

Computational Fluid Dynamics Fundamentals Training Course

7 – 11 June 2027
London

AGILE LEADERS
Training Center

Computational Fluid Dynamics Fundamentals Training Course

Ref: 1877_240061 **Date:** 7 – 11 June 2027 **Location:** London **Fees:** 6500 Euro

Overview

Computational Fluid Dynamics Fundamentals Training Course is a one-week foundation course for mechanical, process, aerospace, energy, and engineering professionals who leave with a CFD Analysis Assurance Workbook. Participants connect governing equations, domains, meshes, boundary conditions, physical models, solver settings, convergence, verification, validation, and result interpretation. The course turns software steps into a traceable engineering analysis. Agile Leaders Training Center presents computational fluid dynamics fundamentals through disciplined modelling and evidence review.

Who Should Attend

- Engineering functions beginning fluid-flow and heat-transfer simulation work
- Design teams responsible for evaluating flow, pressure, temperature, or forces
- Process teams responsible for investigating operating and geometry changes
- Research functions responsible for comparing calculations with experimental evidence
- Technical reviewers responsible for assessing CFD assumptions and results

The course assumes participants can use engineering units, algebra, and basic fluid mechanics, and it leaves out advanced solver development, specialized multiphase models, and software certification.

Departments and Industries

The course supports departments and industries that use simulated flow evidence for design, operation, or technical review.

- Mechanical, process, aerospace, energy, and thermal engineering
- Manufacturing, automotive, marine, and building services
- Chemicals, power generation, water, and environmental systems
- Product development, research laboratories, and technical assurance
- Operations improvement, projects, and engineering consulting

Learning Objectives

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

- Analyze a flow problem and define a computational domain
- Build and evaluate an appropriate numerical mesh
- Apply material models and boundary conditions
- Compare laminar, turbulent, steady, and transient assumptions
- Evaluate residuals, balances, convergence, and mesh sensitivity
- Use verification and validation evidence to interpret CFD results

Course Agenda

Day 1: Frame the CFD Problem

- Engineering Question and Quantity-of-Interest Worksheet
- Mass, Momentum, and Energy Conservation Map
- Computational Domain and Simplification Decision Sheet
- Steady, Transient, Incompressible, and Compressible Assumption Matrix
- CFD Analysis Assurance Workbook Structure

Day 2: Build and Assess the Mesh

- Geometry Cleanup and Fluid-Volume Extraction Checklist
- Structured, Unstructured, Hexahedral, and Tetrahedral Mesh Comparison
- Skewness, Orthogonality, Aspect Ratio, and Growth-Rate Quality Review
- Boundary-Layer Inflation and Near-Wall Resolution Plan
- Local Refinement and Mesh-Sensitivity Study Design

Day 3: Define Physics and Solver Setup

- Material Property and Reference Condition Register
- Inlet, Outlet, Wall, Symmetry, and Periodic Boundary Map
- Laminar, RANS, and Turbulence Model Selection Guide
- Pressure-Velocity Coupling and Spatial Discretization Matrix
- Initialization, Time Step, Relaxation, and Solver-Control Sheet

Day 4: Monitor, Verify, and Interpret

- Residual, Imbalance, and Quantity Monitor Dashboard
- Convergence, Stability, and Numerical Diffusion Diagnostic Map
- Grid Independence and Discretization Error Review
- Verification, Validation, and Experimental Comparison Matrix
- Contour, Vector, Streamline, and Integral Report Checklist

AGILE LEADERS
Training Center

Day 5: CFD Analysis Practice

- Exercise: Define a Domain and Boundary-Condition Set
- Exercise: Diagnose Mesh Quality and Refinement Needs
- Exercise: Select Physics and Solver Controls
- Exercise: Evaluate Convergence and Validation Evidence
- Capstone Exercise: Complete the CFD Analysis Assurance Workbook

Practical Exercises

The course uses suggested activities based on simplified flow cases, mesh reports, solver histories, and reference evidence.

- Suggested activity: convert an engineering question into domain, assumption, and output requirements
- Suggested activity: inspect mesh quality and design a refinement study
- Suggested activity: configure boundary conditions, models, and convergence monitors
- Suggested activity: review a CFD result for conservation, independence, validation, and reporting quality

FAQs

Who suits computational fluid dynamics fundamentals training, and what does it assume?

Computational fluid dynamics fundamentals training suits engineering, design, process, research, and technical-review functions. It assumes basic fluid mechanics, algebra, and engineering units.

How does computational fluid dynamics fundamentals training differ from advanced CFD training?

Computational fluid dynamics fundamentals training develops a complete introductory workflow and review discipline. Advanced CFD training addresses specialized physics, numerical methods, solver customization, high-performance computing, or research-level model development.

Why is mesh quality important in computational fluid dynamics?

Mesh quality affects how geometry, gradients, boundary layers, and conservation equations are represented. Distorted or insufficient cells can reduce accuracy, destabilize the solution, and hide important flow behavior.

How is convergence assessed in a CFD analysis?

Convergence is assessed with residual trends, stable quantities of interest, conservation balances, iteration or time-step behavior, and evidence that the selected stopping criteria match the engineering question.

AGILE LEADERS
Training Center

What is the difference between CFD verification and validation?

Verification checks whether equations and numerical procedures are solved with controlled error. Validation compares the model and its predictions with appropriate physical evidence for the intended use.

Conclusion

Participants take back a CFD Analysis Assurance Workbook linking the engineering question, assumptions, mesh, physics, solver settings, convergence, verification, validation, and reported results. The workbook makes modelling choices and limitations visible. It supports defensible simulation decisions while approved procedures, software documentation, and competent engineering review remain governing.