



IFS ACADEMY

Training For The Future!!

Introduction to Computational Fluid Dynamics using ANSYS Workbench & FLUENT Course Curriculum (Duration: 60 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
- o. Pinch

- **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. Heat transfer modeling
- k. Theory of Discretization
- l. Convergence settings and monitoring
- m. Post processing

Section 3:

- 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,

Visit Us At: www.ifsacademy.org