



ANSYS Fluent Getting Started Guide



ANSYS, Inc.
Southpointe
2600 ANSYS Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
<http://www.ansys.com>
(T) 724-746-3304
(F) 724-514-9494

Release 17.0
January 2016

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

Copyright and Trademark Information

© 2015 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, AIM 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, 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	vii
1. Introduction to ANSYS Fluent	1
1.1. Program Capabilities	4
1.2. ANSYS Fluent Documentation	5
1.2.1. Accessing the Documentation Files Using the ANSYS Help Viewer	5
1.2.2. Accessing the PDF Documentation	6
1.2.2.1. Downloading and Extracting the PDF Documentation Files	7
1.2.2.2. Navigating the PDF Files	7
1.2.2.3. Printing the PDF Files	7
1.3. Known Limitations in ANSYS Fluent 17.0	7
2. Basic Steps for CFD Analysis using ANSYS Fluent	17
2.1. Steps in Solving Your CFD Problem	17
2.2. Planning Your CFD Analysis	17
3. Guide to a Successful Simulation Using ANSYS Fluent	23
4. Starting and Executing ANSYS Fluent	25
4.1. Starting ANSYS Fluent	25
4.1.1. Starting ANSYS Fluent Using Fluent Launcher	25
4.1.1.1. Setting General Options in Fluent Launcher	27
4.1.1.2. Single-Precision and Double-Precision Solvers	29
4.1.1.3. Setting Parallel Options in Fluent Launcher	30
4.1.1.4. Setting Remote Options in Fluent Launcher	31
4.1.1.5. Setting Scheduler Options in Fluent Launcher	32
4.1.1.6. Setting Environment Options in Fluent Launcher	34
4.1.2. Starting ANSYS Fluent on a Windows System	35
4.1.3. Starting ANSYS Fluent on a Linux System	36
4.1.4. Command Line Startup Options	36
4.1.4.1. Graphics Options	38
4.1.4.2. Meshing Mode Option	39
4.1.4.3. Parallel Options	39
4.1.4.4. Postprocessing Option	39
4.1.4.5. SGE Options	40
4.1.4.6. LSF Option	40
4.1.4.7. Version and Release Options	40
4.1.4.8. System Coupling Options	40
4.1.4.9. Other Startup Options	41
4.2. Running ANSYS Fluent in Batch Mode	41
4.2.1. Background Execution on Linux Systems	41
4.2.2. Background Execution on Windows Systems	43
4.2.3. Batch Execution Options	43
4.3. Switching Between Meshing and Solution Modes	44
4.4. Checkpointing an ANSYS Fluent Simulation	45
4.5. Cleaning Up Processes From an ANSYS Fluent Simulation	46
4.6. Exiting ANSYS Fluent	47
Glossary of Terms	49

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 \(p. 17\)](#), describes the steps involved in solving a CFD problem and questions to consider.
- [Guide to a Successful Simulation Using ANSYS Fluent \(p. 23\)](#), provides specific guidelines that help ensure your CFD simulation is a success.
- [Starting and Executing ANSYS Fluent \(p. 25\)](#), describes options and alternatives to starting, running, and exiting ANSYS Fluent. It also provides instructions for remote execution and batch execution.
- [Glossary of Terms \(p. 49\)](#), 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, output 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 Migration Manual](#) contains information about transitioning from the previous release of Fluent Meshing, including descriptions of new features and text command list changes.
- [ANSYS Fluent Meshing User's Guide](#) contains detailed information about creating 3D meshes using the meshing mode of Fluent.

Related video help can be found on the [ANSYS How To Videos](#) page.

- [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 Customization 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.

The latest updates of the ANSYS Fluent tutorials are available on the ANSYS Customer Portal. To access tutorials and their input files on the ANSYS Customer Portal, go to <http://support.ansys.com/training>.

- The latest updates of the ANSYS Fluent Meshing tutorials are available on the ANSYS Customer Portal. To access tutorials and their input files on the ANSYS Customer Portal, go to <http://support.ansys.com/training>.
- **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.
- **ANSYS Fluent Advanced Add-On Modules** contains information about the usage of the different advanced Fluent add-on modules, which are applicable for specific modeling needs.
 - **Part I: ANSYS Fluent Adjoint Solver** 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.
 - **Part II: ANSYS Fluent Battery Module** contains information about the background and usage of Fluent's Battery Module that allows you to analyze the behavior of electric batteries.
 - **Part III: ANSYS Fluent Continuous Fiber Module** 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.
 - **Part V: ANSYS Fluent Fuel Cell Modules** 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.
 - **Part VI: ANSYS Fluent Magnetohydrodynamics (MHD) Module** 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.
 - **Part VII: ANSYS Fluent Population Balance Module** 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 ANSYS Fluent Using a Load Manager](#) contains information about using third-party load managers with ANSYS Fluent.
 - [Part I: Running ANSYS Fluent Under LSF](#) contains information about using Fluent with Platform Computing's LSF software, a distributed computing resource management tool.
 - [Part II: Running ANSYS Fluent Under PBS Professional](#) contains information about using Fluent with Altair PBS Professional, an open workload management tool for local and distributed environments.
 - [Part III: Running ANSYS Fluent Under SGE](#) contains information about using Fluent with Univa Grid Engine (formerly Sun Grid Engine) 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. To access documentation files on the ANSYS Customer Portal, go to <http://support.ansys.com/documentation>.

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 **Contacts > 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 > Customer Portal**. 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. To use this feature, enter relevant text (error message, etc.) in the Knowledge Resources Search box and click the magnifying glass icon. These Knowledge Resources provide solutions and guidance on how to resolve installation and licensing issues quickly.

NORTH AMERICA

All ANSYS Products except Esterel, Apache and Reaction Design products

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

Toll-Free Telephone: 1.800.711.7199 (Please have your Customer or Contact ID ready.)

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://support.ansys.com>) and select the appropriate option.

National Toll-Free Telephone: (Please have your Customer or Contact ID ready.)

German language: 0800 181 8499

English language: 0800 181 1565

Austria: 0800 297 835

Switzerland: 0800 564 318

International Telephone: (Please have your Customer or Contact ID ready.)

German language: +49 6151 152 9981

English language: +49 6151 152 9982

Email: support-germany@ansys.com

UNITED KINGDOM

All ANSYS Products

Web: Go to the ANSYS Customer Portal (<http://support.ansys.com>) 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 and Mechanical Products

Telephone: +81-3-5324-7305

Email:

Mechanical: japan-ansys-support@ansys.com

Fluent: japan-fluent-support@ansys.com;

CFX: japan-cfx-support@ansys.com;

Polyflow: japan-polyflow-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 Products

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

Telephone: +91 1 800 209 3475 (toll free) or +91 20 6654 3000 (toll) (Please have your Customer or Contact ID ready.)

FRANCE**All ANSYS Products**

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

Toll-Free Telephone: +33 (0) 800 919 225 **Toll Number:** +33 (0) 170 489 087 (Please have your Customer or Contact ID ready.)

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://support.ansys.com>) and select the appropriate option.

Toll-Free Telephone: (0) 800 777 83 **Toll Number:** +32 2 620 0152 (Please have your Customer or Contact ID ready.)

Email: support-benelux@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://support.ansys.com>) and select the appropriate option.

Telephone: +46 (0) 10 516 49 00

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://support.ansys.com>) and select the appropriate option.

Spain: Toll-Free Telephone: 900 933 407 **Toll Number:** +34 9178 78350 (Please have your Customer or Contact ID ready.)

Portugal: Toll-Free Telephone: 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://support.ansys.com>) and select the appropriate option.

Toll-Free Telephone: 800 789 531 **Toll Number:** +39 02 00621386 (Please have your Customer or Contact ID ready.)

Email: support-italy@ansys.com

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

TAIWAN, REPUBLIC OF CHINA

Telephone: 866 22725 5828

KOREA

Telephone: 82-2-3441-5000

CHINA

Toll-Free Telephone: 400 819 8999 **Toll Number:** +86 10 82861715

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

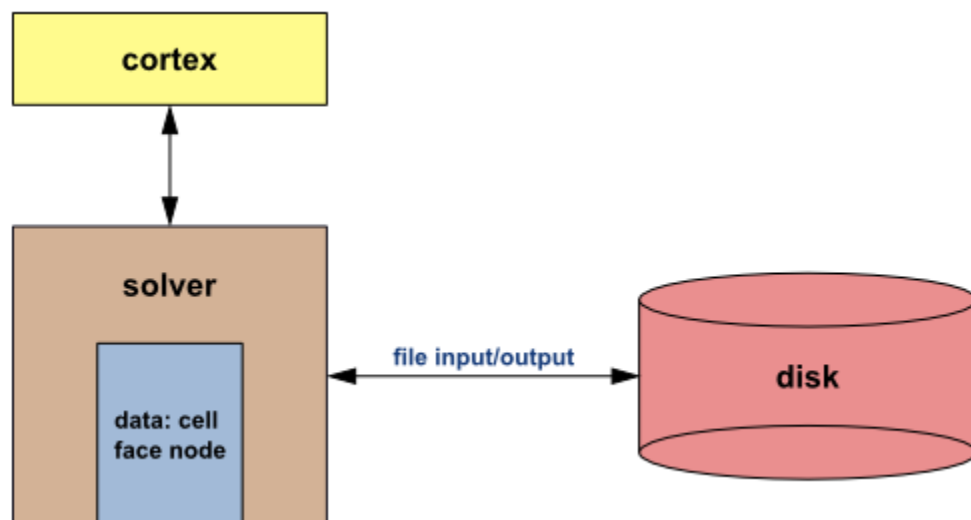
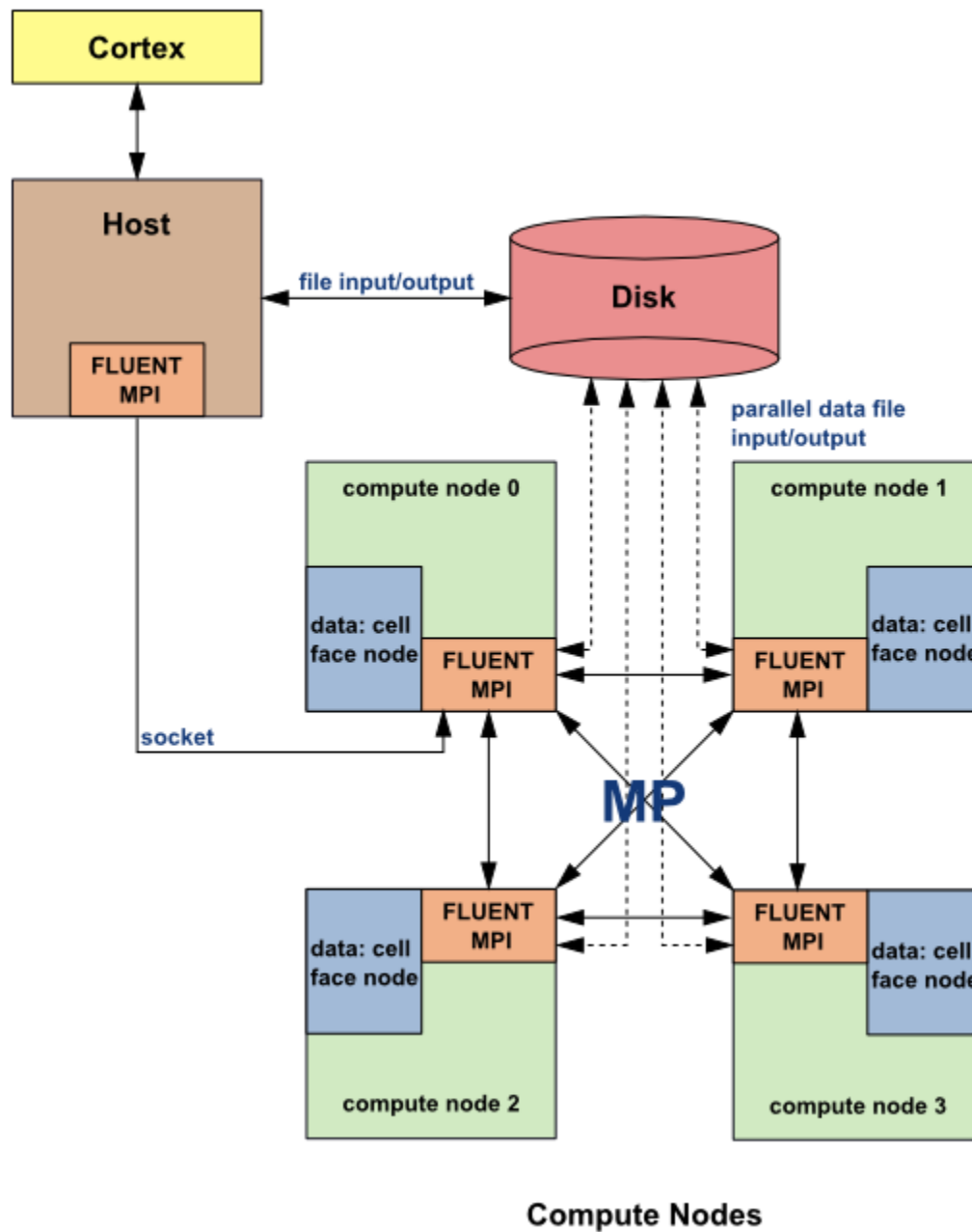


Figure 1.2: Parallel ANSYS Fluent Architecture



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](#)
- [1.3. Known Limitations in ANSYS Fluent 17.0](#)

1.1. Program Capabilities

When in meshing mode, ANSYS Fluent functions as a robust unstructured-volume-mesh generator (see [Meshing Mode 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, 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
- Adjoint solver module (documented separately)
- Battery module (documented separately)
- Continuous fiber module (documented separately)
- Fuel cell modules (documented separately)
- Magnetohydrodynamics (MHD) module (documented separately)
- Population balance module (documented separately)
- Fluent as a Server (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 or through PDF files. 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. The PDF files can be obtained from the ANSYS Customer Portal.

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.

1.2.1. Accessing the Documentation Files Using the ANSYS Help Viewer

1.2.2. Accessing the PDF Documentation

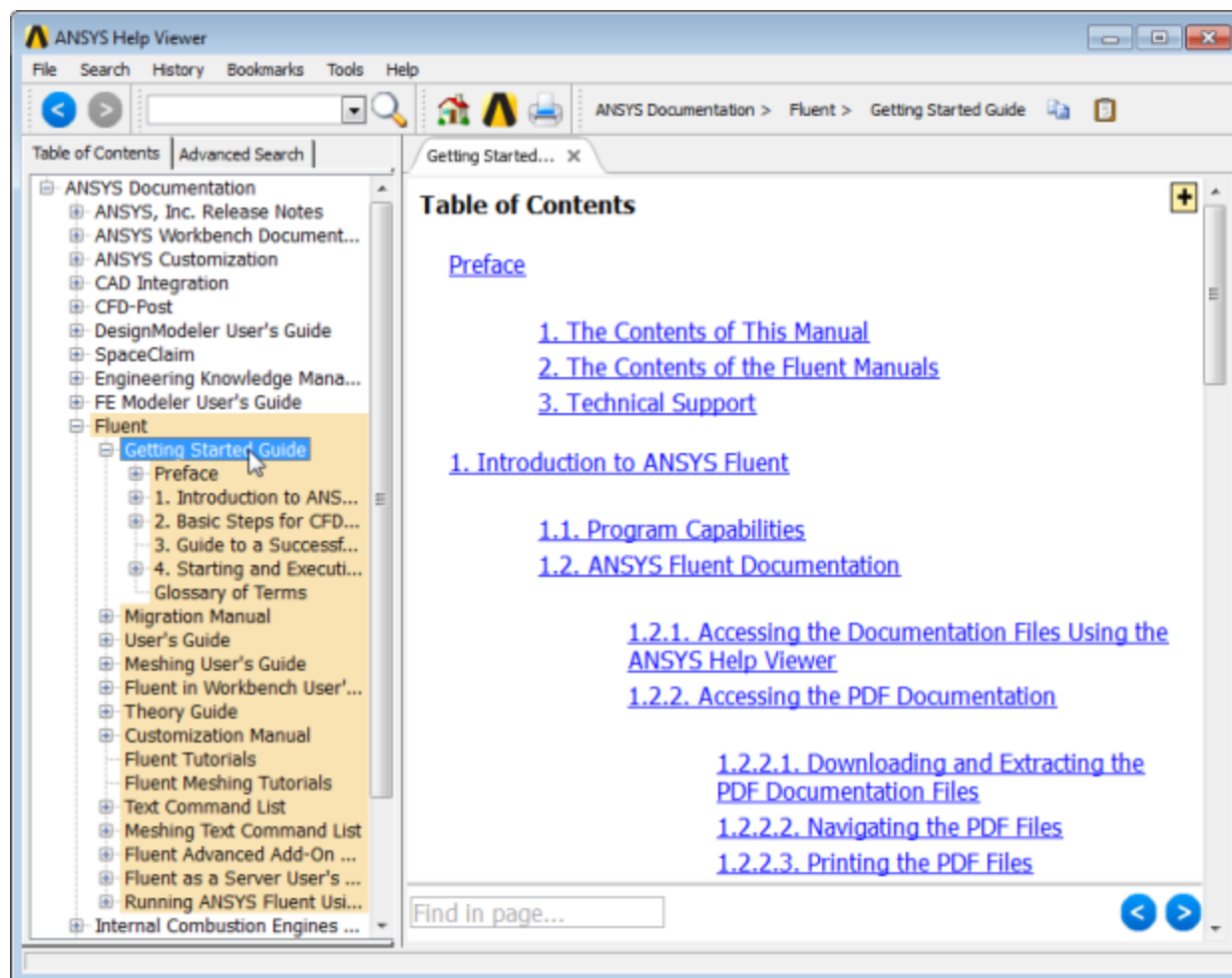
1.2.1. Accessing the Documentation Files Using the ANSYS Help Viewer



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

Start > Program Files > ANSYS 17.0 > Help > ANSYS Help 17.0

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  next to **Fluent**.
3. To display the table of contents for the selected document, click the document title.
4. To access a specific content, you can do one of the following:
 - In the tree, click  next to a title to expand the tree hierarchy and then click the title of the desired section.
 - In the Table of Contents, click an item to display the corresponding information.
5. To find specific information, use a basic search window (on the toolbar) or advanced search feature. See [Searching in the Using Help](#) for details.

For additional information on the ANSYS Help Viewer, see [Using Help](#).

1.2.2. Accessing 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. Downloading and Extracting the PDF Documentation Files

1.2.2.2. Navigating the PDF Files

1.2.2.3. Printing the PDF Files

1.2.2.1. Downloading and Extracting the PDF Documentation Files

You can download the PDF documentation from the ANSYS Customer Portal (support.ansys.com/doc-downloads).

1.2.2.2. 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.

1.2.2.3. 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**.

1.3. Known Limitations in ANSYS Fluent 17.0

This section lists limitations that are known to exist in ANSYS Fluent. Where possible, suggested work-arounds are provided.

- File import/export (for a list of supported files, refer to the table in this section, under **Third-party software**)
 - If you change the **File Storage Options** settings in the **Autosave** dialog box, the solution history will be lost.
 - Data export to Mechanical APDL result file is not available on the linux64 platform. (Mechanical APDL data export to .cdb file is available on all platforms.)
 - When exporting EnSight Case Gold files for transient simulations, the solver cannot be switched between serial and parallel, and the number of compute nodes cannot be changed for a given parallel run. Otherwise, the exported EnSight Case Gold files for each time step will not be compatible.
 - EnSight export with topology changes is not supported.
 - To properly view Fieldview Unstructured (.fvuns) results from a parallel ANSYS Fluent simulation
 - Mesh files must be exported from the parallel solver via the TUI command `fieldview-unstruct-grid`.
 - Mesh and data files should all be exported from parallel ANSYS Fluent sessions with the same number of nodes.
 - Tecplot file import does not support the Tecplot360 file format.

- Mesh
 - Boundary zone extrusion is not possible from faces that have hanging nodes.
 - For simulations that involve the Fluent, Mechanical, and Meshing applications, meshing problems can arise in instances where there are multiple regions and contacts between them. In Fluent, a zone can only exist in a single contact region. The Mechanical and Meshing applications both use a different approach concerning contact regions when compared to Fluent.
 - ANSYS Fluent does not support FSI data mapping of edges and, therefore, it is not supported in 2D.
 - Dynamic gradient adaption based on custom field functions is not supported when running ANSYS Fluent in parallel.
- Models
 - ANSYS Fluent supports the Chemkin II format for Oppdif flamelet import only.
 - The surface-to-surface (S2S) radiation model does not work with moving/deforming meshes.
 - The DPM work pile algorithm is not compatible with the wall film boundary condition.
 - The shell conduction model is not applicable on moving walls.
 - The heat exchanger model is not compatible with mesh adaption.
 - The Fluent/ANSYS Reaction Design KINetics coupling is not available on the win64 platform.
 - DO-Energy coupling is recommended for large optical thickness cases (> 10) only.
 - FMG initialization is not available with the shell conduction model.
 - FMG initialization is not compatible with the unsteady solver.
 - The MHD module is not compatible with Eulerian multiphase models.
 - Bounded 2nd order discretization in time is not compatible with moving and deforming meshes.
 - When simulating porous media, the value of the **Porosity** (defined in the **Fluid** dialog box) cannot be 0 or 1 (that is, it must be in between these values) if the non-equilibrium thermal model is enabled.
 - When simulating porous media, the non-equilibrium thermal model is not supported with radiation and/or multiphase models.
 - The junction of a wall with shell conduction enabled and a non-conformal coupled wall is not supported. Such a junction will not be thermally connected, that is, there will be no heat transfer between the shell and the mesh interface wall.
 - After you enable the Eulerian Wall Film model, Fluent will not allow you to save the mesh modifications, such as separating cells, extruding face zones, and changing the cell zones type. If you want to modify the mesh in Fluent, be sure to complete all mesh operations prior to enabling the Eulerian Wall Film model.
 - The Transition SST model (also known as the γ - Re_θ model) is not Galilean invariant and should therefore not be applied to surfaces that move relative to the coordinate system for which the velocity field is

computed; for such cases, the Intermittency Transition model (also known as the γ model) should be used instead.

- In simulations that use the discrete phase model, particle mass may be lost when simulation of transient particles released with constant parcel size is combined with auto-save of case files.
- Use of the Lagrangian wall-film model in combination with hanging-node adaption in a parallel Fluent session may cause the Fluent session to fail.
- User-defined wall functions are not compatible with the Eulerian multiphase formulation and cannot be used.
- The view factor files generated as part of a surface-to-surface radiation model calculation for version 16.0 or 16.1 of Fluent may not be compatible with newer versions if the **Matching** option was enabled for a mapped interface. For any case file with such a setup, you must recompute the view factors in the newer version to ensure correct results.
- Parallel processing
 - These features are currently unavailable in the parallel solver:
 - Discrete transfer radiation model (DTRM)
 - Data export to non-native formats other than EnSight, FIELDVIEW, Tecplot, and the generic heat flux data file
 - Mellanox OFED version 1.5.3–1.0.0 is known to cause random crashes or startup issues in Fluent when using core counts greater than approximately 64. The last known stable Mellanox version is 1.5.2. You may use either of the following as a workaround:
 - Revert to OFED 1.5.2.
 - Set the value of `log_num_mtt` to 24 in the `mlx4_core` driver on all machines. For more information refer to Solution #2024818 from the Knowledge Resources at the ANSYS Customer Portal ([ANSYS Customer Portal](#)).
 - Platform MPI does not support running in parallel in an overload condition (using more cores than are physically available on a machine). For performance reasons, running in an overload condition is generally not recommended. However, if it is required to run overloaded, then you can use Intel MPI (`-mpi=intel`) or Open MPI (`-mpi=openmpi`).
 - The version of Platform MPI that is distributed with Fluent is now limited to a maximum of 4096 processes for a single simulation. For higher core counts, an additional license is required. Refer to [IBM Platform MPI with High \(>4096\) Process Count in the Configuring High Performance Computing Guide](#) for more information.
 - The PARALLEL INDEPENDENT mode for Hierarchical Data Format file I/O is known to exhibit slow write performance. On parallel file systems, consider using the PARALLEL COLLECTIVE mode when writing HDF files. On other network file systems, consider using the HOST or NODE0 mode.
 - Note that on systems using large pages for memory allocation (such as Cray), the virtual memory usage reported by Fluent may be much higher than actual memory used. In this case resident memory (also reported by Fluent) is a more reliable guide.
 - Degenerate contact points are known to cause topological mesh connectivity issues in parallel. Degenerate contact points are nodes that are shared by 2 or more cell zones that do not share faces. Fluent can detect

degenerate contact points in serial via a mesh check with the mesh check verbosity set greater than 0. If a case has generate contact points then you must use the following command before reading the case for proper parallel handling of such contact points.

```
(rpsetvar 'parallel/add-dgcp-to-int-or-corner?)
```

- The Eulerian Wall-film model is not compatible with the DPM Domain option of the hybrid parallel DPM tracking. For such model combination, the **Use DPM Domain** option must be disabled in the **Parallel** tab of the **Discrete Phase Model** dialog box.

- Platform support and drivers

- On Windows Server OS, ANSYS Fluent supports only MS MPI for parallel runs. Installing any other unsupported MPI libraries (Platform MPI or Intel MPI) will result in conflicts.
- On Windows 7 and later, installing ANSYS Fluent on any drive other than C: may result in issues arising from spaces in the pathname not getting converted to short file names. This is the result of a change in the default value for `NtfsDisable8dot3NameCreation` starting with Windows 7. If you need to install ANSYS Fluent on any drive other than C: you must run the following command *prior* to installing ANSYS Fluent:

```
fsutil 8dot3name set <driveletter> 0
```

where *<driveletter>* is the target drive letter including the colon (for example, D:).

- The minimum OS requirements for Linux are SLES 11 SP2 or RHEL 6.
- The pathname length to the cproep.so library (including the lib name) is limited to 80 characters. (Linux Opteron cluster using Infiniband interconnect only.)
- On Linux platforms, including a space character in the current working directory path is not supported.
- Visit the ANSYS Customer Portal ([ANSYS Customer Portal](#)) for the latest Windows graphics FAQ. Version 2.0 or higher of .NET Framework must be installed in order to run ANSYS Fluent on the winx64 platform.
- If you are installing ANSYS Fluent 17.0 on a Windows machine that already has one or more previous versions of ANSYS Fluent, then after installing Platform and Intel MPI libraries from the prerequisites, make sure to delete the environment variables MPI_ROOT (for Platform MPI) and I_MPI_ROOT (for Intel MPI). Otherwise there will be a conflict while running previous ANSYS Fluent versions in parallel mode.
- Remote Solver Facility (RSF) is no longer supported in ANSYS Fluent.
- Itanium platform (Inia64) is no longer supported.
- The coupling between Ansoft and Fluent in Workbench is not supported on Windows Vista (32 bit and 64 bit). On Linux, the coupling is supported for RHEL 6 and SUSE Linux Enterprise Server 11.
- When running ANSYS Fluent on a Linux machine directly or through a remote connection (such as NICE DCV or RealVNC), some dialog boxes may open in the background and be obscured from view. If you encounter this issue, you can define the environment variable `KEEP_DIALOG_ON_TOP`. This forces new dialog boxes to open in the foreground of your display.
- Fluent uses several TCP/IP ports for parallel communications and error handling. Port conflicts with other programs trying to use the same ports are handled by Fluent and generate warnings similar to the following:

```
428: mpt_accept: warning: incorrect exercise message "GET /" from 10.1.0.188 on port 56564
```

Long running large parallel sessions are more prone to generating such warnings, but these are generally safe for you to ignore.

- When using the default MPI (PCMPI) on some newer hardware the following message may appear when exiting the Fluent session.

```
hwloc has encountered what looks like an error from the operating system.
```

```
object (Socket cpuset 0xff00ffff) intersection without inclusion!
Error occurred in topology.c line 758
```

```
Please report this error message to the hwloc user's mailing list,
along with any relevant topology information from your platform.
```

This message is from the hardware locality library used by Platform MPI. You can set the following environment variable to hide this message, or switch to Intel MPI.

```
HWLOC_HIDE_ERRORS=1
```

- Remote display

- For proper mouse-button functionality with a two-button mouse when running ANSYS Fluent through a Remote Desktop Protocol (RDP) connection to a Windows 8 machine, define the environment variable CX_WIN8_MB_TWO on the remote machine.
- Connecting or disconnecting a VPN network while running a Fluent simulation will result in a failure because of changes to the network interface.

- Solver-Meshing

- The absolute and relative velocity formulations may yield different results in cases where a strong reversal of flow exists at a pressure outlet boundary.
- The non-iterative time advancement (NITA) solver is applicable with only a limited set of models. See the ANSYS Fluent User's Guide for more details.
- NITA (using fractional time step method) is not compatible with porous media.
- The following models are not available for the density-based solvers:
 - Volume-of-fluid (VOF) model
 - Multiphase mixture model
 - Eulerian multiphase model
 - Non-premixed combustion model
 - Premixed combustion model
 - Partially premixed combustion model
 - Composition PDF transport model
 - Soot model
 - Rosseland radiation model

- Melting/solidification model
- Enhanced Coherent Flamelet model
- Inert model: transport of inert species (EGR in IC engines)
- Dense discrete phase model
- Shell conduction model
- Floating operating pressure
- Spark ignition and auto-ignition models
- Physical velocity formulation for porous media
- Selective multigrid (SAMG)
- The pressure-based coupled solver is not available with the following features:
 - Fixed velocity
- On some Linux platforms, pressing **Ctrl+C** will not interrupt the solution. A suggested workaround is to use the checkpoint mechanism in Fluent to save files and/or exit Fluent. ([Checkpointing an ANSYS Fluent Simulation in the *Fluent Getting Started Guide*](#))
- In certain cases with tetrahedral or hybrid meshes, the use of the Least-Squares Cell Based gradient method in combination with the cell-to-cell limiter may cause divergence. If this is observed, it is recommended that you either change the gradient method to Green-Gauss Node Based or change the limiter type to the cell-to-face limiter.
- User-defined functions (UDFs)
 - Interpreted UDFs cannot be used while running in parallel with an Infiniband interconnect. The compiled UDF approach should be used in this case.
- Graphics, Reporting, and Postprocessing
 - Monitors may continue to print/plot values, even if the zones on which they are defined are deactivated.
 - If you are autosaving multiple scenes on a Windows machine, the **Headlight** lighting effect may inconsistently change its state (on/off). This can be avoided by rendering each scene in a separate graphics window.
 - When using the `opengl` display driver, displaying filled contours in a 2D case will cause Fluent to terminate abnormally if the surfaces selected in the Contours dialog box contain more than 2000 1D facets. As a workaround, you can use the `msw` driver (Windows) or `x11` driver (Linux) or disable **Filled** in the Contours dialog box.
 - If you experience any “hanging” / “freezing” issues or incomplete annotations with the **Mouse-Annotate** function on a Windows machine, you can define the environment variable `FLUENT_NATIVE_HOOPS_EVENTS=1`. (The environment variable must be specified prior to beginning a new Fluent session for it to take effect.)

- Mean and root-mean-squared-error (RMSE) quantities of custom field functions are only available for mixtures. In previous releases it was possible to specify these quantities for phases, which was an incorrect behavior. This behavior is no longer allowed in R16.0 or later releases. If you are running a pre-R16.0 case set to output such quantities in R17.0, you may get a segmentation error. To avoid the error, redefine the previously defined monitors reporting mean or RMSE quantities of phases.
- Fluent in Workbench
 - Coupling between Fluent and Q3D is not supported.
 - When the parallel version of Fluent or Icepak is coupled with HFSS, Maxwell 3D, or Q3D Extractor in Workbench, the solver may report imported surface losses that are slightly higher than the actual losses. The serial version of the solver does not have this limitation.
- Fluent as a Server
 - When running Fluent with the `-aaS` option, if you have a mesh with a very low cell count and have set a large number of iterations to be stored using the **Residual Monitors** dialog box (see [Storing Residual History Points in the Fluent User's Guide](#)), you will see a relative degradation of performance. Reducing the number of stored iterations will reduce this degradation.
- Third-party software
 - Fluent-Platform LSF integration is not supported on the MS Windows platform.
 - Fluent-SGE integration is supported only on Linux platforms.
 - Wave and GT-POWER coupling are available only with stand-alone ANSYS Fluent and not in the Workbench environment.
 - Supported versions of third party software are listed below:

Third Party Software	Supported Version
Abaqus	6.14 ODB Library: 6.8.2
Altair HYPERMESH	5.1
AVS	5.0
CGNS	3.1.3
CHEMKIN	2.4
Data Explorer	4.2
Ensignt	9.1.2
EnSight 6 (TUI only)	EnSight 6
EnSight Case Gold	9.1.2
FAST	1.3
Fieldview	13.2
GT-POWER	7.5
HOOPS	19.31
HP/Platform MPI	09.01.02.00
I-DEAS	IDEAS NX Series 11
Intel MPI	4.1.3.045*

Third Party Software	Supported Version
LSTC-DYNA	970.0
MPCCI	3.0.5
NASTRAN	Bulk data input file - MSC.NASTRAN 2010
NIST	7.2 + (9.1)
Open MPI	1.6.5
PATRAN	3.0
PTC MECHANICA	PTC/Mechanica Wildfire 4.0
Sundials	2.5.0
TECPLOT	Tecplot file format, version 11.2
VKI	4.3.0
WAVE	2014.1

* By default, Intel MPI does not support heterogeneous environments (that is, a mix of operating system versions or hardware) for Windows. As a workaround, you can use the following environment setting:

```
I_MPI_PLATFORM zero
```

- Other
 - In parallel, custom vectors cannot be created from components that are custom field functions themselves.
 - The IRIS Image and HPGL hard copy formats are no longer supported in ANSYS Fluent.
 - When using ANSYS Fluent with the Remote Solve Manager (RSM):
 - Only one copy of a saved project that is in the pending state can reconnect successfully.
 - System Coupling is not supported.
 - Ansoft Coupling is not supported.
 - UDFs are supported with limitations as detailed in [Submitting Fluent Jobs to RSM in Workbench User's Guide](#).
 - The turbo-averaged contour plot in turbomachinery postprocessing may give an unexpected contour region in a selected topology.
 - The **Inverse Distance** and **Least Squares** profile interpolation methods are not applicable when a profile is attached to cell zones.
 - Heat exchanger networks are not supported in HDF-formatted case and data files.
 - When opening the Help Viewer from Fluent in Linux, you may receive an error message in the Linux console. This can result when another user has created the installation and run Fluent, thus creating a registry file; if you then run this same installation, there will be a permissions conflict. As a workaround, remove the registry file:

```
path/ansys_inc/v170/Tools/mono/Linux64/etc/mono/registry
```


(where *path* is the directory in which you have placed the release directory). Then change the permissions for the Mono platform in order to remove write access from the directory:

```
path/ansys_inc/v170/Tools/mono/Linux64/etc/mono
```

- On Windows, mesh reading into a serial ANSYS Fluent session may fail if you use more than 20 million cells per core.
- For non-conformal interfaces, the use of the **Matching** option with the **Mapped** option is not recommended, especially if the extents of **Interface Zone 1** and **Interface Zone 2** do not coincide.

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 DesignModeler 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 use 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 the smallest flux through the 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

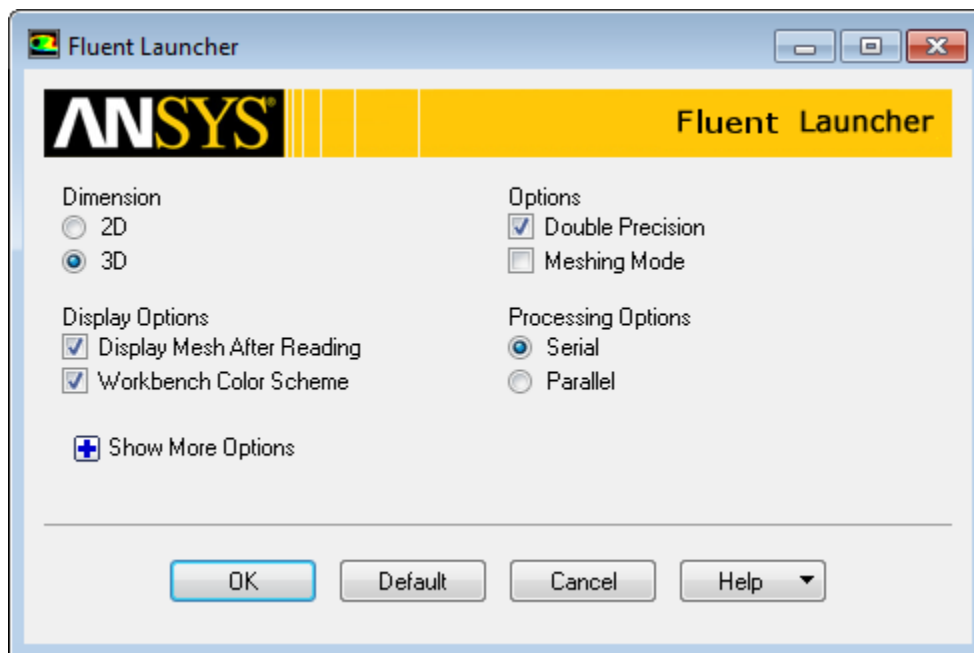
4.1. Starting ANSYS Fluent

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

You can interactively specify ANSYS Fluent dimension, display, processing and other options using the Fluent Launcher.



To start the Fluent Launcher, do one of the following:

- Start ANSYS Fluent from the Linux or Windows command line with no arguments.
- Start ANSYS Fluent from the Windows Start menu.

- Start ANSYS Fluent from the Windows desktop or Quick Launch bar.

Any options set in the Fluent Launcher will be retained for your next session.

Specify the **Dimension** of the simulation you intend to perform.

Important

For **Meshing Mode**, you must select **3D**. **Meshing Mode** is not available if **2D** is selected.

Select your required **Display Options**.

- Optionally, choose to **Display Mesh After Reading** (disabled by default). This option is applicable only to volume meshes and not surface meshes. All of the boundary zones will be displayed except for the interior zones of 3D geometries.

Note

You can override this option on a file-by-file basis using the **Display Mesh After Reading** option in the **Select File** dialog box that opens when you are reading in a file.

- Choose either the default **Workbench Color Scheme** (blue gradient background) or classic, black background for the graphics windows.

Select your required **Options**.

- Choose to perform solution calculations in **Double-Precision** mode, if desired. (Default is single-precision mode) See [Single-Precision and Double-Precision Solvers \(p. 29\)](#) to help with your decision.

Note

Meshing Mode is always run in **Double Precision**. This option applies for Solution Mode only.

- Choose to start ANSYS Fluent in **Meshing Mode**, if desired. (Default is Solution Mode.) See the [Fluent Meshing User's Guide](#) for further details about using Fluent in meshing mode.
- Choose an appropriate **Use Job Scheduler** option, if applicable. 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 \(p. 32\)](#), as well as [Setting Parallel Scheduler Options in Fluent Launcher](#) in the *User's Guide*.

Note

This option is available only if **Parallel** is selected under **Processing Options**.

- 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 \(p. 31\)](#) for details).

Note

This option is available only if **Parallel** is selected under **Processing Options**.

Select your required **Processing Options**.

- Use **Serial** to restrict the meshing/solution calculations to a single processor core.
- Use **Parallel** to allow multiple simultaneous processes. You will be asked to specify the number of processes for both Meshing (if enabled) and Solver. See [Setting Parallel Options in Fluent Launcher \(p. 30\)](#). When parallel is enabled, several additional parallel-specific items are available for selection in the **Parallel** ribbon tab.

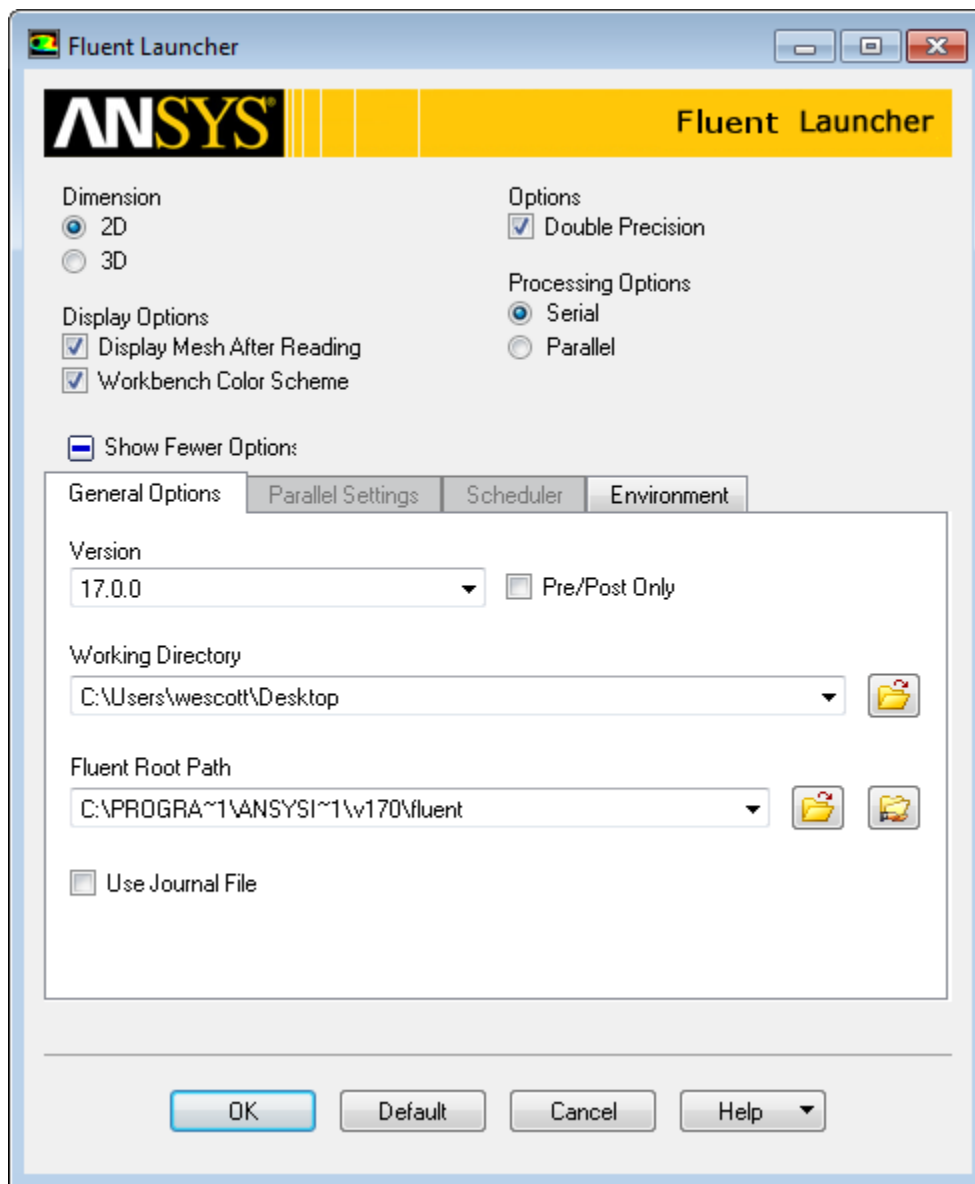
Select **Show More Options** to expand the Fluent Launcher window to reveal more options. ([Figure 4.1: The General Options Tab of Fluent Launcher \(p. 28\)](#)). Note that once Fluent Launcher expands, the **Show More Options** button becomes the **Show Fewer Options** button, allowing you to hide the 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


Set file and path options using the **General Options** tab in Fluent Launcher.

Figure 4.1: The General Options Tab of Fluent Launcher


1. Select a **Version** from the drop-down list. All of the available versions of ANSYS Fluent that exist in your ANSYS Fluent installation will be displayed.
2. Select **Pre/Post Only** to run ANSYS Fluent with only the setup and postprocessing capabilities available. The default ANSYS Fluent full solution mode allows you to set up, solve and postprocess a problem, while **Pre/Post Only** will not allow you to perform calculations.
3. Specify the path of your current working directory using the **Working Directory** field or click  to browse through your directory structure.


Note

a Uniform Naming Convention (UNC) path cannot be set as a working directory. You need to map a drive to the UNC path (Windows only)

4. 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. Try to use the UNC path if applicable.

Note

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.

5. To use a journal file to automatically load the case, compile any user-defined functions, iterate until the solution converges, and write results to an output file,
 - a. Enable **Use Journal File**.
 - b. Specify your journal file path and name, or click  to browse through your directory structure to locate the 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

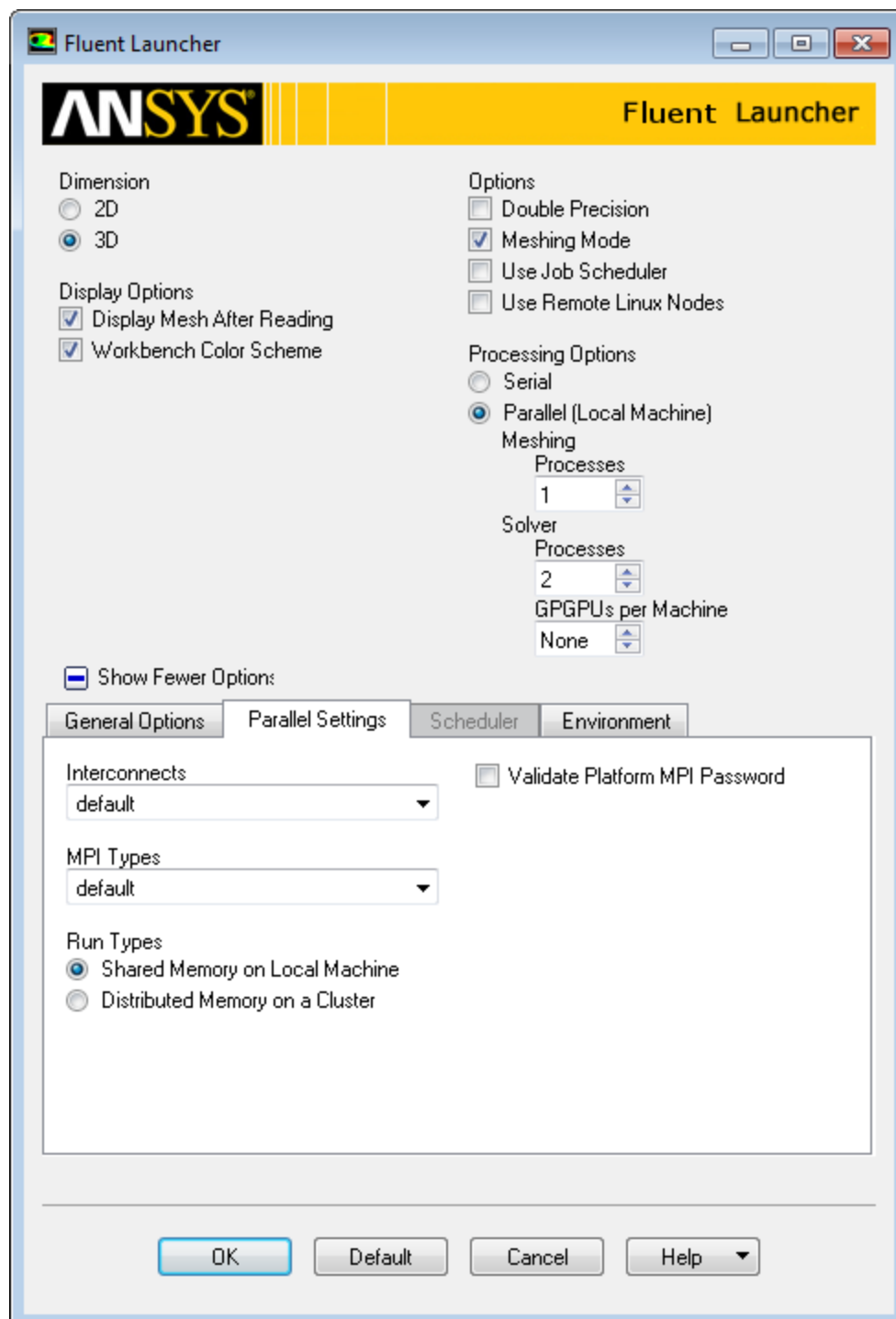
ANSYS Fluent allows only a period to be used as a decimal separator. If your system is set to a European locale that uses a comma separator (for example, Germany), fields that accept numeric input may accept a comma, but may ignore everything after the comma. If your system is set to a non-European locale, 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 allows you to specify settings for running ANSYS Fluent in parallel. This tab is only available if you have selected **Parallel** under **Processing Options**.

Figure 4.2: The Parallel Settings Tab of Fluent Launcher



Once you select **Parallel**,

1. Enter the number of processes to be used for meshing under **Meshing Processes**, if enabled.
2. Enter the number of processes to be used for solution under **Solver Processes**. When you change to solution mode, additional processes will be spawned as necessary to bring the total number of processes to this value. This must be set to a value greater than or equal to **Meshing Processes**.
3. If your machine is equipped with General Purpose Graphics Processing Units you can also specify **GPGPUs per Machine**.

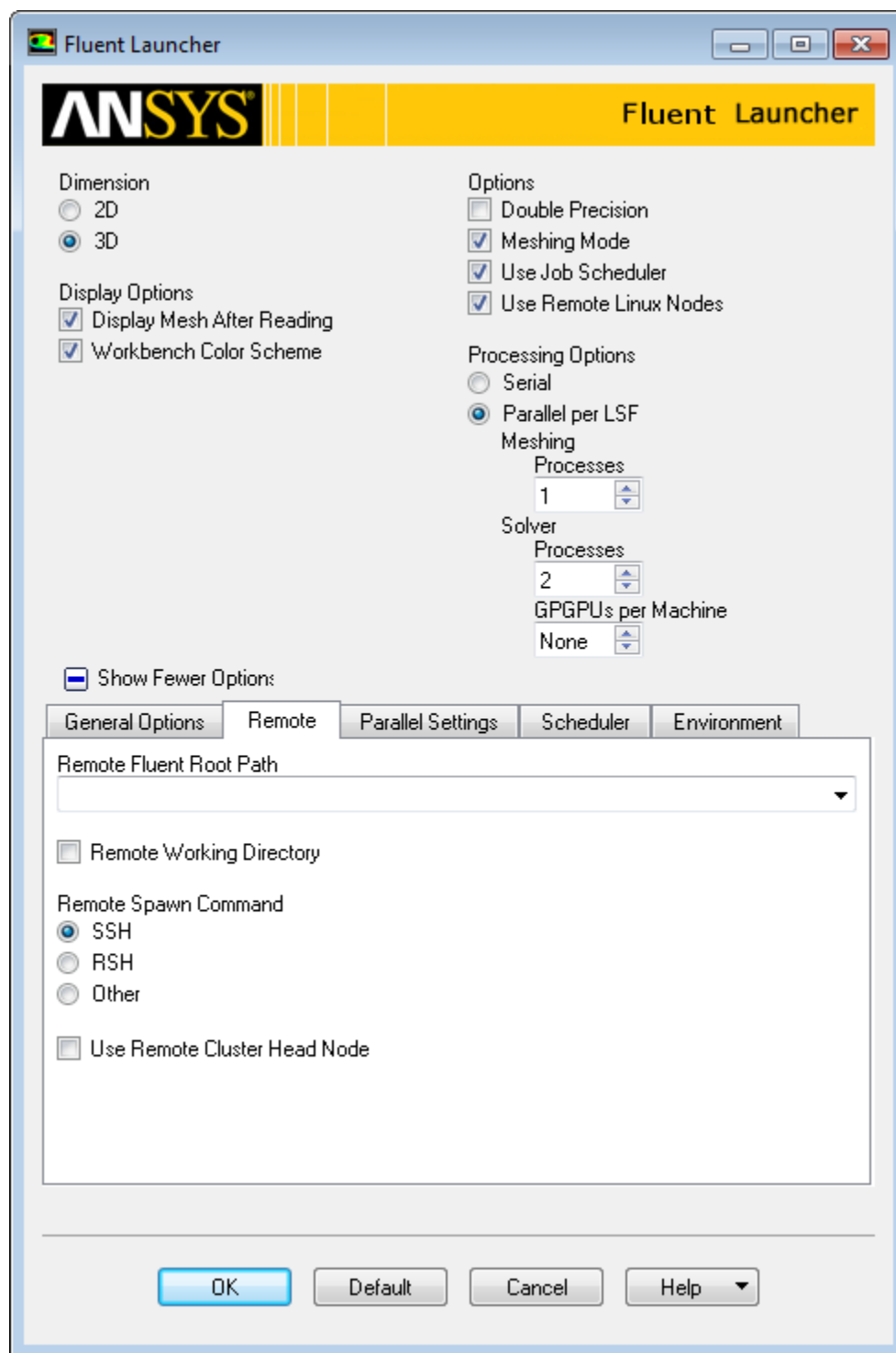
Note

With Parallel processing enabled, some global file read and write operations are affected. Unavailable options will have their menu entry grayed out.

Refer to [Parallel Processing in the *Fluent User's Guide*](#) for details on parallel processing using Fluent and [Starting Parallel ANSYS Fluent Using Fluent Launcher in the *Fluent User's Guide*](#) for additional details parallel process configuration options on this tab.

4.1.1.4. Setting Remote Options in Fluent Launcher

The **Remote** tab ([Figure 4.3: The Remote Tab of Fluent Launcher \(p. 32\)](#)) allows you to specify settings for running ANSYS Fluent parallel simulations on Linux clusters, via the Windows interface.

Figure 4.3: The Remote Tab of Fluent Launcher

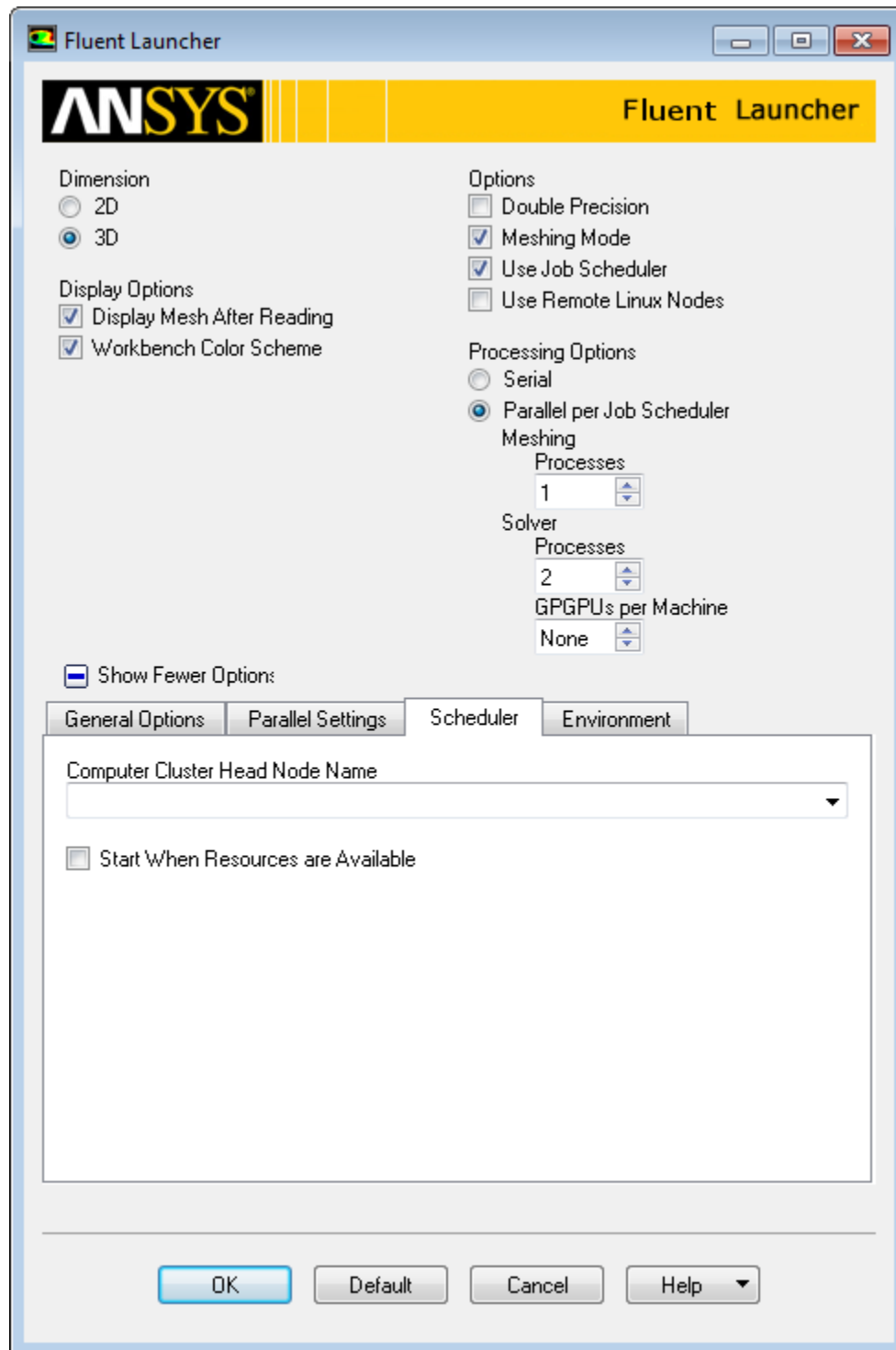
For additional information about this tab, see [Setting Additional Options When Running on Remote Linux Machines](#) in the [User's Guide](#).

4.1.1.5. Setting Scheduler Options in Fluent Launcher

The **Scheduler** tab (Figure 4.4: The Scheduler Tab of Fluent Launcher (Windows 64 Version)(p. 33)) allows you to specify settings for running ANSYS Fluent with various job schedulers (for example, the Microsoft

Job Scheduler for Windows, or LSF, SGE, and PBS Pro on Linux). This tab is available if you have selected **Use Job Scheduler** under **Options**.

Figure 4.4: The Scheduler Tab of Fluent Launcher (Windows 64 Version)

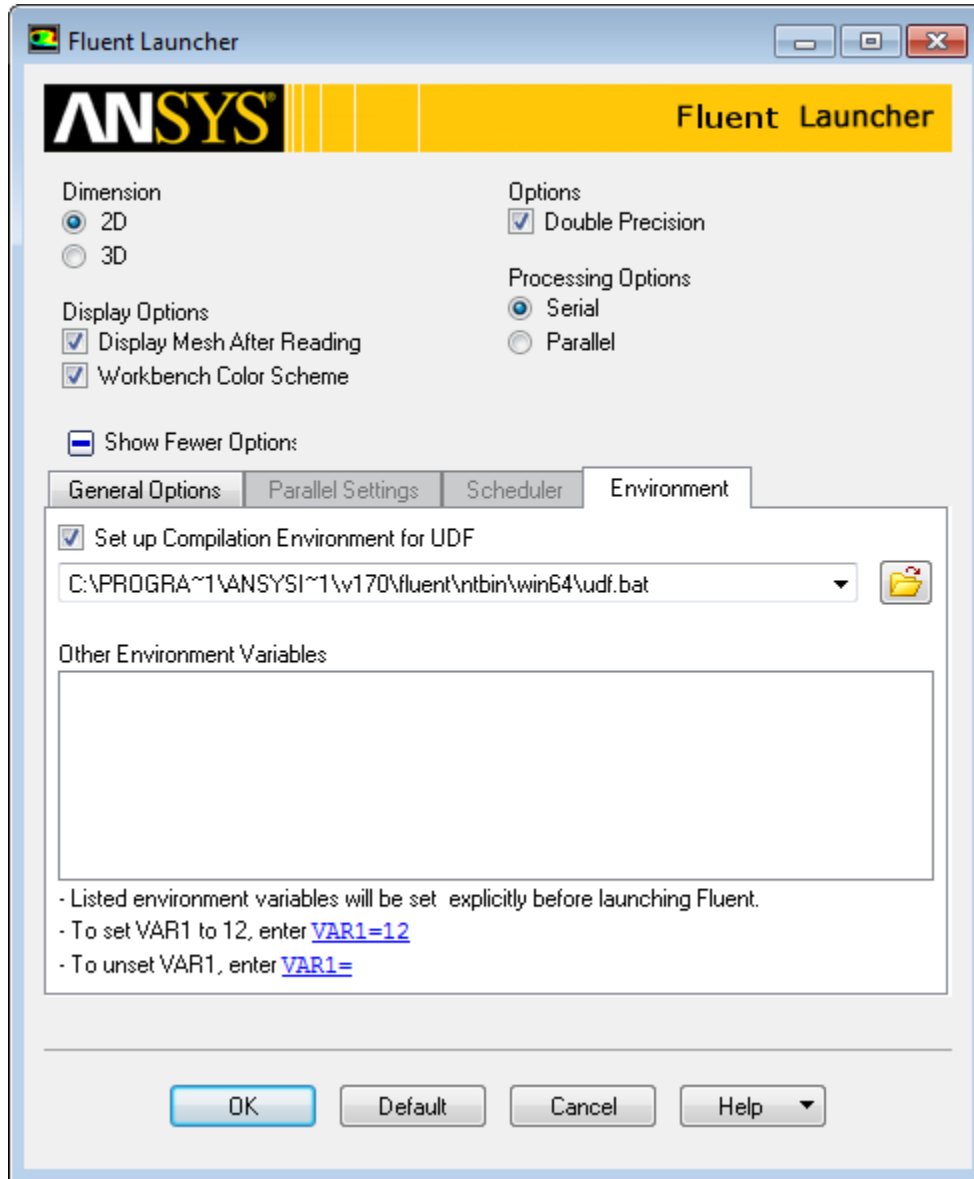


For additional information about this tab, see [Setting Parallel Scheduler Options in Fluent Launcher](#) in the [User's Guide](#).

4.1.1.6. Setting Environment Options in Fluent Launcher

The **Environment** tab (Figure 4.5: The Environment Tab of Fluent Launcher (p. 34)) 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 4.5: The Environment Tab of Fluent Launcher



Specify a batch file that contains UDF compilation environment settings by selecting the **Set up 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 directory 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 [Fluent Customization 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:

- From the Windows **Start** menu, click **Start > All Programs > ANSYS 17.0 > Fluid Dynamics > Fluent 17.0**

This option starts Fluent Launcher (see [Starting ANSYS Fluent Using Fluent Launcher](#)). The Fluent Launcher may also be accessed via an icon on your desktop or in the Quick Launch bar.

Note

If the default “ANSYS 17.0” 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 17.0** program group.

- From a Command Prompt window, type `fluent version`, where `version` is replaced with one of the four options specifying the dimension and precision of the solver.
 - `2d` for the 2D, single-precision solver.
 - `3d` for the 3D, single-precision solver.
 - `2ddp` for the 2D, double-precision solver.
 - `3ddp` for the 3D double-precision solver.

Important

Be sure the path to your ANSYS Fluent home directory is in your command search path environment variable by executing the `setenv.exe` program located in the ANSYS Fluent directory (for example, `C:\Program Files\ANSYS Inc\v170\fluent\nt-bin\win64`).

Tip

You can also start the parallel version of ANSYS Fluent, or start ANSYS Fluent in meshing mode from the Command Prompt.

- To start the parallel version on `x` processors, type `fluent version -tx`, replacing `version` with the desired solver version and `x` with the number of processors. For example, `fluent 3d -t4` to run the 3D version on 4 processors.
 - To start in meshing mode, add the command line option `-meshing`. For example, `fluent 3d -meshing` to start in meshing mode.
 - Both parallel and meshing mode may be combined. You must specify the number of meshing processes using `-tmy`. For example, `fluent 3ddp -meshing -tm4 -t8` will start ANSYS Fluent in meshing mode with 4 meshing processes. When switched to solution mode, the solver will be 3D, double precision and run 8 processes.
-

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 `fluent version`, where `version` is replaced with one of the four options specifying the dimension and precision of the solver.
 - `2d` for the 2D, single-precision solver.
 - `3d` for the 3D, single-precision solver.
 - `2ddp` for the 2D, double-precision solver.
 - `3ddp` for the 3D double-precision solver.

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 and `x` with the number of processors. For example, `fluent 3d -t4` will 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.

If you are logged onto a Linux machine remotely, ANSYS Fluent picks the `x11` driver by default. If your remote desktop software supports `opengl` driver, you can launch ANSYS Fluent using the `opengl` driver by either of the following two methods.

- Use `-driver opengl` as a command line option.
- Define `HOOPS_PICTURE` as an environment variable, either in the Linux console or under the **Environment** tab of the Fluent Launcher.

For example:

- Linux: `setenv HOOPS_PICTURE opengl/:2.0`
- Fluent Launcher: `HOOPS_PICTURE=opengl/:2.0`

Where `:2.0` is the value of the Linux `DISPLAY` environment variable.

4.1.4. Command Line Startup Options

[Table 4.1: Available Command Line Options for Linux and Windows Platforms \(p. 36\)](#) lists the available command line arguments for Linux and Windows. More detailed descriptions of these options can be found in the following sections.

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

Option	Platform	Description
<code>-aas</code>	all	Start Fluent in server mode

Option	Platform	Description
-cc	all	Use the classic color scheme
-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
-g	all	Run without the GUI or graphics
-gpgpu=ngpgpus	Linux and Win64 only	Specify that ngpgpus GPGPUs per machine should be used for AMG acceleration. Only available in parallel.
-gr	all	Run without graphics
-gu	all	Run without the GUI but with graphics. You cannot interact with the displayed graphics objects.
-help	all	Display command line options
-hidden	Windows only	Run in minimized mode
-host_ip=host:ip	Linux only	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
-meshing	all	Start Fluent in meshing mode (you must specify Fluent as either 3d or 3ddp)
-mpi	all	Specify MPI implementation
-mpitest	all	Launches an MPI program to collect network performance data and prints to console (Linux) or to the working directory (Windows)
-nm	all	Do not display mesh after reading
-pbs	Linux only	Launches ANSYS Fluent under PBS Professional
-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 ethernet or ib
-r	all	List all releases installed in the current directory
-rsh	all	Use rsh for remote connection rather than the default ssh

Option	Platform	Description
-rsh=<command>	Windows only	Use <command> for remote connection rather than the default ssh for running on a remote Linux cluster.
-rx	all	Specify release number
-schost	all	Specify a host machine for system coupling
-sctest	all	Specify a port on the host machine for system coupling
-sctest	all	Specify a unique name for a system coupling participant
-sge	Linux only	Run ANSYS Fluent under Sun Grid Engine
-sgeq 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
-tmx	all	Specify the number of processors for meshing 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 -g` will run Cortex without graphics and without the graphical user interface. This option is useful if 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 open graphics windows and display graphics objects. You cannot interact with the displayed graphics objects.

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 -meshing` 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*.

`-gpgpu=ngpgpus` specifies the number of general purpose graphics processing units (GPGPUs) per machine to be used for AMG acceleration. For more information, see [Using General Purpose Graphics Processing Units \(GPGPUs\) With the Algebraic Multigrid \(AMG\) Solver](#) in the *Fluent 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).

`-rsh` specifies that RSH should be used to spawn remote processes, instead of the default `ssh`.

`-rsh=<command>` specifies that `<command>` should be used to spawn remote processes. (Windows only)

`-ssh` specifies that SSH should be used to spawn remote processes. (Beginning with ANSYS Fluent R16.0, SSH is used by default. This option is included primarily for backward compatibility with existing launch scripts, etc.)

`-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*.

`-tmx` specifies that `x` processors are to be used for meshing. This value must be less than or equal to the number of processes specified with `-tx`.

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 `-sgeq queue`, `-sgeckpt ckpt_obj`, and `-sgepe fluent_pe min_n-max_n`.

For a detailed explanation of these options, see [Part III: Running ANSYS 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 [Part I: Running ANSYS 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.

`-schost=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).

`-sctest=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.

`-sctest=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 -schost=host name in quotes -sctest=port number -sctest=name of the solver in quotes
```

For instance:

```
fluent 3d -schost="machine1.domain.com" -sctest=1234 -sctest="Solution1"
```

Once ANSYS Fluent loads the case, initialize the solution using the following command:

```
s i i
```

Once your case is initialized, 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 described in [Customizing the Graphical User Interface](#) 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.

There are additional Qt command line startup options for modifying the graphical stylesheet and more, which can be found in Qt documentation.

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 \(p. 41\)](#) provides guidance for running in batch/background on Linux systems, and [Background Execution on Windows Systems \(p. 43\)](#) 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 run ANSYS Fluent interactively.
- `-g` indicates that the program is to be run without the GUI or graphics (see [Starting ANSYS Fluent \(p. 25\)](#)).
- `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, 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 run 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 runs ANSYS Fluent completely hidden and non-interactively.

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, 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.

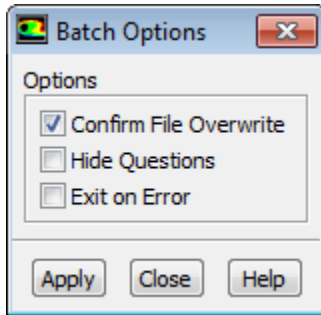
4.2.3. Batch Execution Options

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 4.6: The Batch Options Dialog Box](#) (p. 44)).



File → Batch Options...

Figure 4.6: 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 disabled by default.

If you are running in batch mode through the command prompt with **Exit on Error** enabled, Fluent will return a zero value if it ran successfully. Fluent will return a non-zero value if it did not run successfully. The two types of errors that would cause a non-zero exit are if Fluent was unable to check out a license or if there was an error during the execution of a command.

Note that in Windows you must start Fluent with the `-wait` command line option.

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 [Part III: Running ANSYS Fluent Under SGE](#) or [Part I: Running ANSYS 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 must 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 directory with a filename 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 directory. 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 directory.

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** ribbon tab. 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 \(p. 49\)](#)
- [case files \(p. 49\)](#)
- [cell types \(p. 50\)](#)
- [computational fluid dynamics \(CFD\) \(p. 50\)](#)
- [console \(p. 51\)](#)
- [convergence \(p. 51\)](#)
- [cortex \(p. 51\)](#)
- [data files \(p. 51\)](#)
- [dialog boxes \(p. 51\)](#)
- [discretization \(p. 51\)](#)
- [GUI \(p. 51\)](#)
- [mesh \(p. 51\)](#)
- [models \(p. 51\)](#)
- [node \(p. 51\)](#)
- [postprocessing \(p. 51\)](#)
- [residuals \(p. 51\)](#)
- [skewness \(p. 51\)](#)
- [solvers \(p. 52\)](#)
- [terminal emulator \(p. 52\)](#)
- [TUI \(p. 52\)](#)

adaption

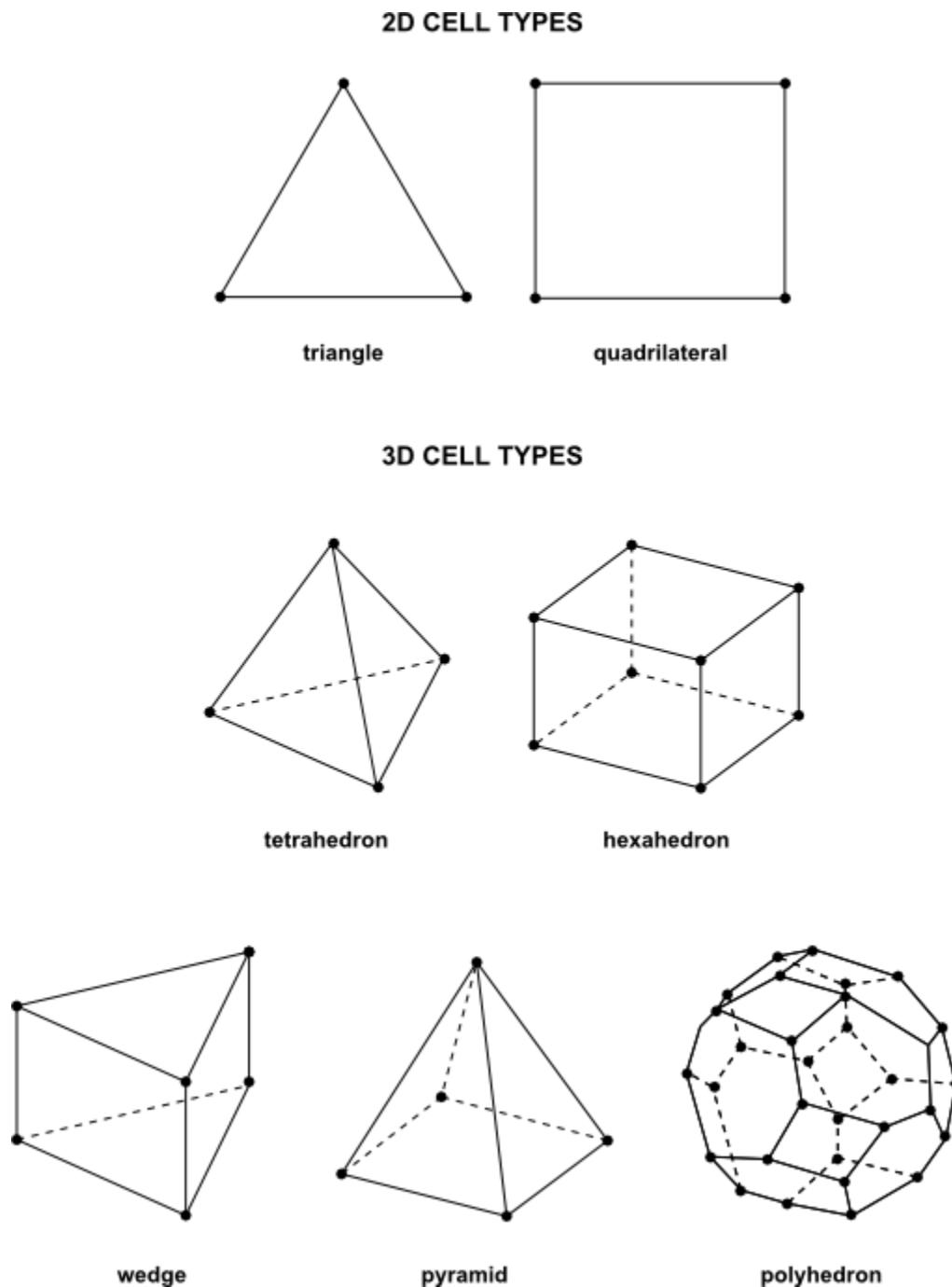
A technique useful in improving overall mesh quality. The solution-adaptive mesh refinement feature of ANSYS Fluent allows you to refine and/or coarsen your mesh based on geometric and numerical solution data. In addition, ANSYS Fluent provides tools for creating and viewing adaption fields customized 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 composed of tetrahedral, hexahedral, pyramid, wedge, or polyhedral cells (or a combination of these).

Figure 9: 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. 51) 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. 51) .
TUI	The text user interface, which consists of textual commands that can be entered into the terminal emulator.