ANSYS Fluent Advanced Computational Fluid Dynamics (CFD) and Heat flow

Target Audience

This course is intended for engineering students, researchers, and professionals who already have a foundational understanding of CFD and want to advance their skills in numerical accuracy, turbulence modelling, external aerodynamics, rotating machinery, heat transfer, multiphase flows, and transient simulations. It is particularly suited for learners aiming to apply CFD to complex engineering problems in aerospace, automotive, turbomachinery, and thermal-fluid systems.

Course Outcomes

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

- Apply advanced CFD concepts and understand numerical accuracy issues.
- Generate high-quality meshes and perform mesh independence studies.
- Select and implement turbulence models for different engineering applications.
- Conduct external aerodynamics simulations and calculate lift/drag forces.
- Model rotating machinery flows using advanced CFD techniques.
- Perform heat transfer and conjugate heat transfer simulations.
- Simulate multiphase and transient flows with industry relevance.
- Post-process results effectively and validate simulations.
- Execute a complete CFD workflow through a capstone project.

Course Objectives

The objectives of this course are to:

- Provide deeper insights into advanced CFD fundamentals and numerical methods.
- Train learners in advanced meshing strategies and turbulence modeling.
- Introduce external aerodynamics and rotating machinery simulations.
- Develop expertise in heat transfer, multiphase, and transient flow simulations.
- Enhance skills in advanced post-processing and validation techniques.
- Reinforce learning through hands-on labs and a student-friendly capstone project.

Course Outline

The course comprises **64 hours** of theory and labs and is divided into **10 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 5 – Rotating Machinery CFD

- Rotating Flow Modeling: Moving Reference Frame (MRF), rotational effects
- Applications: Pumps, fans, turbomachinery
- Hands-on Lab: Fan simulation (velocity distribution, pressure rise)

Module 6 – Heat Transfer CFD

- Thermal Modeling: Energy equation, convection, conjugate heat transfer
- Applications: Electronics cooling, heat exchangers, cooling ducts
- Hands-on Lab: Heated plate cooling simulation (temperature distribution, heat transfer coefficient evaluation)

Module 7 – Multiphase Flow Simulation

- Concepts: Free surface flows, particle transport
- Models: Volume of Fluid (VOF), mixture model
- Hands-on Lab: Water-air free surface flow (filling tank simulation)

Module 8 – Transient Flow Simulation

- Transient Concepts: Time stepping, CFL condition
- Vortex Dynamics: Vortex shedding, wake oscillations
- Hands-on Lab: Flow over cylinder (vortex shedding frequency, Strouhal number analysis)

Module 9 – Advanced Post-Processing & Validation

- Visualization: Contours, streamlines, vorticity plots
- Data Extraction: Lift and drag, pressure drop, velocity profiles
- Validation: Comparing with theory, grid independence checks
- Hands-on Lab: Extract aerodynamic coefficients, pressure drop analysis

Module 10 – Capstone Project

- Students perform complete CFD workflow: Geometry → Mesh → Solver setup → Physics → post-processing → Reporting