

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 14.5
October 2012

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

Copyright and Trademark Information

© 2012 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, EKM, Engineering Knowledge Manager, CFX, FLUENT, HFSS and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

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

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the [legal information](#) in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

Preface	v
1. The Contents of This Manual	v
2. The Contents of the FLUENT Manuals	v
3. Technical Support	vi
1. Introduction to ANSYS FLUENT	1
1.1. Program Capabilities	4
1.2. ANSYS FLUENT Documentation	5
1.2.1. Accessing the ANSYS FLUENT Documentation	5
1.2.1.1. Accessing the Documentation Files Using the ANSYS Help Viewer	5
1.2.1.2. Downloading and Extracting the PDF Documentation Files	6
1.2.2. Viewing and Printing the PDF Documentation	6
1.2.2.1. Navigating the PDF Files	6
1.2.2.2. Printing the PDF Files	7
2. Basic Steps for CFD Analysis using ANSYS FLUENT	9
2.1. Steps in Solving Your CFD Problem	9
2.2. Planning Your CFD Analysis	9
3. Guide to a Successful Simulation Using ANSYS FLUENT	15
4. Starting and Executing ANSYS FLUENT	17
4.1. Starting ANSYS FLUENT	17
4.1.1. Starting ANSYS FLUENT Using FLUENT Launcher	17
4.1.1.1. Setting General Options in FLUENT Launcher	19
4.1.1.2. Single-Precision and Double-Precision Solvers	21
4.1.1.3. Setting Parallel Options in FLUENT Launcher	21
4.1.1.4. Setting Remote Options in FLUENT Launcher	22
4.1.1.5. Setting Scheduler Options in FLUENT Launcher	23
4.1.1.6. Setting Environment Options in FLUENT Launcher	24
4.1.2. Starting ANSYS FLUENT on a Windows System	25
4.1.3. Starting ANSYS FLUENT on a Linux System	26
4.1.4. Command Line Startup Options	26
4.1.4.1. Graphics Options	28
4.1.4.2. Meshing Mode Option	28
4.1.4.3. Parallel Options	28
4.1.4.4. Postprocessing Option	29
4.1.4.5. SGE Options	29
4.1.4.6. LSF Option	29
4.1.4.7. Version and Release Options	29
4.1.4.8. System Coupling Options	30
4.1.4.9. Other Startup Options	30
4.2. Running ANSYS FLUENT in Batch Mode	31
4.2.1. Background Execution on Linux Systems	31
4.2.2. Background Execution on Windows Systems	32
4.2.3. Batch Execution Options	33
4.3. Switching Between Meshing and Solution Modes	34
4.4. Checkpointing an ANSYS FLUENT Simulation	34
4.5. Cleaning Up Processes From an ANSYS FLUENT Simulation	35
4.6. Exiting ANSYS FLUENT	36
Glossary of Terms	37

Preface

This preface is divided into the following sections:

1. [The Contents of This Manual](#)
2. [The Contents of the FLUENT Manuals](#)
3. [Technical Support](#)

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.

A brief description of what is in each chapter follows:

- [Introduction to ANSYS FLUENT \(p. 1\)](#), provides an overview of the capabilities of ANSYS FLUENT and details about the available documentation.
- [Basic Steps for CFD Analysis using ANSYS FLUENT \(p. 9\)](#), describes the steps involved in solving a CFD problem and questions to consider.
- [Guide to a Successful Simulation Using ANSYS FLUENT \(p. 15\)](#), provides specific guidelines that help ensure your CFD simulation is a success.
- [Starting and Executing ANSYS FLUENT \(p. 17\)](#), describes options and alternatives to starting, running, and exiting ANSYS FLUENT. It also provides instructions for remote execution and batch execution.
- [Glossary of Terms \(p. 37\)](#), contains a listing of terms commonly used throughout the documentation.

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 and provides details about starting, running, and exiting the program.
- [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 User's Guide](#) contains detailed information about running a simulation using the solution mode of 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.
- [ANSYS FLUENT Meshing User Guide](#) contains detailed information about creating 3D meshes using the meshing mode of FLUENT.
- [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 examples of various flow problems with detailed instructions, commentary, and postprocessing of results.
- [ANSYS FLUENT Meshing Tutorials](#) contains a number of examples of general mesh-generation techniques used in ANSYS FLUENT Meshing.
- [FLUENT Text Command List](#) contains a brief description of each of the commands in FLUENT's solution mode text interface.
- [ANSYS FLUENT Meshing Text Command List](#) contains a brief description of each of the commands in FLUENT's meshing mode 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 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.
- [FLUENT as a Server User's Guide](#) contains information about the usage of FLUENT as a Server which allows you to connect to a FLUENT session and issue commands from a remote client application.
- [Running FLUENT Under LSF](#) contains information about using FLUENT with Platform Computing's LSF software, a distributed computing resource management tool.
- [Running FLUENT Under PBS Professional](#) contains information about using FLUENT with Altair PBS Professional, an open workload management tool for local and distributed environments.
- [Running FLUENT Under SGE](#) contains information about 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**.

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: support.ansys.com.

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 (CADFEM)

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

Austria: 0800 297 835

Switzerland: 0800 546 318

International Telephone:

German language: +49 6151 152 9981

English language: +49 6151 152 9982

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: Please have your Customer or Contact ID ready.

UK: 0800 048 0462

Republic of Ireland: 1800 065 6642

Outside UK: +44 1235 420130

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

All ANSYS, Inc. Products

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

Telephone: +91 1 800 209 3475 (toll free) or +91 20 6654 3000 (toll)

Fax: +91 80 6772 2600

Email:

FEA products: feasup-india@ansys.com;

CFD products: cfdsup-india@ansys.com;

Ansoft products: ansoftsup-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 **Toll Number:** +33 (0) 170 489 087

Email: support-france@ansys.com

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: +34 900 933 407 (Spain), +351 800 880 513 (Portugal)

Email: support-spain@ansys.com, support-portugal@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.

You can read your mesh into ANSYS FLUENT, or, for 3D geometries, create your mesh using the meshing mode of FLUENT (see the [FLUENT Meshing User's Guide](#) for further details). All remaining operations are performed within the solution mode of FLUENT, including 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: Serial ANSYS FLUENT Architecture \(p. 2\)](#) and [Figure 1.2: Parallel ANSYS FLUENT Architecture \(p. 3\)](#) illustrate the serial and parallel ANSYS FLUENT architectures.

Figure 1.1: Serial ANSYS FLUENT Architecture

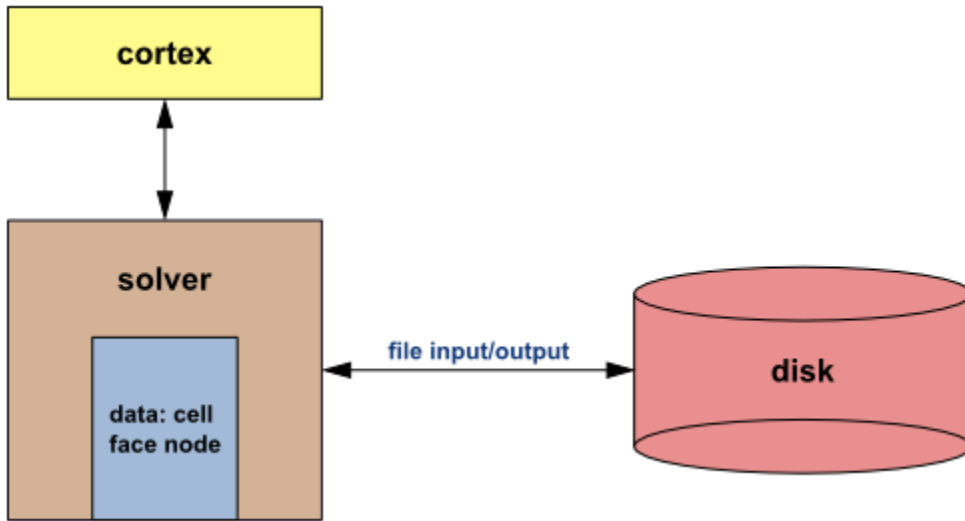
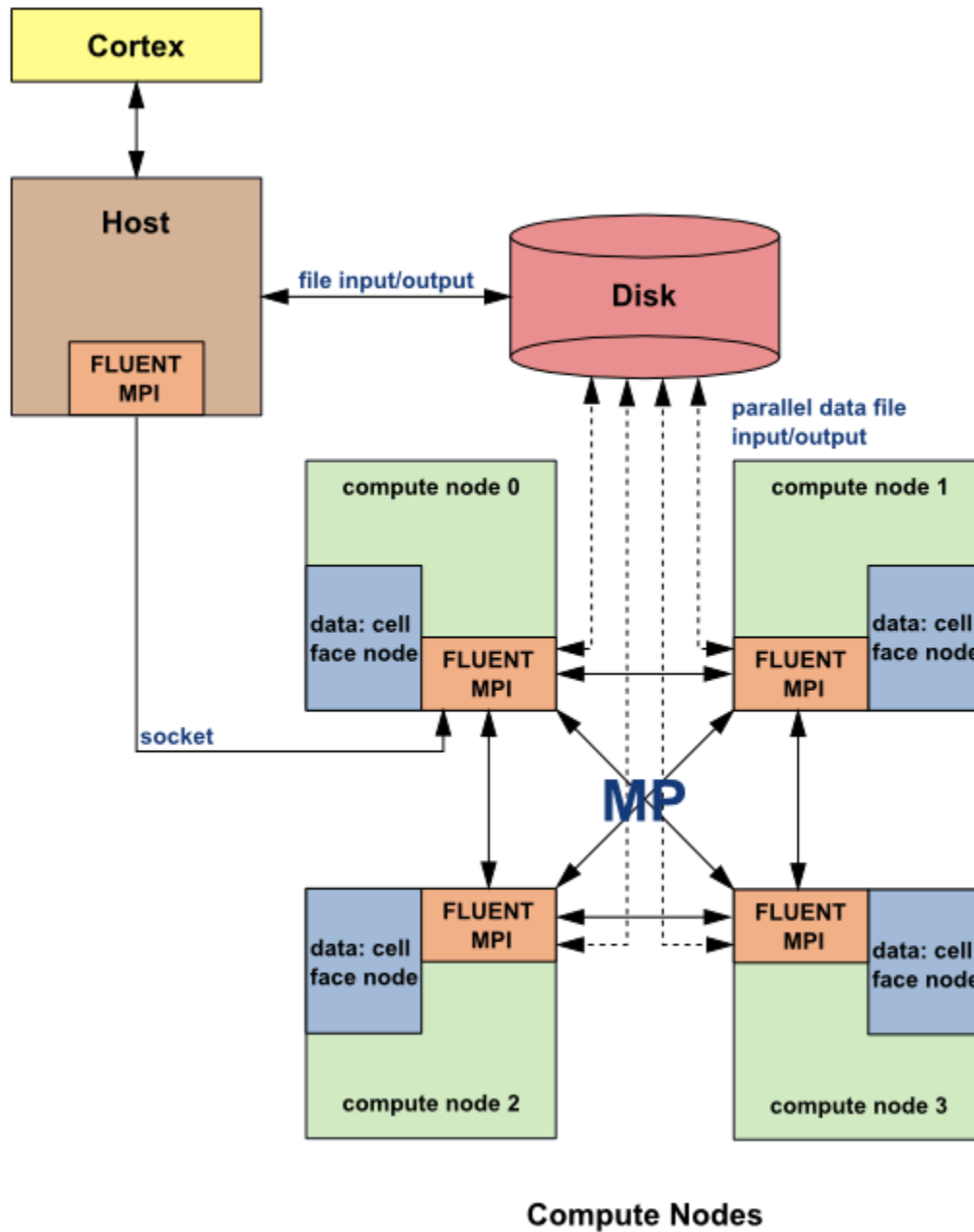


Figure 1.2: Parallel ANSYS FLUENT Architecture



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](#)

[1.2. ANSYS FLUENT Documentation](#)

1.1. Program Capabilities

When in meshing mode, ANSYS FLUENT functions as a robust unstructured-volume-mesh generator (see [Program Capabilities](#) in the [FLUENT Meshing User's Guide](#) for further details). When in solution mode, FLUENT allows you to simulate the following:

- 2D planar, 2D axisymmetric, 2D axisymmetric with swirl (rotationally symmetric), and 3D flows
- Flows on 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 calculations for dispersed phase (particles/droplets/bubbles), including coupling with continuous phase and spray modeling
- Cavitation model simulations
- Melting/solidification applications using the phase change model
- 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 moving frames using multiple reference frame (MRF) and sliding mesh options
- Mixing-plane model simulations of rotor-stator interactions, torque converters, and similar turbomachinery applications with options for mass conservation and swirl conservation
- Dynamic mesh model simulations for domains with moving and deforming meshes
- Volumetric sources of mass, momentum, heat, and chemical species
- Simulations that use a material property database

- Simulations customized by 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, ANSYS 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. 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 available from the installation area.

[1.2.1. Accessing the ANSYS FLUENT Documentation](#)

[1.2.2. Viewing and Printing the PDF Documentation](#)

1.2.1. Accessing the ANSYS FLUENT Documentation

You can access the ANSYS FLUENT documentation through the ANSYS help viewer.

[1.2.1.1. Accessing the Documentation Files Using the ANSYS Help Viewer](#)

[1.2.1.2. Downloading and Extracting the PDF Documentation Files](#)

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 14.5 > 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 **ANSYS** in the left-hand panel.
2. Expand the ANSYS FLUENT documentation set by clicking the icon next to **ANSYS**.
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. Downloading and Extracting the PDF Documentation Files

You can download the PDF documentation from the ANSYS Customer Portal (support.ansys.com/doc-downloads). These files must be downloaded and extracted to the following location so you can launch them from within your ANSYS product **Help** menu.

On Windows:

```
path\ANSYS Inc\v145\commonfiles\help\en-us\pdf\
```

where *path* is the folder where you installed your ANSYS product (by default, the path is C:\Program Files).

On Linux:

```
path/ansys_inc/v145/commonfiles/help/en-us/pdf/
```

where *path* is the directory where you installed your ANSYS product.

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](#)

[1.2.2.2. Printing the PDF Files](#)

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](#)

[2.2. Planning Your CFD Analysis](#)

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 Design-Modeler and ANSYS Meshing within ANSYS Workbench or you can use a separate CAD system for geometry modeling and mesh generation. When meshing 3D geometries, you can also use the meshing mode of FLUENT. 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](#).

For 3D geometries, you can create the mesh using the meshing mode of FLUENT; otherwise, you must generate the initial mesh (whatever the element types used) outside of FLUENT or use one of the CAD systems for which mesh import filters exist. When in solution mode, FLUENT can be used to adapt all types of meshes (except for polyhedral), in order to resolve large gradients in the flow field.

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 ANSYS 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 in FLUENT.

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. In particular, you should be sure that the minimum reported cell volume is not negative.
- Look at maximum cell skewness (for example, using the **Compute** button in the **Contours** dialog box after initializing the model). As a rule of thumb, the skewness should be below 0.98. You can also use the **Report Quality** function to calculate the minimum cell orthogonality. You can find more details about mesh quality considerations in [Mesh Quality in the FLUENT User's Guide](#).

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.

Chapter 4: Starting and Executing ANSYS FLUENT

This chapter provides instructions for starting and executing ANSYS FLUENT.

- 4.1. Starting ANSYS FLUENT
- 4.2. Running ANSYS FLUENT in Batch Mode
- 4.3. Switching Between Meshing and Solution Modes
- 4.4. Checkpointing an ANSYS FLUENT Simulation
- 4.5. Cleaning Up Processes From an ANSYS FLUENT Simulation
- 4.6. Exiting ANSYS FLUENT

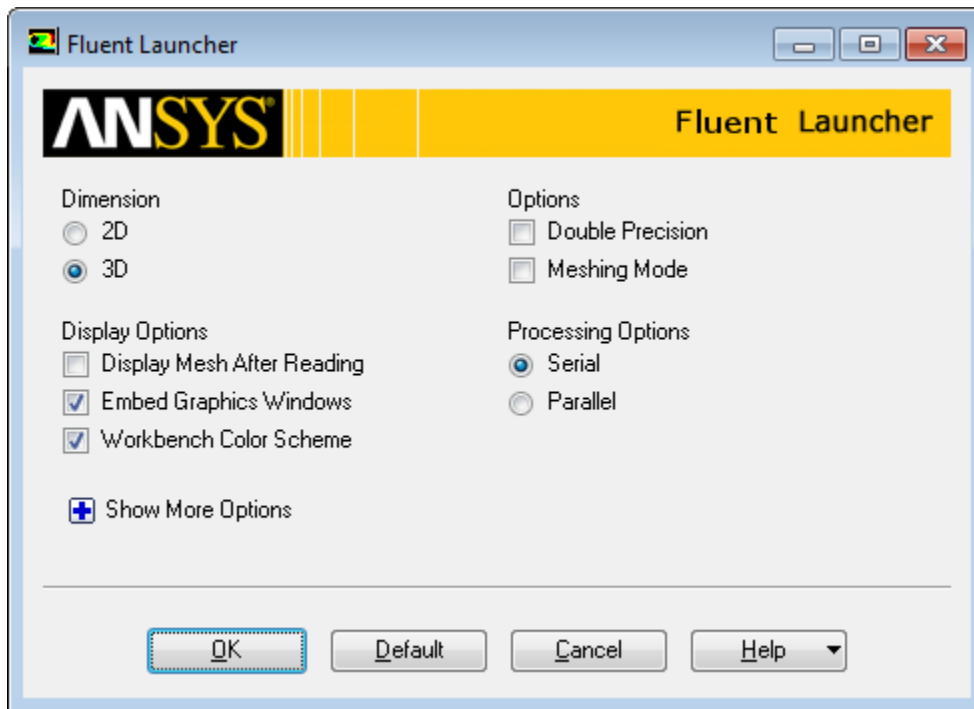
4.1. Starting ANSYS FLUENT

The following sections describe how start ANSYS FLUENT:

- 4.1.1. Starting ANSYS FLUENT Using FLUENT Launcher
- 4.1.2. Starting ANSYS FLUENT on a Windows System
- 4.1.3. Starting ANSYS FLUENT on a Linux System
- 4.1.4. Command Line Startup Options

4.1.1. Starting ANSYS FLUENT Using FLUENT Launcher

When you start ANSYS FLUENT from the Linux or Windows command line with no arguments, from the Windows Programs menu, or from the Windows desktop, FLUENT Launcher will appear ([Figure 4.1: FLUENT Launcher \(p. 18\)](#)). In the FLUENT Launcher, you can specify the dimensionality of the problem (2D or 3D), as well as other options (for example, whether you want a single-precision or double-precision calculation):

Figure 4.1: FLUENT Launcher

Under **Dimension**, select **2D** for two-dimensional problems, or select **3D** for the three-dimensional problems.

The **Display Options** allow you to make decisions related to the graphics windows:

- You can choose to have ANSYS FLUENT automatically display the mesh immediately after reading a mesh or case file by using the **Display Mesh After Reading** option (disabled by default). All of the boundary zones will be displayed, except for the interior zones of 3D geometries. Note that your decision regarding this option can be overridden after you have launched ANSYS FLUENT: simply change the status of the **Display Mesh After Reading** option in the **Select File** dialog box that opens when you are reading in the mesh or case file.
- You can choose to have the graphics windows embedded within the ANSYS FLUENT application window by using the **Embed Graphics Windows** option (enabled by default), rather than having floating graphics windows.
- You can choose to use the default **Workbench Color Scheme** in the graphics windows (that is, a blue background), rather than the classic black background.

Under **Options**, you can do the following:

- You can choose to run ANSYS FLUENT in double-precision mode by enabling the **Double-Precision** option (by default, you start ANSYS FLUENT in single-precision mode).
- You can choose to start ANSYS FLUENT in meshing mode (rather than the default solution mode) by enabling the **Meshing Mode** option. See the [FLUENT Meshing User's Guide](#) for further details about the meshing mode.
- You can choose to use various job schedulers using the **Use Job Scheduler** option (for example, the Microsoft Job Scheduler for Windows, or LSF, SGE, and PBS Pro on Linux). For more information about using

FLUENT Launcher with job schedulers, see [Setting Scheduler Options in FLUENT Launcher \(p. 23\)](#), as well as [Setting Parallel Scheduler Options in FLUENT Launcher](#) in the *User's Guide*.

- You can choose to run parallel simulations on Linux clusters, via the Windows interface using the **Use Remote Linux Nodes** option (see [Setting Remote Options in FLUENT Launcher \(p. 22\)](#) for details).

Under **Processing Options**, select **Serial** for the serial solver, or select **Parallel** to run the solver in parallel (see [Parallel Processing](#) in the *User's Guide*).

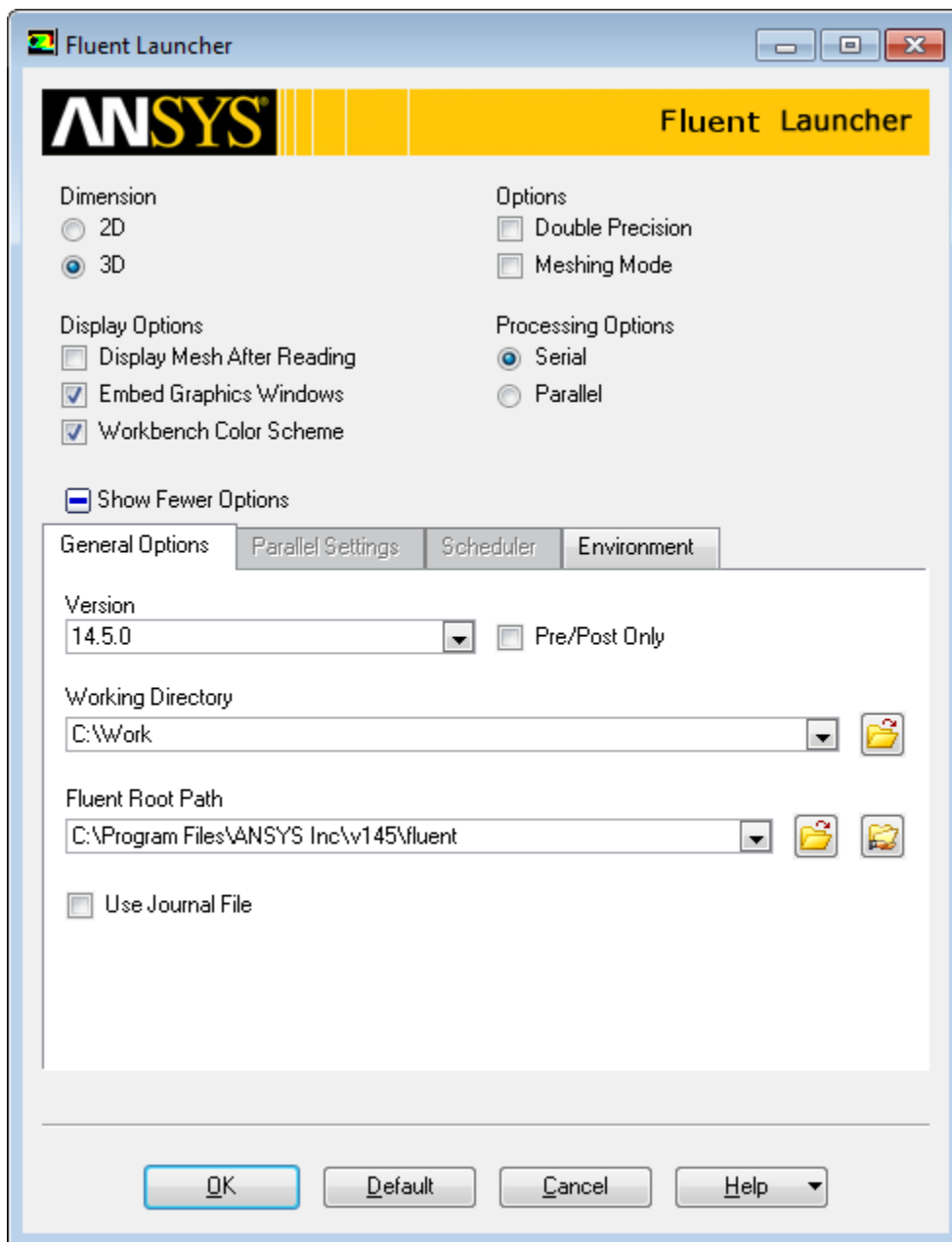
If you select the **Show More Options** button, FLUENT Launcher expands to reveal more options ([Figure 4.2: The General Options Tab of FLUENT Launcher \(p. 20\)](#)). Note that once FLUENT Launcher expands, the **Show More Options** button becomes the **Show Fewer Options** button, allowing you to hide the additional options.

Important

FLUENT Launcher also appears when you start ANSYS FLUENT within ANSYS Workbench. For more information, see the separate [ANSYS FLUENT in Workbench User's Guide](#).


4.1.1.1. Setting General Options in FLUENT Launcher




The **General Options** tab allows you to specify generic settings for running ANSYS FLUENT.

Figure 4.2: The General Options Tab of FLUENT Launcher

1. Specify the version of ANSYS FLUENT by selecting the appropriate option in the **Version** drop-down list. The drop-down list contains all of the available versions of ANSYS FLUENT that exist in your ANSYS FLUENT installation.

In addition, you start ANSYS FLUENT in full solution mode. You can choose to run ANSYS FLUENT where only the set up or postprocessing capabilities are available by selecting the **Pre/Post Only** check box. The full ANSYS FLUENT solution mode allows you to set up, solve and postprocess a problem, while **Pre/Post Only** allows you to set up or postprocess a problem, but will not allow you to perform calculations.

2. Specify the path of your current working folder using the **Working Directory** field or click  to browse through your directory structure. Note that a Uniform Naming Convention (UNC) path cannot be set as a working folder, and you need to map a drive to the UNC path (Windows only).

3. Specify the location of the ANSYS FLUENT installation on your system using the **Fluent Root Path** field, or click  to browse through your directory structure to locate the installation folder; try to use the UNC path if applicable—the  button automatically converts a local path to a UNC path if any matching shared directory is found (Windows only). Once set, various fields in FLUENT Launcher (for example, parallel settings, etc.) are automatically populated with the available options, depending on the ANSYS FLUENT installations that are available.
4. Specify the path and name of a journal file by selecting the **Use Journal File** check box and entering the journal file location and name, or click  to browse through your directory structure to locate the file. Using the journal file, you can automatically load the case, compile any user-defined functions, iterate until the solution converges, and write results to an output file.

4.1.1.2. Single-Precision and Double-Precision Solvers

Both single-precision and double-precision versions of ANSYS FLUENT are available on all computer platforms. For most cases, the single-precision solver will be sufficiently accurate, but certain types of problems may benefit from the use of a double-precision version. Several examples are listed below:

- If your geometry has features of very disparate length scales (for example, a very long, thin pipe), single-precision calculations may not be adequate to represent the node coordinates.
- If your geometry involves multiple enclosures connected via small-diameter pipes (for example, automotive manifolds), mean pressure levels in all but one of the zones can be quite large (since you can set only one global reference pressure location). Double-precision calculations may therefore be necessary to resolve the pressure differences that drive the flow, since these will typically be much smaller than the pressure levels.
- For conjugate problems involving high thermal-conductivity ratios and/or high-aspect-ratio meshes, convergence and/or accuracy may be impaired with the single-precision solver, due to inefficient transfer of boundary information.
- For multiphase problems where the population balance model is used to resolve particle size distributions, which could have statistical moments whose values span many orders of magnitude.

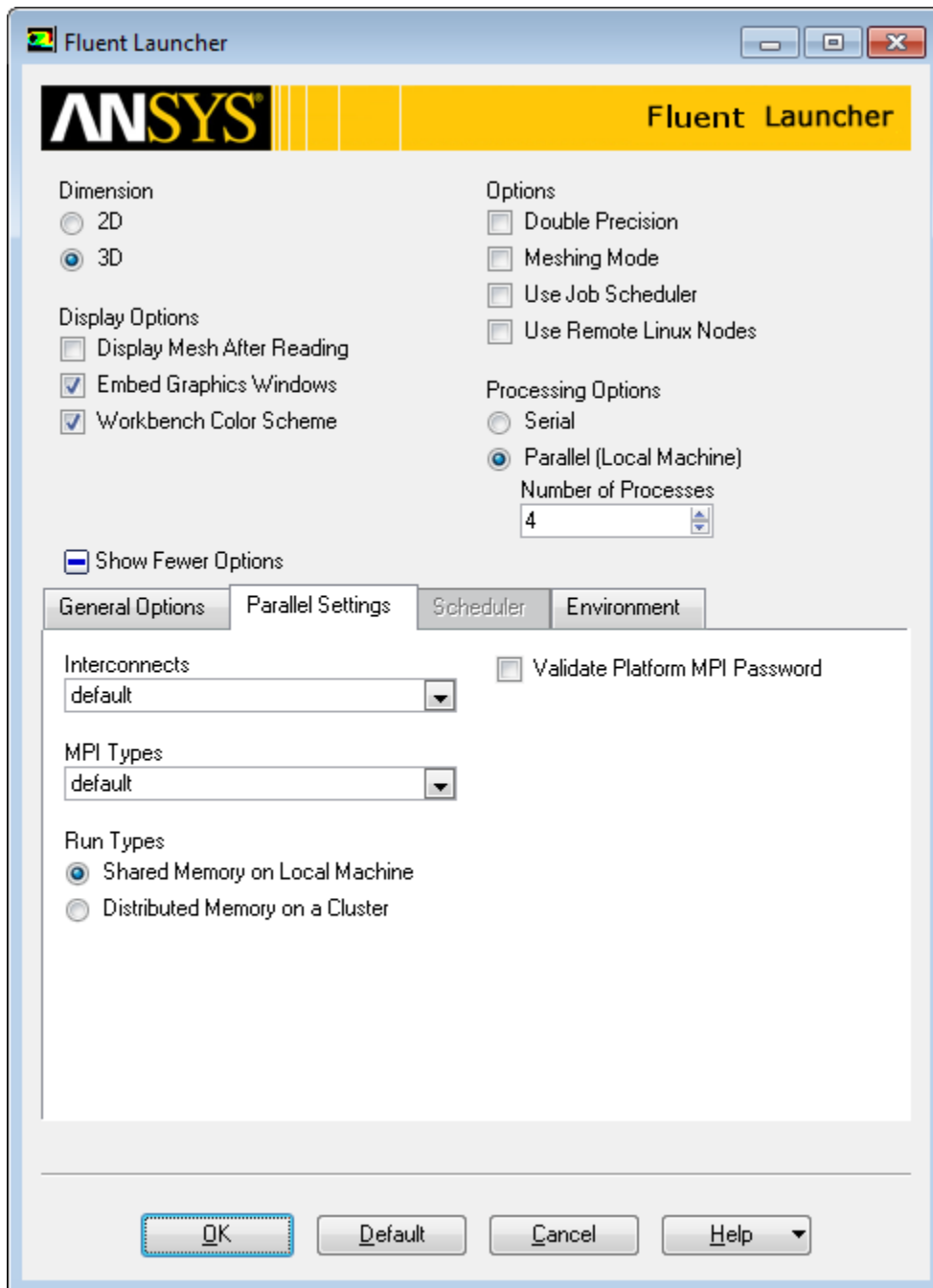
Note

ANSYS FLUENT allows only a period to be used as a decimal separator. If your system is set to a European locale that uses a comma separator (for example, Germany), fields that accept numeric input may accept a comma, but may ignore everything after the comma. If your system is set to a non-European locale, numeric fields will not accept a comma at all.

ANSYS Workbench accepts commas as decimal delimiters. These are translated into periods when data is passed to ANSYS FLUENT.

4.1.1.3. Setting Parallel Options in FLUENT Launcher

The **Parallel Settings** tab ([Figure 4.3: The Parallel Settings Tab of FLUENT Launcher \(p. 22\)](#)) allows you to specify settings for running ANSYS FLUENT in parallel. This tab is only available if you have selected **Parallel** under **Processing Options**. Once you select **Parallel**, you can specify the number of processes using the **Number of Processes** field.

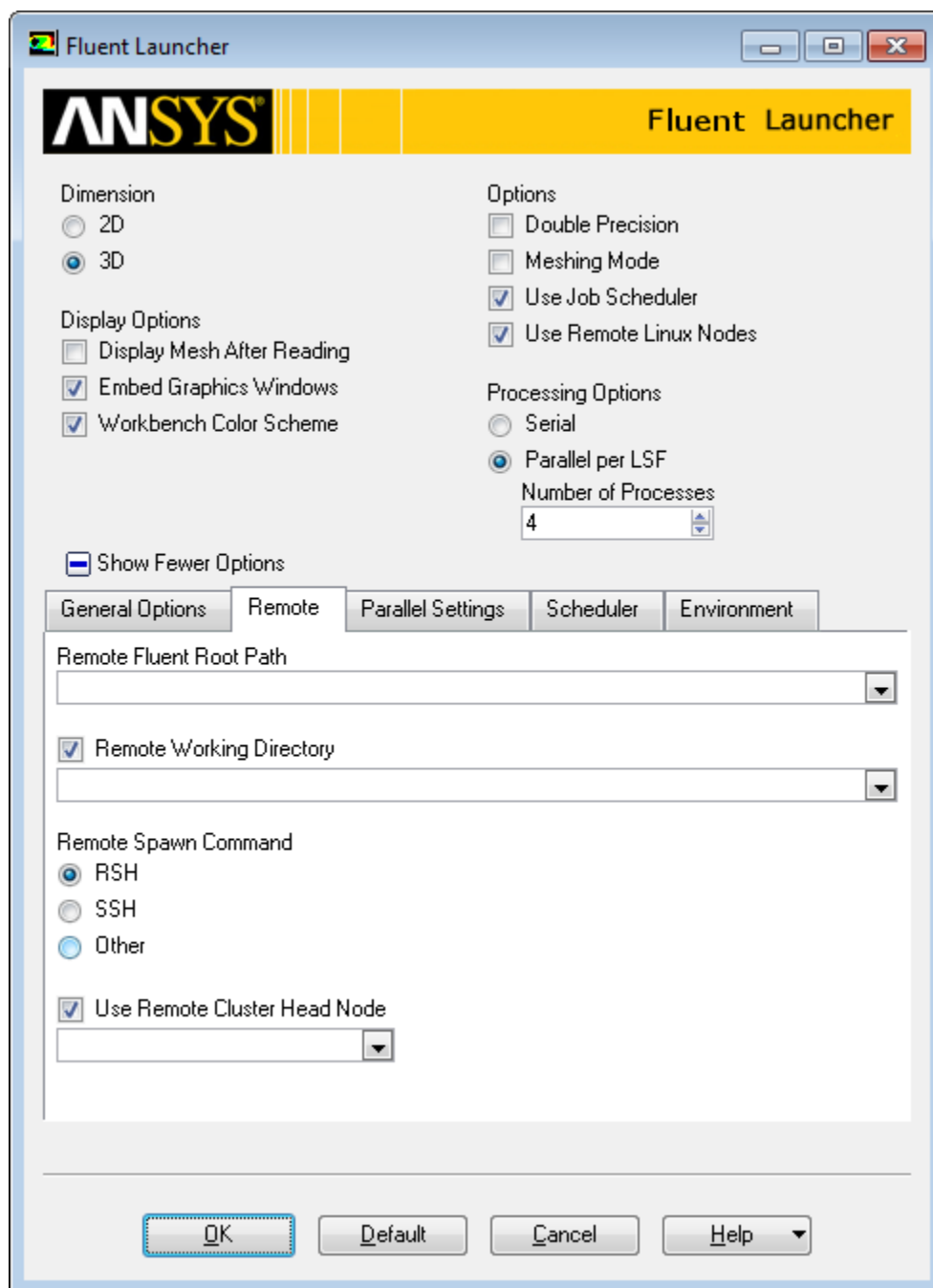
Figure 4.3: The Parallel Settings Tab of FLUENT Launcher

For additional information about this tab, see [Starting Parallel ANSYS FLUENT Using FLUENT Launcher](#) in the *User's Guide*.

4.1.1.4. Setting Remote Options in FLUENT Launcher

The **Remote** tab (Figure 4.4: The Remote Tab of FLUENT Launcher (p. 23)) allows you to specify settings for running ANSYS FLUENT parallel simulations on Linux clusters, via the Windows interface.

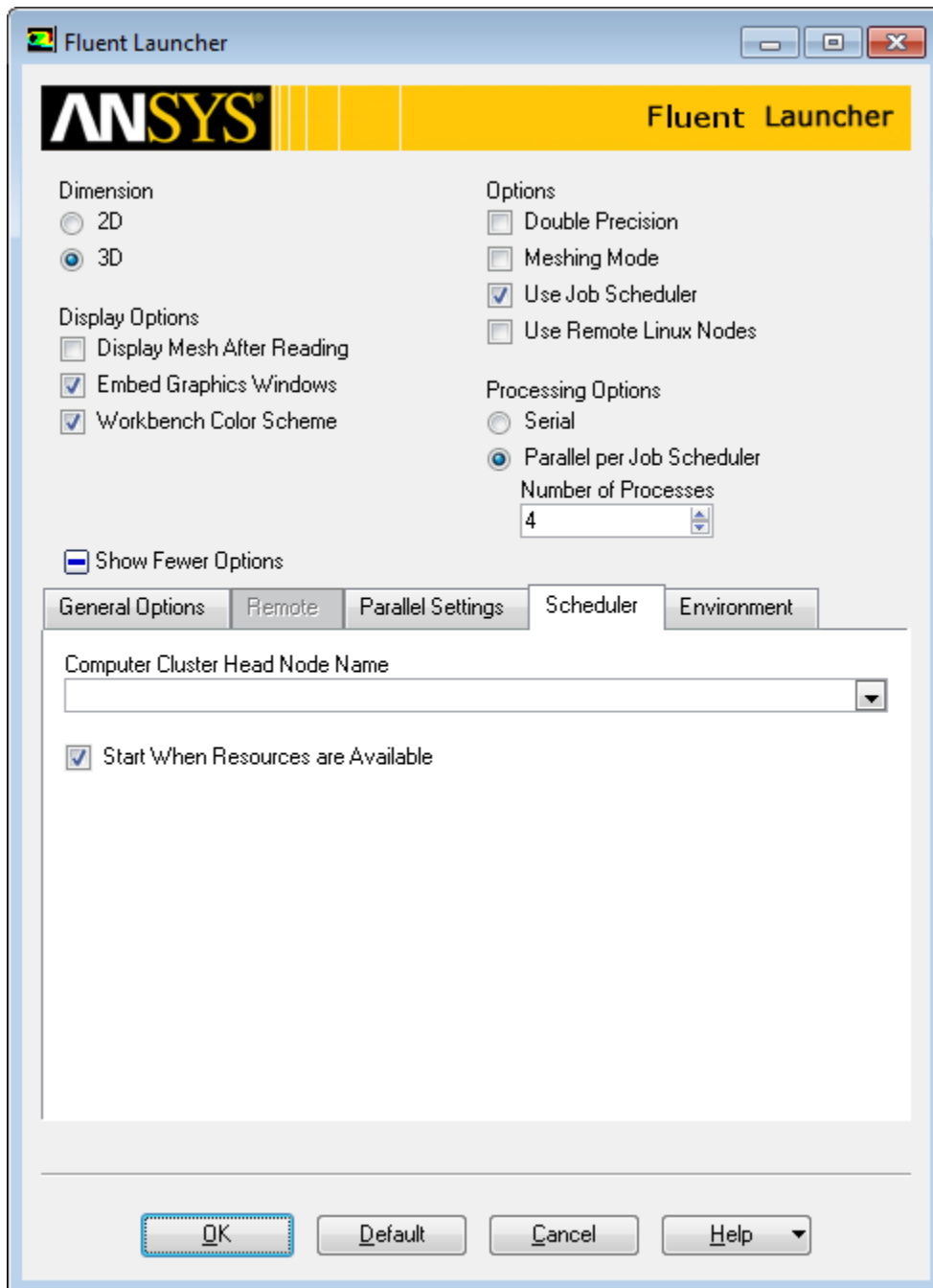
Figure 4.4: The Remote Tab of FLUENT Launcher



For additional information about this tab, see [Setting Additional Options When Running on Remote Linux Machines](#) in the *User's Guide*.

4.1.1.5. Setting Scheduler Options in FLUENT Launcher

The **Scheduler** tab (Figure 4.5: [The Scheduler Tab of FLUENT Launcher \(Windows 64 Version\)](#) (p. 24)) allows you to specify settings for running ANSYS FLUENT with various job schedulers (for example, the Microsoft Job Scheduler for Windows (64-bit only), or LSF, SGE, and PBS Pro on Linux). This tab is available if you have selected **Use Job Scheduler** under **Options**.

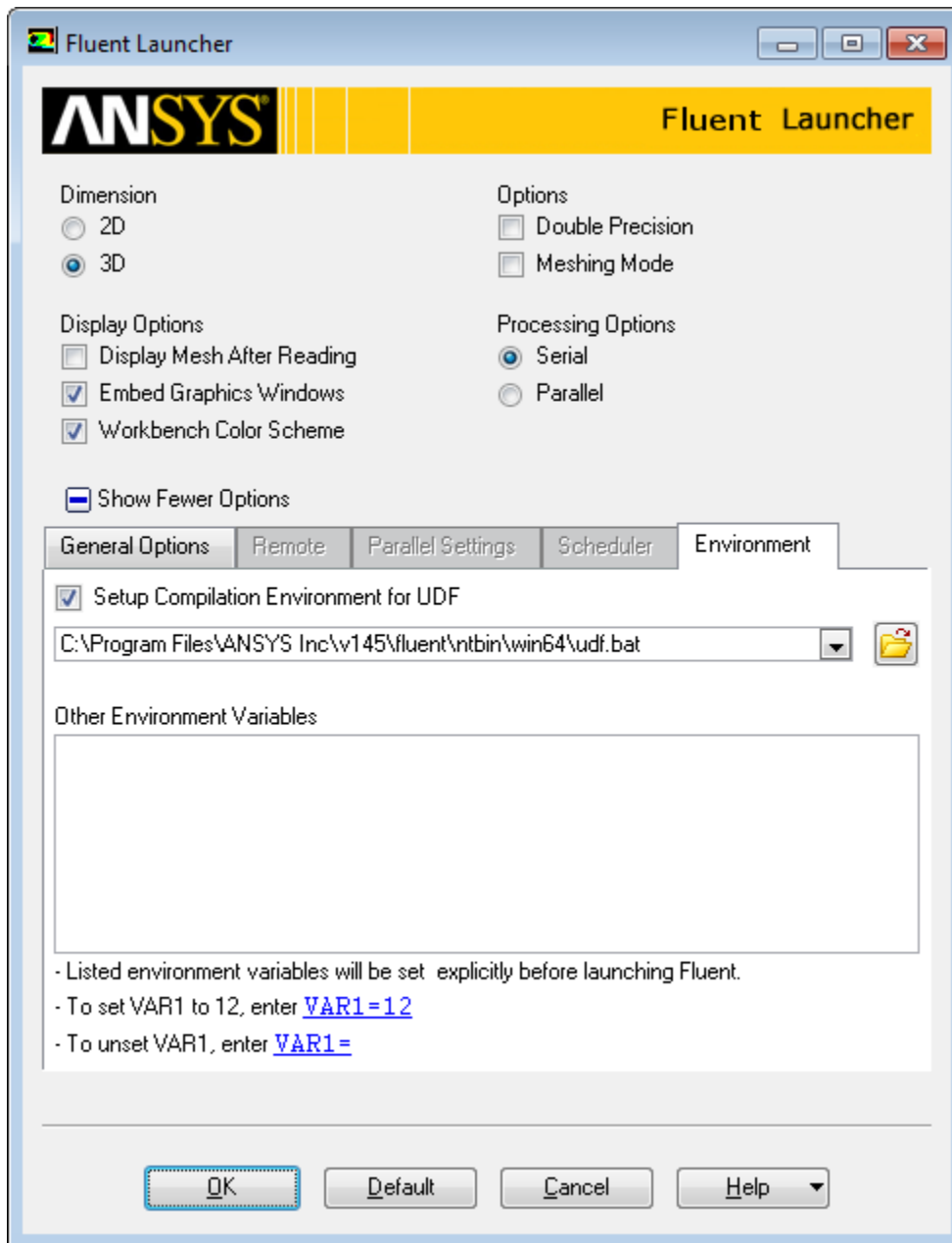
Figure 4.5: The Scheduler Tab of FLUENT Launcher (Windows 64 Version)

For additional information about this tab, see [Setting Parallel Scheduler Options in FLUENT Launcher](#) in the *User's Guide*.

4.1.1.6. Setting Environment Options in FLUENT Launcher

The **Environment** tab (Figure 4.6: [The Environment Tab of FLUENT Launcher \(p. 25\)](#)) allows you to specify compiler settings for compiling user-defined functions (UDFs) with ANSYS FLUENT (Windows only). The **Environment** tab also allows you to specify environment variable settings for running ANSYS FLUENT.

Figure 4.6: The Environment Tab of FLUENT Launcher



Specify a batch file that contains UDF compilation environment settings by selecting the **Setup Compilation Environment for UDF** check box (enabled by default). Once selected, you can then enter a batch file name in the text field. By default, FLUENT Launcher uses the `udf.bat` file that is located in the folder where ANSYS FLUENT is installed. It is recommended that you keep the default batch file, which is tested with the latest MS Visual Studio C++ compilers at the time of the ANSYS FLUENT release date. For more information about compiling UDFs, see the separate ANSYS FLUENT [UDF Manual](#).

Under **Other Environment Variables**, enter or edit license file or environment variable information in the text field. Using the **Default** button resets the default value(s).

4.1.2. Starting ANSYS FLUENT on a Windows System

There are two ways to start ANSYS FLUENT on a Windows system:

- Click the Start button, select the All Programs menu, select the ANSYS 14.5 menu, select the Fluid Dynamics menu, then select the FLUENT program item. (Note that if the default “ANSYS 14.5” program group name was changed when ANSYS FLUENT was installed, you will find the FLUENT menu item in the program group with the new name that was assigned, rather than in the ANSYS 14.5 program group.) This option starts FLUENT Launcher (see [Starting ANSYS FLUENT Using FLUENT Launcher](#)).
- Start from a Command Prompt window by typing `fluent 2d` (for the 2D single-precision solver), `fluent 3d` (for the 3D single-precision solver), `fluent 2ddp` (for the 2D double-precision solver), or `fluent 3ddp` (for the 3D double-precision solver) at the prompt. Before doing so, however, you must first modify your user environment so that the Command utility will find `fluent`. You can do this by executing the `setenv.exe` program located in the ANSYS FLUENT home directory (for example, `C:\Program Files\ANSYS Inc\v145\fluent\ntbin\win64`). This program will add the ANSYS FLUENT folder to your command search path.

From the Command Prompt window, you can also start the parallel version of ANSYS FLUENT. To start the parallel version on `x` processors, type `fluent version -tx` at the prompt, replacing `version` with the desired solver version (`2d`, `3d`, `2ddp`, or `3ddp`) and `x` with the number of processors (for example, `fluent 3d -t4` to run the 3D version on 4 processors). For information about the parallel version of ANSYS FLUENT for Windows, see [Starting Parallel ANSYS FLUENT on a Windows System](#) in the *User's Guide*.

4.1.3. Starting ANSYS FLUENT on a Linux System

There are two ways to start ANSYS FLUENT on a Linux system:

- Start FLUENT from the command line without specifying a version, and then use FLUENT Launcher to choose the appropriate version along with other options. See [Starting ANSYS FLUENT Using FLUENT Launcher](#) for details.
- Start the appropriate version from the command line by typing `fluent2d` (for the 2D single-precision solver), `fluent3d` (for the 3D single-precision solver), `fluent 2ddp` (for the 2D double-precision solver), or `fluent 3ddp` (for the 3D double-precision solver) at the prompt.

You can also start the parallel version of ANSYS FLUENT from the command line. To start the parallel version on `x` processors, type `fluent version -tx` at the prompt, replacing `version` with the desired solver version (`2d`, `3d`, `2ddp`, or `3ddp`) and `x` with the number of processors (for example, `fluent 3d -t4` to run the 3D version on 4 processors). See [Starting Parallel ANSYS FLUENT on a Linux System](#) in the *User's Guide* for more information about starting the parallel solvers.

4.1.4. Command Line Startup Options

To obtain information about available startup options, you can type `fluent -help` before starting up FLUENT. [Table 4.1: Available Command Line Options for Linux and Windows Platforms \(p. 26\)](#) lists the available command line arguments for Linux and Windows. More detailed descriptions of these options can be found in the following sections.

Table 4.1: Available Command Line Options for Linux and Windows Platforms

Option	Platform	Description
<code>-aas</code>	all	Start FLUENT in server mode
<code>-cc</code>	all	Use the classic color scheme

Option	Platform	Description
-ccp x	Windows only	Use the Microsoft Job Scheduler where x is the head node name.
-cnf=x	all	Specify the hosts or machine list file
-driver	all	Sets the graphics driver (available drivers vary by platform – opengl or x11 or null (Linux) – opengl or msw or null (Windows))
-env	all	Show environment variables
-fgw	all	Disables the embedded graphics
-g	all	Run without the GUI or graphics (Linux); Run with the GUI minimized (Windows)
-gr	all	Run without graphics
-gu	all	Run without the GUI but with graphics (Linux); Run with the GUI minimized but with graphics (Windows)
-help	all	Display command line options
-hidden	Windows only	Run in batch mode
-host_ip=host:ip	all	Specify the IP interface to be used by the host
-i journal	all	Reads the specified journal file
-lsf	Linux only	Run ANSYS FLUENT using LSF
-mesher	all	Start ANSYS FLUENT in meshing mode
-mpi=	all	Specify MPI implementation
-mpitest	all	Will launch an MPI program to collect network performance data
-nm	all	Do not display mesh after reading
-pcheck	Linux only	Checks all nodes
-post	all	Run the ANSYS FLUENT post-processing-only executable
-p<ic>	all	Choose the interconnect <ic>= default or myr or inf
-r	all	List all releases installed
-rx	all	Specify release number
-schost	all	Specify a host machine for system coupling
-scport	all	Specify a port on the host machine for system coupling
-scname	all	Specify a unique name for a system coupling participant
-sge	Linux only	Run ANSYS FLUENT under Sun Grid Engine

Option	Platform	Description
-sge queue	Linux only	Name of the queue for a given computing grid
-sgeckpt ckpt_obj	Linux only	Set checkpointing object to ckpt_obj for SGE
-sgepe fluent_pe min_n-max_n	Linux only	Set the parallel environment for SGE to fluent_pe, min_n and max_n are number of min and max nodes requested
-tx	all	Specify the number of processors x

4.1.4.1. Graphics Options

`fluent -driver` allows you to specify the graphics driver to be used in the session. For example, on Linux you can specify `fluent -driver opengl`, `fluent -driver x11`, or `fluent -driver null`. These options are described in detail in [Hiding the Graphics Window Display](#) in the [User's Guide](#). On Windows you can specify `fluent -driver opengl` or `fluent -driver msw` to enable graphics display. Using `msw` instead of `opengl` instructs ANSYS FLUENT to use the Operating Systems Windows driver rather than the hardware OpenGL driver.

`fluent -cc` will run Cortex using the classic black background color in the graphics window.

`fluent -fgw` will run Cortex without the graphics window embedded in the application (for example, "floating").

`fluent -g` will run Cortex without graphics and without the graphical user interface. This option is useful if you are not on an X Window display or if you want to submit a batch job.

`fluent -gr` will run Cortex without graphics. This option can be used in conjunction with the `-i journal` option to run a job in "background" mode.

`fluent -gu` will run Cortex without the graphical user interface but will display the graphics window(s). (On Windows systems, `fluent -gu` will run ANSYS FLUENT, keeping it in a minimized window; if you maximize the window, the GUI *will* be available.)

To start FLUENT and immediately read a journal file, type `fluent -i journal`, replacing `journal` with the name of the journal file you want to read.

`fluent -nm` will run Cortex without displaying the mesh in the graphics window.

4.1.4.2. Meshing Mode Option

`fluent -mesher` specifies that FLUENT opens in meshing mode rather than the default solution mode. See the [FLUENT Meshing User's Guide](#) for further details about the meshing mode.

4.1.4.3. Parallel Options

These options are used in association with the parallel solver.

`-ccp x` (where `x` is the name of the head node) runs the parallel job through the Microsoft Job Scheduler as described in [Starting Parallel ANSYS FLUENT with the Microsoft Job Scheduler](#) in the [User's Guide](#).

`-cnf=x` (where `x` is the name of a hosts file) spawns a compute node on each machine listed in the hosts file. Otherwise, you can spawn the processes as described in [Starting Parallel ANSYS FLUENT on a Windows System Using Command Line Options](#) or [Starting Parallel ANSYS FLUENT on a Linux System Using Command Line Options](#) in the User's Guide.

`-host_ip=host:ip` specifies the IP interface to be used by the host process (Linux only).

`-mpi=mpi` specifies the MPI to be used. You can skip this flag if you choose to use the default MPI.

`-mpitest` runs the `mpitest` program instead of ANSYS FLUENT to test the network.

`-p<ic>` specifies the use of parallel interconnect `<ic>`, where `<ic>` can be any of the interconnects listed in [Starting Parallel ANSYS FLUENT on a Windows System Using Command Line Options](#) or [Starting Parallel ANSYS FLUENT on a Linux System Using Command Line Options](#) in the User's Guide.

`-pcheck` checks the network connections before spawning compute nodes (Linux only).

`-tx` specifies that `x` processors are to be used. For more information about starting the parallel version of ANSYS FLUENT, see [Starting Parallel ANSYS FLUENT on a Windows System](#) or [Starting Parallel ANSYS FLUENT on a Linux System](#) in the User's Guide.

4.1.4.4. Postprocessing Option

`fluent -post` will run a version of FLUENT that allows you to set up a problem or perform postprocessing, but will not allow you to perform calculations. Running ANSYS FLUENT for pre- and postprocessing requires you to use the `-post` flag on startup. To use this option on Linux, launch ANSYS FLUENT by adding the `-post` flag after the version number, for example,

```
fluent 3d -post
```

To use this same feature from the graphical interface on Windows or Linux, select the **Pre/Post** option in the **General** tab of FLUENT Launcher, as described in [Starting ANSYS FLUENT Using FLUENT Launcher](#).

4.1.4.5. SGE Options

The `-sge` option runs ANSYS FLUENT under Sun Grid Engine (SGE) software, and allows you to use the features of this software to manage your distributed computing resources. Other options that can be employed in conjunction with `-sge` are `-sge`, `requested`, `-sgeq queue`, `-sgeckpt ckpt_obj`, and `-sgepe fluent_pe min_n-max_n`.

For a detailed explanation of these options, see [Running FLUENT Under SGE](#).

4.1.4.6. LSF Option

The `-lsf` option allows you to run ANSYS FLUENT under Platform Computing Corporation's LSF software, and thereby take advantage of the checkpointing features of that load management tool. For further details about using the `-lsf` option, see [Running FLUENT Under LSF](#).

4.1.4.7. Version and Release Options

Typing `fluent version -r`, replacing `version` with the desired version, will list all releases of the specified version.

`fluent -rx` will run release `x` of ANSYS FLUENT. You may specify a version as well, or you can wait and specify the version when prompted.

`fluent -env` will list all environment variables before running ANSYS FLUENT.

4.1.4.8. System Coupling Options

The following command line options (in either Windows or Linux) can be used when ANSYS FLUENT is involved in a system coupling simulation.

`-schostr=x` (where `x` is the name of the host machine, in quotes) specifies the host machine on which the coupling service is running (to which the co-simulation participant/solver must connect).

`-schoport=y` (where `y` is the port number) specifies the port on the host machine upon which the coupling service is listening for connections from co-simulation participants.

`-schofname=z` (where `z` is the name of the participant, in quotes) specifies the unique name used by the co-simulation participant to identify itself to the coupling service (see [System Coupling Server File \(`scServer.scs`\)](#) in the [System Coupling Guide](#) for more information).

The general syntax for invoking ANSYS FLUENT for system coupling is:

```
fluent 3d -schostr=host name in quotes -schoport=port number -schofname=name of the solver in quotes
```

For instance:

```
fluent 3d -schostr="machine1.domain.com" -schoport=1234 -schofname="Solution1"
```

Once ANSYS FLUENT loads and initializes the case, start the system coupling by typing the following command in the ANSYS FLUENT text user interface (TUI):

```
(sc-solve)
```

For more information, see [Performing System Coupling Simulations Using FLUENT in Workbench](#) in the [FLUENT in Workbench User's Guide](#), as well as the [System Coupling Guide](#).

4.1.4.9. Other Startup Options

There are other startup options that are not listed when you type the `fluent -help` command. These options can be used to customize your graphical user interface. For example, to change the ANSYS FLUENT window size and position you can either modify the `.Xdefaults` file (Linux only) described in [Customizing the Graphical User Interface \(Linux Systems Only\)](#) in the [User's Guide](#), or you can simply type the following command at startup:

```
fluent [version] [-geometry] [XXxYY+00-50]
```

where `XX` and `YY` are the width and height in pixels, respectively, and `+00-50` is the position of the window.

Therefore, typing `fluent 3d -geometry 700x500+20-400` will start the 3D version of ANSYS FLUENT, sizing the ANSYS FLUENT console to 700x500 pixels and positioning it on your monitor screen at `+20-400`.

4.2. Running ANSYS FLUENT in Batch Mode

ANSYS FLUENT can be used interactively, with input from and display to your computer screen, or it can be used in a batch or background mode in which inputs are obtained from and outputs are stored in files. Generally you will perform problem setup, initial calculations, and postprocessing of results in an interactive mode. However, when you are ready to perform a large number of iterative calculations, you may want to run ANSYS FLUENT in batch or background mode. This allows the computer resources to be prioritized, enables you to control the process from a file (eliminating the need for you to be present during the calculation), and also provides a record of the calculation history (residuals) in an output file. While the procedures for running ANSYS FLUENT in a batch mode differ depending on your computer operating system, [Background Execution on Linux Systems \(p. 31\)](#) provides guidance for running in batch/background on Linux systems, and [Background Execution on Windows Systems \(p. 32\)](#) provides guidance for running in batch/background on Windows systems.

For additional information, see the following sections:

- [4.2.1. Background Execution on Linux Systems](#)
- [4.2.2. Background Execution on Windows Systems](#)
- [4.2.3. Batch Execution Options](#)

4.2.1. Background Execution on Linux Systems

To run ANSYS FLUENT in the background in a C-shell (csh) on a Linux system, type a command of the following form at the system-level prompt:

```
fluent 2d -g < inputfile > & outputfile &
```

or in a Bourne/Korn-shell, type:

```
fluent 2d -g < inputfile > outputfile 2>&1 &
```

In these examples,

- `fluent` is the command you type to execute ANSYS FLUENT interactively.
- `-g` indicates that the program is to be run without the GUI or graphics (see [Starting ANSYS FLUENT \(p. 17\)](#)).
- `inputfile` is a file of ANSYS FLUENT commands that are identical to those that you would type interactively.
- `outputfile` is a file that the background job will create and which will contain the output that ANSYS FLUENT would normally print to the screen (for example, the menu prompts and residual reports).
- `&` tells the Linux system to perform this task in background and to send all standard system errors (if any) to `outputfile`.

The file `inputfile` can be a journal file created in an earlier ANSYS FLUENT session, or it can be a file that you have created using a text editor. In either case, the file must consist only of text interface commands (since the GUI is disabled during batch execution). A typical `inputfile` is shown below:

```
; Read case file
rc example.cas
; Initialize the solution
/solve/initialize/initialize-flow
; Calculate 50 iterations
it 50
; Write data file
wd example50.dat
; Calculate another 50 iterations
```

```
it 50
; Write another data file
wd example100.dat
; Exit FLUENT
exit
yes
```

This example file reads a case file `example.cas`, initializes the solution, and performs 100 iterations in two groups of 50, saving a new data file after each 50 iterations. The final line of the file terminates the session. Note that the example input file makes use of the standard aliases for reading and writing case and data files and for iterating. (`it` is the alias for `/solve/iterate`, `rc` is the alias for `/file/read-case`, `wd` is the alias for `/file/write-data`, etc.) These predefined aliases allow you to execute commonly used commands without entering the text menu in which they are found. In general, ANSYS FLUENT assumes that input beginning with a `/` starts in the top-level text menu, so if you use any text commands for which aliases do not exist, you must be sure to type in the complete name of the command (for example, `/solve/initialize/initialize-flow`). Note also that you can include comments in the file. As in the example above, comment lines must begin with a `;` (semicolon).

An alternate strategy for submitting your batch run, as follows, has the advantage that the `outputfile` will contain a record of the commands in the `inputfile`. In this approach, you would submit the batch job in a C-shell using:

```
fluent 2d -g -i inputfile >& outputfile &
```

or in a Bourne/Korn-shell using:

```
fluent 2d -g -i inputfile > outputfile 2>&1 &
```

4.2.2. Background Execution on Windows Systems

To run ANSYS FLUENT in the background on a Windows system, the following commands can be used:

```
fluent 3d -g -i journal
fluent 3d -g -wait -i journal
fluent 3d -hidden -i journal
```

In these examples,

- `fluent` is the command you type to execute ANSYS FLUENT interactively.
- `-g` indicates that the program is to be run minimized in the task bar.
- `-i journal` reads the specified journal file.
- `-wait` is the command you type in a DOS batch file or some other script in a situation where the script must wait until ANSYS FLUENT has completed its run.
- `-hidden` is similar to the `-wait` command, but also executes ANSYS FLUENT completely hidden and noninteractively.

To get an output (or transcript) file while running ANSYS FLUENT in the background on a Windows system, the journal file must contain the following command to write a transcript file:

```
; start transcript file
/file/start-transcript outputfile.trn
```

where the `outputfile` is a file that the background job will create and which will contain the output that ANSYS FLUENT would normally print to the screen (for example, the menu prompts and residual reports).

See [Creating and Reading Journal Files](#) in the *User's Guide* for details about journal files. See [Creating Transcript Files](#) in the *User's Guide* for details about transcript files.

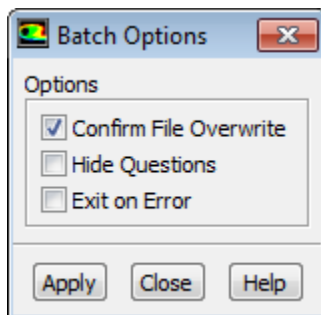
4.2.3. Batch Execution Options

During a typical session, ANSYS FLUENT may require feedback from you in the event of a problem it encounters. ANSYS FLUENT usually communicates problems or questions through the use of **Error** dialog boxes, **Warning** dialog boxes, or **Question** dialog boxes. While executing ANSYS FLUENT in batch mode, you may want to suppress this type of interaction in order to, for example, create journal files more easily.

There are three common batch configuration options available to you when running ANSYS FLUENT in batch mode. You can access these options using the **Batch Options** dialog box ([Figure 4.7: The Batch Options Dialog Box](#) (p. 33)).

File → **Batch Options...**

Figure 4.7: The Batch Options Dialog Box



The **Batch Options** dialog box contains the following items:

Confirm File Overwrite

determines whether ANSYS FLUENT confirms a file overwrite. This option is turned on by default.

Hide Questions

allows you to hide **Question** dialog boxes. This option is turned off by default.

Exit on Error

allows you to automatically exit from batch mode when an error occurs. This option is turned off by default.

Note that these options are also available in the `file/set-batch-options` command in the text interface.

`file` → `set-batch-options`


Any combination of these options can be turned on or off at any given time prior to running in batch mode.

Important

Batch option settings are *not* saved with case files. They are meant to apply for the duration of the current ANSYS FLUENT session only. If you read in additional mesh or case files during this session, the batch option settings will not be altered. As batch options are not saved with case files, journal files developed for use in batch mode should begin by enabling the desired batch option settings (if different from the default settings).

4.3. Switching Between Meshing and Solution Modes

You can switch from the meshing mode of FLUENT to the solution mode by clicking the **Switch to**

Solution  button, located by default in the top left corner of the application window. The mesh from your meshing mode session will be transferred and read in the new solution mode session.

You can switch from the solution mode of FLUENT to the meshing mode by using the `switch-to-meshing-mode` text command. Note that this text command is only available for 3D sessions, before you have read a mesh or case file.

4.4. Checkpointing an ANSYS FLUENT Simulation

The checkpointing feature of ANSYS FLUENT allows you to save case and data files while your simulation is running. While similar to the autosave feature of ANSYS FLUENT ([Automatic Saving of Case and Data Files](#) in the [User's Guide](#)), which allows you to save files throughout a simulation, checkpointing allows you slightly more control in that you can save an ANSYS FLUENT job even after you have started the job and did not set the autosave option. Checkpointing also allows you to save case and data files and then exit out of ANSYS FLUENT. This feature is especially useful when you need to stop an ANSYS FLUENT job abruptly and save its data.

There are two different ways to checkpoint an ANSYS FLUENT simulation, depending upon how the simulation has been started.

1. ANSYS FLUENT running under LSF or SGE

ANSYS FLUENT is integrated with load management tools like LSF and SGE. These two tools allow you to checkpoint any job running under them. You can use the standard method provided by these tools to checkpoint the ANSYS FLUENT job.

For more information on using ANSYS FLUENT and SGE or LSF, see [Running FLUENT Under SGE](#) or [Running FLUENT Under LSF](#), respectively.

2. Independently running ANSYS FLUENT

When not using tools such as LSF or SGE, a different checkpointing mechanism can be used when running an ANSYS FLUENT simulation. You can checkpoint an ANSYS FLUENT simulation while iterating/time-stepping, so that ANSYS FLUENT saves the case and data files and then continues the calculation, or so that ANSYS FLUENT saves the case and data files and then exits.

- Saving case and data files and continuing the calculation:

On Linux, create a file called `check-fluent`, that is,

```
/tmp/check-fluent
```

On Windows, create a file called `check-fluent.txt`, that is,

```
C:\temp\check-fluent.txt
```

- Saving case and data files and exiting ANSYS FLUENT:

On Linux, create a file called `exit-fluent`, that is,

```
/tmp/exit-fluent
```

On Windows, create a file called `exit-fluent.txt`, that is,

```
C:\temp\exit-fluent.txt
```

The saved case and data files will have the current iteration number appended to their file names.

ANSYS FLUENT offers an alternate way to checkpoint an unsteady simulation. While the default behavior is to checkpoint the simulation at the end of the current iteration, for unsteady simulations you have the option of completing all of the iterations in the current time-step before checkpointing. This can be set by entering the following Scheme command prior to running the unsteady simulation:

```
(ckpt/time-step?
 #t)
```

Now when you save the checkpoint file (as described previously), the case and data file will be saved at the end of the current time-step and named accordingly. To switch back to the default checkpointing mechanism at the end of the current iteration, use the following Scheme command:

```
(ckpt/time-step? #f)
```

Important

Note that the `(ckpt/time-step? #t)` command will have the effect only in the case of an unsteady simulation.

Note

It is recommended that you do *not* use checkpointing when using ANSYS FLUENT in Workbench. However, if checkpointing is necessary, the `exit-fluent/exit-fluent.txt` file can be used and the file will be checked in its default location (the `FFF/FLU` system directory containing the `*.set` file). If ANSYS FLUENT is calculating, then the existence of the file is equivalent to an **interrupt** command. Similarly, the `check-fluent/check-fluent.txt` file can be used to save the project on demand when ANSYS FLUENT is calculating.

4.5. Cleaning Up Processes From an ANSYS FLUENT Simulation

ANSYS FLUENT lets you easily remove extraneous processes in the event that an ANSYS FLUENT simulation needs to be stopped.

When a session is started, ANSYS FLUENT creates a `cleanup-fluent` script file. The script can be used to clean up all ANSYS FLUENT-related processes. ANSYS FLUENT creates the cleanup-script file in the current working folder with a file name that includes the machine name and the process identification number (PID) (for example, `cleanup-fluent-mymachine-1234`).

If the current directory does not possess the proper write permissions, then ANSYS FLUENT will write the cleanup-script file to your home directory.

If, for example, ANSYS FLUENT is started on a machine called `thor` and the process identification number is `32895`, ANSYS FLUENT will create a cleanup-script called `cleanup-fluent-thor-32895` in the current folder. To run the cleanup-script, and clean up all ANSYS FLUENT processes related to your session, on Linux platforms, type the following command in the console window:

```
sh cleanup-fluent-thor-32895
```

Or, if the shell script already has executable permissions, simply type:

```
cleanup-fluent-thor-32895
```

To clean up extraneous ANSYS FLUENT processes on Windows (serial or parallel), double-click the corresponding batch file (for example, `cleanup-fluent-thor-32895.bat`) that ANSYS FLUENT generates at the beginning of each session.

Important

During a normal run, this file will be deleted automatically after exiting ANSYS FLUENT. In abnormal situations, you may use this batch file to clean up the ANSYS FLUENT processes. Once an ANSYS FLUENT session has been closed, you can safely delete any left over cleanup scripts from your working folder.

Important

If an ANSYS FLUENT session hangs or freezes on Windows, and you want to view the complete contents of the ANSYS FLUENT console output in a transcript file, you should use the `taskkill` command through the DOS command prompt, rather than terminating the ANSYS FLUENT application through the **Windows Task Manager**.

4.6. Exiting ANSYS FLUENT

You can exit ANSYS FLUENT by selecting **Exit** in the **File** pull-down menu. If the present state of the program has not been written to a file, a **Question** dialog box will open to confirm if you want to proceed. You can cancel the exit and write the appropriate file(s) or you can continue to exit without saving the case or data.

Glossary of Terms

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

- adaption (p. 37)
- case files (p. 37)
- cell types (p. 38)
- computational fluid dynamics (CFD) (p. 38)
- console (p. 38)
- convergence (p. 38)
- cortex (p. 39)
- data files (p. 39)
- dialog boxes (p. 39)
- discretization (p. 39)
- GUI (p. 39)
- mesh (p. 39)
- models (p. 39)
- node (p. 39)
- postprocessing (p. 39)
- residuals (p. 39)
- skewness (p. 39)
- solvers (p. 39)
- terminal emulator (p. 39)
- TUI (p. 40)

adaption

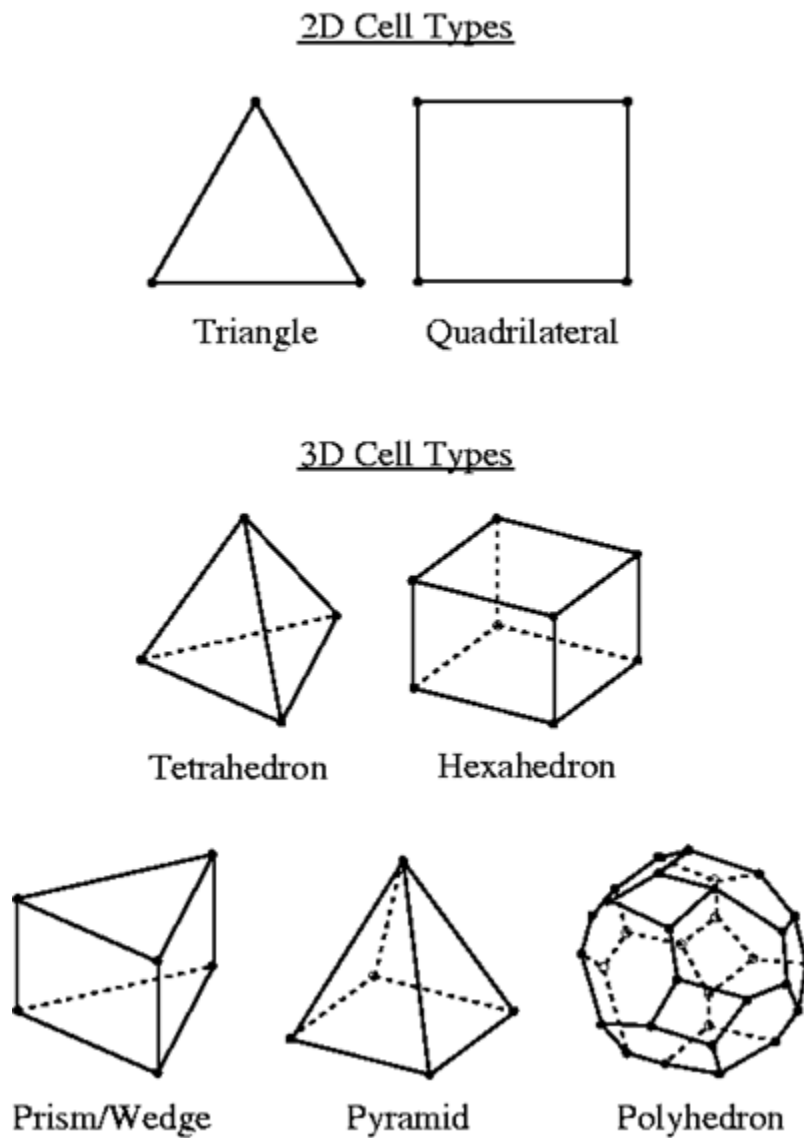
A technique useful in improving overall mesh quality. The solution-adaptive mesh refinement feature of ANSYS FLUENT allows you to refine and/or coarsen your mesh based on geometric and numerical solution data. In addition, ANSYS FLUENT provides tools for creating and viewing adaption fields customized 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 10: 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 (for example, combustion), mechanical movement (for example, 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 (for example, turbulence).
node	The distinct points of a mesh (p. 39) 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. 38) .

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