ANSYS Fluent: Computational Fluid Dynamics Fundamentals

Target Audience

This course is designed for mechanical engineers, aerospace engineers, and simulation professionals who want to gain practical skills in **Computational Fluid Dynamics (CFD)** using ANSYS Workbench and Fluent. It is ideal for learners working in industries such as automotive, aerospace, energy, and manufacturing where fluid flow, heat transfer, and simulation-driven design are critical.

Course Outcomes

By the end of this course, participants will be able to:

- Understand CFD fundamentals, governing equations, and fluid flow types.
- Navigate ANSYS Workbench and Fluent workflows for simulation projects.
- Prepare and simplify CAD geometry for CFD analysis.
- Generate structured and unstructured meshes with quality assessment.
- Configure solver setups including boundary conditions, materials, and solution methods.
- Apply basic physics models for laminar, turbulent, and thermal flows.
- Post-process CFD results with contours, vectors, plots, and reports.
- Validate simulations through mesh independence and convergence studies.
- Document engineering findings with professional reports.

Course Objectives

- Provide a structured foundation in CFD concepts and ANSYS Fluent workflows.
- Train learners in geometry preparation, meshing, and solver setup.
- Develop proficiency in applying physics models for fluid and thermal analysis.
- Enable learners to interpret CFD results and generate engineering reports.
- Reinforce learning through hands-on labs simulating industrial workflows.

Course Outline

The course comprises **32 hours** of theory and labs and is divided into **7 different chapters**. Each chapter will be followed by hands-on lab exercises to reinforce learning and gauge understanding of the topics covered.

Table of Contents

Module 1 – Introduction to Computational Fluid Dynamics

- Introduction to CFD
- Engineering Applications of CFD
- Advantages and Limitations of CFD
- CFD Workflow
- Governing Equations
- Types of Fluid Flow
- Reynolds Number
- Laminar and Turbulent Flow
- Compressible and Incompressible Flow
- Steady and Transient Flow

Module 2 – Introduction to ANSYS Workbench and Fluent

- ANSYS Workbench Overview
- Fluent Workflow
- Project Schematic
- User Interface
- Importing CAD Models
- Units and Coordinate Systems
- Named Selections
- Project Organization

Module 3 – Geometry Preparation

- CAD Cleanup
- Simplifying Geometry
- Suppressing Features
- Fluid Volume Extraction
- Enclosure Creation
- Shared Topology
- Geometry Validation

Module 4 – Meshing Fundamentals

- Introduction to CFD Meshing
- Structured and Unstructured Mesh
- Surface Meshing
- Volume Meshing
- Local Mesh Controls
- Inflation Layers
- Mesh Quality Parameters
- Mesh Statistics
- Mesh Independence Study

Module 5 – Solver Setup

- Pressure-Based Solver
- Density-Based Solver
- Material Properties
- Cell Zone Conditions
- Boundary Conditions
- Operating Conditions
- Solution Methods

- Initialization
- Residual Monitoring
- Convergence Criteria

Module 6 – Basic Physics Models

- Laminar Flow Model
- Introduction to Turbulence Models
- Energy Equation
- Gravity
- Heat Transfer Fundamentals
- Simple Internal Flow Analysis

Module 7 – Post Processing and Reporting

- Contours
- Velocity Vectors
- Streamlines
- Pathlines
- XY Plots
- Surface Integrals
- Force and Moment Reports
- Exporting Results
- Engineering Report Generation
