

Day 1

8:15 – 8:30	Registration
8:30 – 9:15	Introduction & Technical Foundations of FLOW-3D
9:15 – 9:30	Files, Units, and the User Interface
9:30 – 10:15	Results Analysis
10:15 – 10:30	Break
10:30 – 11:00	Exercise 1 – Running and Post-Processing an Example Simulation
11:00 – 11:30	Geometry
11:30 – 12:15	Exercise 2 – Building Geometry
12:15 – 1:00	Lunch (provided by Flow Science)
1:00 – 1:30	Meshing
1:30 – 2:30	Exercise 3 – Building a Mesh
2:30 – 2:45	Break
2:45 – 3:30	Boundary Conditions
3:30 – 4:15	Model Initialization
4:15 – 5:15	Exercise 4 – Setting Boundaries, Initial Conditions, and Physics

Day 2

8:30 – 9:15	Modeling Turbulence
9:15 – 10:00	Exercise 5 – Results Analysis
10:00 – 10:15	Break
10:15 – 11:15	Modeling Variable Density and Multiphase Flow
11:15 – 12:15	Modeling Sediment Scour
12:15 – 1:00	Lunch (provided by Flow Science)
1:00 – 2:00	Exercise 6 – Air Entrainment
1:00 – 1:45	Modeling Cavitation
1:45 – 2:15	Depth-Averaged Flow (Shallow Water), Wind, and Coriolis Effects
2:15 – 2:30	Break
2:30 – 3:15	Modeling Moving Components
3:15 – 4:00	Exercise 7 – Setting up a GMO Mixing Tank (Instructor Led)
4:00 – 4:45	Numerical Options
4:45 – 5:15	Simulation Diagnostics & Troubleshooting

Day 3

8:30 – 9:15	Modeling Porous Media
9:15 – 10:15	Exercise 8 – Saturated Porous Media
10:15 – 10:30	Break
10:30 – 11:45	Minimizing Errors
11:45 – 12:00	Questions & Graduation, End of Class
12:00 – 1:00	Lunch (on your own)
1:00 – 5:00	Consultation

Course Details

The Hydraulics Training course consists of two and a half days of application specific lectures and hands-on work. The intent of this format is to reinforce the lectures with exercises that demonstrate the application of **FLOW-3D** to real-world simulations or validations. The last half of the third day is set aside for consultation on customer problems.

The content of each lecture is given below:

Technical Foundations of **FLOW-3D**:

The technical foundations of **FLOW-3D** are presented. Users will gain an understanding of the governing equations, the structured gridding methodology, TruVOF free surface tracking, and the FAVOR™ method for embedding complex geometries in structured grids.

Files, Units, and the User Interface:

A guided tour of the **FLOW-3D** interface will be provided to help users understand how to manage simulations using the Navigator, recognize the input and output files, create/copy/restart simulations, and run and terminate simulations. A general overview of the interface layout is provided, along with guidance on selecting and applying unit systems.

Results Analysis:

This lecture provides post-processing basics: opening and reloading results files, generating 1-D, 2-D, and 3-D graphical and text output, and visualizing streamlines and path lines. More possible output options will be mentioned that will be covered in detail in later lectures.

Geometry Building:

The concepts of geometry building are covered including the fundamentals of component and subcomponent creation. Various methods of creating geometry will be covered including **FLOW-3D** primitives, CAD files, and topographic data. Techniques for checking and fixing stereolithography CAD files will be discussed. Geometry used to measure flow will also be covered: baffles, sampling volumes, and history probes.

Meshing:

The basics of creating computational meshes will be covered along with best practices for meshing and multi-block meshing. The relationship between mesh and geometry will be discussed, along with approaches to minimize mesh-related error.

Boundary Conditions:

Options for boundary conditions and adding/subtracting fluid within domains will be discussed, including symmetric boundaries, wall boundaries, continuative and outflow boundaries, velocity and volumetric flow rate boundaries, pressure boundaries, wave boundaries, grid overlay boundaries, and mass/mass-momentum sources. Boundary conditions common in hydraulic simulations will be presented for modeling reservoirs, inflow hydrographs, tailwater effects, and situations where upstream depth is unknown. Avoiding boundary over-specification will also be discussed.

Initialization:

Methods of initializing fluid in the simulation will be discussed with the goal of increasing stability and accelerating the simulation to a steady-state. Techniques for initializing pressure, temperature, density, and velocity distributions will be discussed. Defining fluid regions will be illustrated for both simple shapes and .stl geometry.

Physical Models:

The following physical models will be discussed:

Turbulence	Cavitation
Variable density flows	Sediment scour and transport
Buoyant flows	Porous media
Drift-flux (two-phase continuous/discrete flow)	Depth-averaged flow
Particles (with and without drag)	Forces on structures
Air entrainment	Chemistry
Bubble models	Solid motion (gates, valves, machinery, debris)

Numerical Options:

Mathematical options for specifying time steps, pressure solutions, geometry embedding, and physics model solutions will be presented. The focus will be on identifying situations where higher order numerical methods are warranted, increasing simulation speed, and maintaining stability.

Simulation Diagnostics and Troubleshooting:

This lecture describes how to interpret and respond to solver output messages and warnings: when to ignore messages, and when to change numerical options.

Consultation

Attendees may optionally purchase an additional half-day to work on their simulations hands-on with Flow Science staff. This option is recommended for users who are already developing complex simulations, or who wish for the additional hands-on training specific to their application.

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