IFS ACADEMY

Training For The Future!!

Advanced Computational Fluid Dynamics using

ANSYC ICEM CFD & 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 discritized 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: 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 Parametrs
    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 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: +91-20-6400 7296, Email: training@ifsacademy.org,

Visit Us At: www.ifsacademy.org