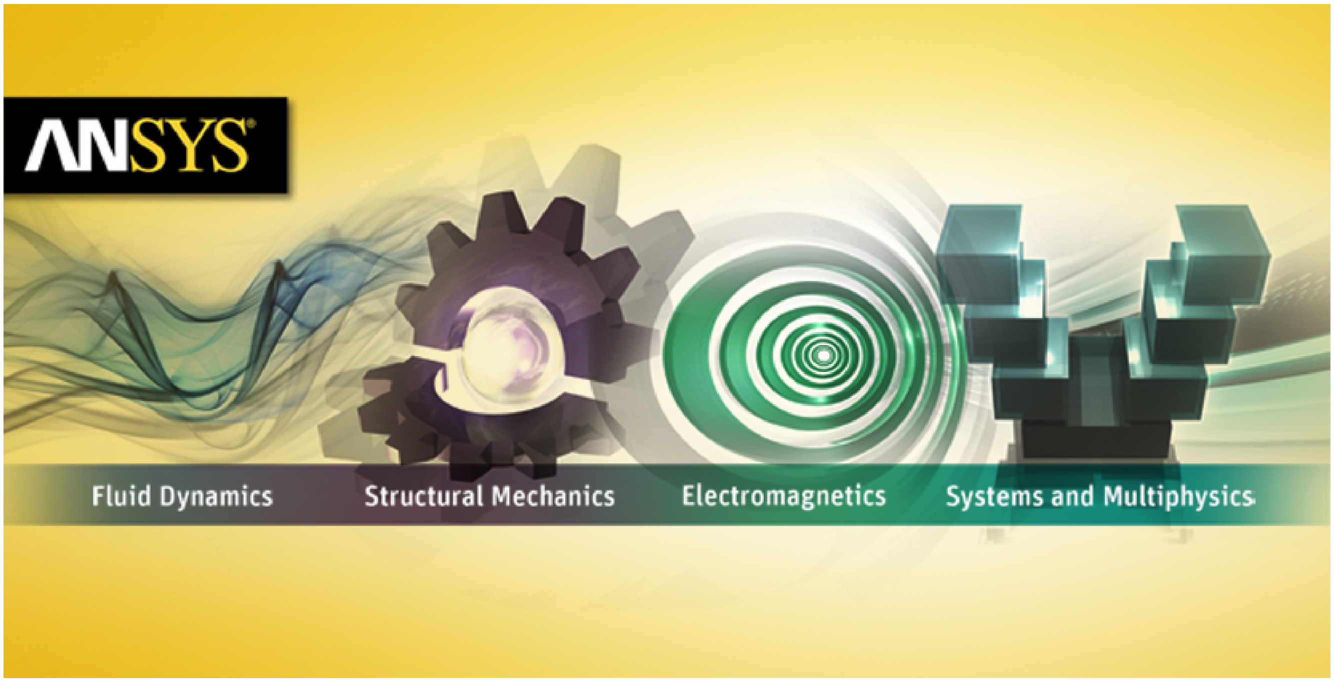


ANSYS FLUENT in ANSYS Workbench User's Guide

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

Release 14.0
November 2011

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

Copyright and Trademark Information

© 2011 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

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

Disclaimer Notice

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

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

U.S. Government Rights

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

Third-Party Software

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

Published in the U.S.A.

Table of Contents

Using This Manual v
1. The Contents of This Manual v
2. The Contents of the FLUENT Manuals v
3. Typographical Conventions vi
4. Technical Support vii
1. Getting Started With FLUENT in Workbench 1
1.1. Introduction to Workbench 1
1.1.1. Limitations 2
1.2. The Workbench Graphical User Interface 2
1.3. Creating FLUENT-Based Systems 4
1.3.1. Creating FLUENT-Based Analysis Systems 4
1.3.2. Creating FLUENT-Based Component Systems 7
1.4. Understanding Cell States with FLUENT in Workbench 8
1.5. Starting FLUENT in Workbench 9
1.5.1. Starting FLUENT from a FLUENT-Based System 9
1.5.2. Specifying FLUENT Launcher Settings Within Workbench 9
1.5.2.1. Specifying FLUENT Launcher Settings Using Cell Properties 10
1.5.2.2. Copying FLUENT Launcher Property Settings 15
1.6. Saving Your Work in FLUENT with Workbench 15
1.7. Exiting FLUENT and Workbench 16
1.8. An Example of a FLUENT Analysis in Workbench 17
1.9. Getting Help for FLUENT in Workbench 20
2. Working With FLUENT in Workbench 23
2.1. Importing Mesh, Case, and Data Files 23
2.2. Using the Update Command 26
2.3. Refreshing FLUENT Input Data 28
2.4. Deleting Solution and Setup Cell Data for FLUENT-Based Systems 29
2.4.1. Using the Clear Generated Data Command from the Solution Cell of FLUENT-Based Systems 29
2.4.2. Using the Reset Command from the Setup and Solution Cells of FLUENT-Based Systems 29
2.4.3. Using the Clear Old Solution Data Command from the Solution Cells of FLUENT-Based Systems 30
2.5. Interrupting, Restarting, and Continuing a Calculation 30
2.6. Connecting Systems in Workbench 33
2.6.1. Connecting Systems by Dragging and Dropping a System from the Toolbox onto Another System 35
2.6.2. Connecting Systems By Dragging and Dropping FLUENT-Based Solution Cells Onto Other Systems 38
2.7. Duplicating FLUENT-Based Systems 40
2.8. Reviewing Mesh Manipulation Operations in FLUENT 40
2.9. Changing the Settings and Mesh in FLUENT 45
2.9.1. Changing Case and Mesh Settings Before Beginning a Calculation 45
2.9.2. Changing Case and Mesh Settings After a Calculation Has Started 47
2.10. Case Modification Strategies with FLUENT and Workbench 50
2.11. Working With Input and Output Parameters in Workbench 51
2.12. Viewing Your FLUENT Data Using ANSYS CFD-Post 54
2.13. Understanding the File Structure for FLUENT in Workbench 55
2.13.1. FLUENT File Naming in Workbench 58
2.14. Working with ANSYS Licensing 59
2.14.1. Shared Licensing Mode 59

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.15. Using FLUENT With the Remote Solve Manager (RSM) 60
2.16. Using Custom Systems 60
2.17. Using Journaling and Scripting with FLUENT in Workbench 61
2.18. Performing System Coupling Simulations Using FLUENT in Workbench 62
2.18.1. Supported Capabilities and Limitations 62
2.18.2. Regions and Variables Available for System Coupling 63
2.18.3. System Coupling Related Settings in FLUENT 63
2.18.4. How FLUENT's Execution is Affected by System Couplings 64
2.18.5. Restarting FLUENT Analyses as Part of System Couplings 64
2.18.5.1. Generating Restart Files 64
2.18.5.2. Executing the Restart Run 64
2.18.6. Running FLUENT as a System Coupling Participant from the Command Line 65
2.18.7. Troubleshooting Two-Way Coupled Analysis Problems 65
2.19. Performing FLUENT and Ansoft Coupling in Workbench 66
A. The FLUENT Menu Under Workbench 69
A.1. **File/Refresh Input Data** 69
A.2. **File/Save Project** 69
A.3. **File/Import/Mesh...** 69
A.4. **File/Import/Case...** 69
A.5. **File/Import/Data...** 70
A.6. **File/Import/Case and Data...** 70
A.7. **File/Export/...** 70
A.8. File/EM Mapping/Volumetric Energy Source... 71
A.8.1. Maxwell Mapping Dialog Box 71
A.9. **Mesh/Recorded Mesh Operations...** 72
Index 73

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using This Manual

This preface is divided into the following sections:

1. The Contents of This Manual
2. The Contents of the FLUENT Manuals
3. Typographical Conventions
4. Technical Support

1. The Contents of This Manual

This document provides information about using the FLUENT application within Workbench.

A brief description of what is in each chapter follows:

- *Getting Started With FLUENT in Workbench* (p. 1), describes an overview of FLUENT within Workbench.
- *Working With FLUENT in Workbench* (p. 23), describes the details of using FLUENT within Workbench.
- *Appendix A* (p. 69), describes the differences in the FLUENT **File** menu within Workbench.

2. The Contents of the FLUENT Manuals

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

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

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FLUENT Magnetohydrodynamics (MHD) Module Manual contains information about the background and usage of FLUENT's Magnetohydrodynamics (MHD) Module that allows you to analyze the behavior of electrically conducting fluid flow under the influence of constant (DC) or oscillating (AC) electromagnetic fields.
FLUENT Migration Manual contains information about transitioning from the previous release of FLUENT, including details about new features, solution changes, and text command list changes.
FLUENT Population Balance Module Manual contains information about the background and usage of FLUENT's Population Balance Module that allows you to analyze multiphase flows involving size distributions where particle population (as well as momentum, mass, and energy) require a balance equation.
Running FLUENT Under LSF contains information about the using FLUENT with Platform Computing's LSF software, a distributed computing resource management tool.
Running FLUENT Under PBS Professional contains information about the using FLUENT with Altair PBS Professional, an open workload management tool for local and distributed environments.
Running FLUENT Under SGE contains information about the using FLUENT with Sun Grid Engine (SGE) software, a distributed computing resource management tool.

3. Typographical Conventions

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

- Different type styles are used to indicate graphical user interface menu items and text interface menu items (for example, **Iso-Surface** dialog box, `surface/iso-surface` command).
- The text interface type style is also used when illustrating exactly what appears on the screen or exactly what you need to type into a field in a dialog box. The information displayed on the screen is enclosed in a large box to distinguish it from the narrative text, and user inputs are often enclosed in smaller boxes.
- A mini flow chart is used to guide you through the navigation pane, which leads you to a specific task page or dialog box. For example,

  Models → **Multiphase** → **Edit...**

  indicates that **Models** is selected in the navigation pane, which then opens the corresponding task page. In the **Models** task page, **Multiphase** is selected from the list. Clicking the **Edit...** button opens the **Multiphase** dialog box.

  Also, a mini flow chart is used to indicate the menu selections that lead you to a specific command or dialog box. For example,

  Define → **Injections...**

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

  `display` → `mesh`

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

  In this manual, mini flow charts usually precede a description of a dialog box or command, or a screen illustration showing how to use the dialog box or command. They allow you to look up information about a command or dialog box and quickly determine how to access it without having to search the preceding material.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

4. Technical Support

If you encounter difficulties while using ANSYS FLUENT, please first refer to the section(s) of the manual containing information on the commands you are trying to use or the type of problem you are trying to solve. The product documentation is available from the online help, or from the ANSYS Customer Portal (www.ansys.com/customerportal).

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

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

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

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

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

NORTH AMERICA

All ANSYS, Inc. Products

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

Toll-Free Telephone: 1.800.711.7199

Fax: 1.724.514.5096

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

GERMANY

ANSYS Mechanical Products

Telephone: +49 (0) 8092 7005-55

Email: support@cadfem.de

All ANSYS Products

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

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

National Toll-Free Telephone:

German language: 0800 181 8499

English language: 0800 181 1565

International Telephone:

German language: +49 6151 3644 300

English language: +49 6151 3644 400

Email: support-germany@ansys.com

UNITED KINGDOM

All ANSYS, Inc. Products

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

Telephone: +44 (0) 870 142 0300

Fax: +44 (0) 870 142 0302

Email: support-uk@ansys.com

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

JAPAN

CFX , ICEM CFD and Mechanical Products

Telephone: +81-3-5324-8333

Fax: +81-3-5324-7308

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

FLUENT Products

Telephone: +81-3-5324-7305

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

Icepak

Telephone: +81-3-5324-7444

Email: japan-icepak-support@ansys.com

Licensing and Installation

Email: japan-license-support@ansys.com

INDIA

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

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

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

Fax: +91 80 2529 1271

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

FRANCE

All ANSYS, Inc. Products

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

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

Email: support-france@ansys.com

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

BELGIUM

All ANSYS Products

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

Telephone: +32 (0) 10 45 28 61

Email: support-belgium@ansys.com

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

SWEDEN

All ANSYS Products

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

Telephone: +44 (0) 870 142 0300

Email: support-sweden@ansys.com

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

SPAIN and PORTUGAL

All ANSYS Products

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

Telephone: +33 1 30 60 15 63

Email: support-spain@ansys.com

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

ITALY

All ANSYS Products

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

Telephone: +39 02 89013378

Email: support-italy@ansys.com

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

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 1: Getting Started With FLUENT in Workbench

This document is designed to provide information about using FLUENT within ANSYS Workbench. Some basic information about using Workbench is provided here, but the majority of the information about using Workbench can be found in the Workbench on-line documentation.

This chapter provides some basic instructions for getting started with using FLUENT in Workbench.

1.1. Introduction to Workbench
1.2. The Workbench Graphical User Interface
1.3. Creating FLUENT-Based Systems
1.4. Understanding Cell States with FLUENT in Workbench
1.5. Starting FLUENT in Workbench
1.6. Saving Your Work in FLUENT with Workbench
1.7. Exiting FLUENT and Workbench
1.8. An Example of a FLUENT Analysis in Workbench
1.9. Getting Help for FLUENT in Workbench

1.1. Introduction to Workbench

ANSYS Workbench combines access to ANSYS applications with utilities that manage the product workflow.

Applications that can be accessed from Workbench include: ANSYS DesignModeler (for geometry creation); ANSYS Meshing (for mesh generation); ANSYS FLUENT or CFX (for setting up and solving fluid dynamics analyses); and ANSYS CFD-Post (for postprocessing the results). In Workbench, a project is composed of a group of systems. The project is driven by a schematic workflow that manages the connections between the systems. From the schematic, you can interact with workspaces that are native to Workbench, such as Design Exploration (parameters and design points), and you can launch applications that are data-integrated with Workbench (such as ANSYS FLUENT or CFX). Data-integrated applications have separate interfaces, but their data is part of the Workbench project and is automatically saved and shared with other applications as needed. This makes the process of creating and running a CFD simulation more streamlined and efficient.

Workbench allows you to construct projects composed of multiple dependent systems that can be updated sequentially based on a workflow defined by the project schematic. For instance, you can construct a project using two connected FLUENT-based systems where the two systems share the same geometry and mesh; and the second system uses data from the first system as its initial solution data. When you have two systems connected in this way, you can modify the shared geometry once and then update the results for both systems with a single mouse click without having to open the ANSYS Meshing application or FLUENT. Some examples of when this is useful include: performing a reacting flow analysis starting from the solution obtained from a cold flow analysis; performing a second order analysis starting from the solution obtained from a first order analysis; and performing a transient simulation starting from the solution obtained from a steady-state analysis.

In addition, Workbench also allows you to copy systems in order to efficiently perform and compare multiple similar analyses. Workbench also provides a parametric modeling capabilities in conjunction

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

with optimization techniques to allow you to efficiently investigate the effects of input parameters on selected output parameters.

For more information, please see the following section:
1.1.1. Limitations

1.1.1. Limitations

The following limitations are known when using FLUENT in Workbench:

- Workbench units and options are not passed to FLUENT.
- The text user interface (TUI) shortcuts, or aliases, for reading case and data files for FLUENT in Workbench are disabled by design. For example, `file read-case`, `f read-case`, and `f r-c` can be used, however `f rc` cannot be used.
- The version of FLUENT used under Workbench must always be the version of FLUENT that was packaged and installed with that version of ANSYS Workbench. It is not possible to use previous versions of FLUENT under Workbench even through a `FLUENT Inc.` environment variable.
- Graphical user interface (GUI) journal files in FLUENT 14.0 are not backwards compatible. That is, Workbench journal files (`*.wbjn` files) created in version 12.0 or version 12.1 that contain FLUENT GUI commands may need to be recreated if they fail due to a change in the FLUENT graphical user interface or a string used therein.
- For older FLUENT in Workbench projects (prior to version 14.0), when making any changes to the mesh using ANSYS Meshing (e.g., renaming a surface), you should first open FLUENT using the **Setup** cell of the Fluid Flow (FLUENT) analysis system in order for FLUENT to be aware of the upstream mesh changes (e.g., when detecting upstream zone name changes).
- Under Workbench, the Microsoft Job Scheduler cannot be used to run FLUENT in serial.
- The coupling between Ansoft and FLUENT in Workbench is not supported on Windows Vista (32 bit and 64 bit), Red Hat 6, and SUSE Linux Enterprise Server 11.
- Changing the meshing method to "Assembly Meshing" methods (e.g., CutCell) from other meshing methods may not preserve the previously defined contact regions. Currently, the FLUENT simulation workflow does not support this type of change.

Note

ANSYS FLUENT only allows a period to be used as a decimal separator. If your system is set to a European locale that uses a comma separator (e.g., 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.

1.2. The Workbench Graphical User Interface

The Workbench graphical user interface (*Figure 1.1* (p. 3)) consists of the Toolbox, the Project Schematic, the Toolbar, and the Menu bar. The most common way to begin work in Workbench is to drag an item, such as a component system (application) or an analysis system, from the Toolbox to the Project Schematic, or to double-click an item to initiate the default action. You will view your component and/or analysis systems – the pieces that make up your analysis – in the Project Schematic, including all con-

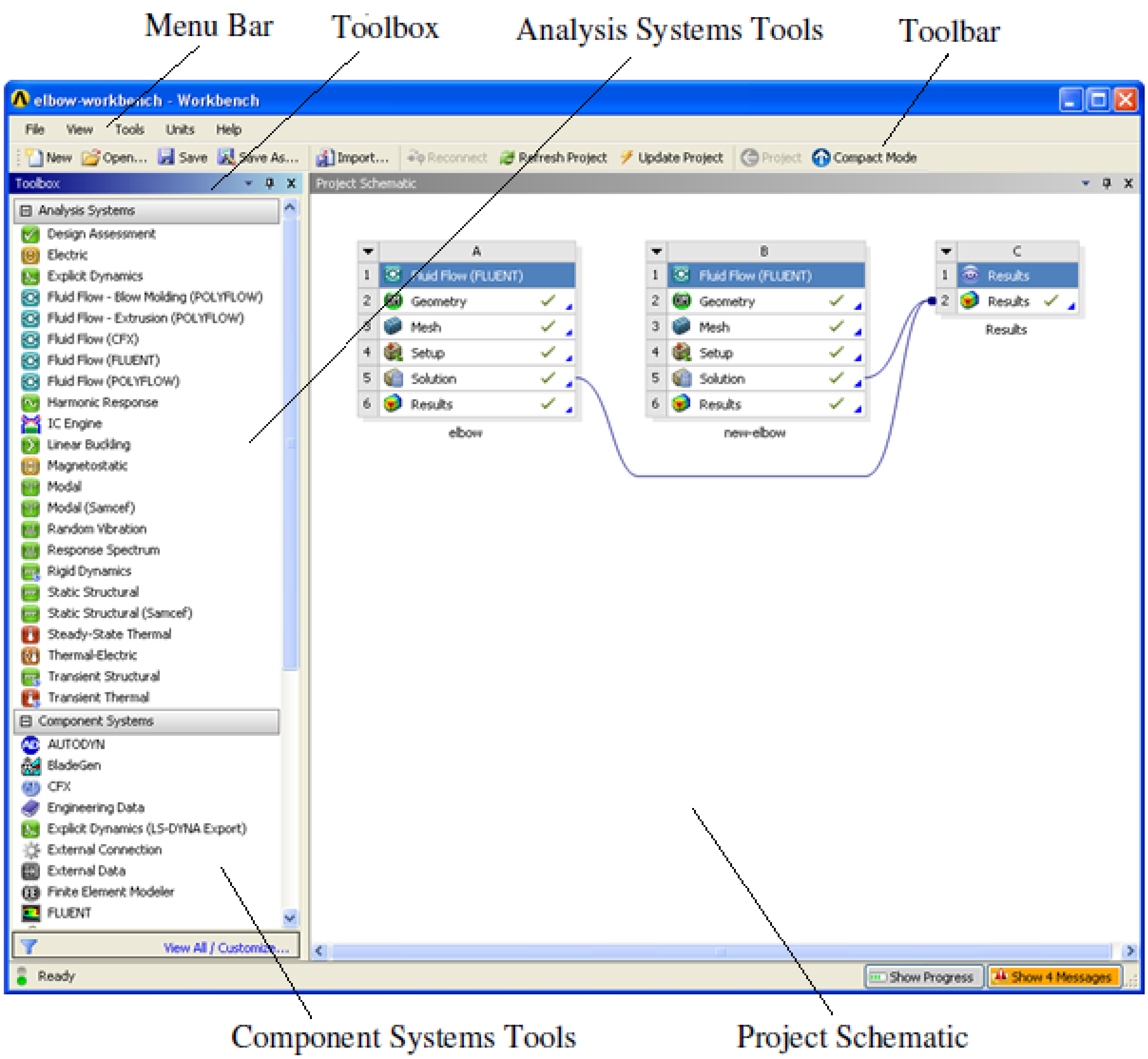
Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

nections between the systems. The individual applications in which you work will display separately from the Workbench graphical interface, but the actions you take in the applications will be reflected in the Project Schematic.

Important

Note that FLUENT can be accessed in Workbench as either a component system or as an analysis system. Details for using both are described throughout this document.

Figure 1.1 The Workbench Graphical User Interface



Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Note that FLUENT in Workbench uses informational, question, and warning dialog boxes that are designed to guide you in various ways as you work through your CFD analysis. Informational dialog boxes display messages that assist you in a specific task, or provide additional information relating to the task at hand. Question dialog boxes present questions concerning a task that is about to be performed, displaying an **OK** and a **Cancel** button in order to allow you to choose from one of two options (to proceed or not to proceed). Warning dialog boxes contain only an **OK** button and are designed to display a cautionary message indicating that you need to be aware that the application is about to change something or has internally changed something to maintain the consistency.

Note

You can set various FLUENT–specific preferences in Workbench (e.g., Launcher settings, color scheme, etc.). For more information, see Configuring ANSYS Workbench in the Workbench User Guide.

1.3. Creating FLUENT-Based Systems

There are two basic types of systems: analysis systems and component systems. The **Fluid Flow (FLUENT)** analysis system allows you to perform a complete CFD analysis and contains cells that allow you to: create geometry, generate a mesh, specify settings in FLUENT, run the FLUENT solver, and visualize the results in CFD-Post.

The FLUENT component system allows you to access the FLUENT application from within Workbench and contains only the cells needed to specify settings in FLUENT and run the FLUENT solver. When using a FLUENT component system, a mesh must be imported into the system or provided through a connection from an upstream system.

Important

A separate cell for results visualization is only needed when using CFD-Post. The post-processing capabilities in FLUENT can be accessed from both the **Fluid Flow (FLUENT)** analysis system and the FLUENT component system.

For more information, please see the following sections:

1.3.1. Creating FLUENT-Based Analysis Systems
1.3.2. Creating FLUENT-Based Component Systems

1.3.1. Creating FLUENT-Based Analysis Systems

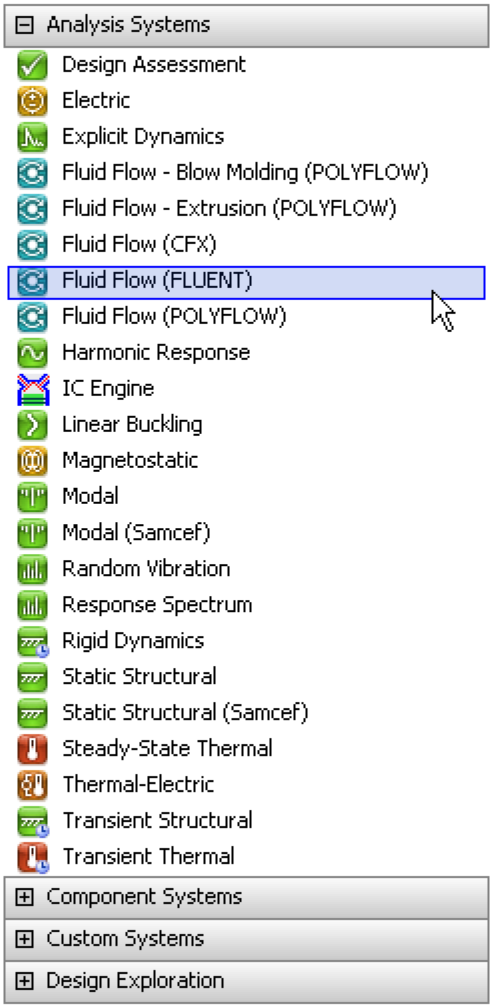
You can create a **Fluid Flow (FLUENT)** analysis system in Workbench by double-clicking on **Fluid Flow (FLUENT)** under **Analysis Systems** in the Toolbox.

Important

You can also create a **Fluid Flow (FLUENT)** analysis system by left-clicking on **Fluid Flow (FLUENT)** under **Analysis Systems** in the Toolbox, and then dragging it onto the Project Schematic.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.2 Selecting the Fluid Flow (FLUENT) Analysis System in Workbench



The new **Fluid Flow (FLUENT)** analysis system appears in the Project Schematic as a box containing several cells (*Figure 1.3* (p. 7)). Each cell corresponds to a typical task you would perform to complete a CFD analysis. The following cells are available in a **Fluid Flow (FLUENT)** analysis system:

Geometry
: allows you to define the geometrical constraints of your analysis. You can use the context menu (by right-clicking on the cell) to import a pre-existing geometry into the system. Double-clicking on the **Geometry** cell opens ANSYS DesignModeler where you can create a new geometry or modify an existing geometry.

Mesh
: allows you to define and generate a computational mesh for your analysis. Double-clicking on the **Mesh** cell opens ANSYS Meshing and loads the current mesh database (or the geometry defined by the **Geometry** cell) if you have not yet begun working on the mesh. Alternatively, you can use the context menu (by right-clicking on the **Mesh** cell) to import a pre-existing FLUENT mesh into the system.

Important

Importing a FLUENT mesh file into the **Mesh** cell results in the **Mesh** cell becoming the starting point for your analysis (and the name of the **Mesh** cell changes to **Imported Mesh**). Therefore, the **Geometry** cell (and data it contains) will be deleted from the system. The deleted **Geometry** cell can be retrieved by selecting **Reset** from the context menu of the **Imported Mesh** cell.

Important

FLUENT meshes imported into the **Mesh** cell cannot be modified by the ANSYS Meshing application.

Setup
allows you to define the boundary conditions, physical models and solver settings for the FLUENT analysis. Double-clicking on the **Setup** cell opens FLUENT and loads the mesh defined by the **Mesh** cell as well as any FLUENT settings that have already been specified. Alternatively, you can use the context menu (by right-clicking on the **Setup** cell) to import a pre-existing FLUENT case or mesh file into the system. After you specify the file you want to import, FLUENT will open and load the file.

Important

If you open FLUENT before defining a mesh, FLUENT will open without loading any files. You can then choose to import files from the **File** menu in FLUENT.

Important

Importing a FLUENT case or mesh file into the **Setup** cell or the FLUENT application results in the **Setup** cell becoming the starting point for your analysis. Therefore, the **Geometry** and **Mesh** cells (and any data they contain) will be deleted from the system.

Solution
allows you to calculate a solution in FLUENT. Double-clicking on the **Solution** cell opens FLUENT and loads the current FLUENT case and data files. If you have not yet performed any calculations, FLUENT will load the mesh file as well as any settings that have been specified.

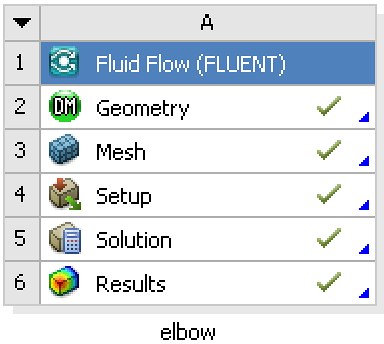
Important

You can also use the **Solution** cell context menu to import a pre-existing FLUENT data file to use for initial solution data. If you have not yet performed any calculations, FLUENT will load this data file in addition to the mesh and settings.

Results
allows you to display and analyze the results of the CFD analysis. Double-clicking on the **Results** cell opens CFD-Post and loads the current FLUENT case and data files as well as the current CFD-Post state file.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.3 A Fluid Flow (FLUENT) Analysis System



Note

While it is possible to apply different names for the **Setup** or the **Solution** cells by right-clicking either cell, and selecting the **Rename** option in the context menu, it is not generally recommended to do so.

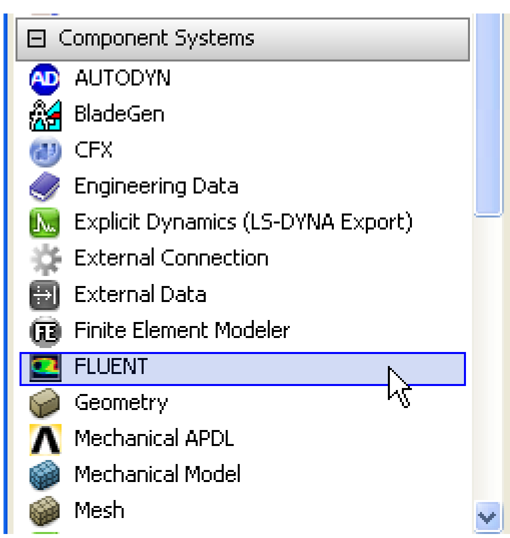
1.3.2. Creating FLUENT-Based Component Systems

Similarly, you can create a FLUENT-based component system in Workbench by double-clicking FLUENT under **Component Systems**.

Important

You can also create a FLUENT component system by left-clicking on FLUENT under **Component Systems** in the Toolbox, and then dragging it onto the Project Schematic.

Figure 1.4 Selecting the FLUENT Component System in Workbench

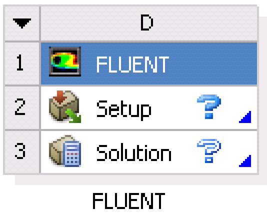


The new FLUENT component system appears in the Project Schematic as a box containing two cells: the **Setup** cell and the **Solution** cell (*Figure 1.5* (p. 8)). The **Setup** and **Solution** cells in a FLUENT component system work in the same manner as described above for the **Fluid Flow (FLUENT)** analysis

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

system. The only difference is that the mesh must originate from a file imported into the **Setup** cell or the FLUENT application, or it must be provided through a connection from an upstream system.

Figure 1.5 A FLUENT Component System



1.4. Understanding Cell States with FLUENT in Workbench

Workbench integrates multiple data-integrated (e.g., FLUENT) and native applications into a single, seamless project flow, where individual cells can obtain data from and provide data to other cells. Workbench provides visual indications of a cell's state via icons on the right side of each cell. Brief descriptions of the each possible state are provided below. For more information about cell states, see the Workbench on-line help:

- Unfulfilled ( ) indicates that required upstream data does not exist. For example, when you first create a new **Fluid Flow (FLUENT)** analysis system, all cells downstream of the **Geometry** cell appear as **Unfulfilled** because you have not yet specified a geometry for the system.

- Refresh Required ( ) indicates that upstream data has changed since the last refresh or update. For example, after you assign a geometry to the **Geometry** cell in your new **Fluid Flow (FLUENT)** analysis system, the **Mesh** cell appears as **Refresh Required** since the geometry data has not yet been passed from the **Geometry** cell to the **Mesh** cell.

- Attention Required ( ) indicates that the current upstream data has been passed to the cell, however, you must take some action to proceed. For example, after you launch FLUENT from the **Setup** cell in a **Fluid Flow (FLUENT)** analysis system that has a valid mesh, the **Setup** cell appears as **Attention Required** because additional data must be entered in FLUENT before you can calculate a solution.

- Update Required ( ) indicates that local data has changed and the output of the cell needs to be regenerated. For example, after you launch ANSYS Meshing from the **Mesh** cell in a **Fluid Flow (FLUENT)** analysis system that has a valid geometry, the **Mesh** cell appears as **Update Required** because the **Mesh** cell has all the data it needs to generate a FLUENT mesh file, but the FLUENT mesh file has not yet been generated.

- Up-To-Date ( ) indicates that an update has been performed on the cell and no failures have occurred (or an interactive calculation has been completed successfully). For example, after FLUENT finishes performing the number of iterations that you request, the **Solution** cell appears as **Up-to-Date**.

- Interrupted ( ) indicates that you have interrupted an update (or canceled an interactive calculation that is in progress). For example, if you select the **Cancel** button in FLUENT while it is iterating, FLUENT completes the current iteration and then the **Solution** cell appears as **Interrupted**.

- Input Changes Pending ( ) indicates that the cell is locally up-to-date, but may change when next updated as a result of changes made to upstream cells. For example, if you change the **Mesh** in an **Up-to-Date Fluid Flow (FLUENT)** analysis system, the **Setup** cell appears as **Refresh Required**, and the **Solution** and **Results** cells appear as **Input Changes Pending**.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Pending () indicates that a batch or asynchronous solution is in progress. When a cell enters the **Pending** state, you can interact with the project to exit Workbench or work with other parts of the project. If you make changes to the project that are upstream of the updating cell, then the cell will not be in an up-to-date state when the solution completes.

If a particular action fails, Workbench provides a visual indication as well. Brief descriptions of the failure states are described below.

- Refresh Failed, Refresh Required () indicates that the last attempt to refresh cell input data failed, and so the cell needs to be refreshed.
- Update Failed, Update Required () indicates that the last attempt to update the cell and calculate output data failed, and so the cell needs to be updated.
- Update Failed, Attention Required () indicates that the last attempt to update the cell and calculate output data failed, and so the cell requires attention.

If an action results in a failure state, you can view any related error messages in the **Messages** window by clicking the **Show Messages** button on the lower right portion of Workbench.

1.5. Starting FLUENT in Workbench

This section describes how to start FLUENT in Workbench using FLUENT-based systems and how you can use FLUENT Launcher within Workbench.

For more information, please see the following sections:

1.5.1. Starting FLUENT from a FLUENT-Based System
1.5.2. Specifying FLUENT Launcher Settings Within Workbench

1.5.1. Starting FLUENT from a FLUENT-Based System

You can start ANSYS FLUENT by double-clicking on the **Setup** cell in a **Fluid Flow (FLUENT)** analysis system or a FLUENT component system. FLUENT launches and loads the **Setup** cell's input data (e.g., mesh) and the **Setup** cell's local data, if it exists (e.g., FLUENT settings). If no mesh has been specified, FLUENT launches and waits for your input.

You can also start ANSYS FLUENT by double-clicking on the **Solution** cell in a **Fluid Flow (FLUENT)** analysis system or a FLUENT component system. FLUENT launches and loads the current case and data files, as well as the **Setup** cell's input data (e.g., mesh), the **Setup** cell's local data, if it exists (e.g., FLUENT settings), and the **Solution** cell's initial data, if it exists. If no mesh has been specified, FLUENT launches and waits for your input.

Important

When FLUENT is launched from the **Setup** cell, it loads only the mesh and settings that served as the starting point for your analysis and are associated with the **Setup** cell. In order to load the current case and data files or the initial data file, you must launch FLUENT from the **Solution** cell.

1.5.2. Specifying FLUENT Launcher Settings Within Workbench

You can use FLUENT in Workbench on either Windows or Linux machines, both interactively and in batch mode. You can also start FLUENT on Linux from a Workbench session running on Windows and

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you can use the same FLUENT-specific project (and related files) in Workbench using both Linux and Windows hardware interchangeably. The information is this section is the same for both Windows and Linux, except where noted.

When you start ANSYS FLUENT from either type of FLUENT-based system within Workbench, FLUENT Launcher will appear by default. Most FLUENT Launcher settings are available, except for the following options:

- **Version** (disabled)
- **Working Directory** (disabled)
- **Use Journal File** (not available)

The **Do not show this panel again** option allows you to bypass FLUENT Launcher for subsequent FLUENT sessions. This option is only available when running FLUENT in Workbench.

Important

When using LSF to schedule a FLUENT run from Workbench, by default the working directory is used for checkpointing. You can specify an alternate directory for checkpointing using the `LSB_CHKPNT_DIR` environment variable.

Important

Note that, when using FLUENT with Workbench on Linux, the **FLUENT Root Path** option is disabled in the **General** tab of FLUENT Launcher.

To start your FLUENT simulation on a Linux cluster from a Workbench session running on Windows, use the **Use Remote Linux Nodes** option in FLUENT Launcher. This option is available when the **Parallel** option is enabled under **Processing Options**. Once this option is enabled, the **Remote** tab is visible when you click the **Show More Options** button.

Important

The **Remote** tab of FLUENT Launcher can only be used for 64-bit Linux machines.

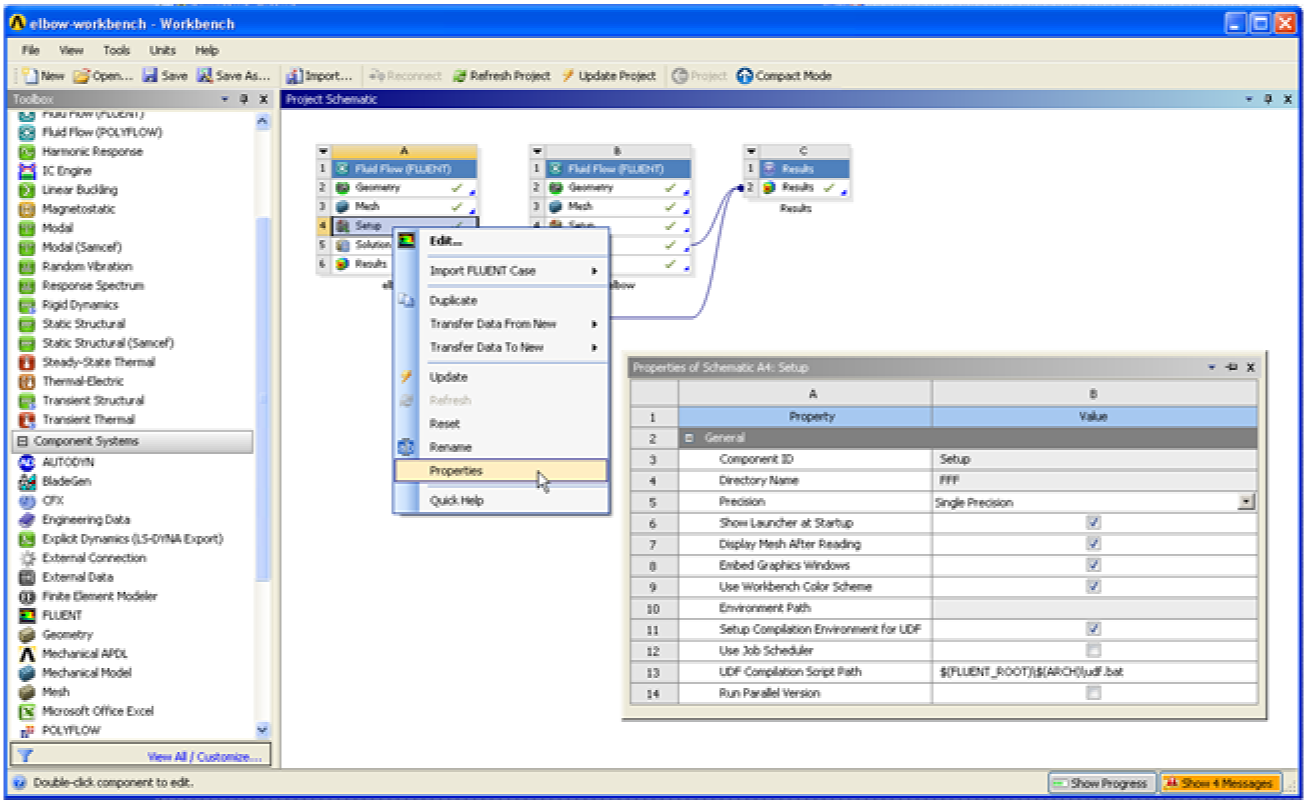
For more information about FLUENT Launcher, see Starting ANSYS FLUENT Using FLUENT Launcher. For more information about using FLUENT Launcher to access remote Linux clusters, see Setting Remote Options in FLUENT Launcher.

1.5.2.1. Specifying FLUENT Launcher Settings Using Cell Properties

You can view the properties of a selected cell in Workbench by selecting the **Properties** option under the **View** menu, or by right-clicking a cell and selecting **Properties** from the context menu. The properties of the selected cell are displayed in the **Properties Window** in Workbench.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.6 Properties for FLUENT-Based Systems in Workbench



The FLUENT-based system **Setup** and **Solution** cells have the following properties that you can set for FLUENT Launcher:

Use Setup Launcher Settings
: (for **Solution** cells only) allows you to specify that the current systems **Solution** cell should use the FLUENT Launcher property settings from the current system's **Setup** cell.

Precision
: allows you to choose either the single-precision or the double-precision solver

Show Launcher at Startup
: allows you to show or hide FLUENT Launcher when FLUENT starts.

Display Mesh After Reading
: allows you to show or hide the mesh after the mesh or case/data is read into FLUENT.

Embed Graphics Windows
: allows you to embed the graphics windows in the FLUENT application window, or to have them free-standing.

Use Workbench Color Scheme
: allows you to use either the classic black background color in the graphics window, or to use the **Workbench** color scheme.

Setup Compilation Environment for UDF
: allows you to specify compiler settings for compiling user-defined functions (UDFs) with FLUENT.

Use Job Scheduler
: allows you to specify settings for running FLUENT with the available job scheduler. On Linux, you can use either LSF, SGE, or PBS Pro, as the job scheduler. On Windows, you can use the Microsoft Job

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scheduler, or, when **Run Parallel Version** and **Use Remote Linux Nodes** are selected, you can use the LSF, SGE, or PBS Pro job schedulers.

Run Parallel Version
allows you to choose to run the parallel version of FLUENT or not.

UDF Compilation Script Path
(if **Setup Compilation Environment for UDF** is selected) allows you to specify the path to the UDF compilation script.

Use Remote Linux Nodes
(available only when **Run Parallel Version** is selected) allows you to run your FLUENT simulation on 64-bit Linux machines.

If **Run Parallel Version** is selected, the following additional properties are available:

Number of Processors
allows you to set the number of processors you wish to use for the parallel calculations (e.g., 2, 4, etc.).

Interconnect
allows you to set the interconnects you wish to use for the parallel calculations (e.g., ethernet, myrinet, infiniband, etc.).

MPI Type
allows you to set the MPI type you wish to use for the parallel calculations (e.g., MPICH2, Platform, etc.).

Use Shared Memory
allows you to specify if shared memory is to be used or not.

Machine Specification
(if **Use Shared Memory** is not selected) allows you to specify a list of machine names, or a file that contains machine names.

Machine List
(if **Use Shared Memory** is not selected and **Machine List** is selected as the **Machine Specification**) allows you to specify a list of machine names to run the parallel job.

Machine Filename
(if **Use Shared Memory** is not selected and **File Containing Machine List** is selected as the **Machine Specification**) allows you to specify the name of the file that contains a list of machines to run the parallel job.

If **Use Remote Linux Nodes** is selected, the following additional properties are available (for 64-bit Linux machines only):

Remote FLUENT Root Path
allows you to specify the remote FLUENT Linux installation root path.

Use Specified Remote Working Directory
allows you to specify a directory other than `temp` directory as the working directory for the remote Linux nodes. When this property is selected, you can specify the directory in the **Remote Working Directory** property that appears.

Remote Spawn Command
allows you to specify the command to connect to the remote node (the default is RSH).

Use Remote Cluster Head Node
allows you to specify the remote node that FLUENT will connect to for spawning (e.g., via `rsh` or `ssh`). When this property is selected, you can specify the remote node in the **Remote Host Name** property that appears.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

When the **PBS Pro** option is selected for the **Job Scheduler**, the host specified in the **Use Remote Cluster Head Node** field should be the PBS Pro submission host.

If **Use Job Scheduler** is selected, the following additional properties are available:

Computer Cluster Head Node Name
allows you to specify the name of the compute cluster head node.

Job Template
(available only when running Windows HPC 2008 Server Scheduler) allows you to create a custom submission policy to define the job parameters for an application. The cluster administrator can use job templates to manage job submission and optimize cluster usage.

Node Group
(available only when running Windows HPC 2008 Server Scheduler) allows you to specify a collection of nodes. Cluster administrators can create groups and assign nodes to one or more groups.

Processor Unit
(available only when running Windows HPC 2008 Server Scheduler) allows you to choose the following:

- **Core** refers to a single, named host in the cluster.
- **Socket** refers to a set of processors with a dedicated memory bus. This is also known as a non-uniform memory access (NUMA) node.
- **Node** refers to an individual CPU on a node. For example, a dual-core processor is considered two cores.

For more information about running FLUENT jobs using the Windows HPC 2008 Server Scheduler, see the Frequently Asked Questions section of the Customer Portal.

Start When Resources Are Available
allows you to start the job when resources are available or not.

Create Job Submission XML
allows you to create the job submission XML file.

Job Submission XML File
(if **Create Job Submission XML** is selected) allows you to specify the name of the job submission XML file.

If **Use Job Scheduler** and **Use Remote Linux Nodes** are selected, the following additional properties are available:

Job Scheduler
allow you to specify the job scheduler for the remote Linux nodes. Available options are **LSF** (the default), **SGE**, or **PBS Pro**.

If **LSF** is selected as the job scheduler, the following additional properties are available:

Use LSF Queue
allows you to use an LSF queue.

LSF Queue
(when **Use LSF Queue** is selected) allows you to specify the name of the LSF queue.

Use Checkpointing
allows you to use checkpointing with LSF.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enable Automatic Checkpointing
(when **Use Checkpointing** is selected) allows you to automatically checkpoint within a specific period of time.

Checkpointing Period
(when **Use Checkpointing** and **Enable Automatic Checkpointing** are selected) allows you to specify a time period for automatic checkpointing.

Important

When using LSF to schedule a FLUENT run from Workbench, by default the working directory is used for checkpointing. You can specify an alternate directory for checkpointing using the `LSB_CHKPNT_DIR` environment variable.

If **SGE** is selected as the job scheduler, the following additional properties are available:

SGE Qmaster
allows you to set the machine in the SGE job submission host list.

SGE Queue
allows you to set the queue where you want to submit your FLUENT jobs. Leave this field blank if you want to use the default queue.

SGE PE
allows you to set the parallel environment where you want to submit your FLUENT jobs. Leave this field blank if you want to use the default parallel environment.

Use SGE Settings
allows you to use SGE settings.

SGE Settings
(when **Use SGE Settings** is selected) allows you to specify SGE settings.

Important

When running FLUENT in Workbench via SGE, there may be instances when not enough time is allotted for FLUENT to start. By default, Workbench performs two steps: starting the FLUENT process and establishing initial contact, with a default wait time of one minute; completing the FLUENT start up process and establishing a final connection (e.g., to spawn parallel nodes, to checkout licenses, etc.), with a default wait time of five minutes. You can change the waiting time increments for each step by defining the `FLUENT_WB_MAX_STARTUP_WAIT` environment variable before starting Workbench. Note that you *cannot* define this environment variable within FLUENT Launcher. The value of this environment variable is equivalent to the wait time in minutes for each start up step. So, the overall wait time is double the value specified for the environment variable. Fractions are allowed as values for the environment variable, and any value less than or equal to zero will let Workbench wait indefinitely for FLUENT to start. Note that while Workbench is waiting for FLUENT to start, the Workbench interface is locked, and if, for some reason FLUENT fails to start, you will have to manually kill the Workbench session.

If **PBS Pro** is selected as the job scheduler, the following additional property is available (on Linux only):

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PBS Submission Host
(optional) allows you to specify the name of the PBS Pro submission host if the machine you are using to run Workbench cannot submit a job to PBS Pro.

These properties are the same as the settings within FLUENT Launcher. For more information about FLUENT Launcher, see the FLUENT User's Guide.

In addition, note that the **Solution** cell contains additional general properties and some specific properties related to the solution process:

- **General**
  - **Component ID**: the name of the cell (not editable).
  - **Directory Name**: the name of the directory (“FFF”) where solution files are located (not editable).
  - **Use Setup Launcher Settings**: enable or disable the use of the **Setup** cell's FLUENT Launcher settings (see above).
- **Solution Process**
  - **Update Option**: enables the solution process to be either **Run in Foreground** (solutions are run within the current Workbench session), **Run in Background** (solutions are run in the background on the local machine), or **Submit to Remote Solve Manager** (solutions are run in the background by submitting the solution to Remote Solve Manager (RSM)). For more information about these options and the Remote Solve Manager, please refer to Submitting Solutions for Local, Background, and Remote Solve Manager (RSM) Processes in the ANSYS Workbench User's Guide.

1.5.2.2. Copying FLUENT Launcher Property Settings

Typically, FLUENT Launcher settings are specified in the FLUENT Launcher dialog box when FLUENTis launched from the **Setup** cell.

By default, the **Solution** cell uses the same FLUENT Launcher property settings as the **Setup** cell. If you want the **Solution** cell’s FLUENT Launcher settings to be different than those specified for the **Setup** cell, you can disable the **Use Setup Launcher Settings** property setting (see *Specifying FLUENT Launcher Settings Within Workbench* (p. 9)) and then set the **Solution** cell’s FLUENT Launcher property settings without impacting the settings for the **Setup** cell.

You can copy the **Setup** cell’s FLUENT Launcher settings to the **Solution** cell by selecting the **Copy Launcher settings to Solution cell** option from the **Setup** cell’s context menu (available only when the **Use Setup Launcher Settings** property setting is disabled). Note that the values that are written are retained, even if you later disable this setting. Likewise, you can also copy the **Solution** cell’s FLUENT Launcher settings to the **Setup** cell by selecting the **Copy Launcher settings to Setup cell** option from the **Solution** cell’s context menu (available only when the **Use Setup Launcher Settings** property setting is disabled).

1.6. Saving Your Work in FLUENT with Workbench

Data that is read into and written by FLUENT when it is run within Workbench is split into two parts:

- Setup data
- Solution data

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup data is the data used to start a simulation over from the beginning. This data is associated with the **Setup** cell and includes the mesh (`.msh`) file and the settings (`.set`) file.

Important

The settings file is a file used when FLUENT is run under Workbench. It contains the case settings but does not contain mesh data. The settings file and the mesh file are read by FLUENT whenever FLUENT is launched from the **Setup** cell.

Important

Note that sometimes, instead of a mesh file, a case file (`.cas`) is used to represent the mesh. In this situation, FLUENT reads the case file first and then reads the settings file if it exists. Therefore, the settings stored in the settings file will overwrite any settings that might be contained in the case file.

Solution data is the data that results from performing a calculation and is used to restart a simulation from existing data. This data is associated with the **Solution** cell and includes the current case (`.cas`) file and the current data (`.dat`) file.

Important

The case file and the data file are read by FLUENT whenever FLUENT is launched from the **Solution** cell.

When working in Workbench, your work in FLUENT is automatically saved as needed. For example, whenever you close FLUENT or save your Workbench project, your unsaved data is automatically saved.

You can save your Workbench project directly from FLUENT in two ways:

1. Select the **Save Project** option under the **File** menu within FLUENT.

   File → **Save Project**
2. Select the **Save Project** option from the **Write a File** icon in the FLUENT toolbar.

Alternatively, you can also save your Workbench project by selecting the **Save** option under the **File** menu within Workbench or by selecting the **Save** icon from the Workbench toolbar.

1.7. Exiting FLUENT and Workbench

You can end your FLUENT session by using the **Close FLUENT** option under the **File** menu.

File → **Close FLUENT**

You can end your Workbench session by using the **Exit** option under the **File** menu.

File → **Exit**

All open applications that are associated with your Workbench session, including any open instances of ANSYS FLUENT, are automatically closed upon exiting Workbench. If there is any unsaved data in Workbench or any of the open applications associated with your Workbench session, you will be prompted to save your project before exiting Workbench.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

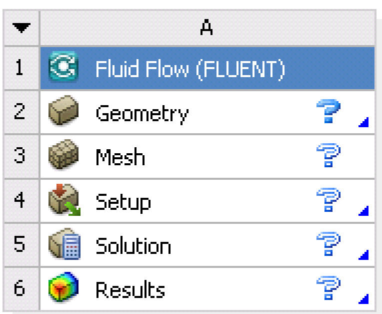
There are several other circumstances in which open instances of FLUENT as well as other applications are automatically closed:

- Whenever you close the current project in Workbench, all open applications are automatically closed.
- Whenever you open a different project in ANSYS Workbench, all open applications associated with the original project are automatically closed.
- Whenever a system is deleted, all open applications associated with that system are automatically closed.
- Whenever data is reset or cleared from a cell, all open applications associated with that cell are automatically closed.

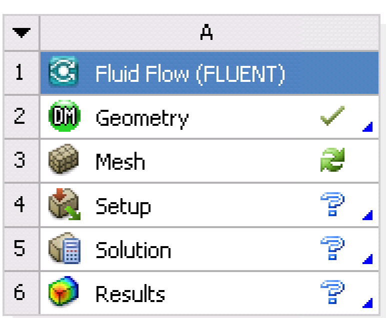
1.8. An Example of a FLUENT Analysis in Workbench

This example describes when the files that are generated and used by FLUENT are written and how the cell states change as you work with a FLUENT-based system in Workbench.

1. Add a new **Fluid Flow (FLUENT)** analysis system to the Project Schematic. The state of the **Geometry** cell is **Attention Required** and that the states for the **Mesh**, **Setup**, **Solution**, and **Results** cells are **Unfulfilled**.

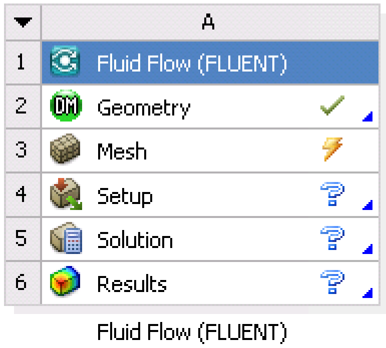


2. Import a geometry file by using the context menu on the **Geometry** cell. The state of the **Geometry** cell becomes **Up-to-Date** and the state of the **Mesh** cell becomes **Refresh Required**.

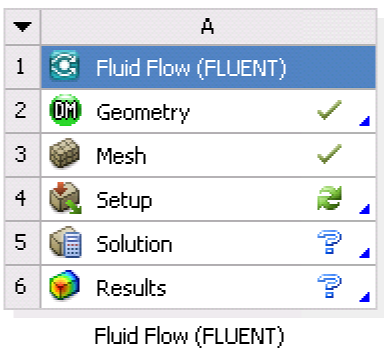


Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

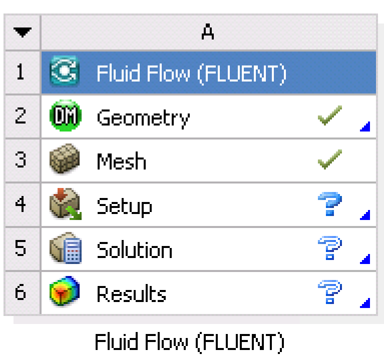
3. Double-click the **Mesh** cell. The ANSYS Meshing application launches and loads the geometry file. The state of the **Mesh** cell becomes **Update Required**.



4. In the ANSYS Meshing application, specify settings for the mesh, then select the **Update** command. The mesh is generated, the mesh (`.msh`) file is written, the state of the **Mesh** cell becomes **Up-to-Date**, and the state of the **Setup** cell becomes **Refresh Required**.

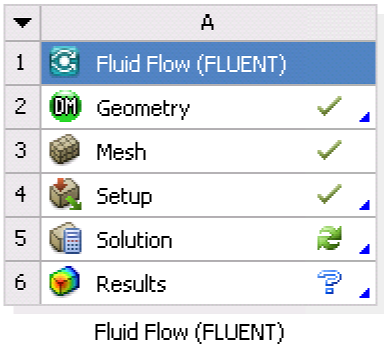


5. Double-click the **Setup** cell. FLUENT launches and loads the mesh file. The state of the **Setup** cell becomes **Attention Required**.

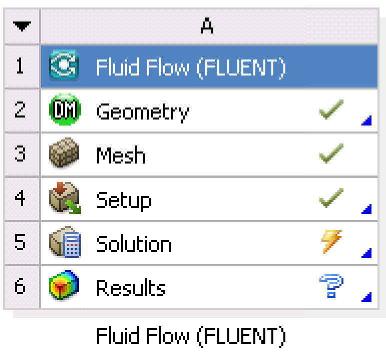


6. In FLUENT, specify boundary conditions, initialize the solution, and enter a non-zero number of iterations on the **Run Calculation** task page. The state of the **Setup** cell becomes **Up-to-Date**, and the state of the **Solution** cell becomes **Refresh Required**.

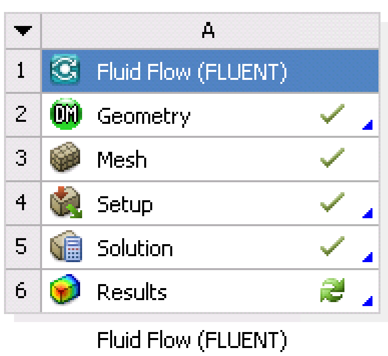
Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7. In the FLUENT application, select the **Calculate** button. The settings (`.set`) file is written and iterations begin. The state of the **Solution** cell becomes **Update Required**.



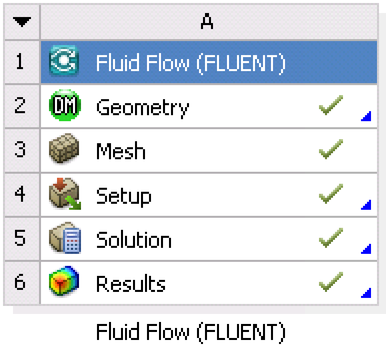
Iterations are completed, or the solution meets the convergence criteria. The state of the **Solution** cell becomes **Up-to-Date** and the state of the **Results** cell becomes **Refresh Required**.



8. Double-click the **Results** cell. CFD-Post launches.

   The case (`.cas`) and data (`.dat`) files are written, CFD-Post loads the case and data files, and the state of the **Results** cell becomes **Up-to-Date**.

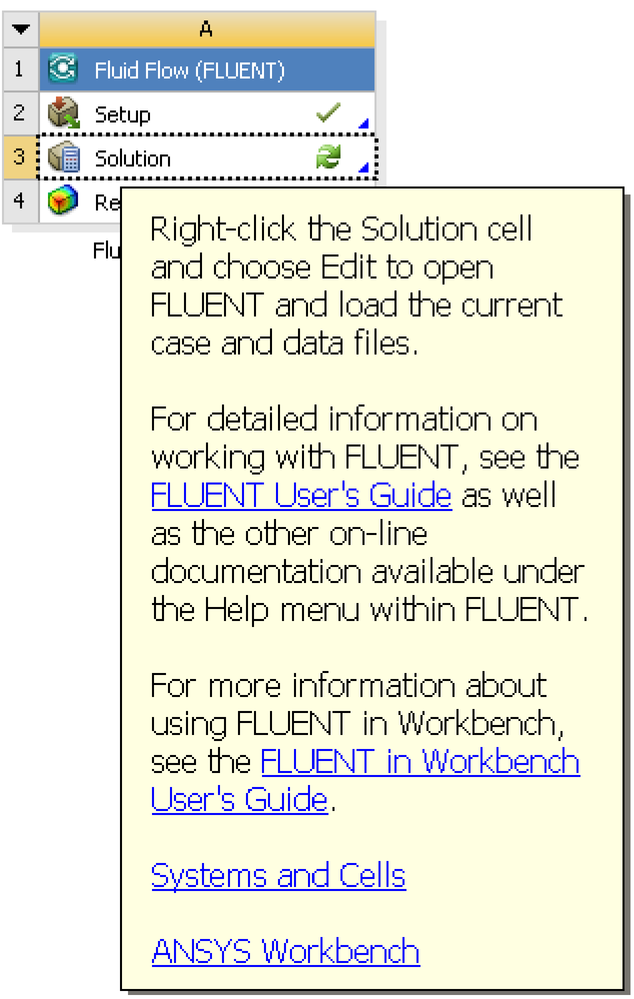
Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



1.9. Getting Help for FLUENT in Workbench

Workbench offers three levels of help:

- Quick help - available for most cells in a system. Click the blue triangle in the bottom right corner of the cell to see a brief help description on that cell. For FLUENT-based systems, FLUENT-specific quick help is available for the **Setup** and **Solution** cells, providing you with instructions for proceeding further. For example:



Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Sidebar or context-sensitive help - available at any time by clicking F1.
- Online help - available from the **Help** menu, or from any of the links in the quick help or sidebar help.

For more information about Workbench help, see the on-line documentation.

FLUENT documentation and help is available from the FLUENT menu once the FLUENT application is running. The documentation is automatically installed when you install Workbench.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 2: Working With FLUENT in Workbench

This chapter provides instructions for using FLUENT in Workbench.

2.1. Importing Mesh, Case, and Data Files
2.2. Using the Update Command
2.3. Refreshing FLUENT Input Data
2.4. Deleting Solution and Setup Cell Data for FLUENT-Based Systems
2.5. Interrupting, Restarting, and Continuing a Calculation
2.6. Connecting Systems in Workbench
2.7. Duplicating FLUENT-Based Systems
2.8. Reviewing Mesh Manipulation Operations in FLUENT
2.9. Changing the Settings and Mesh in FLUENT
2.10. Case Modification Strategies with FLUENT and Workbench
2.11. Working With Input and Output Parameters in Workbench
2.12. Viewing Your FLUENT Data Using ANSYS CFD-Post
2.13. Understanding the File Structure for FLUENT in Workbench
2.14. Working with ANSYS Licensing
2.15. Using FLUENT With the Remote Solve Manager (RSM)
2.16. Using Custom Systems
2.17. Using Journaling and Scripting with FLUENT in Workbench
2.18. Performing System Coupling Simulations Using FLUENT in Workbench
2.19. Performing FLUENT and Ansoft Coupling in Workbench

2.1. Importing Mesh, Case, and Data Files

You can directly import a FLUENT case or mesh file into a FLUENT-based system by right-clicking on the **Setup** cell and selecting the **Import FLUENT Case...** option from the context menu. You will be prompted for a specific case or mesh file. After you select a file, FLUENT launches and loads the file you specified. If you import a case file, the physical model settings, boundary condition information, material properties and solver settings are also loaded into FLUENT.

When you import a case file in which the number of iterations (or time-steps) specified in the **Run Calculation** task page is greater than `0`, the state of the **Setup** cell becomes **Up-to-Date** and the state of the **Solution** cell becomes **Refresh Required**.

When you import a mesh file or a case file in which the number of iterations (or time-steps) specified in the **Run Calculation** task page is `0`, the state of the **Setup** cell becomes **Attention Required** because additional information must be specified in FLUENT before you can proceed.

If you import a case or mesh file into a system that contains a **Mesh** cell, you are informed that the imported information will not be compatible with the information coming from the **Mesh** cell and asked whether the **Mesh** cell (and **Geometry** cell, if it also exists) should be deleted and replaced by the mesh from the imported file.

If you import a case or mesh file into a system that has a connection to a **Mesh** cell in an upstream system, you are informed that the mesh is being provided to the current system from an upstream

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

system and asked whether the connection should be deleted and the mesh replaced with the mesh from the imported file.

Important

If a case or mesh file is imported into a **Setup** cell that already has settings information associated with it, the settings information is deleted when the new file is imported.

You can also directly import a FLUENT data file into a FLUENT-based system by right-clicking on the **Solution** cell and selecting the **Import Initial Data...** option from the context menu. You will be prompted for a data file. Importing a data file does not affect the state of the system. This option is not available if FLUENT is already open or an incoming connection is providing data.

In addition, you can directly import a FLUENT data set (e.g., legacy FLUENT data, or a large data set solved on an external cluster) into a FLUENT-based system by right-clicking on the **Solution** cell and selecting the **Import Final Data...** option from the context menu. You will be prompted for a data file. Importing a data file in this manner causes the **Solution** cell to be **Up-to-Date**, and allows you to import data directly into a FLUENT-based system without the need to perform any additional solution tasks in Workbench, i.e., you can immediately perform any post-processing of the data using ANSYS CFD-Post, or you can directly transfer that data to another system (e.g., to perform one-way fluid-structure interaction (FSI) analysis). Note that after the state of the **Solution** cell is marked as **Up-to-Date**, then this option is no longer available. This option is also not available if FLUENT is already open or an incoming connection is providing data.

Important

For unsteady cases, if the last time-step data file is imported as a final data set, then ANSYS CFD-Post can load all the previous time-step data as well, provided:

- All the previous time-step related (case/data) files are kept at same location as that of final data, and that they have the same base file name as the last data file (assuming that the case file also has a consistent base file name).
- ANSYS CFD-Post directly reads this information from the data file. The FLUENT-based system's **Solution** cell will not extract any of the solution history and further calculation will result in loss of this information. Note that performing additional calculations will change the solution history but it does *not* effect the data that is loaded into ANSYS CFD-Post if the additional data is output with the same base file name as the imported data file.
- Because of this reason, only the case/data file pairs originally generated by FLUENT can be imported in **Setup** cell or the **Solution** cell as final case/data pairs.
  - If the case file imported into the **Setup** cell is not same (at least the name) as that of original case file, ANSYS CFD-Post will not be able to load the data.
  - If you want to import a pair of case/data files with a different name, then you need to perform at least one iteration in FLUENT in Workbench, automatically saving this new set of data once FLUENT is closed or the Workbench is saved. Alternatively, you can manually rename the case/data file sets to have the same base file name prior to importing the final data set.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Final data files that are the result of calculations performed in FLUENT contain information related to solution history. This solution history information is not retained if FLUENT imports the data file as its final data, or if further calculations are later performed in FLUENT (even though ANSYS CFD-Post will still be able to read the solution history information from the imported data file).

For example, for an unsteady problem, if you calculate five solution points (`test-1.cas`, `test-1.dat`, ... etc.) using standard FLUENT (outside of Workbench), the entire solution would be available in the solution history (i.e., any solution point can be read by FLUENT using the **Solution Files** option under the **File** menu). In addition, all the solution points are also available to ANSYS CFD-Post, for post-processing. If you subsequently run FLUENT in Workbench and import the last case and data set (e.g., `test-5.cas` and `test-5.dat`) from the **Solution** cell as the final data, then only the imported data set is available in FLUENT's solution history, however, ANSYS CFD-Post still has access to all of the solution history (as long as the other solution files are contained in the same directory as the imported data file). After importing the final data set, if you continue calculations, then the solution history of the imported data file (i.e., `test-1.cas`, `test-1.dat`...`test-4.cas`, `test-4.dat`). is no longer available to ANSYS CFD-Post.

Note

It should be noted that only FLUENT data files (`.dat` or `.dat.gz`) are registered as part of FLUENT solution history. A FLUENT data file can have additional data quantities (chosen by you) stored in it along with other standard quantities.

When case and “initial” data files are imported into FLUENT, they are treated as start-up files and not results files. Therefore, when you import a case and data file from a previous calculation, the state of the **Solution** cell becomes **Refresh Required** and not **Up-to-Date**.

Important

In order to post-process the results from an existing set of case and “initial” data files in ANSYS CFD-Post, at least one iteration (or time-step) must be performed in FLUENT from within Workbench in order to bring the state of the **Solution** cell to **Up-to-Date**. Alternatively, if you do not intend to perform any calculations, you could create a **Results** component system, double-click its **Results** cell to open ANSYS CFD-Post and then load the FLUENT data file using the **Load Results** option under the **File** menu in ANSYS CFD-Post.

Alternatively, you can also import pre-existing mesh, case, and data files from within FLUENT by using one of the following options:

File → Import → Case...

File → Import → Data...

File → Import → Case and Data...

When mesh, case, and data files are imported from within FLUENT using these commands, the behavior is exactly the same as when files are imported from the Project Schematic.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In addition, you can also import the mesh from pre-existing mesh and case files from within FLUENT by using the following option:

File → Import → Mesh...

When using this command, you are given two options:

Discard Case, Read New Mesh
- this discards any settings information currently in FLUENT and imports the specified file. If the specified file is a case file, the settings information from that case file is also imported.

Replace Mesh
- this preserves the settings information currently in FLUENT and imports only the mesh from the specified file.

2.2. Using the Update Command

The **Update** command is available from the context menu of all cells, from the context menu for the system, and from the Workbench Toolbar, the Workbench **Tools** menu, and the context menu for the Project Schematic.

When selected from a cell, the **Update** command updates the current cell and all upstream cells that must be updated to bring the current cell **Up-to-Date**. When a cell is updated, any new upstream data is passed to it before performing the update command.

When selected from the system, the **Update** command updates all of the out-of-date cells in the current system, as well as any cells in upstream systems that must be updated to bring the current system **Up-to-Date**.

When selected from the Workbench Toolbar, the Workbench **Tools** menu, or the context menu for the Project Schematic, the **Update** command updates all out-of-date cells in the project.

When updating the **Solution** cell in a FLUENT system, the following steps take place:

1. FLUENT launches in the background.
2. FLUENT performs either the number of iterations (or time-steps) specified in the settings or case file or the number of iterations required to reach convergence.
3. FLUENT writes the case and data files.
4. FLUENT exits.

Important

When an update is performed on a **Solution** cell and a FLUENT session from the same system as that **Solution** cell is open, the calculation will be performed in that open FLUENT session. This may not result in the action that you intended, therefore it is recommended that you close any open FLUENT sessions before executing an **Update** command. For example, if you have specified initial data using either a connection from an upstream solution cell or by importing initial data into the **Solution** cell and you open FLUENT from the **Setup** cell of the system, the initial data is not loaded (see *Starting FLUENT from a FLUENT-Based System* (p. 9)). If you subsequently perform an **Update**, the calculation will be performed in the open FLUENT session and the initial data you specified will not be used as the starting point.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Update** command is particularly useful when you make changes to upstream data that impact downstream data. For example, if you start with an **Up-to-Date Fluid Flow (FLUENT)** analysis system and then modify the mesh in the ANSYS Meshing application, you can simply select **Update** from the system's context menu to generate the new results.

When performing an **Update**, you can specify whether the **Solution** cell in a FLUENT-based system should be updated starting from the current data file or from the initial data. This is specified by selecting or deselecting the **Update from Current Solution Data if Possible** option from the context menu of the **Solution** cell. This option is enabled when there is a check mark to its left. This option is always enabled by default.

The **Update from Current Solution Data if Possible** option for the **Solution** cell in a FLUENT-based system will always have the highest priority. Here, the 'current solution data' refers to the updated solution of the first design point (DP0) or the most recently computed design point, according to the design point update order (option 1 below) as follows:

Currently, the following options are available:

1. Using the **Parameter Set** property view in Workbench (Design Point Update Order):
   - **Update from Current**, or
   - **Update Design Points in Order**
2. Using the options from the **Solution** cell context menu in the FLUENT-based system:
   - If the **Update from Current Solution Data if Possible** option is enabled (true), then:
     - In this case, the setting for the **Parameter Set** property **Update Design Points in Order** (if specified) will be ignored.
     - In this case, if the **Solution** cell has an upstream connection, then it will be ignored for all design points, except the initial design point, DP0
   - If the **Update from Current Solution Data if Possible** option is disabled (false), then:
     - In this case, the setting for the **Parameter Set** property **Update from Current** (if specified) will be ignored, and the initial data file (or upstream data file) or basic initialization will be used.

Initial data can be specified in four ways:

- On the **Solution Initialization** task page in FLUENT.
- Import an initial data file into the **Solution** cell or the FLUENT application (see *Importing Mesh, Case, and Data Files* (p. 23)).
- Create a connection from an upstream **Solution** cell (see *Connecting Systems in Workbench* (p. 33)).
- Specify a case modification strategy (see *Case Modification Strategies with FLUENT and Workbench* (p. 50)).

If you have imported an initial data file or created a connection to specify initial data, that data will override the initialization method specified on the **Solution Initialization** task page. If you have a case modification strategy defined, the initialization step in that strategy will always be performed regardless of whether initial data is specified in any other way.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

If the **Solution** cell in a FLUENT-based system is connected to the **Solution** cell in an upstream system, the **Import Initial Data...** option is not available from the context menu of the downstream **Solution** cell. If you want to import initial data into the downstream **Solution** cell, you must delete the connection first. Similarly, if the **Solution** cell already has imported initial data, a connection cannot be made to that cell from the **Solution** cell in an upstream system. Furthermore, you cannot create a connection from the **Solution** cell in an upstream system to any **Solution** cell in which you have ever imported initial data.

Important

When you select **Update** from the **Setup** cell in a FLUENT-based system and a FLUENT session is not already open, a FLUENT session launches in background mode. After the **Update** is complete, the background session of FLUENT remains open and waits for you to update the **Solution** cell. To close the open background session of FLUENT without updating the **Solution** cell, you can select the **Close FLUENT Session** option from the **Setup** cell's context menu.

Important

There is also an **Update** command in the ANSYS Meshing application which generates the mesh and creates the input files required by downstream cells. The **Generate Mesh** command in the ANSYS Meshing application generates the mesh but does not produce any input files. If a connection is made from an up-to-date **Mesh** cell, the state of the **Mesh** cell may become **Update Required**, indicating that the ANSYS Meshing application needs to generate an additional input file. This file can be generated by selecting the **Update** option from the context menu of the **Mesh** cell. If you try to open FLUENT before the **Mesh** cell is updated, a warning message is displayed informing you that you must update the **Mesh** cell before you can start FLUENT, since the mesh file required for FLUENT does not yet exist.

Important

Whether you edit the project through the **Setup** or **Solution** cells, your project's post-processing settings in FLUENT (including any surface definitions that you may have created), are saved at the start of iteration or when you close FLUENT. When FLUENT is later opened from either cell, these new settings will be available. Changes made to post-processing settings in FLUENT do not affect cell state.

2.3. Refreshing FLUENT Input Data

You can refresh the input data for a cell by right-clicking on the cell and selecting the **Refresh** option from the context menu. The **Refresh** command passes modified upstream data to the cell but does not conduct any long-running operations to regenerate the cell's output data.

For example, you can refresh the mesh by right-clicking on the **Setup** cell in Workbench and selecting the **Refresh** option from the context menu. The state of the **Setup** cell becomes **Update Required**. It will become **Up-to-Date** the next time you launch FLUENT from the **Setup** cell, or if you select the **Update** option from the context menu of the **Setup** cell.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can refresh the input data for the **Setup** cell in a FLUENT-based system by using either the **Refresh** command from the cell's context menu or by selecting the **Refresh Input Data** option in the FLUENT **File** menu if FLUENT is already open.

Important

Selecting the **Update** option from the context menu performs a **Refresh** command (if needed) before performing the **Update** command. You do not need to perform a **Refresh** and an **Update** in two separate steps.

Important

If you open FLUENT after making a modification to the mesh and without refreshing the input data, you will be asked whether you want to load the modified mesh before FLUENT launches.

Important

If FLUENT is open and you make a modification to the mesh, you will be informed that the upstream mesh has changed and asked whether you want to load the new mesh before proceeding.

2.4. Deleting Solution and Setup Cell Data for FLUENT-Based Systems

The following sections describes how to delete solution and setup data for your FLUENT-based systems by using the **Clear Generated Data** command, the **Reset** command, and the **Clear Old Solution Data** command.

2.4.1. Using the Clear Generated Data Command from the Solution Cell of FLUENT-Based Systems
2.4.2. Using the Reset Command from the Setup and Solution Cells of FLUENT-Based Systems
2.4.3. Using the Clear Old Solution Data Command from the Solution Cells of FLUENT-Based Systems

2.4.1. Using the Clear Generated Data Command from the Solution Cell of FLUENT-Based Systems

For either type of FLUENT-based system, you can remove all past and current generated case and data files from a **Solution** cell by right-clicking on the **Solution** cell and selecting the **Clear Generated Data** option from the context menu. All past and current generated case and data files associated with the cell are deleted and the FLUENT application is closed if it is open. If the **Solution** cell is **Up-to-Date**, it will become **Update Required** when the **Clear Generated Data** command is executed.

2.4.2. Using the Reset Command from the Setup and Solution Cells of FLUENT-Based Systems

For either type of FLUENT-based systems, you can remove all local and generated data from the **Setup** cell or from the **Solution** cell by right-clicking on either cell and selecting the **Reset** option from the context menu.

For **Setup** cells, the **Reset** option removes the **Setup** cell's references to the mesh file, deletes the settings file associated with the **Setup** cell, sets the cell property values to their defaults, and closes the FLUENT application if it is open. If the **Setup** cell is **Up-to-Date**, it will become **Refresh Required** when the **Reset** command is executed.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For **Solution** cells, the **Reset** option deletes all past and current case and data files (*not* including imported initial data files) associated with the cell, sets the cell property values to their defaults, and closes the FLUENT application if it is open. If the **Solution** cell is **Up-to-Date**, it will become **Refresh Required** when the **Reset** command is executed.

2.4.3. Using the Clear Old Solution Data Command from the Solution Cells of FLUENT-Based Systems

For either type of FLUENT-based systems, you can remove all older local and generated data from the **Solution** cell by right-clicking on the cell and selecting the **Clear Old Solution** option from the context menu.

For **Solution** cells, the **Clear Old Solution Data** option retains only the most recent solution files associated with the **Solution** cell, and removes older case and data files that are not part of the current solution history. This option is only available if there are already solutions associated with the cell, in addition to the most recent solution.

Note

Archived projects will only contain the most current solution files, and all solution files that are associated with the current run (i.e., solution history).

Note

While importing design points, the next design point (`dp1`) will contain only the last data file of the current solution from the last design point (`dp0`).

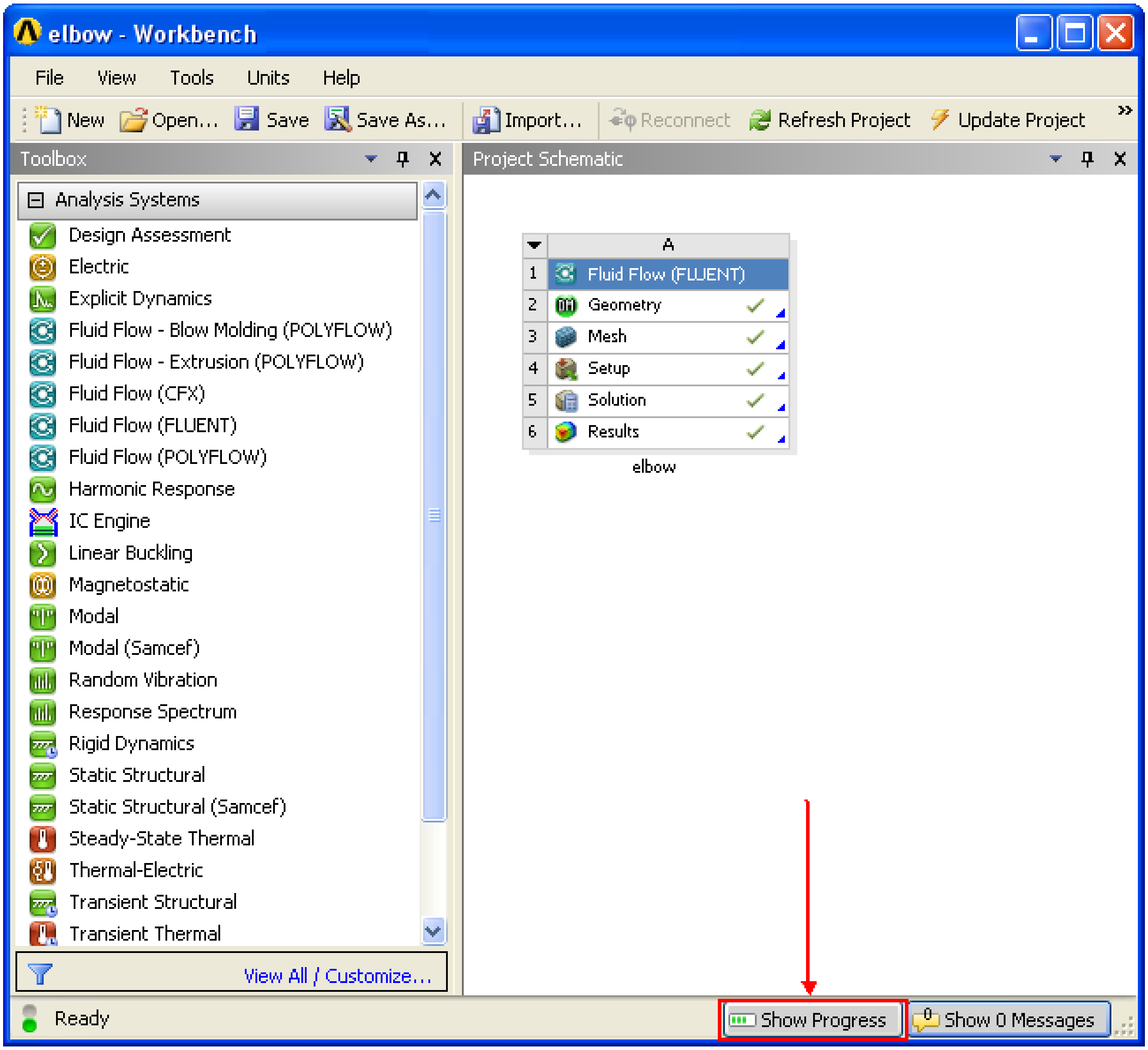
2.5. Interrupting, Restarting, and Continuing a Calculation

You can interrupt the calculation in an interactive FLUENT session by using the **Cancel** button or by typing **Ctrl** and **c** at the same time in the FLUENT console window.

You can also interrupt a FLUENT calculation in the background by using the Progress Monitor in Workbench. The Progress Monitor is useful if you would like some visual feedback on the progress of your calculations. Typically, the Progress Monitor is hidden, but can be displayed at the bottom of the Project Schematic by toggling the **Show Progress** button.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

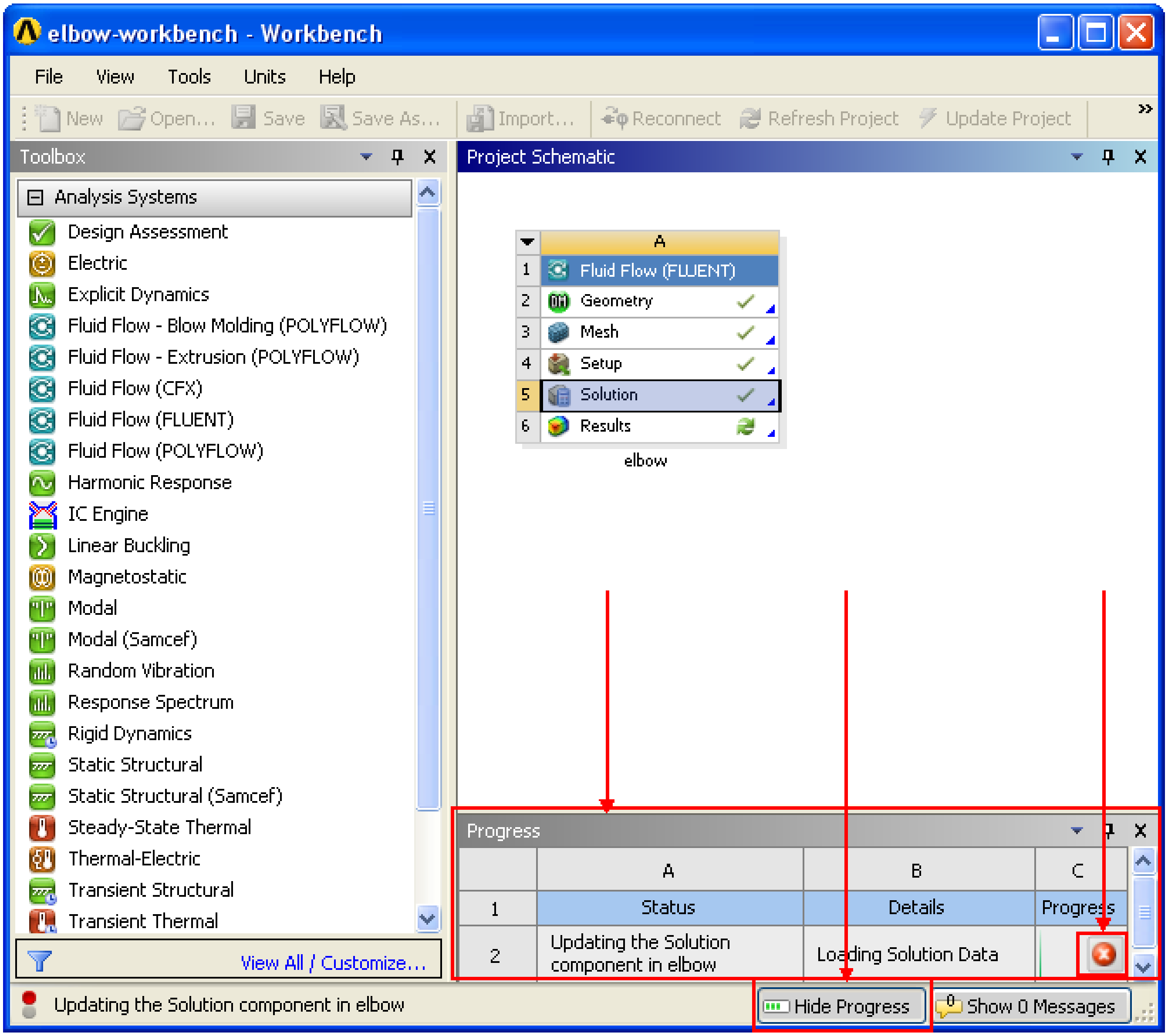
Figure 2.1 The Show Progress Button in Workbench



When the Progress Monitor is displayed, the **Show Progress** button becomes the **Hide Progress** button, so the button can be used to toggle the display of the Progress Monitor.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

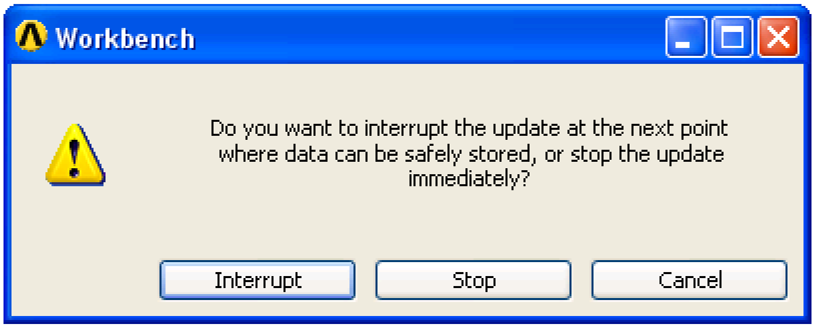
Figure 2.2 The Hide Progress Button, the Progress Monitor and the Stop Icon in Workbench



To interrupt an on-going FLUENT calculation from Workbench, select the  icon in the Progress Monitor (see *Figure 2.2* (p. 32)). This will display a prompt, asking if you would like to interrupt or stop the calculations.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 2.3 The Workbench Prompt



Selecting **Interrupt** stops the calculation at the next point where data can be safely stored for later use. Selecting **Stop** stops the calculation immediately without concern for whether data associated with the current action can be stored.

Important

When the calculation is interrupted from within an interactive FLUENT session, it always stops the data at the next point where data can be safely stored for later use.

When a calculation is interrupted, the state of the **Solution** cell becomes **Interrupted**. If a background calculation is interrupted, FLUENT writes the case and data file and then closes.

You can restart a previously interrupted FLUENT calculation from within Workbench by right-clicking on the **Solution** cell and selecting the **Continue Calculation** option from the context menu. This will allow you to restart the calculation from the current case and data files, performing the total number of iterations (or time-steps) specified in the case file.

If you interrupt a calculation, review the results, and decide that the solution is converged, you can force the **Solution** cell state to be **Up-to-Date** by right-clicking on the **Solution** cell and selecting the **Accept Interrupted Solution as Up-to-Date** option from the context menu.

2.6. Connecting Systems in Workbench

Workbench allows you to create connections between multiple systems that enable the systems to access the same data. This is useful, for instance, when you want to compare the results from multiple FLUENT-based systems in the same ANSYS CFD-Post session. In this case, you would connect the various **Solution** cells to one **Results** cell (either in one of your FLUENT-based systems or in a separate **Results** system). When you double-click on that **Results** cell, the results from all connected systems will be loaded into ANSYS CFD-Post at the same time.

Workbench supports two different types of connections:

- Connections that share data are used when the inputs and outputs of the two connected cells are identical. Shared data connections can only be created between two cells of the same type. A shared data connection is represented on the Project Schematic by a line with a square on its right (target) side (see *Figure 2.4* (p. 35)).
- Connections that transfer data are used when the output of one cell is used as the input to the connected cell. Transfer data connections are usually created between two cells of different types. One exception is that a transfer data connection can be used between the **Solution** cells of two FLUENT-based systems when you want to use the current data from one system as the initial data for the other system. A

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

transfer data connection is represented in the Project Schematic by a line with a circle on its right (target) side (see *Figure 2.4* (p. 35)).

There are four ways to create connected systems in Workbench.

- Left-click a cell in one system, then drag and drop it onto a compatible cell in another system.
- Left-click a system in the Toolbox, then drag and drop it onto a compatible system in the Project Schematic.
- Create a duplicate system (see *Duplicating FLUENT-Based Systems* (p. 40)).
- Right-click on a cell and select one of the options under **Transfer Data From New...** or **Transfer Data To New...** (these options are not available for all cells). Transferring data from the **Solution** cell to a new FLUENT system's **Setup** cell transfers the mesh (but not the settings) to the **Setup** cell.

When you highlight a system in the Toolbox, Workbench highlights all of the compatible drop targets in the Project Schematic. As you move the mouse over a drop target, it is highlighted in red and a message appears in the Project Schematic that informs you what the result will be if you drop the system onto that target.

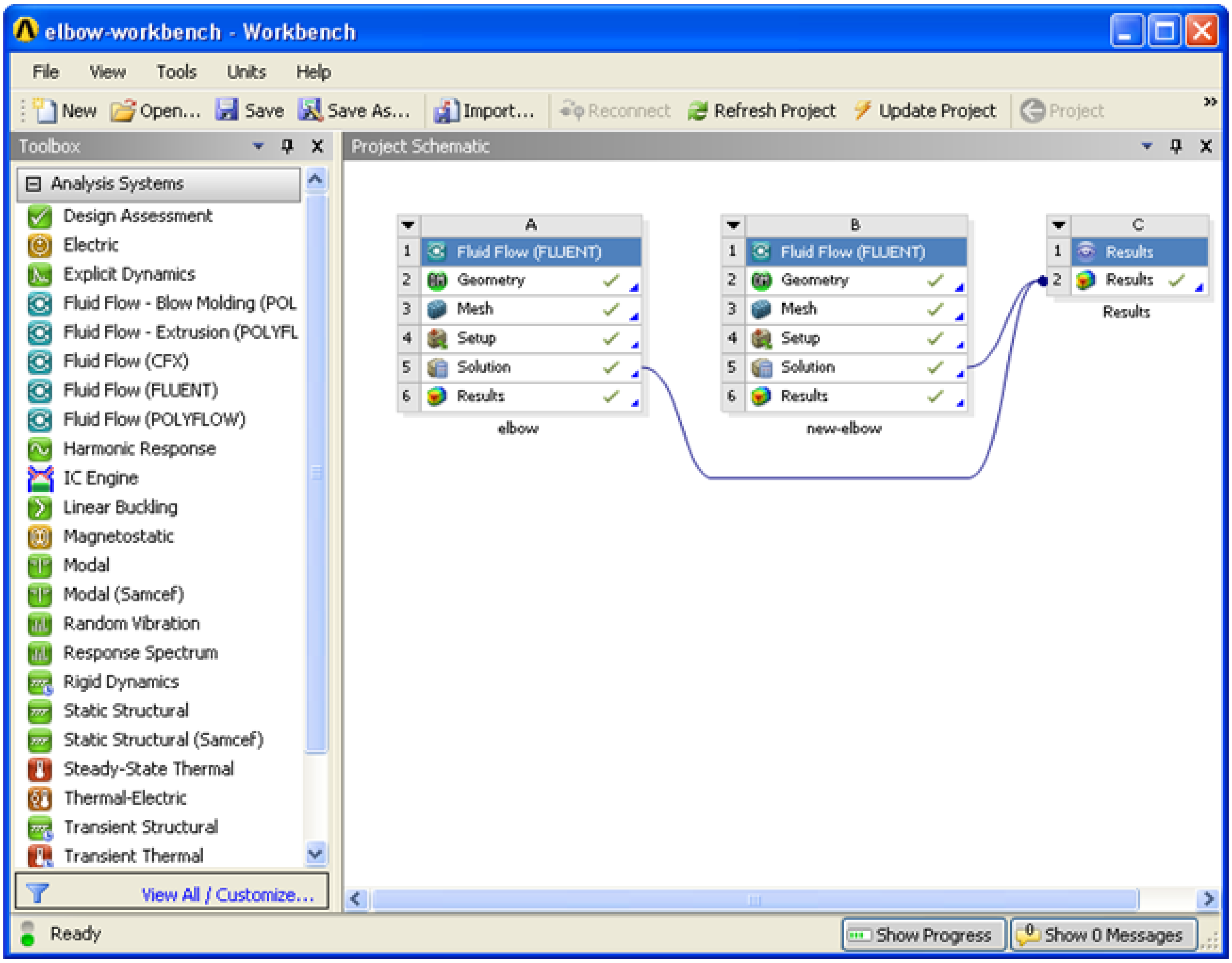
Important

There are usually several compatible drop targets on empty space in the Project Schematic. Dropping the system onto one of these targets will create a standalone system in that location.

Similarly, when you highlight a cell and begin to drag it, Workbench highlights all of the compatible drop targets in the Project Schematic. As you move the mouse over a drop target, it is highlighted in red and a message appears in the Project Schematic that informs you what the result will be if you drop the cell onto that target.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 2.4 Connected Systems Within Workbench



For more information about connecting systems, see the Workbench on-line help, as well as the following sections:

2.6.1. Connecting Systems by Dragging and Dropping a System from the Toolbox onto Another System
2.6.2. Connecting Systems By Dragging and Dropping FLUENT-Based Solution Cells Onto Other Systems

2.6.1. Connecting Systems by Dragging and Dropping a System from the Toolbox onto Another System

The following example demonstrates the procedure for creating connected systems by dragging a system from the Toolbox and dropping it onto a compatible system in the Project Schematic.

1. Starting from a project with an up-to-date **Mesh** component system, select the FLUENT-based component system from the Toolbox; the compatible drop targets are highlighted in green.

2. Drag the system over the Project Schematic and pause over the **Mesh** cell of the **Mesh** component system; the **Mesh** cell target is highlighted in red and a message informs you that selecting that target will transfer the data from cell A3 to the new system.

3. Drop the system on the drop target and a transfer data connection is created between the **Mesh** cell A3 and the **Setup** cell B1.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

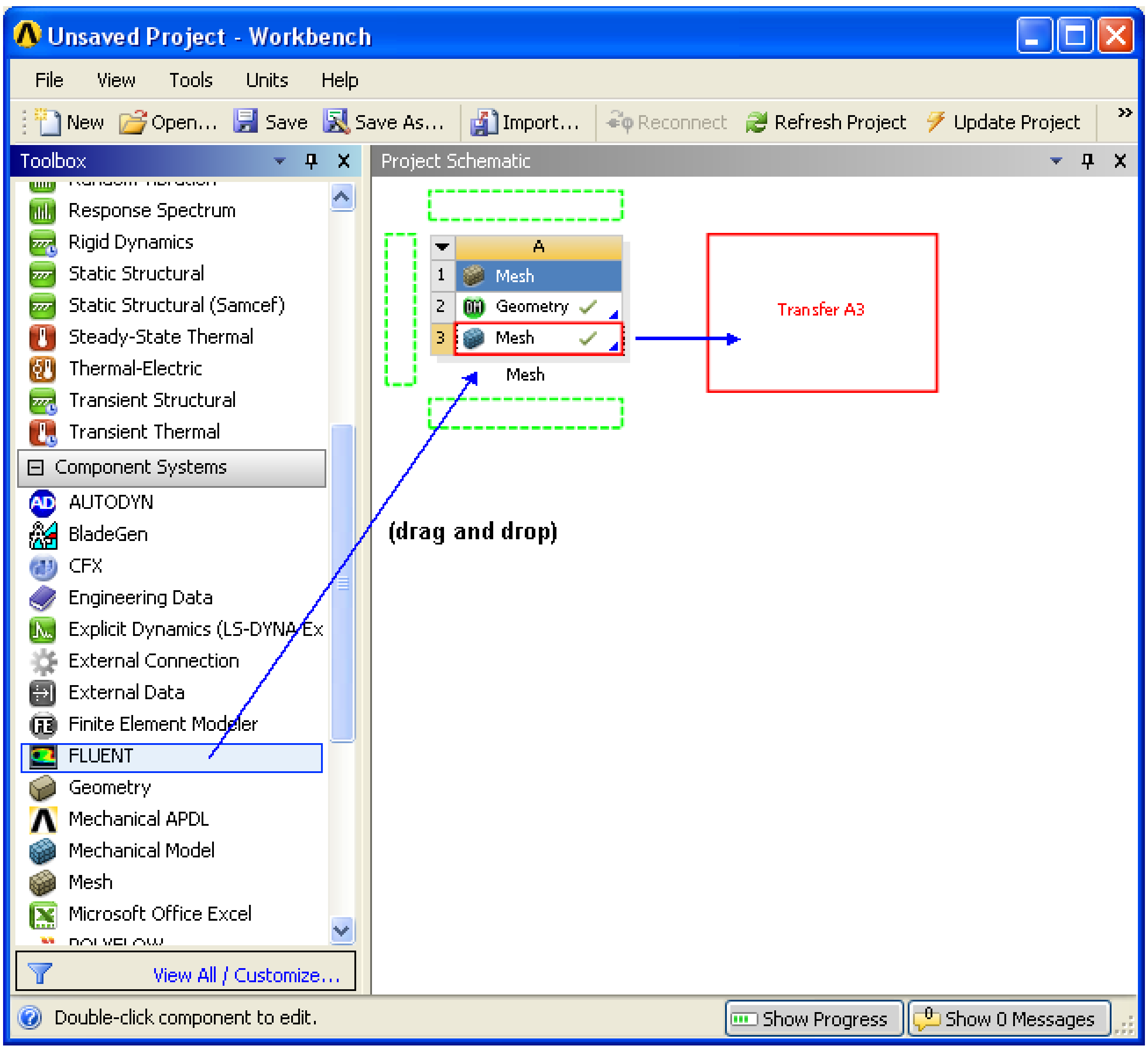
Note that **Mesh** cell A3 becomes **Update Required**, this is because the input data for the new system has not yet been generated by the ANSYS Meshing application.

Important

If you try to open FLUENT before the **Mesh** cell is updated, a warning message is displayed informing you that you must update the **Mesh** cell before you can start FLUENT, since the mesh file required for FLUENT does not yet exist.

4. Right-click **Mesh** cell A3 and select **Update**.
5. Double-click **Setup** cell B1; FLUENT launches and loads the mesh from cell A3.

Figure 2.5 Applying the Mesh Settings to a New FLUENT-Based Component System by Dragging and Dropping Systems

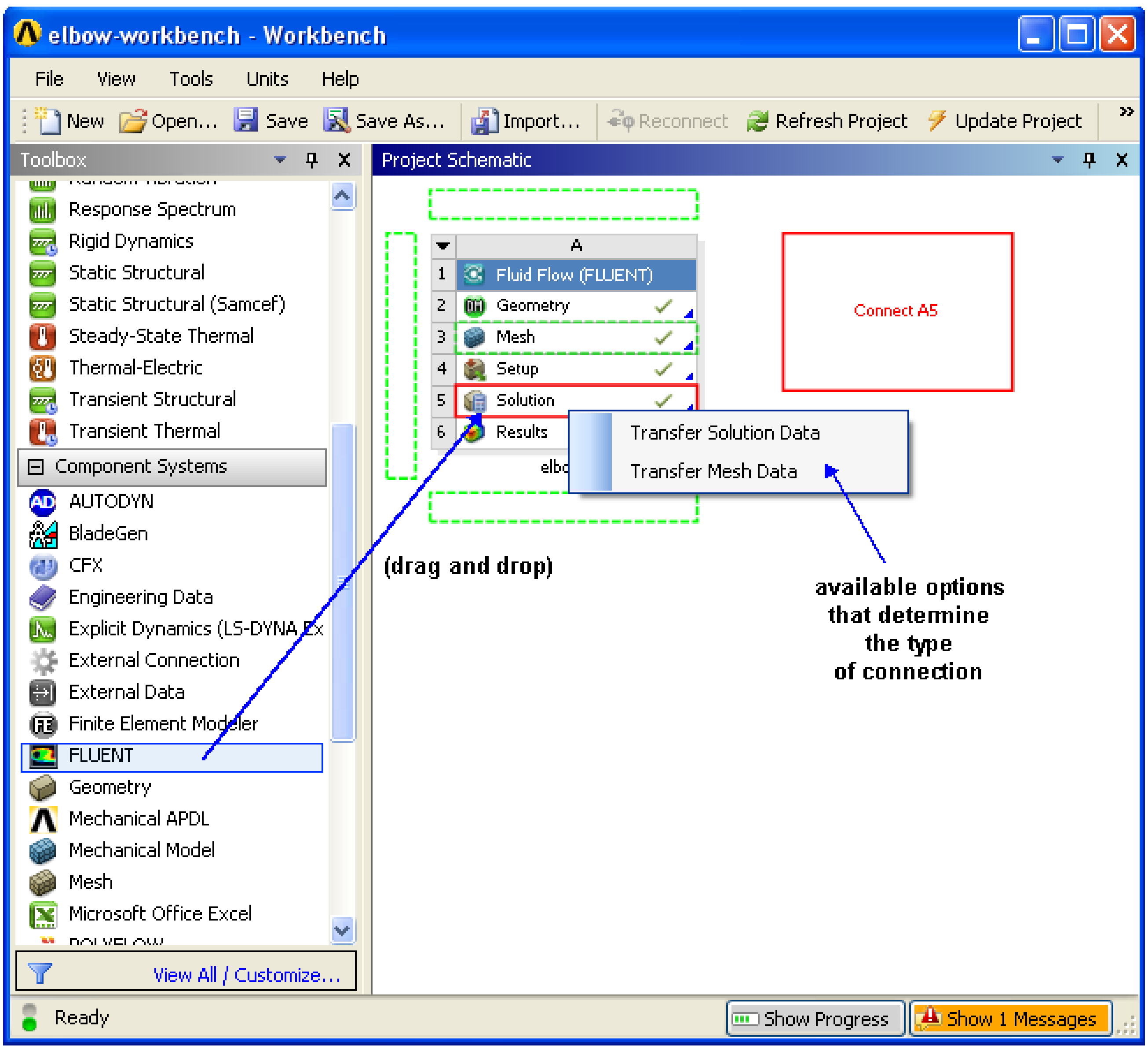


Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the previous example, a transfer data connection was created. Shared data connections can also be creating by dragging a system from the Toolbox and dropping it onto a compatible system in the Project Schematic. The type of connection that Workbench creates depends on which drop target you select. The red preview messages in the Project Schematic inform you of the type of connection(s) that will result from your action.

When a FLUENT-based component system is dragged from the Toolbox onto the **Solution** cell of an existing FLUENT-based analysis system, you are presented with two choices: **Transfer Solution Data** or **Transfer Mesh Data**. The connection that is made between the two systems is based on the option selected.

Figure 2.6 Transferring Solution Data or Mesh Data to a New FLUENT-Based Component System by Dragging and Dropping Systems



Important

The mesh from the case file associated with the **Solution** cell of a FLUENT-based analysis system or component system can be transferred to the **Setup** cell of a FLUENT-based component system only and not the **Setup** cell of a FLUENT-based analysis system.

2.6.2. Connecting Systems By Dragging and Dropping FLUENT-Based Solution Cells Onto Other Systems

The following figures demonstrate the procedure for creating a transfer data connection by dragging a **Solution** cell from a FLUENT-based system and dropping it onto a compatible cell in another system:

Figure 2.7 An Example of Two Unconnected Systems

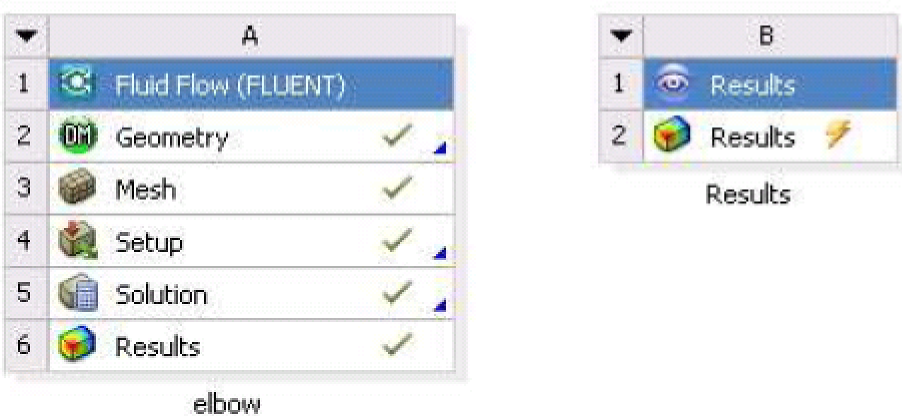
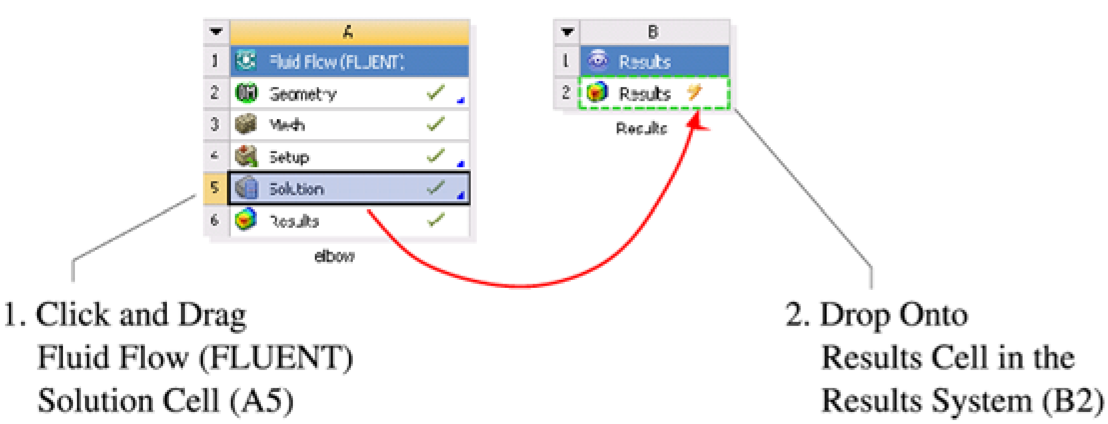
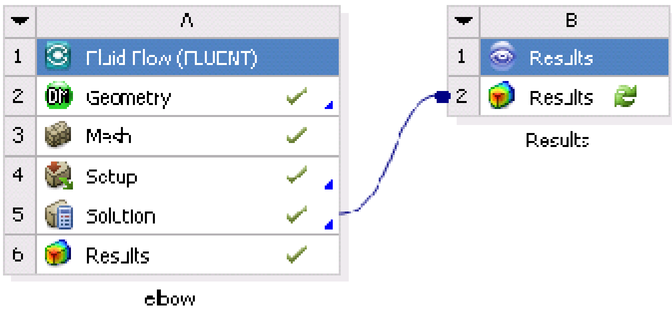


Figure 2.8 An Example of Dragging and Dropping a Solution Cell Onto Another Compatible Cell



Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 2.9 An Example of Two Connected Systems



The following table lists the compatible drop targets for the **Solution** cell from a FLUENT-based system:

Table 2.1 Connecting Systems By Dragging and Dropping Cells Between Systems

From Cell	To Cell
Solution	FLUENT: Setup FLUENT: Solution Fluid Flow (FLUENT): Solution Fluid Flow (FLUENT): Results CFX: Results Fluid Flow (CFX): Results Vista TF: Results Results: Results
	Static Structural (ABAQUS): Setup Static Structural (Samcef): Setup Shape Optimization (ANSYS): Setup Static Structural: Setup Steady-State Thermal: Setup Transient Structural: Setup Thermal-Electric: Setup Transient Thermal: Setup

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.7. Duplicating FLUENT-Based Systems

Workbench allows you to create a duplicate of a system so that you can set up multiple, similar systems and analyze them at the same time. For instance, if you would like to study the differences in the fluid flow between two slightly different geometries, then you can create, set up, and solve a single fluid flow analysis system, duplicate the entire system, change the geometry in the duplicate system and perform another fluid flow analysis on the new geometry.

You can create a duplicate of a FLUENT-based system by performing the following steps:

1. In the Project Schematic, right-click the system header to open the system's context menu.
2. Select **Duplicate** from the context menu.

   A copy of the original FLUENT-based system is created in the Project Schematic.

All data associated with the FLUENT-based system, except for any case, data, and initial data files associated with the **Solution** cell, are copied to the new system. The states of the **Geometry**, **Mesh**, and **Setup** cells in the new system will be the same as the states of the cells in the original system. The state of the **Solution** and **Results** cells in the new system will be different than those of the original system if the original system had case and data files associated with its **Solution** cell.

In addition, you can use the **Duplicate** command to create a duplicate of a FLUENT-based system in which the data in the **Geometry** cells or the data in both the **Geometry** cells and the **Mesh** cells is shared between the two systems rather than copied.

To create a duplicate system in which the geometry is shared between the original and new system:

1. In the Project Schematic, right-click the **Mesh** cell in the system you want to duplicate to open the context menu.
2. Select **Duplicate** from the context menu.

   A copy of the original FLUENT-based fluid flow system is created in the Project Schematic. A shared data connection is created between the **Geometry** cell in the original system and the **Geometry** cell in the new system.

To create a duplicate system in which both the geometry and the mesh are shared between the original and new system:

1. In the Project Schematic, right-click the **Setup** cell or any cell below it in the system you want to duplicate to open the context menu.
2. Select **Duplicate** from the context menu.

   A copy of the original FLUENT-based fluid flow system is created in the Project Schematic. Two shared data connections are created: one between the **Geometry** cell in the original system and the **Geometry** cell in the new system, and the other between the **Mesh** cell in the original system and the **Mesh** cell in the new system.

2.8. Reviewing Mesh Manipulation Operations in FLUENT

While working within FLUENT in Workbench, you are able to perform and preserve basic mesh manipulation operations in FLUENT. The mesh manipulation operations are saved as part of the Workbench project in the **Setup** cell such that the operations are explicitly applied by Workbench once FLUENT loads the mesh (and before reading the settings file) and automatically reapplied the next time an

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Update is performed. This allows you to more easily update the system if the upstream mesh has changed, and also, for example, when parameters are modified (see *Changing the Settings and Mesh in FLUENT* (p. 45)).

You are able to perform and preserve the following mesh operations in FLUENT in Workbench:

- scaling
- rotation
- translation
- smoothing
- swapping
- merging
- fusing
- zone/domain renaming
- zone/domain reordering
- zone type changing (the final zone type is dependent on the settings file. This is recorded primarily to address occasional mesh changes triggered by a change in the zone type)
- creating periodic zones
- polyhedra operations (convert to polyhedra, convert skewed cell to polyhedra)

Important

By contrast, the following mesh operations are *not* recorded within FLUENT in Workbench:

- separating faces and cells
- zone operations such as activating, deactivating, deleting, and replacing zones, as well as appending case and data files
- replacing meshes
- adapting meshes, specifically using
  - boundaries
  - gradients
  - iso-values
  - regions
  - Yplus/Ystar
  - anisotropic
  - volume

See Modifying the Mesh in the User's Guide for more information about manipulating the mesh in FLUENT.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

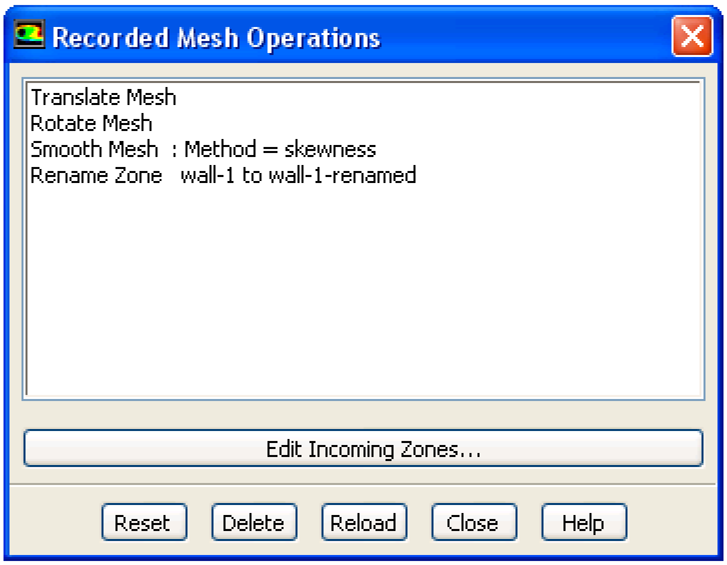
These common mesh manipulation operations can be reviewed while working with FLUENT in Workbench using the **Recorded Mesh Operations** option in the **Mesh** menu.

Mesh → **Recorded Mesh Operations...**

This displays the **Recorded Mesh Operations** dialog box (*Figure 2.10* (p. 42)).

*When FLUENT is started from the **Solution** cell, the **Recorded Mesh Operations** option in the **Mesh** menu is disabled.*

Figure 2.10 The Recorded Mesh Operations Dialog Box



The **Recorded Mesh Operations** dialog box allows you to view the mesh manipulation operations that are currently stored. You can also see new operations, or see if an operation has failed due to an incompatible upstream mesh. In the **Recorded Mesh Operations** dialog box, the following controls are available:

- **Reset** — resets the stored mesh operations, clearing out all existing mesh operations, and accepting the current zones as expected incoming zones.
- **Delete** — deletes the selected stored mesh operation.
- **Reload** — saves *only* the current set of mesh operations and incoming zone information, and reloads the **Setup** cell.

*Note the **Reset**, **Delete**, and the **Reload** buttons are disabled when there are no mesh operations listed in the **Recorded Mesh Operations** dialog box.*

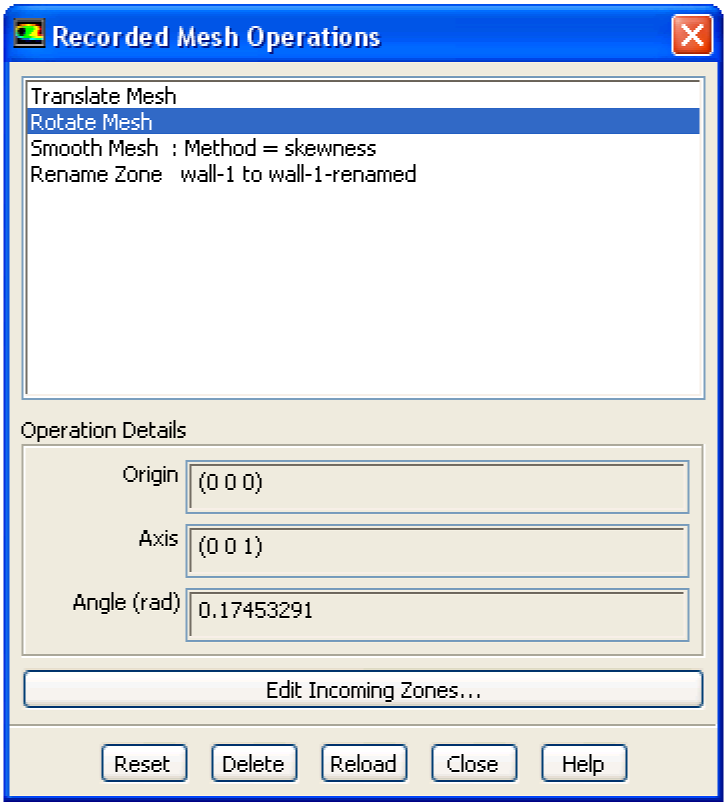
Important

You should note that using **Reset/Delete/Reload** is discouraged in cases with topological mesh transformations because of potential mismatches between recorded operations and the settings file.

In the **Recorded Mesh Operations** dialog box, select a listed operation to expand the dialog box in order to review the details of the selected mesh operation. For example. *Figure 2.11* (p. 43) illustrates how you can review the details of the rotation operation upon a mesh.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

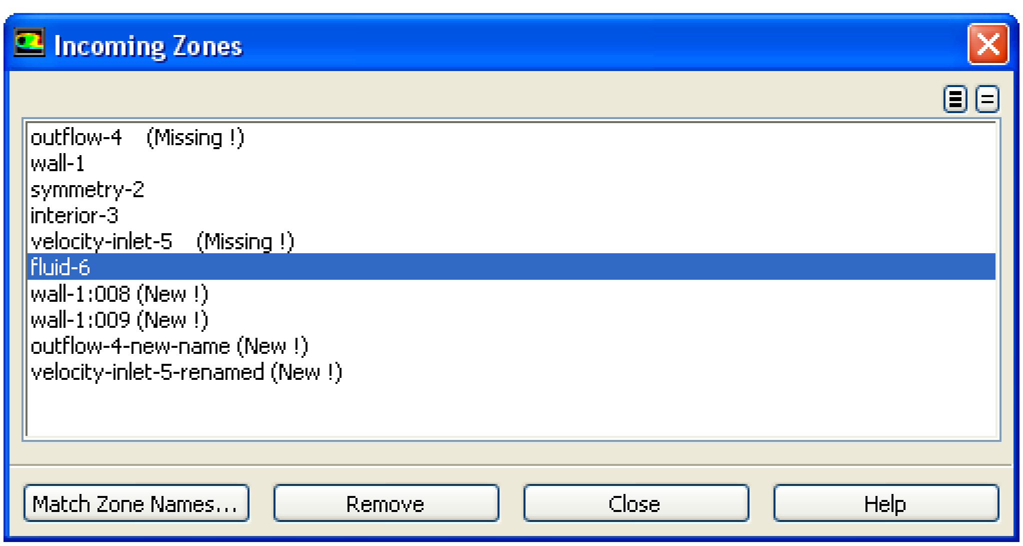
Figure 2.11 Reviewing the Details of Rotating the Mesh in the Recorded Mesh Operations Dialog Box



For a selected operation, specific details about the mesh manipulation operation can be reviewed under **Operation Details** in the **Recorded Mesh Operations** dialog box. Note that when rotating the mesh, the angle is provided in radians, and when translating the mesh, the distance is provided in meters.

You can also view (and minimally edit) the names of incoming (i.e., upstream) zone names by clicking the **Edit Incoming Zones...** button in the **Recorded Mesh Operations** dialog box. This displays the **Incoming Zones** dialog box (*Figure 2.12* (p. 43)).

Figure 2.12 The Incoming Zones Dialog Box

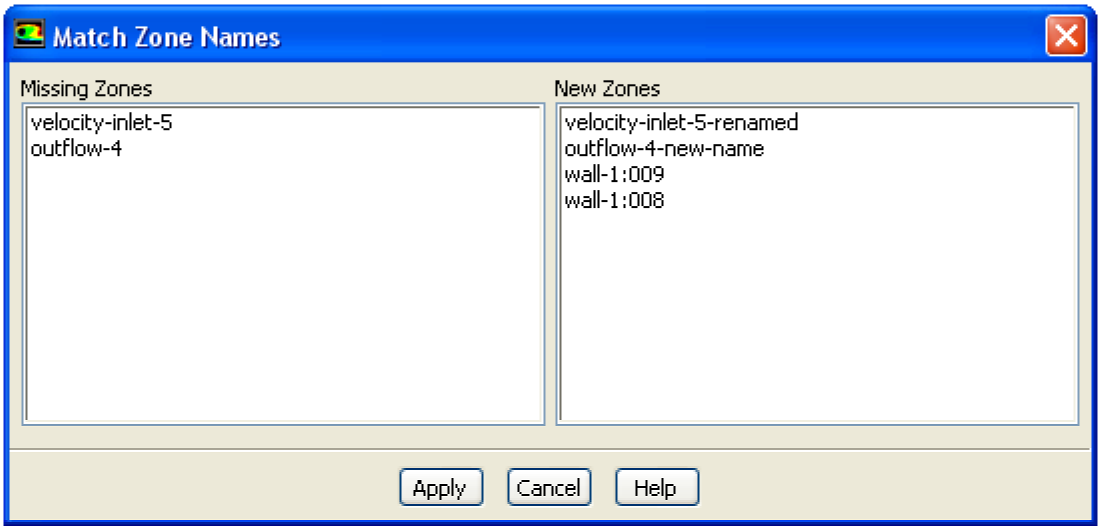


Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using the **Incoming Zones** dialog box, you can see new or missing zones, remove zones, or match zone names. When FLUENT is closed, *all* of the zones shown in the **Incoming Zones** dialog box are included in the list of zones that FLUENT expects in the upstream mesh. This list is used to provide you with warnings about any new or missing zones. The stored mesh operations and settings will continue to behave as expected depending upon what zones they are operating upon.

To correct any issues with missing or new incoming zone names, click the **Match Zone Names...** button in the **Incoming Zones** dialog box to open the **Match Zone Names** dialog box.

Figure 2.13 The Match Zone Names Dialog Box



The **Match Zone Names** dialog box contains a listing of any missing zones or new zones in their respective columns. Simply select a zone in the **Missing Zones** list and select a corresponding zone in the **New Zones** list, and click **Apply**.

You can use the **Match Zone Names** dialog box to manually manage upstream topological changes (such as zone name changes). For instance, if a zone name has changed upstream, you can use this dialog box to match the corresponding zone names, effectively renaming the zone in FLUENT. In this case, you should perform a **Reload** after renaming the old zone name, if you do not want to change anything in the settings file. Another example would be if a zone is no longer present in the upstream mesh, then you can delete it using the **Incoming Zones** dialog box. In this case, reloading would not work since the settings file will also need to be written. So, you would delete the zone names that are no longer expected and then continue setting other details, solving, and closing FLUENT as you would normally do.

Important

After matching all zone names, it is recommended to use the **Reload** button in the **Recorded Mesh Operations** dialog box.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note

For older FLUENT in Workbench projects (prior to version 14.0), when making any changes to the mesh using ANSYS Meshing (e.g., renaming a surface), you should first open FLUENT using the **Setup** cell of the Fluid Flow (FLUENT) analysis system in order for FLUENT to be aware of the upstream mesh changes (e.g., when detecting upstream zone name changes).

2.9. Changing the Settings and Mesh in FLUENT

In order to use the **Update** command (see *Using the Update Command* (p. 26)) when changes are made to your project, FLUENT must know which settings changes should be stored as part of the **Setup** cell's data (and therefore used during an update) and which settings changes should only be reflected in the results data that is associated with the **Solution** cell.

Important

Please note that there are certain changes to the mesh that you can perform within FLUENT that are able to be recorded and reviewed and are treated as settings. In addition, there are other changes to the mesh within FLUENT that are not recorded, and, as such, are not saved as settings, and cannot be automatically re-applied when the **Update** command is used. For more information about recording and reviewing mesh manipulation operations in FLUENT in Workbench, see *Reviewing Mesh Manipulation Operations in FLUENT* (p. 40).

In order to address these issues, if you make certain changes to the mesh and/or settings in FLUENT, you may be prompted when you attempt to calculate, close FLUENT, or save the project from FLUENT. The dialog boxes that appear (described in more detail below) allow you to select an action or to cancel the operation.

Important

If you save the project from an application other than FLUENT or from the Project Schematic, you will not be prompted; FLUENT will automatically perform the default action for each dialog box described below.

For more information, please see the following sections:
2.9.1. Changing Case and Mesh Settings Before Beginning a Calculation
2.9.2. Changing Case and Mesh Settings After a Calculation Has Started

2.9.1. Changing Case and Mesh Settings Before Beginning a Calculation

Important

Note that in the descriptions below, mesh changes performed within FLUENT only include those mesh modifications that are *not* recorded in FLUENT in Workbench. In addition, changes to the settings can also include any mesh modifications that *can* be recorded in FLUENT in Workbench. For more information about recording and reviewing mesh manipulation operations in FLUENT in Workbench, see *Reviewing Mesh Manipulation Operations in FLUENT* (p. 40).

If the mesh has not been changed using tools in FLUENT, any changes to the settings are automatically saved to the settings (`.set`) file.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the mesh has been changed using tools in FLUENT and the mesh was imported into the FLUENT application or the system's **Setup** cell, any changes to the settings are automatically saved to the settings (`.set`) file. In addition, a case file is saved and registered to the **Setup** cell to represent the modified mesh.

If the mesh has been changed using tools in FLUENT and the mesh was provided to the system's **Setup** cell by an upstream cell, the saved settings may not be compatible with the original mesh available to the **Setup** cell.

Since the mesh was provided to the system's **Setup** cell by an upstream cell, the original mesh cannot be replaced by the modified mesh without also making changes to the Project Schematic. Since there are several ways in which the schematic can be modified, FLUENT does not provide a way to do this automatically.

Any changes to the settings will be saved to the settings file. The action you requested when you were prompted will proceed. If you had selected to calculate, the iterations (or time-steps) will be performed on the modified mesh. If you had selected to close FLUENT or save the project, the modified mesh will be stored in the case file that is written as a result of either of those actions.

Important

If you open FLUENT from the **Setup** cell, or update or refresh the **Setup** or **Solution** cell from the Project Schematic, the modified mesh will be replaced with the mesh provided by the upstream cell. To calculate using the modified mesh, either open FLUENT from the **Solution** cell or select **Continue Calculation** from the **Solution** cell's context menu.

Important

If you specified settings before you changed the mesh, you must also verify that those settings are consistent with the modified mesh.

Important

If you want to use the modified mesh as the starting point for another analysis, simply create a new FLUENT-based system and import the new case file that contains the modified mesh into its **Setup** cell.

You can avoid performing the mesh modification (which cannot be recorded by FLUENT) and create a case modification strategy that automatically performs the desired mesh modification steps (as well as any settings changes that are dependent on the modified mesh) before calculating (see *Case Modification Strategies with FLUENT and Workbench* (p. 50)). This will allow you to automatically repeat the desired mesh modifications every time you perform an **Update** or restart the calculation from the **Setup** cell.

Important

If you plan to make modifications to your mesh in FLUENT (such as scaling, rotating, or converting to polyhedra) before performing any calculations, and you do not require any change made upstream of the FLUENT **Setup** cell to be propagated to the mesh in the future, you should import the mesh you plan to modify directly into the **Setup** cell of a FLUENT-based analysis or component system.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

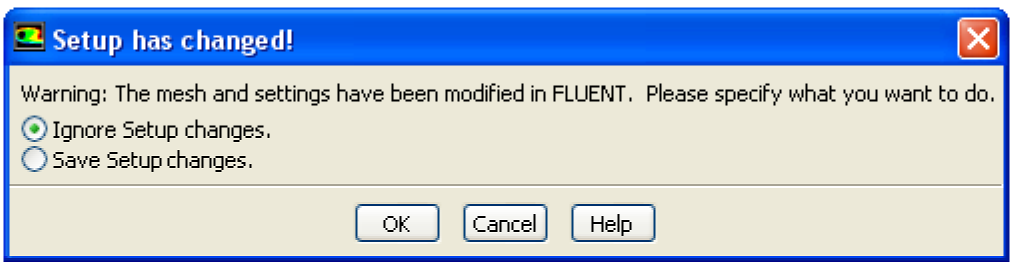
2.9.2. Changing Case and Mesh Settings After a Calculation Has Started

Important

Note that in the descriptions below, mesh changes performed within FLUENT only include those mesh modifications that are *not* recorded in FLUENT in Workbench. In addition, changes to the settings can also include any mesh modifications that *can* be recorded in FLUENT in Workbench. For more information about recording and reviewing mesh manipulation operations in FLUENT in Workbench, see *Reviewing Mesh Manipulation Operations in FLUENT* (p. 40).

If changes to the settings have been made and the mesh has not been changed using tools in FLUENT, you are prompted with the following dialog box:

Figure 2.14 The Setup has changed! Dialog



Since changes were made to the settings after you began the calculation, you have to specify whether or not the settings changes should be saved to the settings file.

You can choose any of the following actions:

- Select the **Ignore Setup changes** option in the warning dialog box. The modified settings will not be saved to the settings file. The action you requested when you were prompted will proceed. If you had selected to calculate, the iterations (or time-steps) will start using the new settings. If you had selected to close FLUENT, or save the project, the new settings will be stored in the case file that is written as a result of either of those actions.

  Important

  If you open FLUENT from the **Setup** cell, or update or refresh the **Setup** or **Solution** cell from the Project Schematic, the settings will be replaced with the settings in the settings file that is associated with the **Setup** cell. To calculate using the new settings either open FLUENT from the **Solution** cell or select **Continue Calculation** from the **Solution** cell's context menu.

  Important

  If you want to use the new settings as the starting point for another analysis, simply create a new FLUENT-based system and import the new case file that contains the new settings into its **Setup** cell.

- Select the **Save Setup changes** option in the warning dialog box. The modified settings will be saved to the settings file. The action you requested when you were prompted will proceed. If you had selected

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

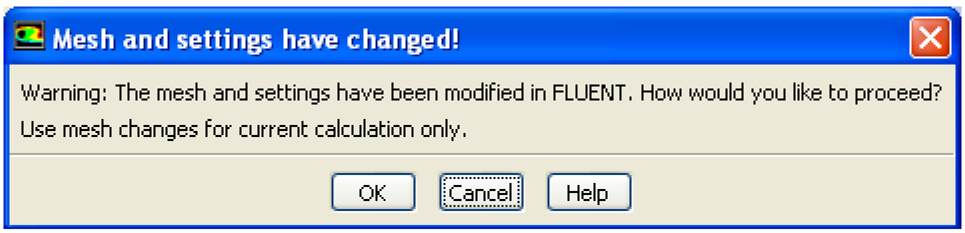
to calculate, the iterations (or time-steps) will start using the new settings. If you had selected to close FLUENT, or save the project, the new settings will also be stored in the case file that is written as a result of either of those actions.

- Select **Cancel** in the warning dialog box. Create a new duplicate system (see *Duplicating FLUENT-Based Systems* (p. 40)), modify the settings, and connect the **Solution** cells for the two systems so that the calculations will be performed in sequence.
- Select **Cancel** in the warning dialog box and create a case modification strategy that automatically performs the desired settings changes after the appropriate number of iterations (or time-steps); see *Case Modification Strategies with FLUENT and Workbench* (p. 50) for more details.

The last two approaches allow you to automatically repeat setting changes after a specified number of iterations (or time-steps) every time you perform an **Update** or restart the calculation from the **Setup** cell.

If the mesh has been changed using tools in FLUENT, and the mesh includes changes made using dynamic or sliding mesh), you are prompted with the following dialog box:

Figure 2.15 The Mesh and settings have changed! Dialog

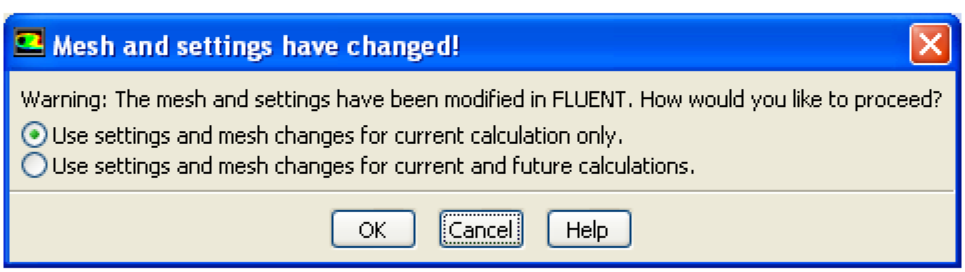


Important

Note that when FLUENT modifies the mesh, FLUENT does not allow you to save the modified settings and/or mesh operations once the calculation has started.

Otherwise, if the mesh has been changed using tools in FLUENT, you are prompted with the following dialog box:

Figure 2.16 The Mesh and settings have changed! Dialog



You need to specify whether or not any changes you made to the settings after you began the calculation should be saved to the settings file. Note that in cases where unrecorded mesh operations are performed in FLUENT, those mesh operations will be ignored in future solution runs.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Whenever you modify the mesh in FLUENT, you also make changes to some settings in FLUENT. Therefore, mesh modifications always result in setting changes.

Important

The original mesh cannot be replaced by the modified mesh after the calculation has begun.

You can choose any of the following actions:

- Select the **Use settings and mesh changes for current calculations only** option in the warning dialog box. The modified settings will not be saved to the settings file. The action you requested when you were prompted will proceed. If you had selected to calculate, the iterations (or time-steps) will start using the new settings and the modified mesh. If you had selected to close FLUENT, or save the project, the new settings and the modified mesh will be stored in the case file that is written as a result of either of those actions.

  Important

  If you open FLUENT from the **Setup** cell, or update or refresh the **Setup** or **Solution** cell from the Project Schematic, the settings and the modified mesh will be replaced with the mesh and settings in the files that are associated with the **Setup** cell. To calculate using the new settings and the modified mesh, either open FLUENT from the **Solution** cell or select **Continue Calculation** from the **Solution** cell's context menu.

  Important

  If you want to use the new settings and the modified mesh as the starting point for another analysis, simply create a new FLUENT-based system and import the new case file that contains the new settings and the modified mesh into its **Setup** cell.

- Select the **Use settings and mesh changes for current and future calculations** option in the warning dialog box. The modified settings will be saved to the settings file. The action you requested when you were prompted will proceed. If you had selected to calculate, the iterations (or time-steps) will start using the new settings and the modified mesh. If you had selected to close FLUENT, or save the project, the new settings and the modified mesh will also be stored in the case file that is written as a result of either of those actions.

  Important

  If you open FLUENT from the **Setup** cell, or update or refresh the **Setup** or **Solution** cell from the Project Schematic, the new settings will be used in conjunction with the mesh in the file that is associated with the **Setup** cell. Since those settings may have been specified after the mesh was modified, you must verify that the new settings are consistent with the original mesh.

- Select **Cancel** in the warning dialog box and create a case modification strategy that automatically performs the desired settings changes and mesh modification steps after the appropriate number of

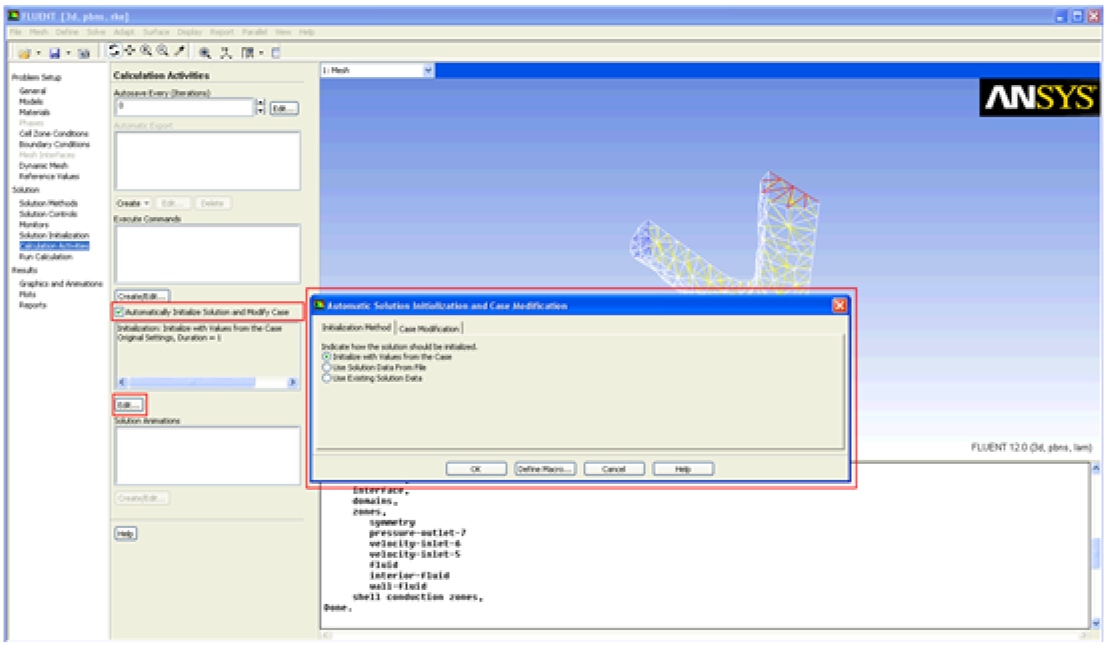
Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

iterations (or time-steps); see *Case Modification Strategies with FLUENT and Workbench* (p. 50) for more details. This allows you to automatically repeat the desired setting changes and mesh modification steps after a specified number of iterations (or time-steps) every time you perform an **Update** or restart the calculation from the **Setup** cell.

2.10. Case Modification Strategies with FLUENT and Workbench

FLUENT allows you to specify your own initialization and start-up routines using automatically executed strategies that you can define using the **Automatic Solution Initialization and Case Modification** option on the **Calculation Activities** task page within FLUENT. Selecting the option and clicking the **Edit..** button opens the **Automatic Solution Initialization and Case Modification** dialog box.

Figure 2.17 Accessing Solution Strategies in FLUENT



Using the **Automatic Solution Initialization and Case Modification** dialog box, you are able to specify your initialization method on the **Initialization Method** tab and use any text user interface (TUI) command to modify the case after a specified number of time steps or iterations in the **Case Modification** tab. This option replaces the need for some simple journal files, especially for cases where a prescribed start-up and solution routine is used (start with 1st order, switch to 2nd order, turn on reactions, etc.)

When a case modification strategy is defined within FLUENT, it will be used when a system is updated from the Project Schematic in Workbench.

As mentioned in *Changing the Settings and Mesh in FLUENT* (p. 45), mesh manipulations steps executed from within FLUENT are not repeated when an update is performed from the Project Schematic or you start FLUENT from the **Setup** cell. However, for mesh manipulation, you can specify steps that will be executed at the beginning of your calculation if you incorporate them into a case modification strategy.

Solution strategies can also be useful when you need to perform mesh manipulation steps after a specified number of iterations during a calculation.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Whenever you run a case modification strategy interactively, make sure to reload the original mesh and revert to the original settings before re-running the case modification strategy (see the separate FLUENT User's Guide for information on how the original settings can be reset automatically as part of a case modification strategy).

Important

When running FLUENT in Workbench, the behavior when continuing the calculation from the **Solution** cell, when a solution strategy is defined, is different than when additional iterations or time steps are specified in the **Run Calculations** task page. When a solution strategy is defined, FLUENT will complete the iterations or time steps specified by the solution strategy. Once the solution is **Up-to-Date**, continuing the calculation has no effect. On the other hand, when additional iterations or time steps are specified in the **Run Calculations** task page, every time you choose to continue the calculation, FLUENT will try to perform the specified number of iterations or time-steps.

For more information about using these features in FLUENT, see the FLUENT User's Guide.

2.11. Working With Input and Output Parameters in Workbench

Workbench uses parameters and design points to allow you to run optimization and what-if scenarios. You can define both input and output parameters in FLUENT that can be used in your Workbench project. You can also define parameters in other applications including ANSYS DesignModeler and ANSYS CFD-Post. Once you have defined parameters for your system, a **Parameters** cell is added to the system and the **Parameter Set** bus bar is added to your project. Arrows representing input and output parameters connect the bus bar to each system in which you have defined parameters.

Double-click the **Parameter Set** bus bar to open the **Parameters** workspace. The parameters workspace includes the **Outline of All Parameters** table that lists all of the parameters in your project as well as the **Table of Design Points** table in which you can specify design points.

To create a new design point, enter the input parameter values that you want to use for that design point in the **Table of Design Points** in the row with an asterisk (*) in the first column. You can create several design points. Once you have finished specifying design points, you can right-click the row for one design point and select the **Update Selected Design Point** option from the context menu to compute the output parameters for that design point. Alternatively, you can select **Update All Design Points** from the Toolbar to update all of your design points in sequence.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Only the data from the design point in the row labeled **Current** is saved with the project. If you want to post-process the results from a different design point in either ANSYS FLUENT or ANSYS CFD-Post, click the box in the **Exported** column for that design point before you update that design point. Otherwise, the data for that design point is automatically deleted after the output parameters for that design point are updated. If you choose to export a design point, the data associated with that design point is exported to a new project. The new project is located in the same directory as the original project. The name of the project is the same as the name of the original project, except that it is appended with `_dpn`, where `n` is the row number that corresponds to the design point in the original project's **Table of Design Points**.

Important

Note that you cannot create, edit, delete, or rename parameters in FLUENT if any iterations (or time-steps) have been performed. If you want to create, edit, delete, or rename parameters in FLUENT for a case with an existing solution, you must first initialize the solution.

For more information about input and output parameters in FLUENT, see the FLUENT User's Guide.

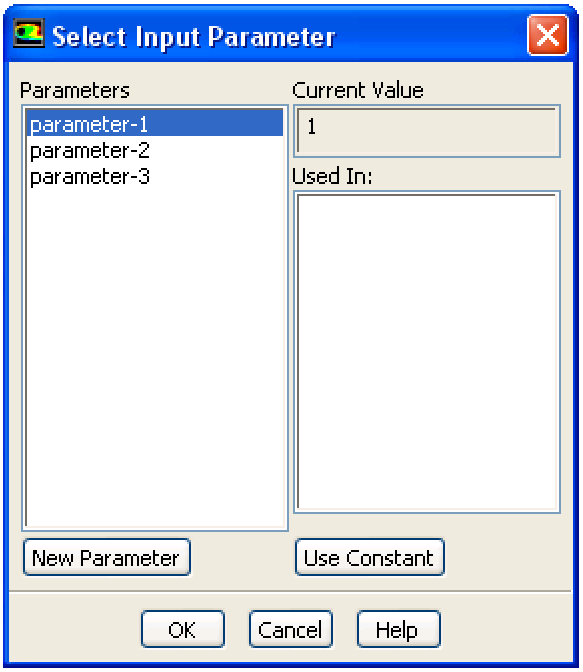
For more information about parameters, design points, what-if scenarios and optimization studies in Workbench, see the Workbench on-line documentation.

Note

Various ANSYS FLUENT setup—related input quantities (of type real and profile) can be assigned to an input parameter (indicated by the **New Input Parameter...** option in the corresponding drop-down list or by a small "p" icon next to the field). Clicking this option or icon displays the **Select Input Parameter** dialog box (see *Figure 2.18* (p. 53)) where you can create and assign input parameters.

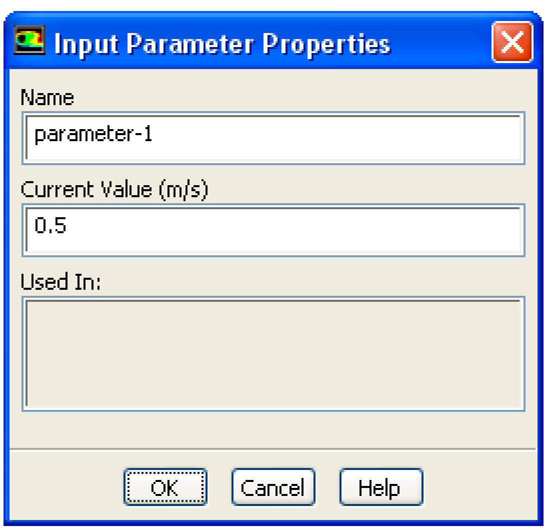
Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 2.18 The Select Input Parameter Dialog Box



The **Select Input Parameters** dialog box contains a list of existing compatible input parameters under **Parameters**. The **Current Value** field contains the value of the currently selected parameter. The **Used In** field lists any variables that are already associated with the currently selected parameter. You can also change the associated parameter or switch it back to a constant (i.e., real) value using the **Use Constant** button. Clicking the **New Parameter** button displays the **Input Parameters Properties** dialog box (see *Figure 2.19* (p. 53)) where you can define a new input parameter along with its value.

Figure 2.19 The Input Parameter Properties Dialog Box



The **Input Parameters Properties** dialog box allows you to enter a new **Name** and **Current Value** for the input parameter, as well as to see any variables that may be associated with the parameter.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note

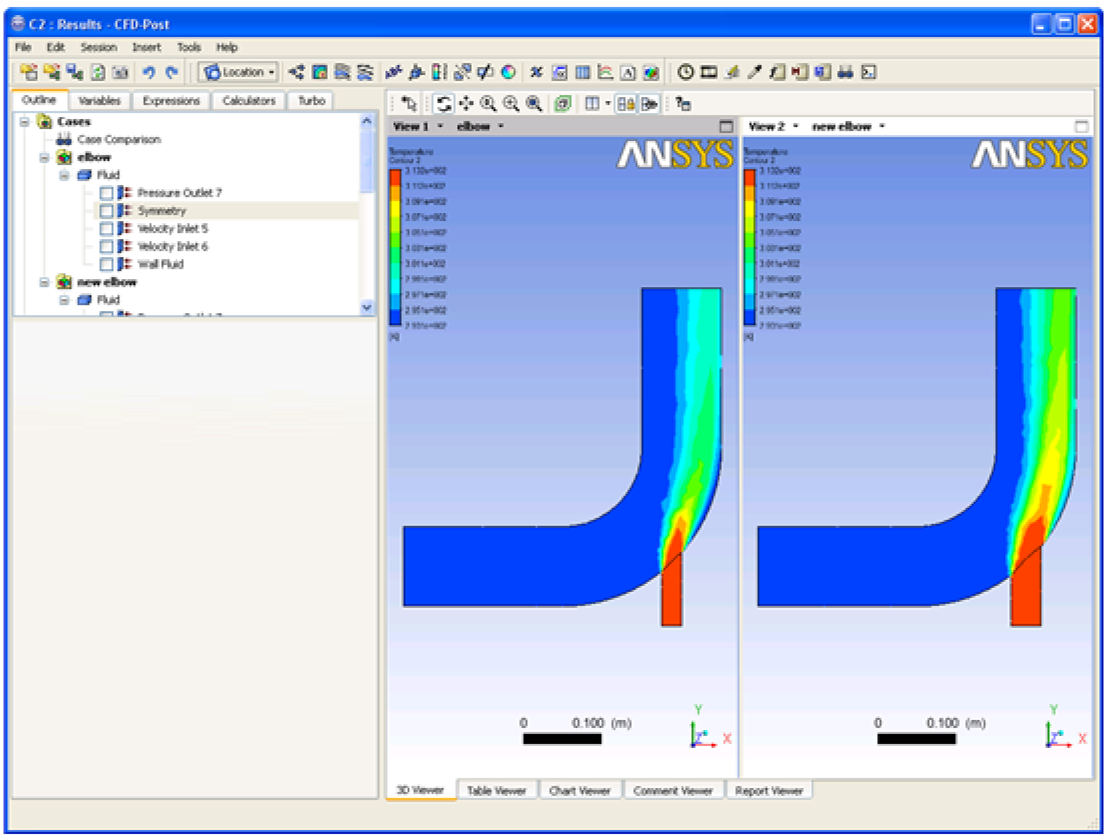
ANSYS FLUENT automatically creates generic default names for new input and output parameters (e.g., `parameter-1`, `parameter-2`, `parameter-3`, etc.) If a parameter is deleted, the default name is not reused. For example, if you have `parameter-1`, `parameter-2`, and `parameter-3`, then delete `parameter-2` and create a new parameter, the default name for the new parameter will be `parameter-4`.

2.12. Viewing Your FLUENT Data Using ANSYS CFD-Post

ANSYS CFD-Post is a post-processing application that helps you visualize the results of your ANSYS FLUENT CFD simulations. For FLUENT-based analysis systems, the **Results** cell provides access to the ANSYS CFD-Post application. In addition, the Toolbox contains a separate **Results** component system (i.e., ANSYS CFD-Post) that you can add to the Project Schematic and connect to FLUENT-based systems.

When a **Results** cell is connected to a FLUENT-based system's **Solution** cell and the **Results** cell's state is either **Refresh Required** or **Update Changes Pending**, you can view the results of your FLUENT calculation in ANSYS CFD-Post by double-clicking the **Results** cell. This will load the FLUENT results into ANSYS CFD-Post.

Figure 2.20 Multiple FLUENT Results Loaded Into ANSYS CFD-Post



Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

In addition to analyzing your results in ANSYS CFD-Post, you can also view the results of your simulation using the standard FLUENT postprocessing tools. For more information, see the separate FLUENT User's Guide.

Important

There are two options for exporting FLUENT files for ANSYS CFD-Post:

1. Standard FLUENT case and data files do not contain all variables, however, you can add additional quantities to your regular FLUENT data file using the **Data File Quantities** dialog box in FLUENT.
2. Lightweight data files are created by exporting ANSYS CFD-Post compatible files. These files can be used to save just the variables of interest.

When you edit a **Results** cell that is connected to the **Solution** cell in a FLUENT-based system, ANSYS CFD-Post always loads the standard FLUENT case and data files. For more information on exporting FLUENT files for ANSYS CFD-Post, see the separate FLUENT User's Guide.

2.13. Understanding the File Structure for FLUENT in Workbench

When you save a Workbench project (e.g., `my-project`), the project is saved with a `.wbpj` extension (e.g., `my-project.wbpj`). Other files associated with the project (through other Workbench applications such as ANSYS FLUENT or CFX) are located in the `dp0` folder within a `_files` folder (e.g., `my-project_files`).

Each system in the Project Schematic has its own directory under the `dp0` directory. The directory is named using the corresponding system identifier (e.g., `FFF` represents a FLUENT-based analysis system; `FLU` represents a FLUENT-based component system; `Post` represents a **Results** component system, etc.). The directory name is appended with a number to distinguish it from the directories for other systems of the same type (with the exception of the directory name for the first system of a specific type which has no number appended to it).

Within each system directory is a folder for each application that is part of the system. This folder is used to store the files generated and used by the application.

In addition to the settings, case and data files, the following files are managed by FLUENT in Workbench:

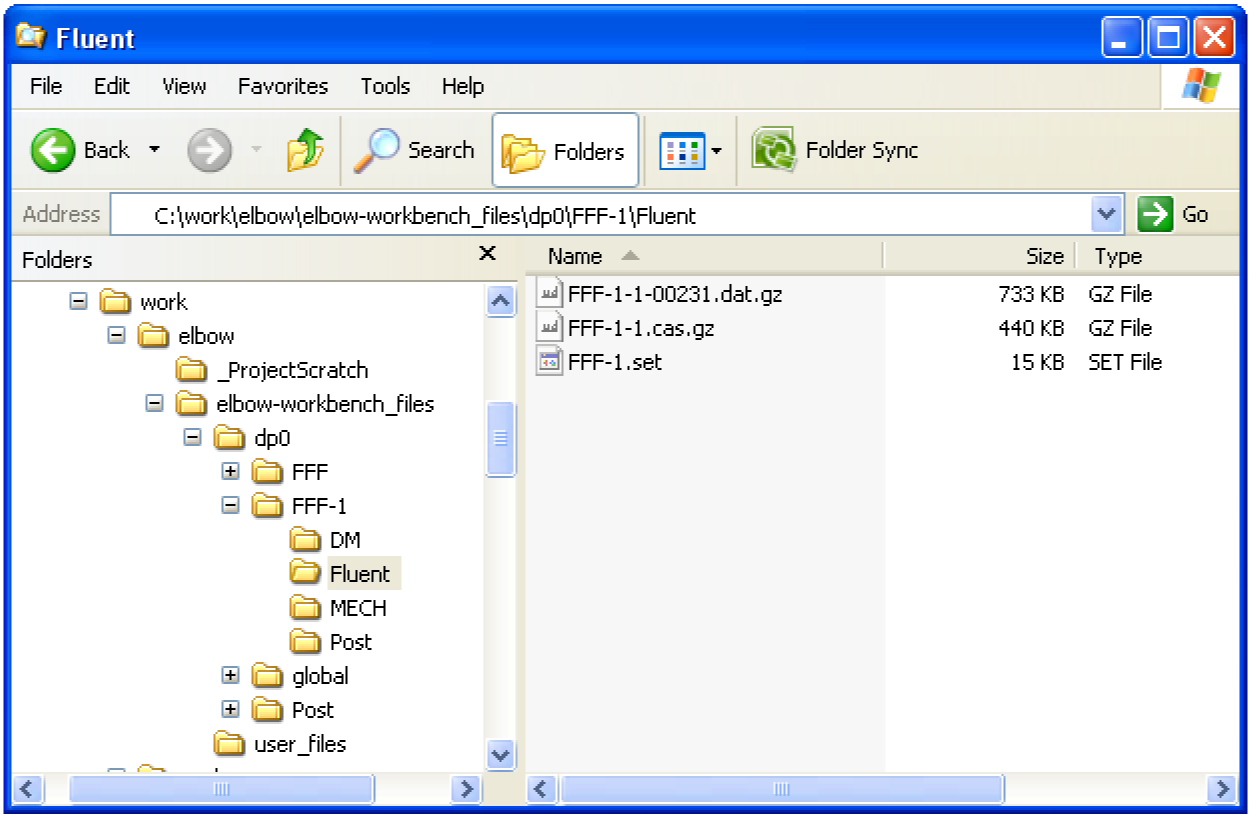
- monitor files
- flamelet files
- view factor files
- non-premixed files (`.pdf`)
- interpreted user-defined (UDF) files
- compiled UDF libraries
- Discrete Transfer Radiation (DTRM) `.ray` files
- mechanism files (`.che` or `.inp`)
- property database files (`thermo.db` and `transport.db`)

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You may use other types of files with FLUENT in Workbench, however, you are responsible for making sure that they are located in the appropriate folder within the project file structure.

The following figure represents an example of the directory structure for Workbench project with two **Fluid Flow (FLUENT)** systems and one **Results** system:

Figure 2.21 Example of the Directory Structure for a FLUENT-Based Project in Workbench

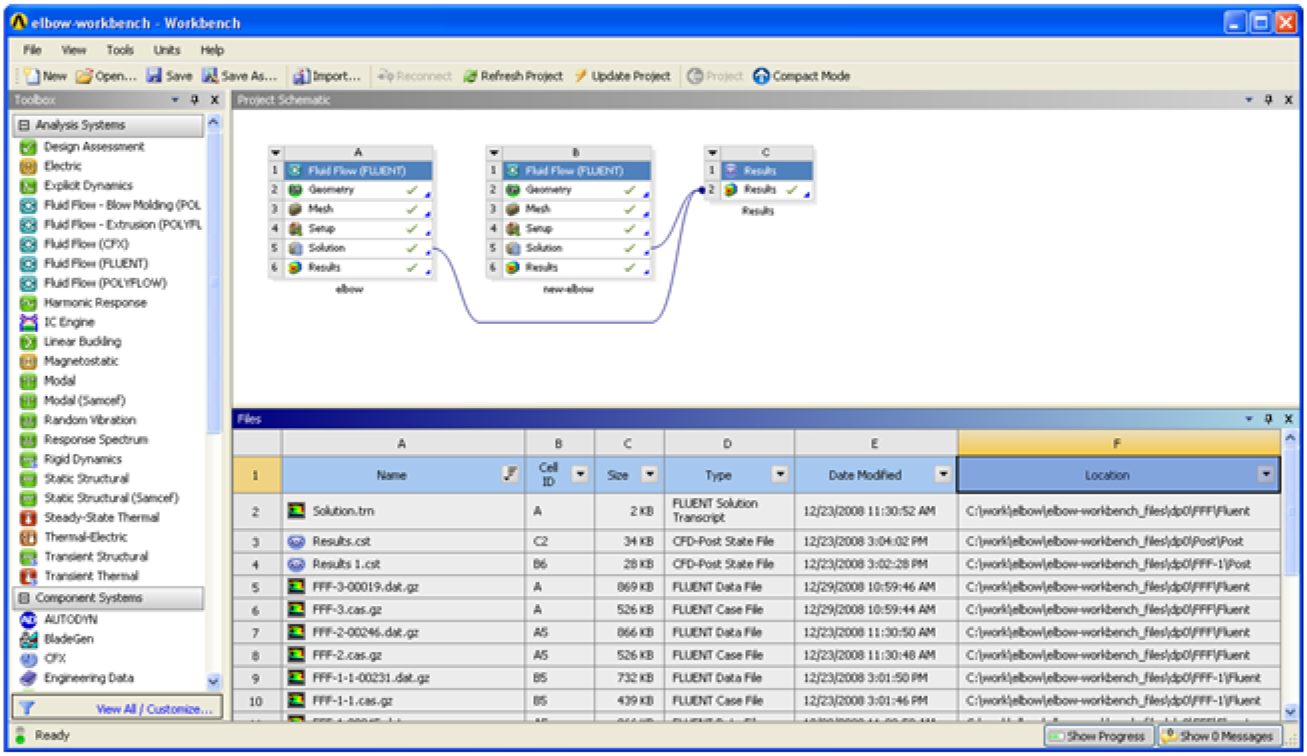


You can view the files associated with your Workbench project by selecting the **Files** option under the **View** menu.

View → **Files**

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 2.22 The Files View for a Project in Workbench



If data is shared between two systems, then files are also shared between the two systems. The shared file will exist in either the directory of the first system that used it, or in a global directory in the design point directory (depending on the type of system that generated the file).

The `_files` folder also contains a `user_files` directory. This directory should be used for any files you create or reference that you would like to store with the project.

Important

In general, you should not modify the structure of the project directories or delete or modify any of the files that Workbench applications have stored in the project directories. However, you may delete FLUENT case and data files that are stored with the project but are no longer needed. You should close FLUENT before deleting FLUENT case and data files from the project directories.

Important

Monitor files are automatically written to the appropriate working directory within the Workbench project files directory. You will not be able to specify a different path for monitor files from within FLUENT. When reading older case files in which a different location is specified for monitor files, a warning message will inform you that the path will be modified.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

If you intend to use a FLUENT journal file that reads or writes files while running under Workbench, the journal file and the files it references should be moved to the appropriate FLUENT folder in the appropriate system folder in the Workbench project working directory. File paths in the journal files should use relative paths to point to the new locations for the files.

Important

If you use models that generate mesh-dependent information the first time they are set up (e.g., DTRM, S2S) and then you change the mesh used in your system, the mesh dependent information (e.g., `.dtrm` file, `.s2s` file) may be incompatible with the new mesh. To resolve this issue, you can:

- Open FLUENT from your system, compute/write the `.dtrm` or `.s2s` output files, and then update the system.
- Use a solution strategy and write the `.s2s` file as a pre-initialization method using the appropriate command (e.g., `(write-sglobs "2d.s2s.gz")`).

Important

If your FLUENT setup involves compiling and loading a user-defined function (UDF), it is recommended that you copy the UDF source code files to the same location as your mesh and settings files before compiling the UDF libraries. This location is available in the Files view in Workbench. You may need to save the project first to create the appropriate files folder within the Workbench project (e.g. `project name \dp0\FFF \Fluent\`). If you need to unload a UDF library for any reason during your simulation, it is recommended that you save the project soon after unloading the library. If you archive a project that includes compiled user defined functions, you will need to recompile the libraries after opening the archived project. To do so, you will need to open FLUENT from the **Setup** cell, go to the **UDF Library Manager** dialog box (**Define** → **User Defined** → **Functions** → **Manage...**) and unload the existing UDF library. Next, rebuild the UDF library locally by going to the **Compiled UDFs** dialog box (**Define** → **User Defined** → **Functions** → **Compiled...**), selecting the archived `.c` and `.h` files, compiling, and loading the new UDF library. Finally, save the project.

For more information, please see the following section:

2.13.1. FLUENT File Naming in Workbench

2.13.1. FLUENT File Naming in Workbench

When running under Workbench, FLUENT automatically saves the settings, case, and data files for your system as needed.

FLUENT names these files using the base name of the mesh file. If a mesh (or case) file is imported, the base name of the imported file is used as the base name for the settings, case, and data files. If the mesh is created in Workbench, the mesh file uses the system's directory name as its base name.

When FLUENT saves a case file, it appends the base name with the run number. When FLUENT next writes a case file, the run number is incremented in order to avoid overwriting the previous case file.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When FLUENT saves a data file, it appends the base name with the same run number as the associated case file and with the iteration (or time-step) number at which it was saved. A new case file is not written every time that a data file is written. The case file is only written when the settings and/or mesh is changed.

The following example shows how file naming works in FLUENT under Workbench. In this example, you have a single **Fluid Flow (FLUENT)** analysis system and you have created the mesh in Workbench.

1. Double-click **Setup** cell. FLUENT launches and loads `FFF.msh`.
2. In ANSYS FLUENT, specify boundary conditions, initialize the solution and enter 10 iterations on the **Run Calculation** task page.
3. In the FLUENT application, select the **Calculate** button.

   `FFF.set` file is written and 10 iterations are completed.
4. Save the project.

   `FFF-1.cas.gz` and `FFF-1-00010.dat.gz` are written.
5. In the FLUENT application, select the **Calculate** button.

   10 additional iterations are completed.
6. Save the project.

   `FFF-1-00020.dat.gz` is written.
7. In the FLUENT application, change the value of the inlet velocity, select the **Calculate** button. When prompted, select the **Store the modified settings for a future calculation and use them this time** option.

   `FFF.set` is written (overwriting the existing file) and 10 additional iterations are completed.
8. Save the project.

   `FFF-2.cas.gz` and `FFF-2-00030.dat.gz` are written.

Important

The same naming convention is used when autosaving with FLUENT under Workbench.

2.14. Working with ANSYS Licensing

When working with Workbench, you have the option to share a single license between applications that use the same license or the option for each application to check out its own license.

For more information, please see the following section:

2.14.1. Shared Licensing Mode

2.14.1. Shared Licensing Mode

When using shared licensing, although a single license is shared between multiple applications, only one application can actively use the license at a time. Therefore, with just a single license, you can have both ANSYS FLUENT and ANSYS CFD-Post open at the same time. If iterations are being performed in the FLUENT session, you cannot do anything in the ANSYS CFD-Post session. However, if the FLUENT session is open and idle, you can work in ANSYS CFD-Post without closing FLUENT. Similarly, you can have multiple sessions of FLUENT open at the same time using just a single license.

If you open an application, it will first check to see if it can use a license that is already checked out. If it can, and that license is available, it will use that license. If the license is not available because it is being used by another application, you will be informed that the required license is not available. You will not be able to use the new application until that license becomes available. If there is not a license checked out that is compatible with the new application, the new application will check out an additional license.

In shared mode, you can have multiple licenses of each type of non-shareable licenses checked out at a time. For example, you can have 1 `acfd` license and 1 `acfd prepost` license checked out at the same time but you cannot have 2 `acfd` licenses checked out at the same time.

In shared mode, you can have multiple licenses of each type of non-shareable licenses checked out at a time. Non-shareable licenses include solver-only licenses and parallel licenses.

For more information about licensing and shared license mode, please see the Workbench on-line documentation.

2.15. Using FLUENT With the Remote Solve Manager (RSM)

The Remote Solve Manager (RSM) is a job queuing system used to distribute tasks that require computing resources. RSM schedules and manages tasks within Workbench and allows tasks to be sent *only* to the local machine to run in background mode, or they can be broken into a series of jobs for parallel processing.

For more information about using FLUENT in Workbench and RSM (with associated limitations), see Submitting Solutions for Local, Background, and Remote Solve Manager (RSM) Processes and Submitting FLUENT Jobs to RSM.

2.16. Using Custom Systems

Workbench allows you to add custom templates and includes some pre-defined custom templates. To use a custom template, double-click the template to add it to the Project Schematic.

Under **Custom Systems** in the Toolbox, there is a predefined FLUENT-based custom system (**FSI: Fluid Flow (FLUENT)** −> **Static Structural** – the single arrow is meant to convey that the interaction is one-way). When you double-click the template to add it to the Project Schematic, Workbench automatically creates a link between the **Geometry** cell in the FLUENT system and the **Geometry** cell in the **Structural** system and between the **Solution** cell in the FLUENT system and the **Setup** cell in the **Structural** system.

You can create your own system template and then save it as a **Custom System** by performing the following steps:

1. Manually create the desired system diagram in the Project Schematic.
2. Right-click the white space in the Project Schematic and select **Add to Custom** from the context menu.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

When you use FLUENT to perform a one-way fluid-structure interaction (FSI) analysis using the approach demonstrated in the predefined FLUENT-based custom system, you can only exchange surface data for force and thermal loads.

Important

When you use FLUENT to perform a one-way fluid-structure interaction (FSI) analysis using the approach demonstrated in the predefined FLUENT-based custom system, and you are performing a multiphase simulation in FLUENT, you cannot exchange thermal load data.

2.17. Using Journaling and Scripting with FLUENT in Workbench

You can keep a history of your interactions within Workbench, that can also include your interactions within FLUENT, by recording your interactions with the program(s) in session journals (also referred to as *journaling*). This is done using the **Scripting** submenu in the Workbench **File** menu:

File → Scripting → Record Journal...

In addition, since the journal files are Python-based scripts, you can edit and/or play back previously recorded journal files, or create new journals manually (also known as *scripting*), that include your interactions within Workbench and, if applicable, any interactions within FLUENT:

File → Scripting → Run Script File...

For more information about recording and using session journals in Workbench, as well as reference documentation containing available commands and properties, see the separate ANSYS Workbench Scripting Guide.

Important

When using the **Send Command** method to directly call a FLUENT text user interface (TUI) command, the TUI command will not be recognized unless you use double quotes around it (e.g., `setup.SendCommand("define model energy no")`). If a string is included in the TUI command, then a backslash is required before the quotes around the string. For example:

```
setup1.SendCommand(Command="(cx-gui-do cx-activate-item \"MenuBar*FileMenu*Close FLUENT\") ")
```

Important

FLUENT internal Scheme commands may not work properly when called directly using the **Send Command** method. Therefore, you should use FLUENT text user interface (TUI) commands instead.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.18. Performing System Coupling Simulations Using FLUENT in Workbench

You can use Workbench to perform coupled simulations using two or more systems (i.e., ANSYS Mechanical (e.g., Transient or Static Structural) and ANSYS FLUENT) using a System Coupling component system. As described in the separate System Coupling Guide, you can set up a one-way or two-way fluid-structure interaction (FSI) design analysis in Workbench by connecting a System Coupling component system to your Mechanical system and to your ANSYS FLUENT fluid flow analysis system.

Connecting the **Setup** cell from an ANSYS FLUENT analysis system to the **Setup** cell for the **System Coupling** component system signals the latter system that the ANSYS FLUENT solver wishes to act as a co-simulation participant in a coupled analysis. Few coupling related settings are required in the ANSYS FLUENT setup, as described in the section below. Instead, most of coupling related settings are made through the **System Coupling** system's **Setup** user interface. Once the coupling setup is complete, the coupled analysis is executed by updating the **System Coupling** system's **Solution** cell, rather than the same cell in the connected co-simulation participant systems.

Additional information can be found in the following sections:

2.18.1. Supported Capabilities and Limitations
2.18.2. Regions and Variables Available for System Coupling
2.18.3. System Coupling Related Settings in FLUENT
2.18.4. How FLUENT's Execution is Affected by System Couplings
2.18.5. Restarting FLUENT Analyses as Part of System Couplings
2.18.6. Running FLUENT as a System Coupling Participant from the Command Line
2.18.7. Troubleshooting Two-Way Coupled Analysis Problems

2.18.1. Supported Capabilities and Limitations

ANSYS FLUENT currently has the following capabilities when involved with the System Coupling:

- Support for the use of triangular & quadrilateral faced interface cell types
- Full dynamic mesh support everywhere except on coupling interface
- Local and distributed parallel
- Moving and deforming mesh specification on boundary wall regions using incremental displacements obtained through System Couplings. (Note the additional **System Coupling** option in the **Dynamic Mesh Zones** dialog box, see Specifying the Motion of Dynamic Zones for more information).
- Output of total force on all boundary wall regions. FLUENT is able to serve the total force variable (viscous and normal) on all wall boundaries (with or without enabling the **System Coupling** option in the **Dynamic Mesh Zones** dialog box).

Important

Note that using System Coupling in conjunction with the Remote Solver manager (RSM) is *not* supported. If attempted, you will receive the following message:

```
Solution updates for FLUENT systems participating in System Couplings
must run in the foreground. This change has been automatically applied.
```


Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.18.2. Regions and Variables Available for System Coupling

The following variables will be available on all boundary wall regions.

Table 2.2 Variables On Boundary Wall Regions

Display Name	Internal Name	Transfer Direction	Data Type	Physical Type
Force	Force	0	VectorXYZ**	Force
Incremental Displacement*	Displacement	1	VectorXYZ**	Length

* The Incremental Displacement variable is only available on walls with the **System Coupling** moving and deforming mesh (MDM) option selected and will qualify as valid coupling regions.

** Represents the force vector $\vec{F}$ (F_{xy}, F_{yz}, F_{zx}) and the incremental displacements vector $\vec{d}$ (Δd_x, Δd_y, Δd_z) respectively.

Note

Forces on wall/wall-shadow pairs are not supported (e.g., two different fluids on either side of a zero thickness internal wall in FLUENT).

2.18.3. System Coupling Related Settings in FLUENT

- Dynamic Mesh
  - The **System Coupling** option must be selected on the desired moving and deforming wall boundaries in order to obtain displacements from other co-simulation participants taking part in the coupled analysis.
- Run Calculation
  - When running as part of a transient coupled analysis, the step size for and duration of the analysis are controlled by the System Coupling service.
    - → The **Time Step Size**(s) specified in the FLUENT setup is currently ignored; the FLUENT solution will be advanced using the time step size specified as part of the System Coupling setup.
    - → The **Number of Time Steps** is also ignored.
  - The specified **Max Iterations/Time Step** corresponds to the maximum number of non-linear solver iterations performed per coupling iteration.

For steady-state, system–coupled FLUENT analyses, the System Coupling analysis settings override the FLUENT calculation settings. The FLUENT system executes (N/M) solver iterations per coupling iteration, where N is the number of solver iterations specified in the FLUENT **Run Calculation** task page, and M is the maximum number of coupling iterations from System Coupling service. Note that a minimum of 5 solver iterations will always be executed per coupling iteration, regardless of the value given by (N/M). For example, if the number of iterations in FLUENT is 50, and the number of coupling iterations in the System Coupling service is 5, then the system–coupled FLUENT system will execute 10 solver iterations per coupling iteration. Before executing the solver iterations, FLUENT gathers the incremental displacement values from System Coupling service, applies the displacements, then updates the mesh.

For pseudo-transient cases, please refer to Performing Pseudo Transient Calculations in the FLUENT User's Guide.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For steady-state dynamic mesh applications, please refer to Steady-State Dynamic Mesh Applications in the FLUENT User's Guide.

2.18.4. How FLUENT's Execution is Affected by System Couplings

ANSYS FLUENT's execution is modified slightly to allow the Coupling Service to manage the evolution of the coupled analysis. These changes are summarized in Process Synchronization and Analysis Evolution in the System Coupling User's Guide.

2.18.5. Restarting FLUENT Analyses as Part of System Couplings

ANSYS FLUENT writes out case and data files that contain information related to the simulation. The case file contains the mesh, boundary and cell zone conditions, solution parameters for the simulation, as well as information about the user interface and graphics environment. The data file contains the values of specified flow field quantities for each mesh element and the convergence history (residuals) for the flow field.

2.18.5.1. Generating Restart Files

Restarts of a system coupling analysis requires compatible restart points to exist in the coupling service and in each of the solvers participating in the analysis. To facilitate this, FLUENT will automatically generate the required case and data files based upon requests received from the System Coupling service. Note that this is in addition to the restart points (or results) manually requested via the **Autosave Every** field in the **Calculation Activities** task page within FLUENT.

2.18.5.2. Executing the Restart Run

Once the coupled analysis run is finished or interrupted, you can restart this run from any of the saved restart points.

Important

The restart point selected in the FLUENT solver must be consistent with the restart points selected for the System Coupling service and other coupling participants.

When the FLUENT solution is updated (or restarted), the case and data files corresponding to the current solution point will be used, if they exist. The last generated case and data files form the current solution point, by default. To specify an alternate current solution point, perform the following steps:

1. Double-click FLUENT's **Solution** cell in Workbench.

   Warning

   Editing (i.e. double-clicking) the FLUENT **Solution** cell will clear all solution data and restore the original mesh and settings.

2. Select **File** → **Solution Files** to open the **Solution Files** dialog box with a list of all available restart points.
3. Select the desired restart point to make its mesh and data the current set used by FLUENT. By default, the latest saved restart point is selected. Note that you can only select one restart point. Click the restart point to select/deselect it.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **Read** to read-in the data for the selected restart point.
5. Once the data is read, click **Close** to close the **Solution Files** dialog box.
6. Close FLUENT by selecting **File** → **Close FLUENT**.

In some cases, FLUENT setup changes are required to avoid failure of the coupled analysis (e.g. iteration controls, under-relaxation factors, etc.). As noted above, the settings that are stored in the case file corresponding to the current solution point will be used for FLUENT restarts. To modify the settings for a restart, a new case file for the restart point must be created as follows:

1. Note the name of the case (and data) file that is loaded when the current restart (or solution) point is read.
2. Make the desired setup changes.
3. Select **File** → **Export** → **Case...** and select the noted case file, and choose **OK** when prompted to overwrite the existing case file.
4. Close FLUENT by selecting **File** → **Close FLUENT**.

If you are presented with a notification that the modified mesh and settings will be used for the current run, choose **OK** to accept and close the dialog box.

2.18.6. Running FLUENT as a System Coupling Participant from the Command Line

System Coupling analyses can be run via the command line (described in Executing System Couplings Using the Command Line in the System Coupling User's Guide). To run ANSYS FLUENT as a coupling participant, execute the following steps:

- Complete the System Coupling–related settings in FLUENT (see *System Coupling Related Settings in FLUENT* (p. 63))
- Generate the case and data files needed to start FLUENT
  - Choose **Write** and **Case & Data** from the **File** menu in the FLUENT graphical interface
- Start the coupling service and obtain the following information from the System Coupling Server (SCS) file:
  - the port and host on which the service is being run, and
  - the identifier (or name) for FLUENT
- Use this SCS information to set the FLUENT–specific system coupling command line options (described in System Coupling Options in the FLUENT User's Guide).
- Note that if FLUENT is run in interactive mode, the interface will be locked from when it reports it has established the connection to the coupling service until when all other coupling participants do the same. This occurs so that all participants may be synchronized at the Initial Synchronization point. For information about synchronization points, see Process Synchronization and Analysis Evolution in the System Coupling User's Guide.

2.18.7. Troubleshooting Two-Way Coupled Analysis Problems

The following files, found in the FLUENT run directory (FFF/Fluent under a Workbench design point directory), may prove useful in troubleshooting coupled analysis problems:

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `cortexerror.log`: This file contains a historical summary of all of the errors that have occurred during all runs executed in the same run directory. With this in mind, ensure that you are considering messages corresponding to the most recent run. An example message is: "Update-Dynamic-Mesh failed. Negative cell volume detected.". This indicates that at the date and time noted, the mesh unexpectedly failed.
- `solution 1.log`: This file includes information regarding the FLUENT boundary conditions at which data from System Couplings was/is used. Please note that, like the `cortexerror.log` file, this file contains a historical summary for all runs executed in the same run directory.
- `solution.trn` (or other transcript file): This file contains a complete summary of the current/latest run's evolution. This is one of the most useful files to determine why the coupled analysis failed. To generate extensive debug output during the analysis, enter the following command when completing the FLUENT problem setup:

  ```
  (rpsetvar 'sc/verbosity 1)
  ```

Please provide all of these files when submitting a request for service to ANSYS personnel.

2.19. Performing FLUENT and Ansoft Coupling in Workbench

FLUENT can be coupled with the Ansoft Maxwell application within Workbench in order to perform a one way electromagnetic-thermal interaction problem. The coupling involves solving an electromagnetic problem in the Ansoft Maxwell application, and mapping the resulting volumetric loss information onto the solid cell zones of your FLUENT mesh. The loss information is mapped onto the cell centroids, and is considered a heat source (load) that is added to the energy equation. This capability is useful for applications such as simulating fluid flow around or inside electromechanical (EM) devices, when the temperature of the device is influenced by electromagnetic or electrical losses.

The overall workflow for an Ansoft-FLUENT analysis is as follows:

1. Create and solve the electromagnetic application using Ansoft Maxwell.
2. Drag and drop a FLUENT–based system and import a case or mesh file into FLUENT, and then double-click the **Setup** cell to start FLUENT.

   Note

   If you import a mesh into FLUENT that was generated using non-SI units, then you should scale and/or translate the mesh before you perform any coupling with Ansoft.

3. Drag and drop the **Solution** cell of the Ansoft system onto the **Setup** cell of FLUENT system to enable the data transfer.
4. Update the **Solution** cell of the Ansoft system.
5. Refresh the **Setup** of FLUENT system.
6. Specify the EM settings in FLUENT (**File** → **EM Mapping**).
7. Set up the FLUENT analysis as you normally would (e.g., specify boundary conditions, solution settings, etc.).
8. Solve the FLUENT analysis.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important

Currently, only the Ansoft Maxwell application is supported (not HFSS or Q3D) for one–way coupling between Ansoft and FLUENT in Workbench.

Important

The coupling between Ansoft and FLUENT in Workbench is not supported on Windows Vista (32 bit and 64 bit), Red Hat 6, and SUSE Linux Enterprise Server 11.

Note

During EM mapping, FLUENT automatically turns on the energy model, if you have not already done so.

Note

In 2D axisymmetrical cases only, Maxwell uses the RZ (or XZ) plane while FLUENT uses the XY plane by default. Therefore, when you export a geometry from Maxwell, you need to rotate the geometry in order for FLUENT to mesh the geometry before EM mapping can take place properly. This is not a concern for regular 2D or 3D cases. For those who use ANSYS DesignModeler, rotate the geometry exported from Maxwell by performing the following steps:

1. Define a new plane based on the XY plane, applying a 90 degree rotation along the X axis, and a 90 degree rotation along the Y axis.
2. Move the geometry from the XY plane to the new plane.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Appendix A. The FLUENT Menu Under Workbench

When FLUENT is running within Workbench, the FLUENT **File** and **Mesh** menus are slightly different. The differences are described in the sections that follow:

A.1. File/Refresh Input Data
A.2. File/Save Project
A.3. File/Import/Mesh...
A.4. File/Import/Case...
A.5. File/Import/Data...
A.6. File/Import/Case and Data...
A.7. File/Export/...
A.8. File/EM Mapping/Volumetric Energy Source...
A.9. Mesh/Recorded Mesh Operations...

The common functionality for stand-alone FLUENT is documented in the separate FLUENT User's Guide.

A.1. File/Refresh Input Data

The **File/Refresh Input Data** menu item allows you to refresh your FLUENT input data from within FLUENT. This option is only enabled if new input data exists, or if a parameter value has changed. For more information, see *Refreshing FLUENT Input Data* (p. 28).

A.2. File/Save Project

The **File/Save Project** menu item allows you to save your current Workbench project within FLUENT, along with your current FLUENT case and/or data files. For more information, see *Saving Your Work in FLUENT with Workbench* (p. 15).

A.3. File/Import/Mesh...

The **File/Import/Mesh...** menu item opens the **Select File** dialog box which allows you to select the appropriate mesh file to be read. For more information see *Importing Mesh, Case, and Data Files* (p. 23).

A.4. File/Import/Case...

The **File/Import/Case...** menu item is used to read in a FLUENT case file (extension `.cas`), or a mesh file (extension `.msh`, `.grd`, `.MSH`, or `.GRD`) that has been saved in the native format for FLUENT. See the User's Guide for details.

The **File/Import/Case...** menu item opens the **Select File** dialog box which allows you to select the appropriate file to be read. For more information see *Importing Mesh, Case, and Data Files* (p. 23).

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A.5. File/Import/Data...

The **File/Import/Data...** menu item is used to read in a FLUENT data file (which has a `.dat` extension) or parallel data file (which has a `.pdat` extension). This menu item will not be available until you read in a case or mesh file. See the separate FLUENT User's Guide for details.

The **File/Import/Data...** menu item opens the **Select File** dialog box which allows you to select the appropriate file to be read. For more information see *Importing Mesh, Case, and Data Files* (p. 23).

A.6. File/Import/Case and Data...

The **File/Import/Case & Data...** menu item is used to read in a FLUENT case file and the corresponding data file (e.g., `myfile.cas` and `myfile.dat`) together. See the separate FLUENT User's Guide for details.

The **File/Import/Case & Data...** menu item opens the **Select File** dialog box which allows you to select the appropriate files to be read. Select the appropriate case file, and the corresponding data file (i.e., the file having the same name with a `.dat` extension) will also be read in. For more information see *Importing Mesh, Case, and Data Files* (p. 23).

A.7. File/Export/...

When running under Workbench, several commands located under **Write** option under the **File** menu have been moved to the **Export** option under the **File** menu. The new commands are:

File → **Export** → **Case**

File → **Export** → **Data**

File → **Export** → **Case & Data**

File → **Export** → **PDF**

File → **Export** → **ISAT Table**

File → **Export** → **Flamelet**

File → **Export** → **Surface Clusters**

File → **Export** → **Profile**

File → **Export** → **Boundary Mesh**

These items are used when you want to manually export a file independent of the project. Files exported in this way are not used by the project unless you later import them into a new system. When you use an **Export** command, you can export the files to the location of your choice with a name of your choice.

There is no need to export files since Workbench always saves the files it needs automatically. These export commands are provided for your convenience when you want to save a specific file for later use.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A.8. File/EM Mapping/Volumetric Energy Source...

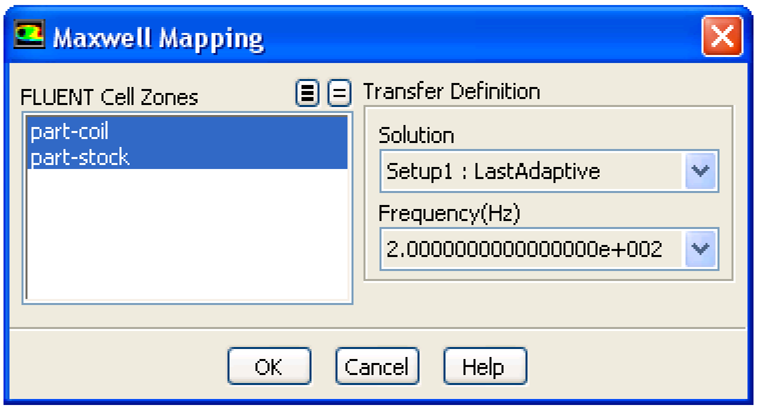
The **File/EM Mapping/Volumetric Energy Source...** menu item opens the *Maxwell Mapping Dialog Box* (p. 71).

Important

EM Mapping is only available in serial ANSYS FLUENT under Workbench. It is not available in parallel ANSYS FLUENT.

A.8.1. Maxwell Mapping Dialog Box

The **Maxwell Mapping** dialog box allows you to map the volumetric loss results from an Ansoft Maxwell electromagnetic simulation onto the centroids of solid cells in the ANSYS FLUENT mesh as a heat source.



Controls

FLUENT Cell Zones
contains a list of solid cell zones from the FLUENT mesh, onto which the loss information can be mapped. For these zones, FLUENT requests the heat source (loss) terms from the Ansoft application. By default all solid zones are selected.

Transfer Definition
contains elements related to the transfer of data, including:

Solution
contains available solution sets. Since the Ansoft application may have multiple solutions, FLUENT will request the generated heat source data for the selected solution.

Frequency
(only available for steady simulations) contains available frequencies. FLUENT will request that the Ansoft application provide the heat source data for the selected frequency.

Start time
(only available for transient simulations) contains the simulation start time. The Ansoft application will request to consider the selected time as the start time.

End time
(only available for transient simulations) contains the simulation end time. The Ansoft application will request to consider the selected time as the end time.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A.9. Mesh/Recorded Mesh Operations...

The **Mesh/Recorded Mesh Operations** menu item allows you to view common mesh manipulation operations that you perform within FLUENT while working in Workbench. For more information, see *Reviewing Mesh Manipulation Operations in FLUENT* (p. 40).

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Index

A
analysis
  example, 17
  systems , 4
ANSYS CFD-Post
  exporting FLUENT files for, 55
  viewing FLUENT data in, 54

C
calculation
  continuing, 30
  interrupting, 30
  restarting, 30
case
  importing files, 23, 69–70
  modification strategies, 50
cells
  FLUENT component systems, 7
  fluid flow FLUENT analysis systems, 5
  properties, 10
  states, 8
clearing generated data , 29
closing, 16
component
  systems, 4
connecting systems, 33
  dragging and dropping from toolbox, 35
  dragging and dropping solution cells, 38
connections
  shared data , 33
  transfer data, 33
continuing a calculation, 30
conventions used in this guide, vi
copying FLUENT Launcher property settings, 15
coupling
  one–way FLUENT and Ansoft, 66
  systems, 62
creating systems, 4
custom systems, 60

D
data
  clearing generated , 29
  exporting FLUENT files for ANSYS CFD-Post, 55
  importing files, 23, 70
  refreshing FLUENT input data, 28, 69
  resetting, 29
  viewing in ANSYS CFD-Post, 54
duplicating systems, 40

E
example, 17
exiting, 16
exporting files, 70
exporting FLUENT files for ANSYS CFD-Post, 55

F
File Menu, 69
file naming conventions, 58
file structure, 55
File/Export..., 70
File/Export/Boundary Mesh, 70
File/Export/Case, 70
File/Export/Case and Data, 70
File/Export/Data, 70
File/Export/Flamelet, 70
File/Export/ISAT Table , 70
File/Export/PDF , 70
File/Export/Profile, 70
File/Export/Surface Clusters, 70
File/Import/Case and Data..., 70
File/Import/Case..., 69
File/Import/Data..., 70
File/Import/Mesh..., 69
File/Refresh Input Data, 69
File/Save Project, 69
files
  exporting, 70
  exporting FLUENT files for ANSYS CFD-Post, 55
  importing
    case, 23, 69–70
    data, 23, 69–70
    mesh, 23, 69–70
FLUENT
  component systems, 4
    cells, 7
  exiting, 16
  fluid flow analysis systems, 4
    cells, 5
  Launcher
    cell properties, 10
    copying property settings, 15
    settings, 9
  starting, 9
fluid-structure interaction, 60

G
graphical user interface, 2

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

H
help, 20

I
importing
    case files, 23, 69–70
    data files, 23, 69–70
    mesh files, 23, 69–70
input data
    refreshing, 28, 69
input parameters, 51
interrupting a calculation, 30
introduction, 1

J
journaling and scripting, 61

L
licensing, 59
limitations, 2

M
manual
    using the, v
Maxwell Mapping dialog box, 71
mesh
    changing in FLUENT, 45
    importing files, 23, 69
    recorded FLUENT operations, 72
mesh operations
    reviewing in FLUENT, 40
Mesh/Recorded Mesh Operations, 72

O
one-way FSI, 60
one–way coupling
    FLUENT and Ansoft, 66
Online Help, 20
output parameters, 51

P
parameters
    input and output, 51
properties, 10

Q
Quick Help, 20

R
recorded mesh operations, 72
refreshing input data , 28, 69
Remote Solve Manager (RSM), 60
resetting data, 29
restarting a calculation, 30

S
saving
    setup data, 15, 69
    solution data, 15, 69
    your work, 15, 69
settings
    changing in FLUENT, 45
setup
    updating, 26
setup data
    saving, 15, 69
shared data connections, 33
shared licensing mode, 59
Sidebar Help, 20
solution
    updating, 26
solution data
    saving, 15, 69
solution strategies, 50
starting FLUENT, 9
    specifying FLUENT Launcher settings, 9
        using cell properties, 10
strategies
    solution, 50
system coupling, 62
    available regions in FLUENT, 63
    capabilities and limitations in FLUENT, 62
    effect on FLUENT execution, 64
    executing restart runs, 64
    FLUENT settings, 63
    generating restart files, 64
    restarting in FLUENT, 64
    running FLUENT on command line, 65
    trouble-shooting, 65
systems
    cells , 5, 7
    connecting, 33
        dragging and dropping from toolbox, 35
        dragging and dropping solution cells, 38
    coupling, 62
    creating, 4
    custom, 60
    duplicating, 40
    FLUENT component, 4
    fluid flow (FLUENT) analysis, 4

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

T

transfer data connections, 33

U

update command, 26
using the manual, v

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 14.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.