ANSYS CFD Fundamentals using Fluent

Target Audience

This course is tailored for engineering students, early-career professionals, and beginners in fluid mechanics who want to gain a practical introduction to CFD. It is ideal for learners with basic knowledge of fluid dynamics who wish to understand how to set up, run, and interpret CFD simulations using ANSYS Workbench for real-world engineering applications.

Course Outcomes

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

- Understand the fundamentals of CFD and its industrial applications.
- Prepare geometry and generate high-quality meshes for CFD simulations.
- Configure solver settings and apply boundary conditions for steady and transient flows.
- Implement turbulence models and compare laminar vs. turbulent regimes.
- Perform heat transfer and multiphysics simulations.
- Post-process results effectively and validate simulations.
- Execute a complete CFD workflow through an industry-inspired capstone project.

Course Objectives

The objectives of this course are to:

- Introduce CFD concepts, governing equations, and fluid mechanics basics.
- Provide hands-on training in geometry preparation, meshing, and solver setup.
- Teach turbulence modeling and heat transfer applications in CFD.
- Develop skills in post-processing, data extraction, and reporting.
- Reinforce learning through practical labs and a capstone project.

Course Outline

The course comprises **40 hours** of theory and labs and is divided into **7 modules**. 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 CFD & Fluid Mechanics Basics

- CFD Overview: What is CFD, industry applications (aerospace, automotive, turbomachinery, HVAC), typical workflow
- Fluid Mechanics Refresher: Flow properties, Reynolds number, laminar vs. turbulent flows, boundary layer concept
- Governing Equations: Continuity, momentum (Navier–Stokes), energy equation
- Hands-on Demo: 2D laminar channel flow (velocity profile, residual convergence)

Module 2 — Geometry Preparation & Meshing

- Geometry Preparation: Importing CAD models, cleanup, removing small features, creating fluid domain
- Meshing Fundamentals: Structured, unstructured, hybrid meshes
- Boundary Layer Meshing: Inflation layers, y+ concept, near-wall resolution
- Mesh Quality: Skewness, orthogonality, aspect ratio
- Hands-on Labs: Pipe flow mesh, airfoil mesh, mesh independence study

Module 3 — Solver Setup & Boundary Conditions

- Solver Selection: Pressure-based vs. density-based solvers
- Simulation Types: Steady-state vs. transient
- Boundary Conditions: Velocity inlet, pressure outlet, mass flow inlet, wall, symmetry
- Wall Modeling: No-slip condition, wall functions
- Initialization & Convergence: Hybrid initialization, residual monitoring, physical monitors
- Hands-on Labs: Flow through a duct, backward-facing step flow

Module 4 — Turbulence Modeling

- Introduction to Turbulence: Reynolds number, flow transition, turbulence physics
- Common Models: k-ε (standard, realizable), k-ω SST
- Model Selection Guidelines: When to use k-ε, k-ω, SST
- Hands-on Lab: Turbulent flat plate, comparison of laminar vs. turbulent results

Module 5 — Heat Transfer & Multiphysics

- Energy Equation: Conduction, convection, radiation basics
- Conjugate Heat Transfer: Fluid–solid coupling
- Engineering Applications: Electronics cooling, heat exchangers, cooling ducts
- Hands-on Labs: Heated plate cooled by airflow, simple heat exchanger simulation

Module 6 — Post-Processing & Result Analysis

- Visualization: Contours, vectors, streamlines, iso-surfaces
- Engineering Data Extraction: Velocity profiles, pressure drop, drag & lift forces
- Validation: Comparing with theory, checking grid independence
- Hands-on Labs: Extract drag coefficient of airfoil, plot pressure drop in pipe

Module 7 — Industry Case Study & Capstone

- Real Engineering Problems: Pump flow analysis, cooling duct airflow, external aerodynamics
- Capstone Project: Airflow through a machine cooling duct (full workflow: geometry → mesh → solver → post-processing → reporting)