IFS ACADEMY

Training For The Future!!

IFS Academy Career Program in Computational Fluid Dynamics using

ANSYS Workbench & FLUENT

Instructor-Led Online Training

Course Curriculum

(Duration: 100 Hrs. including Lectures & Practicals)

- **Engineering the Future:**
  - CAD, CAE (CFD & FEA), and CAM, applications, and scope in industry and research.
- **Introduction of Computational Fluid Dynamics (CFD)**
  - **Governing Equations:** Conservation of mass (continuity), momentum (Navier-Stokes), and energy. Assumptions and simplifications (incompressible vs compressible, steady vs transient).
  - **Compressible vs Incompressible Flow Setups in ANSYS Fluent:** Ideal gas vs constant density models. Selection of appropriate solvers and energy models.
- **Introduction of ANSYS Workbench, FLUENT & ANSYS Products**
  - **Geometry and Meshing:** Geometry creation (basic shapes to complex CAD import) Mesh generation: structured, unstructured, inflation layers Mesh quality and refinement strategies.

- **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

- **SpaceClaim**
  a. What is SpaceClaim?
  b. Launching SpaceClaim
  c. Space Claim Interface
  d. Tree Outline
  e. Details View
  f. Status / Info Bar

g. Toolbars: Selection Tool
h. Toolbars: View Controls

- **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

- **Simulation Setup in ANSYS Fluent:** Setting material properties, Defining cell zones and boundary types, Solver settings: steady/transient, pressure-velocity coupling, Use of monitors and initialization.

- **Post-Processing Workflows:** Contours, vectors, streamlines, surface/volume integrals Report generation (forces, heat transfer, flow rates).

Sr. No.	Name of Projects
1	**Internal Flow Analysis of a Mixing Tee** - Geometry and mesh generation for T-junction - Steady-state laminar/turbulent flow simulation - Velocity and pressure distribution - Effect of inlet velocity ratio - Use of contours and streamlines
2	**Fluid Flow and Heat Transfer in a Mixing Elbow** - Conjugate heat transfer between fluid and solid elbow - Temperature and velocity profile development - Wall heat flux analysis - Use of energy model and thermal boundary conditions
3	**Conjugate Heat Transfer Analysis of an Exhaust Port** - Transient CHT analysis - High-temperature exhaust gas simulation - Boundary layer heat transfer - Realistic thermal boundary (external convection + conduction in solid) - Material property assignment
4	**External Flow Analysis Over a Circular Cylinder** - External laminar/turbulent flow simulation (Re dependent) - Drag and lift coefficient calculation - Wake formation and vortex shedding visualization - Pressure and velocity contours
5	**Aerodynamic Study of Flow Over a NACA 0012 Airfoil** - Airfoil geometry setup with angle of attack variation - Lift and drag force calculation

	- Boundary layer analysis - Mesh refinement near wall - Use of turbulence models (k-epsilon/k-omega)
6	**Flow Analysis Over an Ahmed Body** - External automotive aerodynamics - Y+ and mesh sensitivity study - Flow separation and reattachment - Drag prediction
7	**Thermal and Flow Analysis of a Graphics Card** - Heat sink and cooling flow modeling - Use of real PCB + component heat sources - Natural and forced convection analysis - Contour plots for temperature and velocity - Use of fan boundary condition
8	**Cyclone Separator Simulation Using Discrete Phase Modeling (DPM)** **- Eulerian-Lagrangian approach** **- Injection of discrete particles** **- Cyclonic separation of solid particles** **- Efficiency curve analysis** **- Use of wall and escape boundary conditions**
9	**Addition of Turning Vanes to a Rectangular Duct (HVAC)** - Analyze airflow through a rectangular duct with and without turning vanes. - Study reduction in flow separation and recirculation zones. - Evaluate pressure drop and total pressure recovery. - Improve velocity uniformity at duct outlet. - Optimize number, angle, and position of turning vanes.
10	**Analysis of Shell and Tube Heat Exchangers** - Simulated 3D flow and thermal behavior in shell and tube heat exchanger geometry. - Analyzed heat transfer between hot and cold fluids across tube walls. - Evaluated pressure drop and temperature rise/fall on both shell and tube sides.

****

IFS Academy, Pune

Phone: +91-98228 49628 / 99224 40102, Email: training@ifsacademy.org,

Visit Us At: www.ifsacademy.org