbox
- h. Settings Toolbox
- i. Body Types
- j. Body States
- k. 3D Feature Creation
- l. Extrude, Revolve, Skin/Loft, Mirror, Move, Pattern, Boolean and Slice Operations
- m. 2D Feature Creation
- n. Surfaces from Sketches
- o. Single and Multi-Body Parts
- p. Single, Multi Solid Bodies

#### **4. Geometry Simplification and Repair**

- a. Geometry Cleanup/Repair Introduction
- b. Typical Geometry Issues
- c. Analysis Tools
- d. Repair Tools
- e. Repair: Automation
- f. Face / Edge Delete
- g. Merge
- h. Face Lift
- i. Projection
- j. Body Operation: Sew
- k. Surface Extension

#### **5. Flow Volume Creation**

- a. Fill
- b. Enclosure
- c. Symmetry

#### **6. CAD Connections**

- a. Geometry Properties
- b. CAD Connections: Supported Readers & Plug-Ins/Interfaces
- c. Bi-Directional CAD
- d. Named Selection Manager
- e. Pre-requisites for CAD Connections
- f. Advanced Geometry Options

#### **7. ANSYS Meshing**

- a. What is ANSYS Meshing?
- b. Meshing Fundamentals
- c. Meshing Process in ANSYS Meshing
- d. ANSYS Meshing GUI
- e. Geometry Configuration
- f. Meshing for 3D Geometry
- g. Meshing for 2D Geometry
- h. Global Mesh Controls
- i. Local Mesh Controls
- j. Mesh Generation
- k. Named Selections
- l. Mesh Statistics and Mesh Metrics
- m. Parameterization in ANSYS Meshing
- n. Connections
- o. Selective Meshing
- p. Assembly Meshing

#### **8. Mesh Quality**

- a. Impact of the Mesh Quality
- b. Impact of the Mesh Quality on the Solution
- c. Grid Dependency
- d. Hexa Vs. Tetra
- e. Mesh Statistics and Mesh Metrics
- f. Mesh Quality Metrics

- g. Mesh Quality
- h. Aspect Ratio
- i. Smoothness
- j. Mesh Metric Graph
- k. Section Planes
- l. Mesh Quality Check for Fluent
- m. Factors Affecting Quality
- n. Virtual Topology
- l. Pinch

### **Section 3: 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: +91-20-2543 0338, Email: [training@ifsacademy.org](mailto:training@ifsacademy.org),

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