

SOLIDWORKS FLOW SIMULATION

Target Audience:

This course is designed for mechanical engineers, product designers, thermal analysts, and simulation professionals who want to integrate CFD (Computational Fluid Dynamics) analysis into their design workflow. It is ideal for those involved in fluid flow, heat transfer, and performance optimization of products.

Course Objective:

The objective of this course is to introduce participants to the capabilities of SolidWorks Flow Simulation, enabling them to set up, run, and interpret results for various fluid dynamics and thermal analyses. The course aims to help users make better design decisions by leveraging simulation-driven design.

Course Outcome:

By the end of the course, participants will be able to create and analyze fluid flow and heat transfer problems using SolidWorks Flow Simulation. They will gain hands-on experience in setting boundary conditions, running simulations, and evaluating results to improve product performance.

Prerequisites

Participants should have a working knowledge of SolidWorks CAD, especially in part and assembly modeling. A basic understanding of fluid mechanics and thermodynamics is recommended to fully grasp simulation concepts. Completion of the SolidWorks Essentials course is highly advised.

Course Outline:

The course comprises **40-hours** of theory and labs and is divided into **19** different chapters. Each chapter will be followed by **hands-on lab exercises** to reinforce learning and gauge understanding of the topics covered.

Chapter 1: Introduction

- Overview of Flow Simulation
- Project Setup, Meshing, Boundary Conditions, Goals, and Results

Chapter 2: Flat Plate Boundary Layer

- Modeling, Boundary Setup, Cut & XY Plots
- Comparison with Theory, Cloning, and Exercises

Chapter 3: Flow Past a Sphere and Cylinder

- Time-Dependent Analysis, XY Plots, Frequency Analysis
- Strouhal Number, Theory, and Exercises

Chapter 4: Flow Past an Airfoil

- Equation Goals, Visualization, Batch Runs
- Theory and Practical Exercises

Chapter 5: Rayleigh-Bénard Convection & Taylor-Couette Flow

- 2D Flow, Neutral Stability Theory
- Surface Plots and Exercises

Chapter 6: Pipe Flow

- Laminar & Turbulent Flow Analysis
- Pressure Drop, Velocity Profiles, and Nusselt Number
- Exercise

Chapter 7: Flow Across a Tube Bank

- Setup, Cut Plots, Empirical Comparison
- Exercises and Theory

Chapter 8: Heat Exchanger

- Effectiveness-NTU Method
- Lid Creation, Goals, and Surface Parameters
- Exercise

Chapter 9: Ball Valve

- Valve and Pipe Assembly Modeling
- Simulation, Hydraulic Resistance
- Exercise

Chapter 10: Orifice Plate and Flow Nozzle

- Discharge Coefficient, XY Plots, Flow Trajectories
- Flow Nozzle Theory
- Exercise

Chapter 11: Thermal Boundary Layer

- Low & High Reynolds Number Simulation
- Temperature Profiles and Nusselt Number
- Exercise

Chapter 12: Free Convection on Vertical Plate and Horizontal Cylinder

- Non-Dimensional Profiles
- Theory and Exercises

Chapter 13: Swirling Flow in a Cylindrical Container

- Flow Trajectories and Rotational Flow Analysis
- Exercise

Chapter 14: Flow Past a Model Rocket

- Simulation Setup and Results
- Cut Plots and Theoretical Comparison
- Exercise

Chapter 15: Draining of a Cylindrical Tank

- Initial Conditions, Goal Plots, and Transient Explorer
- Exercise

Chapter 16: Ahmed Body

- External Aerodynamics Simulation
- Batch Runs and Drag Analysis
- Exercise

Chapter 17: Savonius Wind Turbine

- Rotating Region Setup, Mesh Control
- Batch Runs and Theoretical Comparison
- Exercise

Chapter 18: Spinning Propeller

- Rotating Domains, Batch Runs
- Aerodynamic Performance Study

- Exercise

Chapter 19: Supersonic Flow Over a Cone

- Shock Wave Modeling, Density Plots
- Supersonic Flow Theory
- Exercise