IFS ACADEMY
Training For The Future!!

Advanced Computational Fluid Dynamics (CFD) using ANSYS Workbench & ANSYS FLUENT

Instructor-Led Online Training

Course Curriculum

(Duration: 60 Hrs. including Theory, Hands-on Practical Sessions & Industry Projects)

- **Introduction to Engineering Design & Simulation:**
  - CAD, CAE (CFD & FEA), and CAM, applications, and scope in industry and research.
- **Introduction to 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.
  - Reynolds Number, Flow Regimes, Laminar vs Turbulent Flow, Types of Boundary Conditions and Heat Transfer Modes

- **Introduction to ANSYS Workbench, ANSYS Fluent & ANSYS Product Portfolio**
  - **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

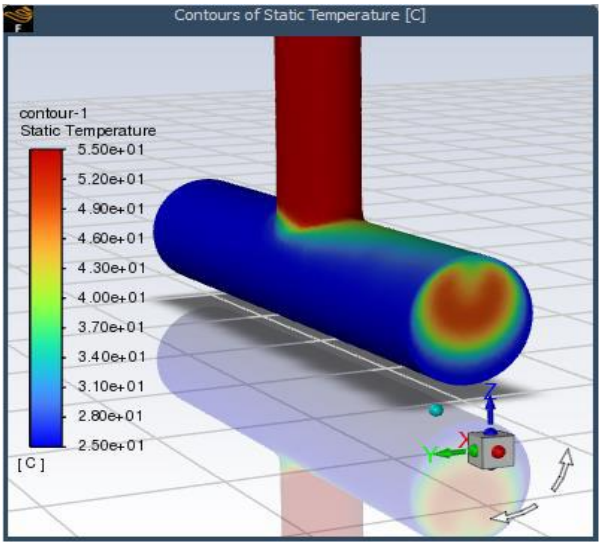
f. Status / Info Bar
g. Toolbars: Selection Tool
h. Toolbars: View Controls

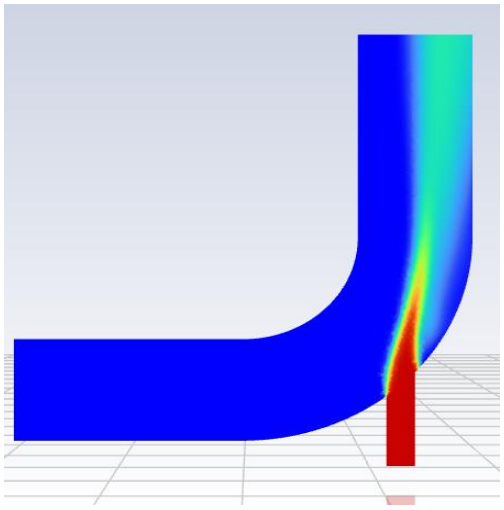
- **Mesh Quality**

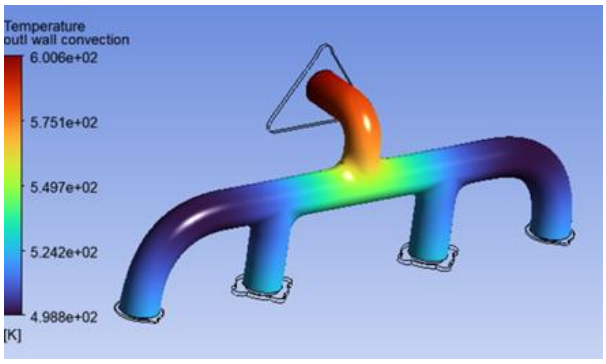
a. Impact of the Mesh Quality
b. Impact of the Mesh Quality on the Solution
c. Grid Dependency
d. Hexa Vs. Tetra
e. Mesh Statistics and Mesh Metrics
f. Mesh Quality Metrics
g. Mesh Quality
h. Aspect Ratio
i. Smoothness
j. Mesh Metric Graph
k. Section Planes
l. Mesh Quality Check for Fluent

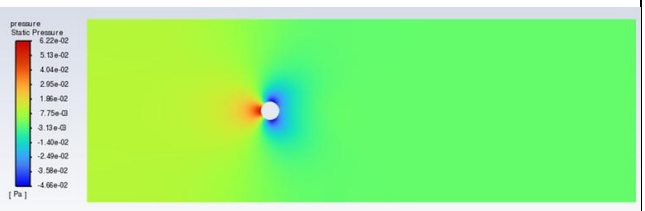
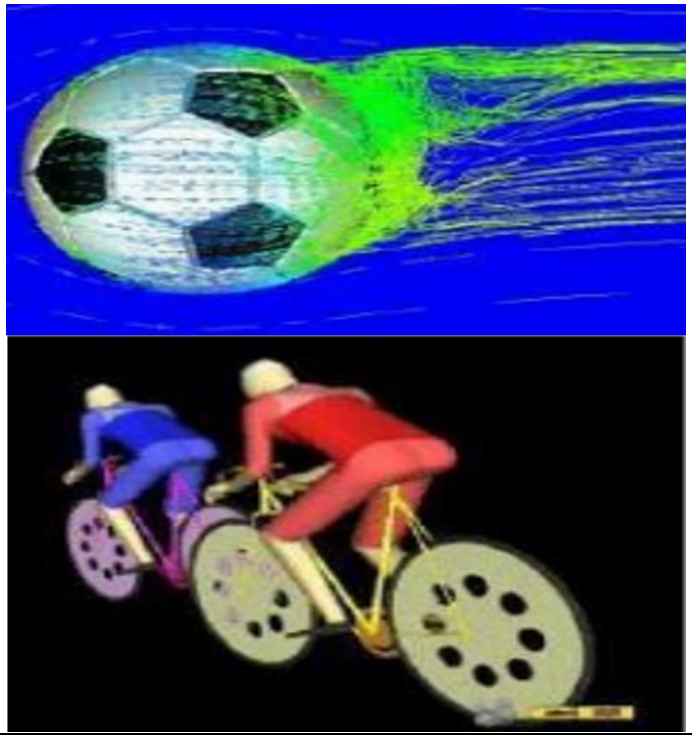
- **Simulation Setup in ANSYS Fluent:** Setting material properties, Defining cell zones and boundary types, Solver settings: steady/transient, pressure-velocity coupling, Use of monitors and initialization.
    - **Post-Processing Workflows:** Contours, vectors, streamlines, surface/volume integrals Report generation (forces, heat transfer, flow rates).

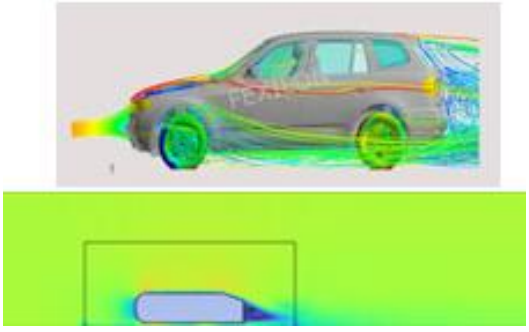
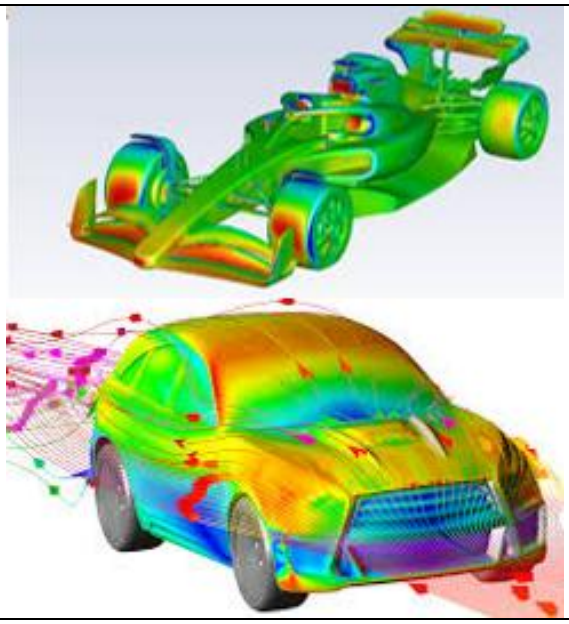
Projects:

Project Title	Applications
Project: 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 -  - 	 

Project Title	Applications
Project 02 **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  - -	 

Project Title	Applications
Project 03 **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 - 	 

Project Title	Applications
Project 04 **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 	

Project Title	Applications
Project 05 **Flow Analysis Over an Ahmed Body** - External automotive aerodynamics - Y+ and mesh sensitivity study - Flow separation and reattachment - Drag prediction 	

****