KOENIG
step forward

AUTODESK
Authorized Training Center

Autodesk CFD Foundations: Design-Oriented Computational Fluid Dynamics

Course Objectives

The objective of this course is to introduce the fundamental concepts of Computational Fluid Dynamics and explain their application within a design-centric CFD workflow. The course aims to develop conceptual clarity in fluid flow and heat transfer phenomena, familiarize learners with the Autodesk CFD environment, and enable correct setup, execution, and interpretation of CFD simulations used in engineering design decision-making.

Target Audience

This program is designed to provide learners with a structured, hands-on introduction to Computational Fluid Dynamics (CFD) using **Autodesk CFD**. This course is intended for mechanical and manufacturing engineers, product design engineers, CAD/CAM professionals, and engineering graduates who wish to build a strong foundational understanding of Computational Fluid Dynamics using Autodesk CFD. It is suitable for learners seeking design-level CFD knowledge without deep mathematical formulation.

Course Outcomes

After completing this **40-hour** course, learners will be able to understand the role, scope, and limitations of CFD in engineering applications, distinguish between different flow and thermal simulation types, prepare CFD-ready geometry, define flow volumes and boundary conditions, configure materials and physics models, execute steady-state simulations, assess convergence behaviour, and interpret CFD results for comparative and trend-based engineering decisions.

Course Outline

Module 1: Fundamentals of Computational Fluid Dynamics

- Definition and scope of Computational Fluid Dynamics
- Role of CFD in modern engineering and product development
- Types of engineering problems solved using CFD
- Internal flow vs external flow applications
- Thermal vs flow-driven simulations
- CFD vs experimental testing (conceptual comparison)
- CFD vs Finite Element Analysis (FEA)
- Strengths and limitations of CFD simulations

- Typical CFD workflow used in industry
- Overview of design-level vs high-fidelity CFD

Module 2: Autodesk CFD Platform Overview and Capabilities

- Introduction to the Autodesk CFD software environment
- Positioning of Autodesk CFD within Autodesk simulation tools
- Types of analyses supported in Autodesk CFD
- Design-centric CFD philosophy
- Typical use cases: airflow, cooling, thermal management
- Solver approach used in Autodesk CFD
- Data flow: geometry → mesh → solver → results
- Understanding Autodesk CFD limitations and intended usage

Module 3: User Interface, Navigation, and Project Management

- User interface layout and workspaces
- Navigation tools: pan, zoom, rotate, view control
- Understanding the project tree and model hierarchy
- Creating and managing CFD projects
- Setting and managing unit systems
- Coordinate systems and orientation
- File handling, saving, and version control practices
- Managing multiple simulation scenarios

Module 4: Applied Fluid Mechanics for CFD Applications

- Physical meaning of velocity, pressure, and flow direction
- Flow continuity concept (qualitative understanding)
- Laminar and turbulent flow behavior (engineering perspective)
- Reynolds number – practical interpretation (non-mathematical)
- Internal flow characteristics (ducts, pipes, enclosures)
- External flow characteristics (basic overview)
- Incompressible flow assumption and its implications

- Common fluid flow assumptions in Autodesk CFD

Module 5: Applied Heat Transfer Concepts for CFD

- Difference between temperature and heat
- Thermal energy transfer mechanisms
- Conduction in solids
- Convection in fluids
- Natural convection vs forced convection
- Heat sources, heat sinks, and ambient conditions
- Thermal resistance concept (qualitative)
- Introduction to conjugate heat transfer
- Typical thermal problems solved using CFD

Module 6: Geometry Preparation and CFD Modeling Strategy

- How CFD interprets CAD geometry
- Solid regions vs fluid regions
- Importance of geometry simplification for CFD
- Identifying non-CFD-friendly features
- Handling small fillets, holes, and sharp edges
- Importing CAD geometry into Autodesk CFD
- Geometry diagnostics and error identification
- Best practices for CFD-ready geometry

Module 7: Flow Volume Creation and Model Validation

- Concept of computational domain
- Importance of defining correct flow volumes
- Internal flow volume creation methodology
- External flow domain concept (basic)
- Identifying and resolving leaks and gaps
- Openings, vents, and flow boundaries
- Model validation before meshing
- Ensuring simulation-ready geometry

Module 8: Mesh Generation and Control

- Purpose of discretization in CFD
- Role of mesh in solution accuracy
- Automatic meshing approach in Autodesk CFD
- Global mesh size control
- Local mesh refinement strategies
- Coarse vs fine mesh impact on results
- Mesh density vs computational cost
- Practical meshing best practices for beginners

Module 9: Boundary Condition Definition – Flow and Thermal

- Purpose of boundary conditions in CFD
- Velocity inlet definition and applications
- Pressure inlet and pressure outlet conditions
- Wall boundary conditions and no-slip concept
- Ambient and environmental conditions
- Temperature boundary conditions
- Heat flux and heat generation definition
- Gravity and buoyancy effects (basic understanding)
- Common boundary condition mistakes

Module 10: Material Assignment and Physics Setup

- Importance of material properties in CFD
- Fluid material definition (air properties)
- Solid material thermal properties
- Assigning materials to regions
- Coupling between fluid and solid domains
- Heat transfer between solid and fluid regions
- Activating gravity and buoyancy effects
- Verifying physics setup before solving

Module 11: Solver Execution, Convergence, and Troubleshooting

- Understanding the CFD solver process
- Steady-state simulation concept
- Starting and monitoring a simulation
- Understanding convergence behavior
- Residuals and solution stability (conceptual)
- Common reasons for non-convergence
- Basic troubleshooting strategies
- When to stop or restart a simulation

Module 12: Post-Processing, Interpretation, and Capstone Application

- Velocity contour visualization
- Pressure distribution analysis
- Temperature contour interpretation
- Streamlines and flow path visualization
- Understanding color scales and legends
- Identifying realistic vs misleading results
- Trend-based engineering decision making
- Design comparison using CFD