

ANSYS Fluent: Applied CFD and Thermal Analysis

Target Audience

This course is intended for engineers and simulation professionals who have a working knowledge of ANSYS Fluent or have completed a beginner-level CFD course. It focuses on enhancing simulation capabilities through advanced meshing techniques, turbulence modeling, transient simulations, heat transfer analysis, rotating machinery, and multiphase flow applications commonly used in industrial product development.

Course Outcomes

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

- Generate advanced meshes for complex geometries.
- Select suitable turbulence models.
- Perform transient CFD simulations.
- Conduct conjugate heat transfer analyses.
- Simulate rotating machinery.
- Perform multiphase flow analyses.
- Validate simulation accuracy.
- Interpret engineering simulation data.
- Apply CFD best practices to industrial problems.
- Produce professional engineering simulation reports.

Course Objectives

- Provide intermediate-level training in ANSYS Fluent for industrial CFD applications.
- Develop proficiency in advanced meshing, turbulence modeling, and transient simulations.
- Train learners in heat transfer, rotating machinery, and multiphase flow analysis.
- Reinforce simulation accuracy through validation and reporting techniques.
- Apply CFD best practices to real-world engineering problems.

Course Outline

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

Table of Contents

Module 1 – Advanced Meshing Techniques

- Advanced Mesh Controls
- Face Sizing
- Body Sizing
- Edge Sizing
- Local Refinement
- Inflation Optimization
- Polyhedral Mesh
- Mesh Quality Improvement
- Mesh Diagnostics

Module 2 – Turbulence Modeling

- Turbulence Fundamentals
- Standard k- ϵ Model
- RNG k- ϵ Model
- Realizable k- ϵ Model
- Standard k- ω Model
- SST k- ω Model
- Near Wall Treatment
- y^+ Concepts
- Turbulence Model Selection

Module 3 – Heat Transfer Analysis

- Conduction

- Convection
- Radiation
- Thermal Boundary Conditions
- Heat Sources
- Conjugate Heat Transfer
- Thermal Performance Evaluation

Module 4 – Transient CFD Analysis

- Time-Dependent Flow
- Time Step Selection
- Initialization Methods
- Solution Controls
- Monitoring Transient Results
- Animation of Results

Module 5 – Rotating Machinery Simulation

- Rotating Reference Frame
- Sliding Mesh
- Fan Simulation
- Pump Simulation
- Turbomachinery Basics

Module 6 – Multiphase Flow

- Introduction to Multiphase Flow
- Volume of Fluid (VOF)
- Mixture Model
- Eulerian Model
- Free Surface Simulation
- Bubble Flow
- Liquid-Gas Applications

Module 7 – Validation and Engineering Reporting

- Mesh Independence Study
- Verification and Validation
- CFD Best Practices
- Engineering Interpretation
- Report Preparation
- Industrial Case Studies