CONCEPT

Computational Fluid Dynamics (CFD): Meshing, Turbulence Modelling and Industrial Flow Simulation

2 – 6 August 2027
Paris - Right Bank business hotel

Computational Fluid Dynamics (CFD): Meshing, Turbulence Modelling and Industrial Flow Simulation

Ref: 476_14472 **Date:** 2 – 6 August 2027 **Location:** Paris - Right Bank business hotel **Fees:** 23500 SAR

Introduction:

Computational fluid dynamics CFD results now drive decisions on pipework, heat transfer equipment, ventilation layouts and gas release studies, yet many models reach design reviews with poorly shaped meshes, a turbulence model chosen by default, unchecked y+ values and residual plots taken as proof of accuracy. This Core Concept course trains mechanical, process and HVAC engineers to set up, run and challenge finite volume simulations in commercial and open-source solvers, from geometry clean-up and meshing through verification and validation to reporting. Participants produce a CFD Simulation Report and Mesh Independence Dossier for an industrial flow case.

Course Objectives:

- Translate an engineering question into a CFD model scope with a defined domain, flow physics, outputs and acceptance criteria
- Prepare CAD geometry and build hexahedral, tetrahedral or polyhedral meshes with inflation layers that meet skewness, orthogonality and aspect ratio targets
- Select k-epsilon, k-omega SST or LES approaches and a matching near-wall treatment based on the y+ achieved on the mesh
- Specify boundary conditions, pressure-velocity coupling, discretisation order and under-relaxation, and judge convergence from residuals, imbalances and monitors
- Demonstrate solution credibility through mesh independence studies and validation against measurements or published correlations
- Post-process velocity, pressure, temperature and concentration fields into a traceable simulation report that reviewers can audit

Target Audience:

- Mechanical engineering functions that assess pressure loss, flow distribution and thermal loads on piping, vessels and rotating equipment
- Process engineering functions that study mixing, separation, heat transfer and flow maldistribution in plant equipment
- HVAC and building services design functions that model air distribution, thermal comfort and contaminant removal in occupied spaces
- Safety and loss prevention engineering functions that model flammable or toxic gas dispersion in congested areas
- Engineering analysis and simulation functions that review CFD studies delivered by consultants and vendors

Course Outline:

Day 1: Governing Equations, Finite Volume Method and the CFD Workflow

- Conservation of Mass, Momentum and Energy Through the Navier-Stokes Equations in Plain Terms
- Finite Volume Discretisation: Control Volumes, Face Fluxes and Interpolation Schemes
- Pre-Processing, Solving and Post-Processing Stages in ANSYS Fluent and OpenFOAM-Type Codes
- Flow Regime Screening with Reynolds, Mach and Prandtl Numbers
- Simulation Scoping Sheet: Decision Question, Domain Extent and Acceptance Criteria

Day 2: Geometry Preparation, Meshing Strategy and Mesh Quality

- CAD Clean-Up: Defeaturing, Sliver Faces, Gaps and Fluid Volume Extraction
- Structured Hexahedral Versus Unstructured Tetrahedral and Polyhedral Cell Types
- Prism Inflation Layers and Local Refinement Zones Around Walls, Jets and Shear Layers
- Mesh Quality Metrics: Skewness, Orthogonal Quality, Aspect Ratio and Cell Growth Rate
- Named Selections and Zone Definitions for Boundary Assignment

Day 3: Turbulence Modelling, Wall Treatment, Boundary Conditions and Solver Settings

- RANS Closures: Standard and Realizable k-Epsilon Versus k-Omega SST
- Near-Wall Treatment: y+ Targets, Viscous Sublayer Resolution and Wall Functions
- Inlet, Outlet, Wall and Symmetry Conditions with Turbulence Intensity and Length Scale Inputs
- Pressure-Velocity Coupling with SIMPLE and Coupled Solvers, Under-Relaxation and Discretisation Order
- Convergence Judgement: Scaled Residuals, Mass and Energy Imbalance and Point Monitors

Day 4: Verification, Validation, Heat Transfer and Multiphase Flow

- Mesh Independence Study with Three Systematically Refined Grids and Richardson Extrapolation
- Validation Against Experimental Data, Pressure Drop Correlations and Plant Measurements
- Conjugate Heat Transfer, Buoyancy and Radiation Model Selection
- Multiphase Approaches at Overview: Volume of Fluid, Eulerian and Discrete Phase Models
- Large Eddy Simulation at Overview and Transient Time-Step Choice by Courant Number

Day 5: Industrial Simulation Labs and CFD Report Build

- Lab: Turbulent Pipe and Elbow Flow with Pressure Loss and Separation Check
- Lab: Shell-Side Heat Exchanger Flow Distribution and Temperature Field
- Lab: Room Ventilation Air Distribution, Age of Air and Comfort Contours
- Lab: Gas Release Dispersion with Species Transport and Concentration Envelopes
- CFD Simulation Report and Mesh Independence Dossier Presentation and Peer Review

Skills You Will Gain:

- CFD Model Scoping
- Geometry Defeaturing
- Mesh Quality Control
- Turbulence Model Selection
- Near-Wall Resolution
- Solver Convergence Diagnosis
- Simulation Verification and Validation
- Flow Field Post-Processing

Why Attend This Course:

- Return with a CFD Simulation Report and Mesh Independence Dossier built on an industrial flow case during the labs
- Stop accepting converged residuals as proof of a correct answer and ask for the evidence that shows accuracy
- Brief consultants and vendors precisely on mesh, turbulence and boundary requirements before a study starts
- Compare simulation practice with mechanical, process, HVAC and safety engineers from oil and gas, utilities, manufacturing and building projects

Conclusion:

A CFD model is only as trustworthy as its mesh, its physics choices and the evidence that its answer does not change with further refinement. The week moves from governing equations and the finite volume method, through geometry preparation and meshing, to turbulence closures, wall treatment, boundary conditions and solver settings, then to verification, validation, heat transfer and multiphase flow. The final day applies the workflow in four industrial labs and assembles a CFD Simulation Report and Mesh Independence Dossier.