



IFS ACADEMY

Training For The Future!!

Training Schedule for Computational Fluid Dynamics (CFD) using ANSYS ICEM CFD & Fluent

Day I

Introduction to CFD

1. Introduction to Simulation
2. Need of Simulation (Old Method and New Method to Optimize the product)
3. Discussion on CFD Applications in different domain and Jobs Opportunities
4. What is CFD?
5. CFD Methodology
6. Need of basic Fluid Dynamics & Heat transfer fundamentals
7. Governing equation of CFD / Navier Stoke equation / Reynolds Transport Theorem
8. CFD Steps – CAD Simplification, Pre-processing, Solving & Post Processing
9. Grid Discretization
10. Interactive learning with **ICEM-CFD** Grid Generation

Day II

Working on ANSYS ICEM CFD & Fluent

1. Grid Generation (ANSYS ICEM CFD)
2. Tetra/Prism and Hexa core meshing
3. Wall Y+ & Boundary layer in Mesh
4. Discussion on the nature of problem and their CFD approach
5. **Fluent** Introduction
6. How to setup the CFD problem in ANSYS Fluent
7. Coupled / Segregated solver
8. Appropriate Boundary Condition
9. Solver Basics
10. Running Simulation
11. Post processing results
12. How to validate the results?
13. Discussion on different type of problems in Industry

Day III

Practicals / Workshops

1. Flow Analysis in Branching Duct
2. Investigate the flow distribution in Inlet manifold
3. Heat Transfer Analysis

Copyright 2010, IFS Academy, Pune

Phone: +91-20-6400 7296, Mob. No.: +91-98228 49628, Email: training@infinitesolutionsindia.com,

Visit Us At: www.ifsacademy.org