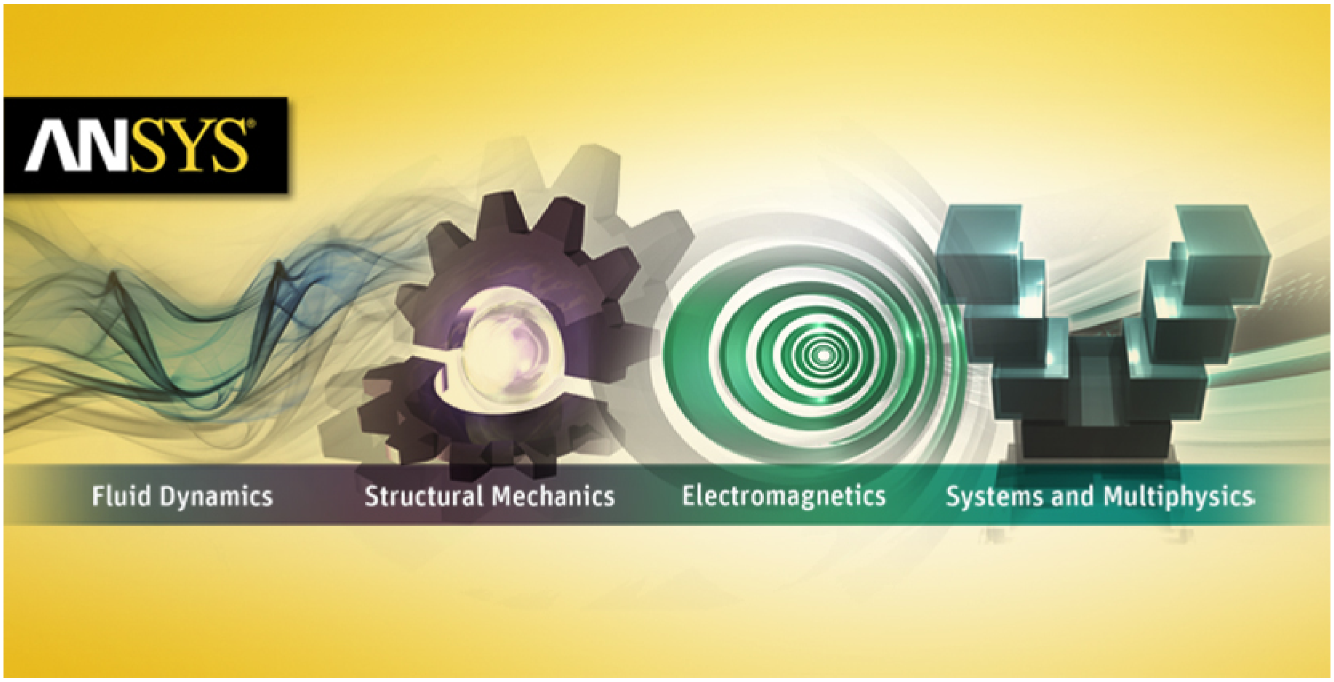


ANSYS FLUENT Getting Started Guide

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 14.0
November 2011

ANSYS, Inc. is certified to ISO 9001:2008.

Copyright and Trademark Information

Disclaimer Notice

U.S. Government Rights

Third-Party Software

Published in the U.S.A.

Table of Contents

Preface

This preface is divided into the following sections:

1. The Contents of This Manual

The ANSYS FLUENT Getting Started Guide highlights some of the features in ANSYS FLUENT and how to get started using the software.

2. The Contents of the FLUENT Manuals

The manuals listed below form the FLUENT product documentation set. They include descriptions of the procedures, commands, and theoretical details needed to use FLUENT products.

FLUENT Getting Started Guide contains general information about getting started with using FLUENT.
FLUENT User's Guide contains detailed information about using FLUENT, including information about the user interface, reading and writing files, defining boundary conditions, setting up physical models, calculating a solution, and analyzing your results.
FLUENT in Workbench User's Guide contains information about getting started with and using FLUENT within the Workbench environment.
FLUENT Theory Guide contains reference information for how the physical models are implemented in FLUENT.
FLUENT UDF Manual contains information about writing and using user-defined functions (UDFs).
FLUENT Tutorial Guide contains a number of example problems with detailed instructions, commentary, and postprocessing of results.
FLUENT Text Command List contains a brief description of each of the commands in FLUENT's text interface.
FLUENT Adjoint Solver Module Manual contains information about the background and usage of FLUENT's Adjoint Solver Module that allows you to obtain detailed sensitivity data for the performance of a fluid system.
FLUENT Battery Module Manual contains information about the background and usage of FLUENT's Battery Module that allows you to analyze the behavior of electric batteries.
FLUENT Continuous Fiber Module Manual contains information about the background and usage of FLUENT's Continuous Fiber Module that allows you to analyze the behavior of fiber flow, fiber properties, and coupling between fibers and the surrounding fluid due to the strong interaction that exists between the fibers and the surrounding gas.
FLUENT Fuel Cell Modules Manual contains information about the background and the usage of two separate add-on fuel cell models for FLUENT that allow you to model polymer electrolyte membrane fuel cells (PEMFC), solid oxide fuel cells (SOFC), and electrolysis with FLUENT.
FLUENT Magnetohydrodynamics (MHD) Module Manual contains information about the background and usage of FLUENT's Magnetohydrodynamics (MHD) Module that allows you to analyze the behavior of electrically conducting fluid flow under the influence of constant (DC) or oscillating (AC) electromagnetic fields.
FLUENT Migration Manual contains information about transitioning from the previous release of FLUENT, including details about new features, solution changes, and text command list changes.

FLUENT Population Balance Module Manual contains information about the background and usage of FLUENT's Population Balance Module that allows you to analyze multiphase flows involving size distributions where particle population (as well as momentum, mass, and energy) require a balance equation.

Running FLUENT Under LSF contains information about the using FLUENT with Platform Computing's LSF software, a distributed computing resource management tool.

Running FLUENT Under PBS Professional contains information about the using FLUENT with Altair PBS Professional, an open workload management tool for local and distributed environments.

Running FLUENT Under SGE contains information about the using FLUENT with Sun Grid Engine (SGE) software, a distributed computing resource management tool.

3. Technical Support

If you encounter difficulties while using ANSYS FLUENT, please first refer to 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. The product documentation is available from the online help, or from the ANSYS Customer Portal (www.ansys.com/customerportal).

If you encounter an error, please write down the exact error message that appeared and note as much information as you can about what you were doing in ANSYS FLUENT.

Technical Support for ANSYS, Inc. products is provided either by ANSYS, Inc. directly or by one of our certified ANSYS Support Providers. Please check with the ANSYS Support Coordinator (ASC) at your company to determine who provides support for your company, or go to www.ansys.com and select **About ANSYS> Contacts and Locations**. The direct URL is: http://www1.ansys.com/customer/public/supportlist.asp. Follow the on-screen instructions to obtain your support provider contact information. You will need your customer number. If you don't know your customer number, contact the ASC at your company.

If your support is provided by ANSYS, Inc. directly, Technical Support can be accessed quickly and efficiently from the ANSYS Customer Portal, which is available from the ANSYS Website (www.ansys.com) under **Support> Technical Support** where the Customer Portal is located. The direct URL is: http://www.ansys.com/customerportal.

One of the many useful features of the Customer Portal is the Knowledge Resources Search, which can be found on the Home page of the Customer Portal.

Systems and installation Knowledge Resources are easily accessible via the Customer Portal by using the following keywords in the search box: `Systems/Installation`. These Knowledge Resources provide solutions and guidance on how to resolve installation and licensing issues quickly.

NORTH AMERICA

All ANSYS, Inc. Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Toll-Free Telephone: 1.800.711.7199

Fax: 1.724.514.5096

Support for University customers is provided only through the ANSYS Customer Portal.

GERMANY

ANSYS Mechanical Products

Telephone: +49 (0) 8092 7005-55

Email: support@cadfem.de

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

National Toll-Free Telephone:

German language: 0800 181 8499

English language: 0800 181 1565

International Telephone:

German language: +49 6151 3644 300

English language: +49 6151 3644 400

Email: support-germany@ansys.com

UNITED KINGDOM

All ANSYS, Inc. Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +44 (0) 870 142 0300

Fax: +44 (0) 870 142 0302

Email: support-uk@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

JAPAN

CFX , ICEM CFD and Mechanical Products

Telephone: +81-3-5324-8333

Fax: +81-3-5324-7308

Email: *CFX:* japan-cfx-support@ansys.com; *Mechanical:* japan-ansys-support@ansys.com

FLUENT Products

Telephone: +81-3-5324-7305

Email: *FLUENT:* japan-fluent-support@ansys.com; *POLYFLOW:* japan-polyflow-support@ansys.com; *FfC:* japan-ffc-support@ansys.com; *FloWizard:* japan-flowizard-support@ansys.com

Icepak

Telephone: +81-3-5324-7444

Email: japan-icepak-support@ansys.com

Licensing and Installation

Email: japan-license-support@ansys.com

INDIA

ANSYS Products (including FLUENT, CFX, ICEM-CFD)

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +91 1 800 233 3475 (toll free) or +91 1 800 209 3475 (toll free)

Fax: +91 80 2529 1271

Email: *FEA products:* feasup-india@ansys.com; *CFD products:* cfdsup-india@ansys.com; *Installation:* installation-india@ansys.com

FRANCE

All ANSYS, Inc. Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Toll-Free Telephone: +33 (0) 800 919 225

Email: support-france@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

BELGIUM

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +32 (0) 10 45 28 61

Email: support-belgium@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

SWEDEN

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +44 (0) 870 142 0300

Email: support-sweden@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

SPAIN and PORTUGAL

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +33 1 30 60 15 63

Email: support-spain@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

ITALY

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +39 02 89013378

Email: support-italy@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

Chapter 1: Introduction to ANSYS FLUENT

ANSYS FLUENT is a state-of-the-art computer program for modeling fluid flow, heat transfer, and chemical reactions in complex geometries.

ANSYS FLUENT is written in the C computer language and makes full use of the flexibility and power offered by the language. Consequently, true dynamic memory allocation, efficient data structures, and flexible solver control are all possible. In addition, ANSYS FLUENT uses a client/server architecture, which enables it to run as separate simultaneous processes on client desktop workstations and powerful compute servers. This architecture allows for efficient execution, interactive control, and complete flexibility between different types of machines or operating systems.

ANSYS FLUENT provides complete mesh flexibility, including the ability to solve your flow problems using unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2D triangular/quadrilateral, 3D tetrahedral/hexahedral/pyramid/wedge/polyhedral, and mixed (hybrid) meshes. ANSYS FLUENT also enables you to refine or coarsen your mesh based on the flow solution.

After a mesh has been read into ANSYS FLUENT, all remaining operations are performed within ANSYS FLUENT. These include setting boundary conditions, defining fluid properties, executing the solution, refining the mesh, and postprocessing and viewing the results.

The ANSYS FLUENT serial solver manages file input and output, data storage, and flow field calculations using a single solver process on a single computer. ANSYS FLUENT also uses a utility called `cortex` that manages ANSYS FLUENT's user interface and basic graphical functions. ANSYS FLUENT's parallel solver enables you to compute a solution using multiple processes that may be executing on the same computer, or on different computers in a network.

Parallel processing in ANSYS FLUENT involves an interaction between ANSYS FLUENT, a host process, and a set of compute-node processes. ANSYS FLUENT interacts with the host process and the collection of compute nodes using the `cortex` user interface utility.

Figure 1.1 (p. 2) and *Figure 1.2* (p. 3) illustrate the serial and parallel ANSYS FLUENT architectures.

Figure 1.1 Serial ANSYS FLUENT Architecture

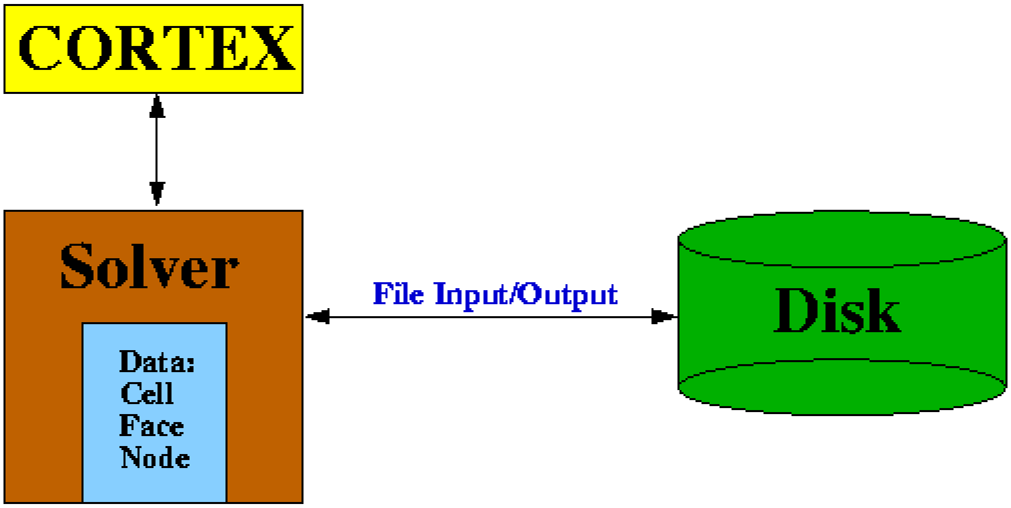
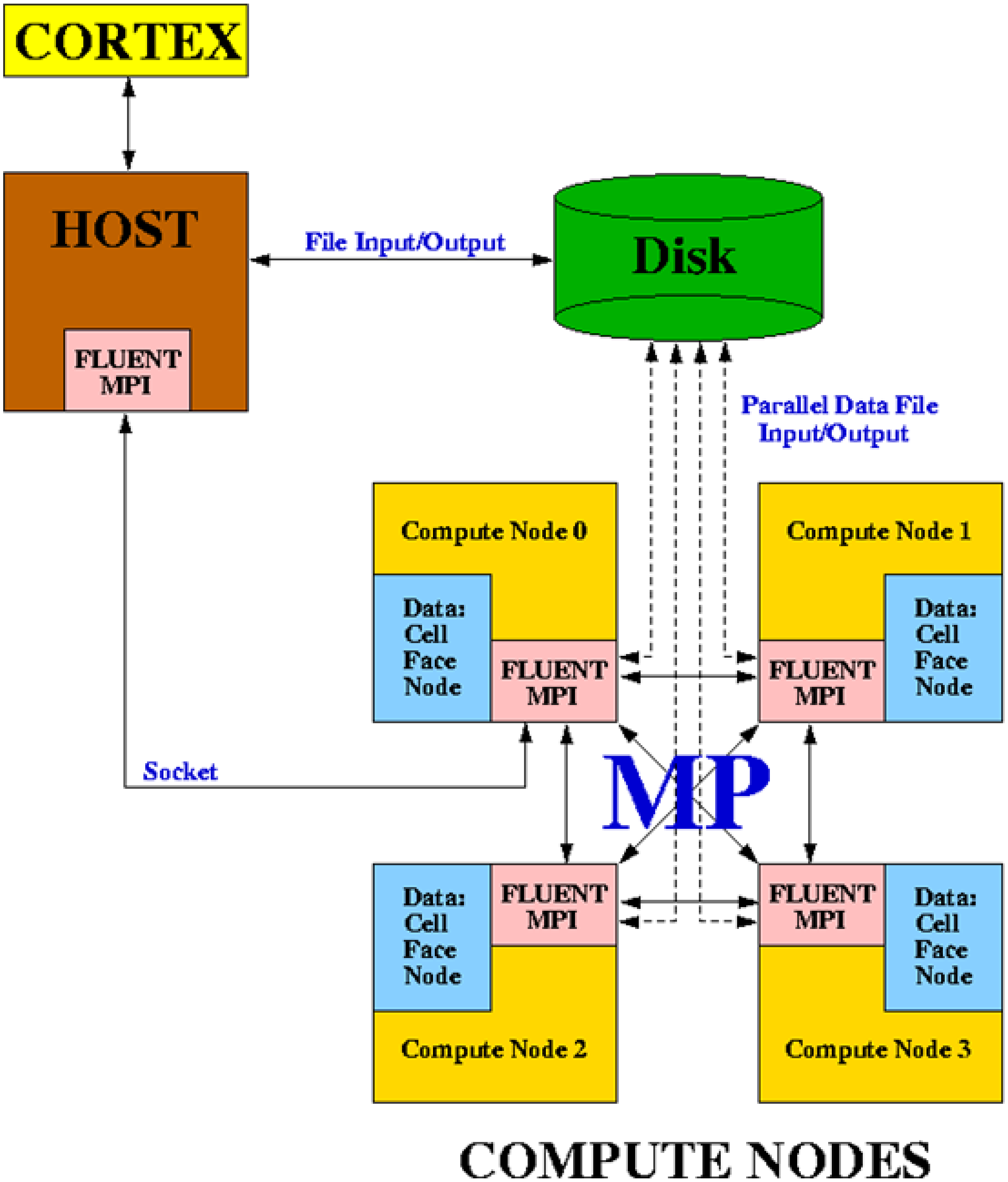


Figure 1.2 Parallel ANSYS FLUENT Architecture



For more information about ANSYS FLUENT's parallel processing capabilities, message passing interfaces (MPI), and so on, refer to "Parallel Processing" in the User's Guide.

All functions required to compute a solution and display the results are accessible in ANSYS FLUENT through an interactive interface.

For more information, see the following sections:

1.1. Program Capabilities

The ANSYS FLUENT solver has the following modeling capabilities:

- 2D planar, 2D axisymmetric, 2D axisymmetric with swirl (rotationally symmetric), and 3D flows
- Quadrilateral, triangular, hexahedral (brick), tetrahedral, prism (wedge), pyramid, polyhedral, and mixed element meshes
- Steady-state or transient flows
- Incompressible or compressible flows, including all speed regimes (low subsonic, transonic, supersonic, and hypersonic flows)
- Inviscid, laminar, and turbulent flows
- Newtonian or non-Newtonian flows
- Ideal or real gases
- Heat transfer, including forced, natural, and mixed convection, conjugate (solid/fluid) heat transfer, and radiation
- Chemical species mixing and reaction, including homogeneous and heterogeneous combustion models and surface deposition/reaction models
- Free surface and multiphase models for gas-liquid, gas-solid, and liquid-solid flows
- Lagrangian trajectory calculation for dispersed phase (particles/droplets/bubbles), including coupling with continuous phase and spray modeling
- Cavitation model
- Phase change model for melting/solidification applications
- Porous media with non-isotropic permeability, inertial resistance, solid heat conduction, and porous-face pressure jump conditions
- Lumped parameter models for fans, pumps, radiators, and heat exchangers
- Acoustic models for predicting flow-induced noise
- Inertial (stationary) or non-inertial (rotating or accelerating) reference frames
- Multiple reference frame (MRF) and sliding mesh options for modeling multiple moving frames
- Mixing-plane model for modeling rotor-stator interactions, torque converters, and similar turbomachinery applications with options for mass conservation and swirl conservation
- Dynamic mesh model for modeling domains with moving and deforming mesh
- Volumetric sources of mass, momentum, heat, and chemical species
- Material property database
- Extensive customization capability via user-defined functions
- Dynamic (two-way) coupling with GT-Power and WAVE
- Magnetohydrodynamics (MHD) module (documented separately)
- Continuous fiber module (documented separately)
- Fuel cell modules (documented separately)
- Population balance module (documented separately)

ANSYS FLUENT is ideally suited for incompressible and compressible fluid-flow simulations in complex geometries. ANSYS FLUENT's parallel solver enables you to compute solutions for cases with very large meshes on multiple processors, either on the same computer or on different computers in a network. ANSYS, Inc. also offers other solvers that address different flow regimes and incorporate alternative physical models. Additional CFD programs from ANSYS, Inc. include CFX, Airpak, ANSYS Icepak, and ANSYS POLYFLOW.

1.2. ANSYS FLUENT Documentation

ANSYS FLUENT documentation is available through ANSYS FLUENT's online help system. The online help system provides access to the ANSYS FLUENT documentation, using the ANSYS help viewer, whether you are working in ANSYS FLUENT or not. Adobe Acrobat PDF versions of the documentation are also provided. This section describes how to access the ANSYS FLUENT documentation outside of ANSYS FLUENT. See the User's Guide for information about accessing the documentation through ANSYS FLUENT.

To view the documentation, you can use the help viewer or the PDF files available from the installation area.

1.2.1. Accessing the ANSYS FLUENT Documentation

You can access the ANSYS FLUENT documentation through the ANSYS help viewer, or by opening the PDF version from your ANSYS, Inc. installation area.

1.2.1.1. Accessing the Documentation Files Using the ANSYS Help Viewer

To start the ANSYS help viewer, go to the following location from the Windows **Start** menu:

Start > Program Files > ANSYS 14.0 > Help > ANSYS Help

The ANSYS help viewer provides access to documentation for most ANSYS products.

To navigate to the ANSYS FLUENT documentation, do the following:

1. Scroll down to **FLUENT** in the left-hand panel.
2. Expand the ANSYS FLUENT documentation set by clicking the icon next to **FLUENT**.
3. Click a document title to display the table of contents for the selected document.
4. To find specific information, you can do any of the following:
    a. In the Contents tab, click an icon next to a title to expand the tree hierarchy, or click an item in the tree hierarchy to display the corresponding information.
    b. In the Search tab, enter a keyword or phrase and click **Search**.

    > **Note**
    >
    > Search results can be sorted by clicking on the column title.

    c. In the Index tab, type an index string.

Note

Index entries that partially match your typing will be displayed. You can continue typing, or scroll through the list to find specific entries.

1.2.1.2. Accessing the PDF Documentation Files in the Installation Area

You can view the PDF documentation by opening the following folder/directory:

On Windows:

path`\ANSYS Inc\v140\commonfiles\help\en-us\pdf\`

where *path* is the directory in which you have installed ANSYS FLUENT (by default, the path is `C:\Program Files`).

On Linux:

path`/ansys_inc/v140/commonfiles/help/en-us/pdf/`

where *path* is the directory in which you have installed ANSYS FLUENT.

This directory contains PDF versions of the ANSYS FLUENT documentation.

1.2.2. Viewing and Printing the PDF Documentation

The PDF documentation files are appropriate for viewing and printing with Adobe Acrobat Reader (version 5.0 or higher), which is available for most Windows and Linux systems. These files are distinguished by a `.pdf` suffix in their file names.

1.2.2.1. Navigating the PDF Files

For the purpose of easier online document navigation, the PDF files contain hyperlinks in the table of contents and index. In addition, hyperlinks have been applied to all cross-references to chapters, sections, figures, tables, bibliography, and index entries.

1.2.2.2. Printing the PDF Files

Adobe Acrobat PDF files are provided for printing all or part of the manuals. While you can also print individual topics from the ANSYS help viewer, the PDF files are recommended when printing long sections since the printout will have a higher quality.

Note that you can select the paper size to which you are printing in Adobe Acrobat Reader by selecting the **File/Print Setup...** menu item and choosing the desired **Paper** size. If the page is too large to fit on your paper size, you can reduce it by selecting the **File/Print...** menu item and choosing the **Reduce to Printer Margins** option under **Page Scaling**.

Chapter 2: Basic Steps for CFD Analysis using ANSYS FLUENT

Before you begin your CFD analysis using ANSYS FLUENT, careful consideration of the following issues will contribute significantly to the success of your modeling effort. Also, when you are planning a CFD project, be sure to take advantage of the customer support available to all ANSYS FLUENT users.

For more information, see the following sections:

2.1. Steps in Solving Your CFD Problem

Once you have determined the important features of the problem you want to solve, follow the basic procedural steps shown below.

1. Define the modeling goals.
2. Create the model geometry and mesh.
3. Set up the solver and physical models.
4. Compute and monitor the solution.
5. Examine and save the results.
6. Consider revisions to the numerical or physical model parameters, if necessary.

Step 2 of the solution process requires a geometry modeler and mesh generator. You can use the geometry and meshing capabilities within ANSYS Workbench, or a separate CAD system for geometry modeling and mesh generation. Alternatively, you can use supported CAD packages to generate volume meshes for import into ANSYS FLUENT (see the User's Guide). For more information on creating geometry and generating meshes using each of these programs, refer to their respective manuals.

The details of the remaining steps are covered in the User's Guide.

2.2. Planning Your CFD Analysis

For each of the problem-solving steps, there are some questions that you need to consider:

- **Defining the Modeling Goals**
  - What results are you looking for, and how will they be used?
    - → What are your modeling options?
    - → What physical models will need to be included in your analysis?
    - → What simplifying assumptions do you have to make?
    - → What simplifying assumptions can you make?
    - → Do you require a unique modeling capability?
      - Could you utilize user-defined functions (written in C)?

- What degree of accuracy is required?
- How quickly do you need the results?
- How will you isolate a piece of the complete physical system?
- Where will the computational domain begin and end?
  - → Do you have boundary condition information at these boundaries?
  - → Can the boundary condition types accommodate that information?
  - → Can you extend the domain to a point where reasonable data exists?
- Can it be simplified or approximated as a 2D or axisymmetric problem?

• **Creating Your Model Geometry and Mesh**

ANSYS FLUENT uses unstructured meshes in order to reduce the amount of time you spend generating meshes, to simplify the geometry modeling and mesh generation process, to enable modeling of more complex geometries than you can handle with conventional, multi-block structured meshes, and to enable you to adapt the mesh to resolve the flow-field features. ANSYS FLUENT can also use body-fitted, block-structured meshes (for example, those used by ANSYS FLUENT 4 and many other CFD solvers). ANSYS FLUENT is capable of handling triangular and quadrilateral elements (or a combination of the two) in 2D, and tetrahedral, hexahedral, pyramid, wedge, and polyhedral elements (or a combination of these) in 3D. This flexibility enables you to pick mesh topologies that are best suited for your particular application, as described in the User's Guide.

You can adapt all types of meshes (except for polyhedral) in ANSYS FLUENT in order to resolve large gradients in the flow field, but you must always generate the initial mesh (whatever the element types used) outside of the solver, or one of the CAD systems for which mesh import filters exist.

The following questions should be considered when you are generating a mesh:

- Can you benefit from other ANSYS, Inc. products such as CFX, ANSYS Icepak, or Airpak?
- Can you use a quad/hex mesh or should you use a tri/tet mesh or a hybrid mesh?
  - → How complex is the geometry and flow?
  - → Will you need a non-conformal interface?
- What degree of mesh resolution is required in each region of the domain?
  - → Is the resolution sufficient for the geometry?
  - → Can you predict regions with high gradients?
  - → Will you use adaption to add resolution?
- Do you have sufficient computer memory?
  - → How many cells are required?
  - → How many models will be used?

• **Setting Up the Solver and Physical Models**

For a given problem, you will need to:

- Import and check the mesh.
- Select the numerical solver (for example, density based, pressure based, unsteady, and so on).
- Select appropriate physical models.

        - → Turbulence, combustion, multiphase, and so on.
    - Define material properties.
        - → Fluid
        - → Solid
        - → Mixture
    - Prescribe operating conditions.
    - Prescribe boundary conditions at all boundary zones.
    - Provide an initial solution.
    - Set up solver controls.
    - Set up convergence monitors.
    - Initialize the flow field.
- **Computing and Monitoring Your Solution**
    - The discretized conservation equations are solved iteratively.
        - → A number of iterations are usually required to reach a converged solution.
    - Convergence is reached when:
        - → Changes in solution variables from one iteration to the next are negligible.
            - Residuals provide a mechanism to help monitor this trend.
        - → Overall property conservation is achieved.
    - The accuracy of a converged solution is dependent upon:
        - → Appropriateness and accuracy of physical models.
        - → Mesh resolution and independence.
        - → Problem setup.
- **Examining and Saving Your Results**

  Examine the results to review the solution and extract useful data.
    - Visualization tools can be used to answer such questions as:
        - → What is the overall flow pattern?
        - → Is there separation?
        - → Where do shocks, shear layers, and so on form?
        - → Are key flow features being resolved?
    - Numerical reporting tools can be used to calculate the following quantitative results:
        - → Forces and moments
        - → Average heat transfer coefficients
        - → Surface and volume integrated quantities
        - → Flux balances
- **Revising Your Model**

Once your solution is converged, the following questions should be considered when you are analyzing the solution:

- Are physical models appropriate?
  - → Is flow turbulent?
  - → Is flow unsteady?
  - → Are there compressibility effects?
  - → Are there 3D effects?
- Are boundary conditions correct?
  - → Is the computational domain large enough?
  - → Are boundary conditions appropriate?
  - → Are boundary values reasonable?
- Is the mesh adequate?
  - → Can the mesh be adapted to improve results?
  - → Does the solution change significantly with adaption, or is the solution mesh independent?
  - → Does boundary resolution need to be improved?

Chapter 3: Guide to a Successful Simulation Using ANSYS FLUENT

The following guidelines can help you make sure your CFD simulation is a success. Before logging a technical support request, make sure you do the following:

1. Examine the quality of the mesh.

   There are two basic things that you should do before you start a simulation:

   - Perform a mesh check to avoid problems due to incorrect mesh connectivity, and so on.
   - Look at maximum cell skewness (for example, using the **Compute** button in the **Contours** dialog box). As a rule of thumb, the skewness should be below 0.98.

   If there are mesh problems, you may have to re-mesh the problem.

2. Scale the mesh and check length units.

   In ANSYS FLUENT, all physical dimensions are initially assumed to be in meters. You should scale the mesh accordingly. Other quantities can also be scaled independently of other units used. ANSYS FLUENT defaults to SI units.

3. Employ the appropriate physical models.
4. Set the energy under-relaxation factor between 0.95 and 1.

   For problems with conjugate heat transfer, when the conductivity ratio is very high, smaller values of the energy under-relaxation factor practically stall the convergence rate.

5. Use node-based gradients with unstructured tetrahedral meshes.

   The node-based averaging scheme is known to be more accurate than the default cell-based scheme for unstructured meshes, most notably for triangular and tetrahedral meshes.

6. Monitor convergence with residuals history.

   Residual plots can show when the residual values have reached the specified tolerance. After the simulation, note if your residuals have decreased by at least 3 orders of magnitude to at least 10^{-3}. For the pressure-based solver, the scaled energy residual must decrease to 10^{-6}. Also, the scaled species residual may need to decrease to 10^{-5} to achieve species balance.

   You can also monitor lift, drag, or moment forces as well as pertinent variables or functions (for example, surface integrals) at a boundary or any defined surface.

7. Run the CFD simulation using second order discretization for better accuracy rather than a faster solution.

   A converged solution is not necessarily a correct one. You should use the second-order upwind discretization scheme for final results.

8. Monitor values of solution variables to make sure that any changes in the solution variables from one iteration to the next are negligible.

9. Verify that property conservation is satisfied.

   After the simulation, note if overall property conservation has been achieved. In addition to monitoring residual and variable histories, you should also check for overall heat and mass balances. At a minimum, the net imbalance should be less than 1% of smallest flux through domain boundary.

10. Check for mesh dependence.

    You should ensure that the solution is mesh-independent and use mesh adaption to modify the mesh or create additional meshes for the mesh-independence study.

11. Check to see that the solution makes sense based on engineering judgment.

    If flow features do not seem reasonable, you should reconsider your physical models and boundary conditions. Reconsider the choice of the boundary locations (or the domain). An inadequate choice of domain (especially the outlet boundary) can significantly impact solution accuracy.

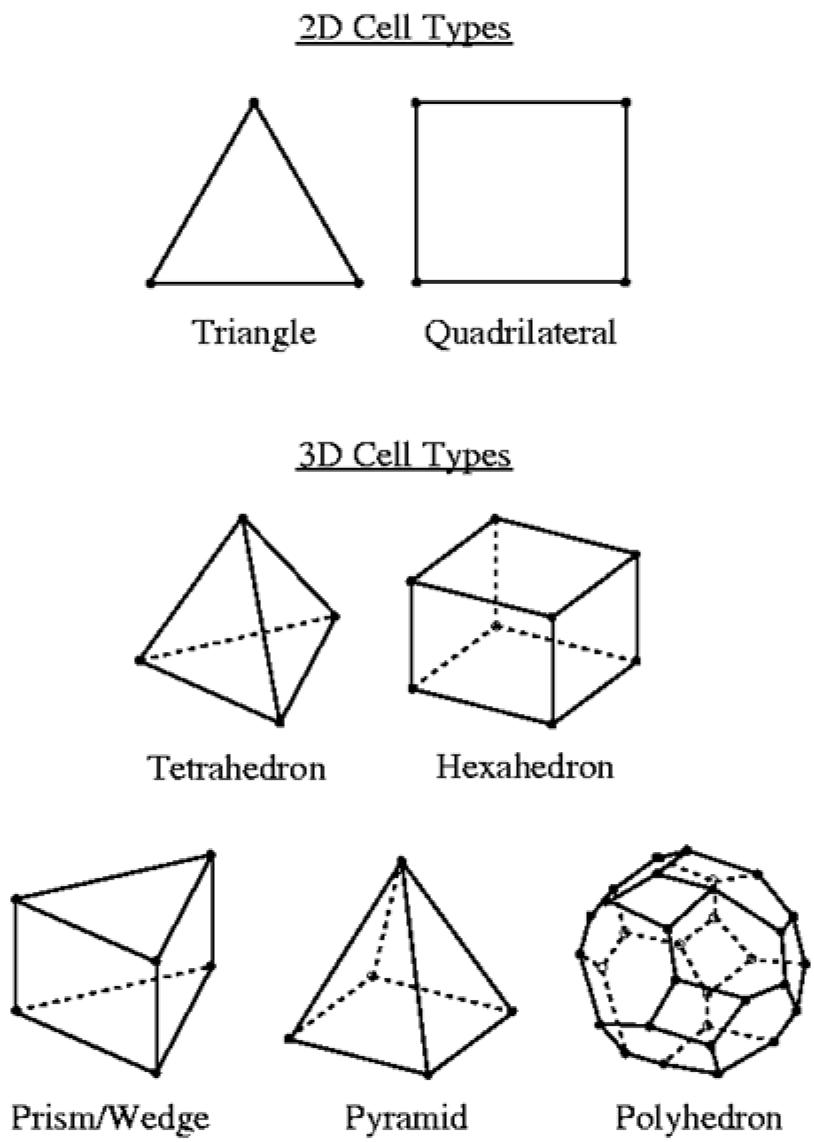
You are encouraged to collaborate with your technical support engineer in order to develop a solution process that ensures good results for your specific application. This type of collaboration is a good investment of time for both yourself and the ANSYS FLUENT support engineer.

Glossary of Terms

This glossary contains a listing of terms commonly used throughout the documentation.

adaption	A technique useful in improving overall mesh quality. The solution-adaptive mesh refinement feature of ANSYS FLUENT allows you to refine and/or coarsen your mesh based on geometric and numerical solution data. In addition, ANSYS FLUENT provides tools for creating and viewing adaption fields customized to particular applications.
case files	Files that contain the mesh, boundary conditions, and solution parameters for a problem. A case file also contains the information about the user interface and graphics environment.
cell types	The various shapes or units that constitute the base elements of a mesh. ANSYS FLUENT can use meshes comprised of tetrahedral, hexahedral, pyramid, wedge, or polyhedral cells (or a combination of these).

Figure 3 Cell Types



computational fluid dynamics (CFD)	The science of predicting fluid flow, heat transfer, mass transfer (as in perspiration or dissolution), phase change (as in freezing or boiling), chemical reaction (e.g., combustion), mechanical movement (e.g., fan rotation), stress or deformation of related solid structures (such as a mast bending in the wind), and related phenomena by solving the mathematical equations that govern these processes using a numerical algorithm on a computer.
console	The console is part of the ANSYS FLUENT application window that allows for text command input and the display of information.
convergence	The point at which the solution is no longer changing with each successive iteration. Convergence criteria, along with a reduction in residuals, also help in determining when a solution is complete. Convergence criteria are pre-set conditions on the residuals that indicate that a certain level of convergence has been achieved. If the residuals for all problem variables fall below the convergence

criteria but are still in decline, the solution is still changing to a greater or lesser degree. A better indicator occurs when the residuals flatten in a traditional residual plot (of residual value vs. iteration). This point, sometimes referred to as convergence at the level of machine accuracy, takes time to reach, however, and may be beyond your needs. For this reason, alternative tools such as reports of forces, heat balances, or mass balances can be used instead.

cortex
: A utility that manages ANSYS FLUENT's user interface and basic graphical functions.

data files
: Files that contain the values of the flow field in each grid element and the convergence history (residuals) for that flow field.

dialog boxes
: The separate windows that are used like forms to perform input tasks. Each dialog box is unique and employs various types of input controls that make up the form.

discretization
: The act of replacing the differential equations that govern fluid flow with a set of algebraic equations that are solved at distinct points.

GUI
: The graphical user interface, which consists of the main ANSYS FLUENT application window, dialog boxes, graphics windows, etc.

mesh
: A collection of points representing the flow field, where the equations of fluid motion (and temperature, if relevant) are calculated.

models
: Numerical algorithms that approximate physical phenomenon (e.g., turbulence).

node
: The distinct points of a *mesh* (p. 15) at which the equations of fluid motion are solved.

postprocessing
: The act of analyzing the numerical results of your CFD simulation using reports, integrals, and graphical analysis tools such as contour plots, animations, etc.

residuals
: The small imbalance that is created during the course of the iterative solution algorithm. This imbalance in each cell is a small, non-zero value that, under normal circumstances, decreases as the solution progresses.

skewness
: The difference between the shape of the cell and the shape of an equilateral cell of equivalent volume. Highly skewed cells can decrease accuracy and destabilize the solution.

solvers
: ANSYS FLUENT has two distinct solvers, based on numerical precision (single-precision vs. double-precision). Within each of these categories, there are solver formulations: pressure based; density based explicit; and density based implicit.

terminal emulator
: See *console* (p. 14).

TUI
: The text user interface, which consists of textual commands that can be entered into the terminal emulator.