Using This Manual

What's In This Manual

The FLUENT User's Guide tells you what you need to know to use FLUENT. It is divided into five volumes:

- Volume 1 contains introductory information, as well as information about the user interface, file import and export, unit systems, grids, boundary conditions, and physical properties.
- Volume 2 contains information about modeling fluid flow and heat transfer.
- Volume 3 contains information about modeling species transport and reacting flows, including pollutant formation.
- Volume 4 contains information about modeling flows that involve multiple phases, including solidification and melting.
- Volume 5 contains information about calculating a solution (including information about parallel processing) and postprocessing the results.

A bibliography and an index for *all five volumes* of the User's Guide are also provided, along with a nomenclature list.

! Under U.S. and international copyright law, Fluent is unable to distribute copies of the papers listed in the bibliography, other than those published internally by Fluent. Please use your library or a document delivery service to obtain copies of copyrighted papers.

A brief description of what's in each chapter follows:

Volume 1

- Chapter 1, Getting Started, describes the capabilities of FLUENT and the way in which it interacts with other Fluent Inc. and third-party programs. It also advises you on how to choose the appropriate solver formulation for your application, gives an overview of the problem setup steps, and presents a sample session that you can work through at your own pace. Finally, this chapter provides information about accessing the FLUENT manuals on CD-ROM or in the installation area.

- Chapter 2, User Interface, describes the mechanics of using the graphical user interface, the text interface, and the on-line help. It also provides instructions for remote and batch execution. (See the separate Text Command List for information about specific text interface commands.)

- Chapter 3, Reading and Writing Files, contains information about the files that FLUENT can read and write, including hardcopy files.

- Chapter 4, Unit Systems, describes how to use the standard and custom unit systems available in FLUENT.

- Chapter 5, Reading and Manipulating Grids, describes the various sources of computational grids and explains how to obtain diagnostic information about the grid and how to modify it by scaling, translating, and other methods. This chapter also contains information about the use of non-conformal grids.

- Chapter 6, Boundary Conditions, explains the different types of boundary conditions available in FLUENT, when to use them, how to define them, and how to define boundary profiles and volumetric sources and fix the value of a variable in a particular region. It also contains information about porous media and lumped parameter models.

- Chapter 7, Physical Properties, explains how to define the physical properties of materials and the equations that FLUENT uses to compute the properties from the information that you input.

Volume 2

- Chapter 8, Modeling Basic Fluid Flow, describes the governing equations and physical models used by FLUENT to compute fluid flow (including periodic flow, swirling and rotating flows, compressible flows, and inviscid flows), as well as the inputs you need to provide to use these models.
- Chapter 9, Modeling Flows in Moving Zones, describes the use of single rotating reference frames, multiple moving reference frames, mixing planes, and sliding meshes in FLUENT.
- Chapter 10, Modeling Turbulence, describes FLUENT's models for turbulent flow and when and how to use them.
- Chapter 11, Modeling Heat Transfer, describes the physical models used by FLUENT to compute heat transfer (including convective and conductive heat transfer, natural convection, radiative heat transfer, and periodic heat transfer), as well as the inputs you need to provide to use these models.

Volume 3

- Chapter 12, Introduction to Modeling Species Transport and Reacting Flows, provides an overview of the models available in FLUENT for species transport and reactions, as well as guidelines for selecting an appropriate model for your application.
- Chapter 13, Modeling Species Transport and Finite-Rate Chemistry, describes the finite-rate chemistry models in FLUENT and how to use them. This chapter also provides information about modeling species transport in non-reacting flows.
- Chapter 14, Modeling Non-Premixed Combustion, describes the non-premixed combustion model and how to use it. This chapter includes details about using prePDF.
- Chapter 15, Modeling Premixed Combustion, describes the premixed combustion model and how to use it.

- Chapter 16, Modeling Partially Premixed Combustion, describes the partially premixed combustion model and how to use it.
- Chapter 17, Modeling Pollutant Formation, describes the models for the formation of NO_x and soot and how to use them.

Volume 4

- Chapter 18, Introduction to Modeling Multiphase Flows, provides an overview of the models for multiphase flow (including the discrete phase, VOF, mixture, and Eulerian models), as well as guidelines for selecting an appropriate model for your application.
- Chapter 19, Discrete Phase Models, describes the discrete phase models available in FLUENT and how to use them.
- Chapter 20, General Multiphase Models, describes the general multiphase models available in FLUENT (VOF, mixture, and Eulerian) and how to use them.
- Chapter 21, Modeling Solidification and Melting, describes FLUENT's model for solidification and melting and how to use it.

Volume 5

- Chapter 22, Using the Solver, describes the FLUENT solvers and how to use them.
- Chapter 23, Grid Adaption, explains the solution-adaptive mesh refinement feature in FLUENT and how to use it.
- Chapter 24, Creating Surfaces for Displaying and Reporting Data, explains how to create surfaces in the domain on which you can examine FLUENT solution data.
- Chapter 25, Graphics and Visualization, describes the graphics tools that you can use to examine your FLUENT solution.

- Chapter 26, Alphanumeric Reporting, describes how to obtain reports of fluxes, forces, surface integrals, and other solution data.
- Chapter 27, Field Function Definitions, defines the flow variables that appear in the variable selection drop-down lists in FLUENT panels, and tells you how to create your own custom field functions.
- Chapter 28, Parallel Processing, explains the parallel processing features in FLUENT and how to use them. This chapter also provides information about partitioning your grid for parallel processing.

What's in the Other Manuals

In addition to this User's Guide, there are several other manuals available to help you use FLUENT and its associated programs:

- The Tutorial Guide contains a number of example problems with complete detailed instructions, commentary, and postprocessing of results.
- The UDF Manual contains information about writing and using user-defined functions (UDFs).
- The Text Command List provides a brief description of each of the commands in FLUENT's text interface.
- The GAMBIT manuals teach you how to use the GAMBIT preprocessor for geometry creation and mesh generation.

Typographical Conventions

Several typographical conventions are used in this manual's text to facilitate your learning process.

- An exclamation point (!) in the margin marks an important note or warning.

- Different type styles are used to indicate graphical user interface menu items and text interface menu items (e.g., Iso-Surface panel, `surface/iso-surface` command).

- The text interface type style is also used when illustrating exactly what appears on the screen or exactly what you need to type into a field in a panel. The information displayed on the screen is enclosed in a large box to distinguish it from the narrative text, and user inputs are often enclosed in smaller boxes.

- A mini flow chart is used to indicate the menu selections that lead you to a specific command or panel. For example,

  [Define] ⟶Boundary Conditions...

  indicates that the Boundary Conditions... menu item can be selected from the Define pull-down menu, and

  [`display`] ⟶`grid`

  indicates that the `grid` command is available in the `display` text menu.

  The words before the arrows invoke menus (or submenus) and the arrows point from a specific menu toward the item you should select from that menu. In this manual, mini flow charts usually precede a description of a panel or command, or a screen illustration showing how to use the panel or command. They allow you to look up information about a command or panel and quickly determine how to access it without having to search the preceding material.

- The menu selections that will lead you to a particular panel are also indicated (usually within a paragraph) using a "/". For example, Define/Materials... tells you to choose the Materials... menu item from the Define pull-down menu.

Mathematical Conventions

- Where possible, vector quantities are displayed with a raised arrow (e.g., $\vec{a}$, $\vec{A}$). Boldfaced characters are reserved for vectors and matrices as they apply to linear algebra (e.g., the identity matrix, $\mathbf{I}$).

- The operator ∇, referred to as grad, nabla, or del, represents the partial derivative of a quantity with respect to all directions in the chosen coordinate system. In Cartesian coordinates, ∇ is defined to be

$$\frac{\partial}{\partial x}\vec{\imath}+\frac{\partial}{\partial y}\vec{\jmath}+\frac{\partial}{\partial z}\vec{k}$$

  ∇ appears in several ways:

  - The gradient of a scalar quantity is the vector whose components are the partial derivatives; for example,

$$\nabla p=\frac{\partial p}{\partial x}\vec{\imath}+\frac{\partial p}{\partial y}\vec{\jmath}+\frac{\partial p}{\partial z}\vec{k}$$

  - The gradient of a vector quantity is a second-order tensor; for example, in Cartesian coordinates,

$$\nabla(\vec{v})=\left(\frac{\partial}{\partial x}\vec{\imath}+\frac{\partial}{\partial y}\vec{\jmath}+\frac{\partial}{\partial z}\vec{k}\right)\left(v_x\vec{\imath}+v_y\vec{\jmath}+v_z\vec{k}\right)$$

    This tensor is usually written as

$$\begin{pmatrix} \frac{\partial v_x}{\partial x} & \frac{\partial v_x}{\partial y} & \frac{\partial v_x}{\partial z} \\ \frac{\partial v_y}{\partial x} & \frac{\partial v_y}{\partial y} & \frac{\partial v_y}{\partial z} \\ \frac{\partial v_z}{\partial x} & \frac{\partial v_z}{\partial y} & \frac{\partial v_z}{\partial z} \end{pmatrix}$$

  - The divergence of a vector quantity, which is the inner product between ∇ and a vector; for example,

$$\nabla\cdot\vec{v}=\frac{\partial v_x}{\partial x}+\frac{\partial v_y}{\partial y}+\frac{\partial v_z}{\partial z}$$

  - The operator $\nabla\cdot\nabla$, which is usually written as ∇^2 and is known as the Laplacian; for example,

$$\nabla^2 T = \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}$$

$\nabla^2 T$ is different from the expression $(\nabla T)^2$, which is defined as

$$(\nabla T)^2 = \left(\frac{\partial T}{\partial x}\right)^2 + \left(\frac{\partial T}{\partial y}\right)^2 + \left(\frac{\partial T}{\partial z}\right)^2$$

- An exception to the use of ∇ is found in the discussion of Reynolds stresses in Chapter 10, where convention dictates the use of Cartesian tensor notation. In this chapter, you will also find that some velocity vector components are written as u, v, and w instead of the conventional v with directional subscripts.

When To Call Your Support Engineer

The Fluent support engineers can help you to plan your CFD modeling projects and to overcome any difficulties you encounter while using FLUENT. If you encounter difficulties we invite you to call your support engineer for assistance. However, there are a few things that we encourage you to do before calling:

- Read the section(s) of the manual containing information on the commands you are trying to use or the type of problem you are trying to solve.
- Recall the exact steps you were following that led up to and caused the problem.
- Write down the exact error message that appeared, if any.
- For particularly difficult problems, save a journal or transcript file of the FLUENT session in which the problem occurred. This is the best source that we can use to reproduce the problem and thereby help to identify the cause.