ANSYS Fluent: Advanced Computational Fluid Dynamics and Industrial Applications

Target Audience

This course is designed for experienced CFD engineers, simulation specialists, research professionals, product development engineers, and engineering consultants who require expertise in advanced CFD methodologies and industrial simulation workflows. Participants should have prior experience with ANSYS Fluent and be comfortable performing standard CFD analyses before undertaking advanced physics simulations and optimization studies.

Course Outcomes

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

- Apply advanced turbulence models for complex flow problems.
- Perform advanced multiphase flow simulations.
- Model combustion and species transport.
- Configure dynamic mesh analyses.
- Develop User Defined Functions (UDFs).
- Perform parametric and optimization studies.
- Utilize High Performance Computing (HPC).
- Validate advanced CFD simulations.
- Solve complex industrial engineering problems.
- Execute complete industrial CFD projects using ANSYS Fluent.

Course Objectives

- Provide advanced-level training in ANSYS Fluent for industrial CFD applications.
- Develop proficiency in turbulence, multiphase, combustion, and dynamic mesh modeling.
- Train learners in UDF development and customization for tailored simulations.
- Enable learners to conduct optimization studies and leverage HPC resources.
- Reinforce industrial workflows through capstone projects and case studies.

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

- Dynamic Mesh Controls
- Motion Definition

Module 5 – User Defined Functions (UDF)

- Introduction to UDF
- UDF Programming Basics
- Interpreted UDF
- Compiled UDF
- Custom Material Properties
- Boundary Condition Customization
- Source Terms
- User-Defined Profiles

Module 6 – CFD Optimization and High Performance Computing

- Parametric Studies
- Design Optimization
- Response Surface Methods
- Batch Simulations
- Parallel Computing
- High Performance Computing (HPC)
- Performance Optimization

Module 7 – Industrial Applications and Capstone Projects

- Automotive Aerodynamics
- Aerospace External Flow
- HVAC Airflow Analysis
- Electronics Cooling
- Heat Exchanger Analysis
- Process Equipment Simulation
- Battery Thermal Management
- Industrial Validation Techniques
- **Capstone Projects:**