



IFS ACADEMY

Training For The Future!!

***Advanced Computational Fluid Dynamics using
ANSYS Workbench & FLUENT
Instructor-Led Online Training
Course Curriculum
(Duration: 60 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 <ul style="list-style-type: none"> - 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 <ul style="list-style-type: none"> - 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 <ul style="list-style-type: none"> - 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 <ul style="list-style-type: none"> - 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 <ul style="list-style-type: none"> - 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)