General Training Schedule

Tuesday

8:00	-	8:15	Registration
8:15	-	8:30	Introduction to Computational Fluid Dynamics (CFD)
8:30	-	9:15	Theory
9:15	-	9:45	The ***FLOW-3D*** user interface
9:45	-	10:00	General settings
10:00	-	10:15	Break
10:15	-	11:00	Creating and viewing geometry
11:00	-	11:30	*Exercise 1: Creating and viewing geometry*
11:30	-	12:15	Lunch (provided by Flow Science)
12:15	-	1:45	Meshing and boundary conditions
1:45	-	2:00	Break
2:00	-	2:45	*Exercise 2: Mesh creation and specification of boundary conditions*
2:45	-	3:45	Initial conditions, material properties, and component properties
3:45	-	4:30	*Exercise 3: Initial conditions, material properties, and component properties*
4:30	-	5:15	Review of Day 1 topics

Wednesday

8:00	-	8:15	Setup and questions
8:15	-	9:15	Viscosity, turbulence, and elasto-visco-plasticity
9:15	-	10:45	Multi-phase flows, air entrainment, solute modeling, and discrete particles
10:45	-	11:00	Break
11:00	-	12:00	Heat transfer
12:00	-	12:45	Lunch (Provided by Flow Science)
12:45	-	1:00	Surface tension
1:00	-	2:30	Bubbles, cavitation, and phase change
2:30	-	2:45	Break
2:45	-	3:30	Porous media
3:30	-	4:15	Non-inertial reference frames and general moving objects
4:15	-	5:15	Review of Day 2 topics

Thursday

8:00	-	8:15	Setup and questions
8:15	-	8:45	Output options
8:45	-	9:45	Numerical options
9:45	-	10:30	Simulation diagnostics and performance issues
10:30	-	10:45	Break
10:45	-	12:00	Postprocessing
12:00	-	1:00	Lunch (on your own)
1:00	-	5:00	Consultation

683 Harkle Road, Suite A Santa Fe, New Mexico 87505
505.982.0088 www.flow3d.com

Course Details

The General Training course consists of 20 hours of lectures and hands-on exercises intended to develop an applied knowledge of computational fluid dynamics, in particular the details of how to use ***FLOW-3D*** to solve a wide variety of real-world problems. There is also an option to spend an additional four hours working one-on-one with the Flow Science support staff during the Consultation session.

Lecture Content:

Introduction to Computational Fluid Dynamics (CFD):

This lecture is a general overview of CFD, discussing the concepts, terminology, and best practices in the field.

Theory:

This lecture covers the technical foundations of ***FLOW-3D***. Users will gain an understanding of the governing equations, the structured gridding methodology, the TruVOF technique, and the FAVOR method used for embedding complex geometries in a structured grid.

The ***FLOW-3D*** user interface:

An interactive tour of the ***FLOW-3D*** user interface, demonstrating everything a user needs to know about navigating through ***FLOW-3D.*** Topics include how to manage simulations using the Navigator; creating, copying, or restarting simulations; manipulating geometry; and running and terminating simulations.

General Settings:

This is a brief lecture that covers general topics like finish conditions, flow modes, acceptable units systems, and the allocation of computational resources.

Creating and viewing geometry:

The concepts of geometry building in ***FLOW-3D*** are covered in this lecture. Major topics include the creation of components and subcomponents using ***FLOW-3D*** primitives, CAD files, and topographic files. Techniques for checking and fixing stereolithography (STL) CAD files will be discussed. Users will learn how to judge adequate levels of mesh resolution so that the geometry is represented accurately without the excessive use of computational resources.

Meshing and boundary conditions:

Topics relating to the effective, efficient design of a computational domain are the primary focus of this lecture. The major components are mesh creation, the specification of boundary conditions, and best practices. The creation of mass sources and mass/momentum sources are also included.

Initial conditions, material properties, and component properties:

Methods for representing the initial state of the simulation are discussed with a focus on physical realism and the efficient use of computational resources. Users will also learn how to use the ***FLOW-3D*** material database to assign fluid and solid properties, edit materials, and merge materials from previous databases.

Physical models (Day 2 lectures):

The second day is devoted to the use of the many physical models available in ***FLOW-3D***. The lectures will cover how to determine what model to use, the appropriate input parameters, and other relevant topics.

Output options:

This covers the many options for outputting data from ***FLOW-3D***. The main focus is the efficient use of computational resources.

Numerical Options:

While the default numerical options in ***FLOW-3D*** will typically produce an accurate and efficient simulation, there are situations where other numerical methods are warranted. The use of these different numerical methods is outlined in this lecture.

Simulation diagnostics and performance issues:

This period is discussion of the available diagnostic tools in ***FLOW-3D*** and how to use them to create an efficient, accurate simulation.

Post-Processing:

This lecture provides post-processing basics such as how to open results files, how to reload results files, and how to generate various graphics including point probes, 1D, 2D, and 3D plots, text output, and neutral file output. Specification of moving and stationary history points will be covered along with streamline and path line generation, vorticity plots, and fluid depth plots. Also, Simulation and Plotting units will be discussed.

Consultation:

The consultation session is designed to allow users to work on their problems with the help of a Flow Science support engineer. It is recommended that users clearly describe the problem they intend to work on by filling out the registration form as completely as possible.

FLOW-3D, TruVOF, and FAVOR are registered trademarks of Flow Science, Inc. in the USA and other countries.

683 Harkle Road, Suite A   Santa Fe, New Mexico 87505
505.982.0088   www.flow3d.com