

# ANSYS FLUENT Tutorial 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 .....	xiii
1. What's In This Manual .....	xiii
2. The Contents of the FLUENT Manuals .....	xiii
3. Where to Find the Files Used in the Tutorials .....	xiv
4. How To Use This Manual .....	xiv
4.1. For the Beginner .....	xv
4.2. For the Experienced User .....	xv
5. Typographical Conventions Used In This Manual .....	xv
<b>1. Introduction to Using ANSYS FLUENT in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow .....</b>	<b>1</b>
1.1. Introduction .....	1
1.2. Prerequisites .....	1
1.3. Problem Description .....	2
1.4. Setup and Solution .....	2
1.4.1. Preparation .....	3
1.4.2. Step 1: Creating a FLUENT Fluid Flow Analysis System in ANSYS Workbench .....	4
1.4.3. Step 2: Creating the Geometry in ANSYS DesignModeler .....	8
1.4.4. Step 3: Meshing the Geometry in the ANSYS Meshing Application .....	19
1.4.5. Step 4: Setting Up the CFD Simulation in ANSYS FLUENT .....	26
1.4.6. Step 5: Displaying Results in ANSYS FLUENT and CFD-Post .....	50
1.4.7. Step 6: Duplicating the FLUENT-Based Fluid Flow Analysis System .....	60
1.4.8. Step 7: Changing the Geometry in ANSYS DesignModeler .....	61
1.4.9. Step 8: Updating the Mesh in the ANSYS Meshing Application .....	64
1.4.10. Step 9: Calculating a New Solution in ANSYS FLUENT .....	67
1.4.11. Step 10: Comparing the Results of Both Systems in CFD-Post .....	69
1.4.12. Step 11: Summary .....	74
<b>2. Parametric Analysis in ANSYS Workbench Using ANSYS FLUENT .....</b>	<b>77</b>
2.1. Introduction .....	77
2.2. Prerequisites .....	78
2.3. Problem Description .....	78
2.4. Setup and Solution .....	82
2.4.1. Preparation .....	83
2.4.2. Step 1: Adding Constraints to ANSYS DesignModeler Parameters in ANSYS Workbench .....	83
2.4.3. Step 2: Setting Up the CFD Simulation in ANSYS FLUENT .....	92
2.4.4. Step 3: Defining Input and Output Parameters in ANSYS FLUENT and Running the Simulation .....	96
2.4.5. Step 4: Postprocessing in ANSYS CFD-Post .....	108
2.4.6. Step 5: Creating Additional Design Points in ANSYS Workbench .....	122
2.4.7. Step 6: Postprocessing the New Design Points in CFD-Post .....	124
2.4.8. Step 7: Summary .....	130
<b>3. Introduction to Using ANSYS FLUENT: Fluid Flow and Heat Transfer in a Mixing Elbow .....</b>	<b>131</b>
3.1. Introduction .....	131
3.2. Prerequisites .....	131
3.3. Problem Description .....	131
3.4. Setup and Solution .....	132
3.4.1. Preparation .....	133
3.4.2. Step 1: Launching ANSYS FLUENT .....	133
3.4.3. Step 2: Mesh .....	136
3.4.4. Step 3: General Settings .....	140
3.4.5. Step 4: Models .....	143

# Using This Manual

---

This preface is divided into the following sections:

1. What's In This Manual
2. The Contents of the FLUENT Manuals
3. Where to Find the Files Used in the Tutorials
4. How To Use This Manual
5. Typographical Conventions Used In This Manual

## 1. What's In This Manual

The ANSYS FLUENT Tutorial Guide contains a number of tutorials that teach you how to use ANSYS FLUENT to solve different types of problems. In each tutorial, features related to problem setup and postprocessing are demonstrated.

The tutorials are written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- *Introduction to Using ANSYS FLUENT in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow* (p. 1)
- *Parametric Analysis in ANSYS Workbench Using ANSYS FLUENT* (p. 77)
- *Introduction to Using ANSYS FLUENT: Fluid Flow and Heat Transfer in a Mixing Elbow* (p. 131)

and that you are familiar with the ANSYS FLUENT navigation pane and menu structure. Some steps in the setup and solution procedure will not be shown explicitly.

*Introduction to Using ANSYS FLUENT: Fluid Flow and Heat Transfer in a Mixing Elbow* (p. 131)

All of the tutorials include some postprocessing instructions, but *Postprocessing* (p. 1059) is devoted entirely to postprocessing.

## 2. The Contents of the FLUENT Manuals

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

[FLUENT Getting Started Guide](#) contains general information about getting started with using FLUENT.

[FLUENT User's Guide](#) contains detailed information about using FLUENT, including information about the user interface, reading and writing files, defining boundary conditions, setting up physical models, calculating a solution, and analyzing your results.

[FLUENT in Workbench User's Guide](#) contains information about getting started with and using FLUENT within the Workbench environment.

[FLUENT Theory Guide](#) contains reference information for how the physical models are implemented in FLUENT.

[FLUENT UDF Manual](#) contains information about writing and using user-defined functions (UDFs).

[FLUENT Tutorial Guide](#) contains a number of example problems with detailed instructions, commentary, and postprocessing of results.

[FLUENT Text Command List](#) contains a brief description of each of the commands in FLUENT's text interface.

[FLUENT Adjoint Solver Module Manual](#) contains information about the background and usage of FLUENT's Adjoint Solver Module that allows you to obtain detailed sensitivity data for the performance of a fluid system.

[FLUENT Battery Module Manual](#) contains information about the background and usage of FLUENT's Battery Module that allows you to analyze the behavior of electric batteries.

[FLUENT Continuous Fiber Module Manual](#) contains information about the background and usage of FLUENT's Continuous Fiber Module that allows you to analyze the behavior of fiber flow, fiber properties, and coupling between fibers and the surrounding fluid due to the strong interaction that exists between the fibers and the surrounding gas.

[FLUENT Fuel Cell Modules Manual](#) contains information about the background and the usage of two separate add-on fuel cell models for FLUENT that allow you to model polymer electrolyte membrane fuel cells (PEMFC), solid oxide fuel cells (SOFC), and electrolysis with FLUENT.

[FLUENT Magnetohydrodynamics \(MHD\) Module Manual](#) contains information about the background and usage of FLUENT's Magnetohydrodynamics (MHD) Module that allows you to analyze the behavior of electrically conducting fluid flow under the influence of constant (DC) or oscillating (AC) electromagnetic fields.

[FLUENT Migration Manual](#) contains information about transitioning from the previous release of FLUENT, including details about new features, solution changes, and text command list changes.

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

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

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

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

### 3. Where to Find the Files Used in the Tutorials

Each of the tutorials uses an existing mesh file. (Tutorials for mesh generation are provided with the mesh generator documentation.) You will find the appropriate mesh file (and any other relevant files used in the tutorial) on the ANSYS Customer Portal. The "Preparation" step of each tutorial will tell you where to find the necessary files. (Note that Tutorials [Postprocessing](#) (p. 1059) and [Parallel Processing](#) (p. 1113) use existing case and data files.)

Some of the more complex tutorials may require a significant amount of computational time. If you want to look at the results immediately, without waiting for the calculation to finish, final solution files are provided in a `solution_files` folder which you can access after extracting the tutorial input archive.

### 4. How To Use This Manual

Depending on your familiarity with computational fluid dynamics and the ANSYS FLUENT software, you can use this tutorial guide in a variety of ways.

## 4.1. For the Beginner

If you are a beginning user of ANSYS FLUENT you should first read and solve Tutorial 1, in order to familiarize yourself with the interface and with basic setup and solution procedures. You may then want to try a tutorial that demonstrates features that you are going to use in your application. For example, if you are planning to solve a problem using the non-premixed combustion model, you should look at [Using the Non-Premixed Combustion Model \(p. 717\)](#).

You may want to refer to other tutorials for instructions on using specific features, such as custom field functions, mesh scaling, and so on, even if the problem solved in the tutorial is not of particular interest to you. To learn about postprocessing, you can look at [Postprocessing \(p. 1059\)](#), which is devoted entirely to postprocessing (although the other tutorials all contain some postprocessing as well).

## 4.2. For the Experienced User

If you are an experienced ANSYS FLUENT user, you can read and/or solve the tutorial(s) that demonstrate features that you are going to use in your application. For example, if you are planning to solve a problem using the non-premixed combustion model, you should look at [Using the Non-Premixed Combustion Model \(p. 717\)](#).

You may want to refer to other tutorials for instructions on using specific features, such as custom field functions, mesh scaling, and so on, even if the problem solved in the tutorial is not of particular interest to you. To learn about postprocessing, you can look at [Postprocessing \(p. 1059\)](#), which is devoted entirely to postprocessing (although the other tutorials all contain some postprocessing as well).

## 5. Typographical Conventions Used In This Manual

Several typographical conventions are used in the text of the tutorials to facilitate your learning process.

- Different type styles are used to indicate graphical user interface menu items and text interface menu items (e.g., **Zone Surface** dialog box, *surface/zone-surface* command).
- The text interface type style is also used when illustrating exactly what appears on the screen or exactly what you must type in the text window or in a dialog box.
- Instructions for performing each step in a tutorial will appear in standard type. Additional information about a step in a tutorial appears in italicized type.
- 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.

The words surrounded by boxes invoke menus (or submenus) and the arrows point from a specific menu toward the item you should select from that menu.

---

# Chapter 1: Introduction to Using ANSYS FLUENT in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

---

This tutorial is divided into the following sections:

- 1.1. Introduction
- 1.2. Prerequisites
- 1.3. Problem Description
- 1.4. Setup and Solution

## 1.1. Introduction

This tutorial illustrates using ANSYS FLUENT fluid flow systems in ANSYS Workbench to set up and solve a three-dimensional turbulent fluid-flow and heat-transfer problem in a mixing elbow. It is designed to introduce you to the ANSYS Workbench tool set using a simple geometry. Guided by the steps that follow, you will create the elbow geometry and the corresponding computational mesh using the geometry and meshing tools within ANSYS Workbench. You will use ANSYS FLUENT to set up and solve the CFD problem, then visualize the results in both ANSYS FLUENT and in the CFD-Post postprocessing tool. Some capabilities of ANSYS Workbench (for example, duplicating fluid flow systems, connecting systems, and comparing multiple data sets) are also examined in this tutorial.

This tutorial demonstrates how to do the following:

- Launch ANSYS Workbench.
- Create a FLUENT fluid flow analysis system in ANSYS Workbench.
- Create the elbow geometry using ANSYS DesignModeler.
- Create the computational mesh for the geometry using ANSYS Meshing.
- Set up the CFD simulation in ANSYS FLUENT, which includes:
  - Setting material properties and boundary conditions for a turbulent forced-convection problem.
  - Initiating the calculation with residual plotting.
  - Calculating a solution using the pressure-based solver.
  - Examining the flow and temperature fields using ANSYS FLUENT and CFD-Post.
- Create a copy of the original FLUENT fluid flow analysis system in ANSYS Workbench.
- Change the geometry in ANSYS DesignModeler, using the duplicated system.
- Regenerate the computational mesh.
- Recalculate a solution in ANSYS FLUENT.
- Compare the results of the two calculations in CFD-Post.

## 1.2. Prerequisites

This tutorial assumes that you have little to no experience with ANSYS Workbench, ANSYS DesignModeler, ANSYS Meshing, ANSYS FLUENT, or CFD-Post, and so each step will be explicitly described.



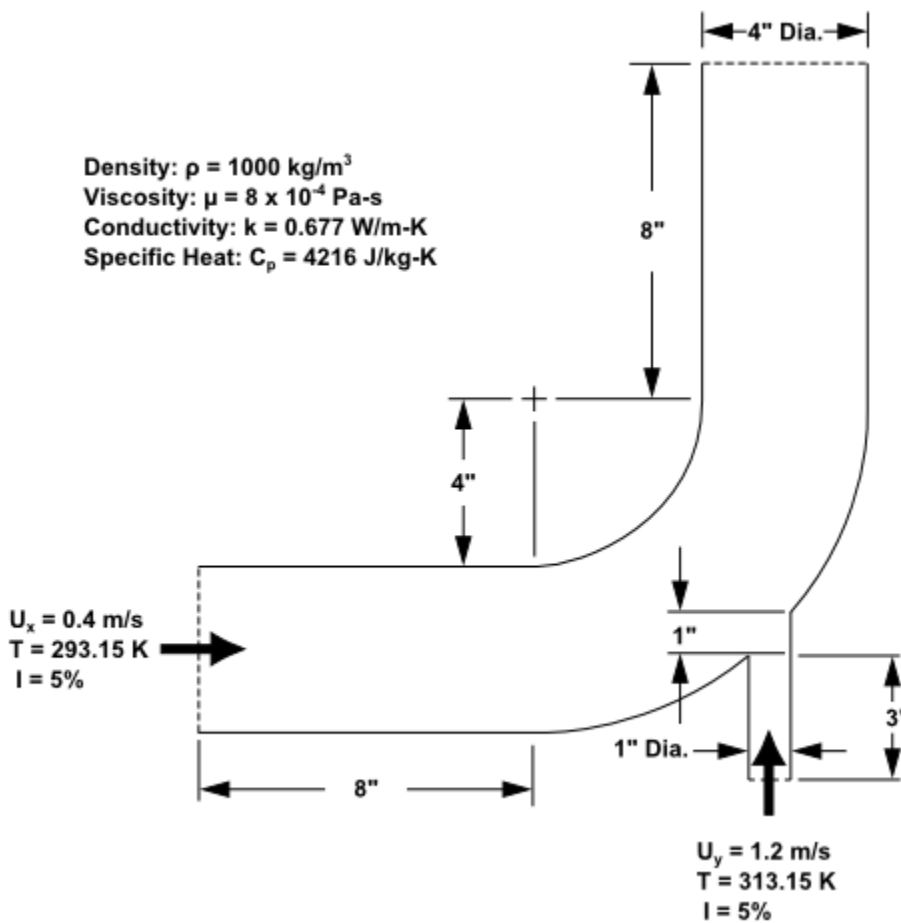
## 1.3. Problem Description

The problem to be considered is shown schematically in *Figure 1.1* (p. 2). A cold fluid at 293.15 K flows into the pipe through a large inlet and mixes with a warmer fluid at 313.15 K that enters through a smaller inlet located at the elbow. The mixing elbow configuration is encountered in piping systems in power plants and process industries. It is often important to predict the flow field and temperature field in the area of the mixing region in order to properly design the junction.

### Note

Because the geometry of the mixing elbow is symmetric, only half of the elbow needs to be modeled.

**Figure 1.1 Problem Specification**



## 1.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

- 1.4.1. Preparation
- 1.4.2. Step 1: Creating a FLUENT Fluid Flow Analysis System in ANSYS Workbench
- 1.4.3. Step 2: Creating the Geometry in ANSYS DesignModeler
- 1.4.4. Step 3: Meshing the Geometry in the ANSYS Meshing Application
- 1.4.5. Step 4: Setting Up the CFD Simulation in ANSYS FLUENT
- 1.4.6. Step 5: Displaying Results in ANSYS FLUENT and CFD-Post

- 1.4.7. Step 6: Duplicating the FLUENT-Based Fluid Flow Analysis System
- 1.4.8. Step 7: Changing the Geometry in ANSYS DesignModeler
- 1.4.9. Step 8: Updating the Mesh in the ANSYS Meshing Application
- 1.4.10. Step 9: Calculating a New Solution in ANSYS FLUENT
- 1.4.11. Step 10: Comparing the Results of Both Systems in CFD-Post
- 1.4.12. Step 11: Summary

## 1.4.1. Preparation

To prepare for running this tutorial, set up a working folder on the computer you will be using. You can also download some files from the ANSYS Customer Portal web page, which you can reach via the **Customer Portal** link on the [ANSYS Website](http://www.ansys.com) at <http://www.ansys.com>.

1. Obtain the Tutorial Inputs Archive from the ANSYS Customer Portal.
  - a. On the [Customer Portal](#) Home page, click **Download Software**. This takes you to the **ANSYS Download Center Wizard**.
  - b. Click **Next Step**.
  - c. Choose a Download Type. Select the **Current Release and Updates** radio button.
  - d. Click **Next Step**.
  - e. Choose the Hardware Platform you are using.
  - f. Click **Next Step**.
  - g. At the bottom of the page, under the heading **ANSYS Documentation and Examples**, check the box next to **ANSYS Fluid Dynamics Tutorial Inputs**.
  - h. Click **Next Step**.
  - i. Click the **ANSYS Fluid Dynamics Tutorial Inputs** link to download the `ANSYS_Fluid_Dynamics_Tutorial_Inputs.zip` file.
  - j. Save and extract the `.zip` file to any convenient location. Tutorial mesh and solution files for each of the ANSYS Fluid Dynamics products are located in `v140\Tutorial_Inputs\Fluid_Dynamics\<product>`.
2. Copy the `elbow-workbench.zip` file from the FLUENT sub-folder of the `ANSYS_Fluid_Dynamics_Tutorial_Inputs.zip` archive.
3. Unzip `elbow-workbench.zip` to your working folder. This file contains a folder, `elbow-workbench`, that holds four items:
  - Two geometry files, `elbow_geometry.agdb` and `elbow_geometry.stp`
  - An ANSYS Workbench project file, `elbow-workbench.wbpj`
  - A project folder (`elbow-workbench_files`) containing supporting files such as solution and results files

---

### Note

ANSYS FLUENT tutorials are prepared using ANSYS FLUENT on a Windows system. The screen shots and graphic images in the tutorials may be slightly different than the appearance on your system, depending on the operating system or graphics card.

## 1.4.2. Step 1: Creating a FLUENT Fluid Flow Analysis System in ANSYS Workbench

In this step, you will start ANSYS Workbench, create a new FLUENT fluid flow analysis system, then review the list of files generated by ANSYS Workbench.

1. Start ANSYS Workbench by clicking the Windows **Start** menu, then selecting the **Workbench 14.0** option in the **ANSYS 14.0** program group.

**Start** → **All Programs** → **ANSYS 14.0** → **Workbench 14.0**

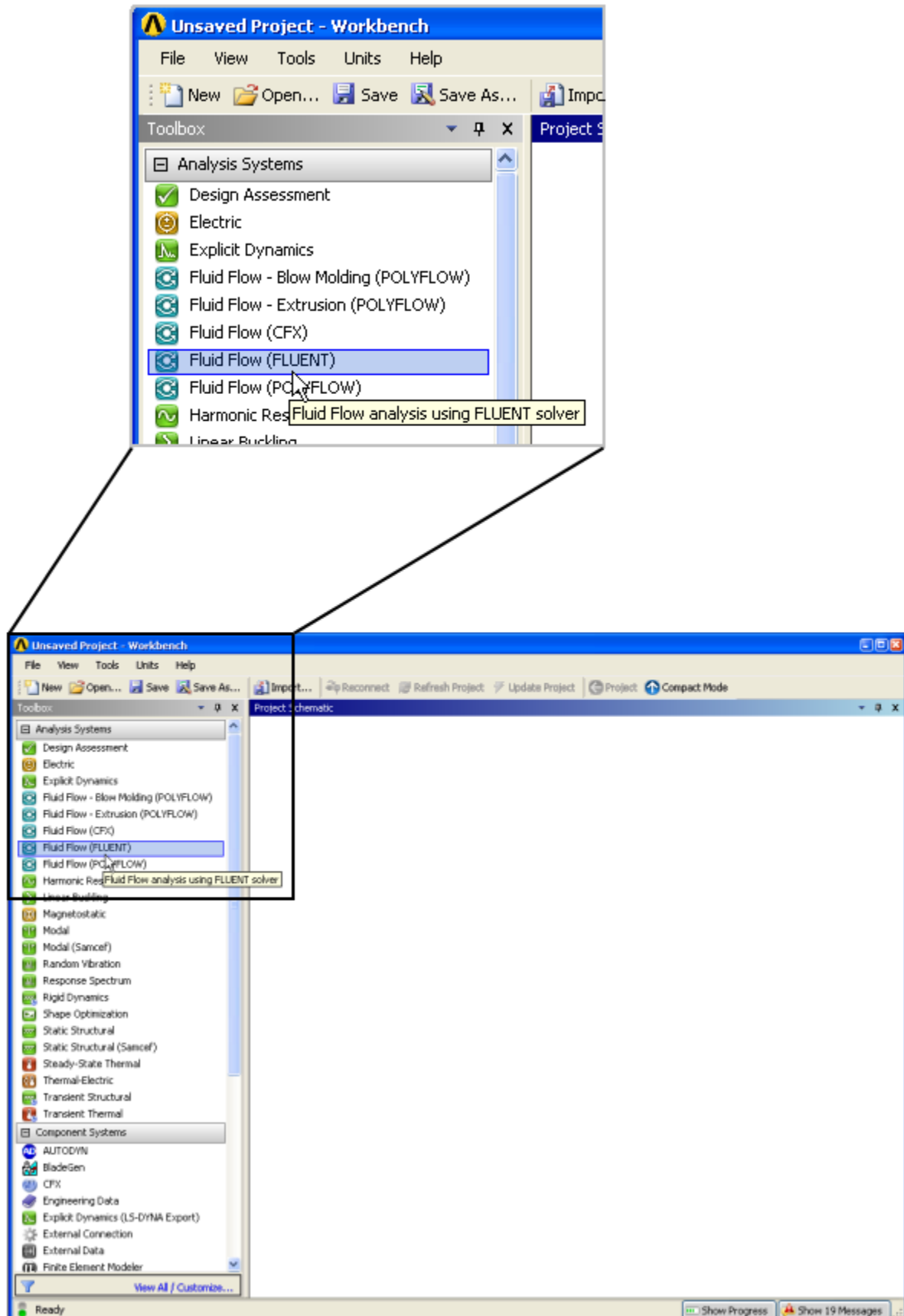
This displays the ANSYS Workbench application window, which has the **Toolbox** on the left and the **Project Schematic** to its right. Various supported applications are listed in the **Toolbox** and the components of the analysis system will be displayed in the **Project Schematic**.

---

### Note

When you first start ANSYS Workbench, the **Getting Started** pop-up window is displayed, offering assistance through the online help for using the application. You can keep the window open, or close it by clicking the 'X' icon in the upper right-hand corner. If you need to access the online help at any time, use the **Help** menu, or press the **F1** key.

2. Create a new FLUENT fluid flow analysis system by double-clicking the **Fluid Flow (FLUENT)** option under **Analysis Systems** in the **Toolbox**.

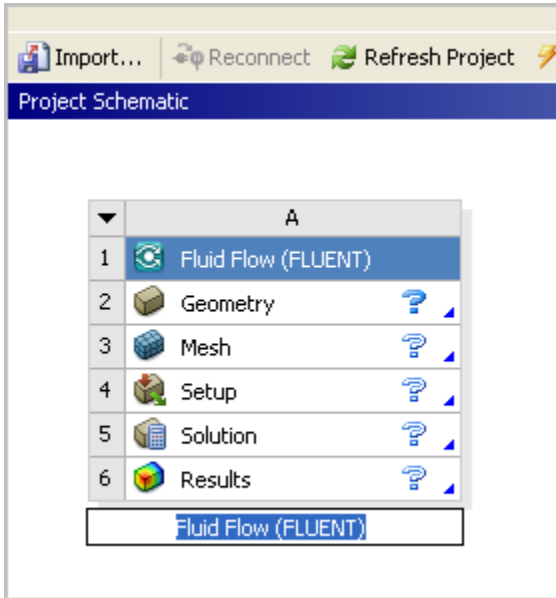
**Figure 1.2 Selecting the Fluid Flow (FLUENT) Analysis System in ANSYS Workbench**

This creates a new FLUENT-based fluid flow analysis system in the **Project Schematic**.

### Tip

You can also drag-and-drop the analysis system into the **Project Schematic**. A green dotted outline indicating a potential location for the new system initially appears in the **Project Schematic**. When you drag the system to one of the outlines, it turns into a red box to indicate the chosen location of the new system.

**Figure 1.3 ANSYS Workbench with a New FLUENT-Based Fluid Flow Analysis System**



3. Name the analysis.
  - a. Double-click the **Fluid Flow (FLUENT)** label underneath the analysis system (if it is not already highlighted).
  - b. Enter `elbow` for the name of the analysis system.
4. Save the project.
  - a. Select the **Save** option under the **File** menu in ANSYS Workbench.

#### **File** → **Save**

This displays the **Save As** dialog box, where you can browse to your working folder and enter a specific name for the ANSYS Workbench project.

- b. In your working directory, enter `elbow-workbench` as the project **File name** and click the **Save** button to save the project. ANSYS Workbench saves the project with a `.wbproj` extension and also saves supporting files for the project.

Note that the fluid flow analysis system is composed of various *cells* (**Geometry**, **Mesh**, etc.) that represent the workflow for performing the analysis. ANSYS Workbench is composed of multiple data-integrated and native applications in a single, seamless project flow, where individual cells can obtain data from other cells and provide data to other cells. As a result of this constant flow of data, a cell's state can quickly change. ANSYS Workbench provides a

visual indication of a cell's state at any given time via icons on the right side of each cell. Brief descriptions of the various states are provided below:

- **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 ANSYS 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 ANSYS 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 an ANSYS FLUENT mesh file, but the ANSYS 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 that an interactive calculation has been completed successfully. For example, after ANSYS 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 ANSYS FLUENT while it is iterating, ANSYS 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**.
- **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.



For more information about cell states, see [Understanding Cell States](#).

5. View the list of files generated by ANSYS Workbench.

ANSYS Workbench allows you to easily view the files associated with your project using the **Files** view. To open the **Files** view, select the **Files** option under the **View** menu at the top of the ANSYS Workbench window.

**View** → **Files**

**Figure 1.4 ANSYS Workbench Files View for the Project After Adding a FLUENT-Based Fluid Flow Analysis System**

Files						
	A	B	C	D	E	
1	Name	Cell ID	Size	Type	Date Modified	Location
2	 elbow-workbench.wbpj		123 KB	ANSYS Project File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01
3	 designPoint.wbdp		25 KB	Design Point File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo


In the **Files** view, you will be able to see the name and type of file, the ID of the cell that the file is associated with, the size of the file, the location of the file, and other information. For more information about the **Files** view, see [Files View](#).

From here, you will create the geometry described in [Figure 1.1 \(p. 2\)](#), and later create a mesh and set up a fluid flow analysis for the geometry.

### 1.4.3. Step 2: Creating the Geometry in ANSYS DesignModeler

For the geometry of your fluid flow analysis, you can create a geometry in ANSYS DesignModeler, or import the appropriate geometry file. In this step, you will create the geometry in ANSYS DesignModeler, then review the list of files generated by ANSYS Workbench.

#### Important

Note the **Attention Required** icon () within the **Geometry** cell for the system. This indicates that the cell requires data (for example, a geometry). Once the geometry is defined, the state of the cell will change accordingly. Likewise, the state of some of the remaining cells in the system will change.

#### Note

If you would rather not create the geometry in ANSYS DesignModeler, you can import a pre-existing geometry by right-clicking the **Geometry** cell and selecting the **Import Geometry** option from the context menu. From there, you can browse your file system to locate the `elbow_geometry.agdb` geometry file that is provided for this tutorial. If you do not have access to ANSYS DesignModeler, you can use the `elbow_geometry.stp` file instead.

To learn how to create a mesh from the geometry you imported, go to [Step 3: Meshing the Geometry in the ANSYS Meshing Application \(p. 19\)](#).

1. Start ANSYS DesignModeler.

In the ANSYS Workbench **Project Schematic**, double-click the **Geometry** cell in the `elbow` fluid flow analysis system. This displays the ANSYS DesignModeler application.

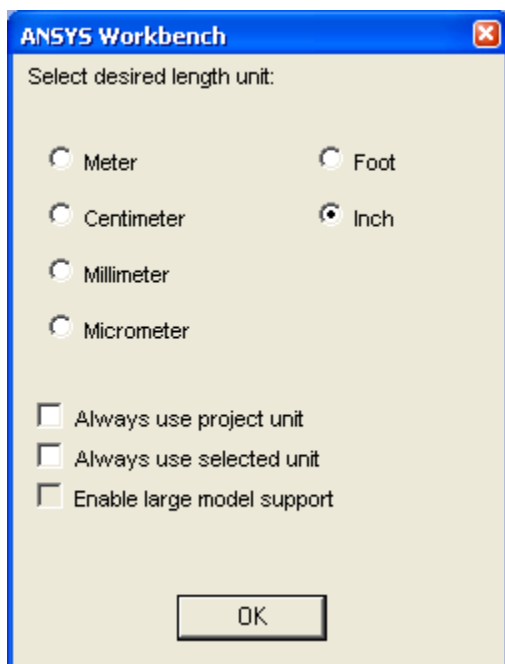
## Tip

You can also right-click the **Geometry** cell to display the context menu, then select **New Geometry...**

2. Set the units in ANSYS DesignModeler.


When ANSYS DesignModeler first appears, you are prompted to select the desired system of length units to work from. For the purposes of this tutorial (where you will create the geometry in inches and perform the CFD analysis using SI units), select **Inch** as the desired length unit and click **OK** to close the prompt.

**Figure 1.5 Setting the Units in ANSYS DesignModeler**

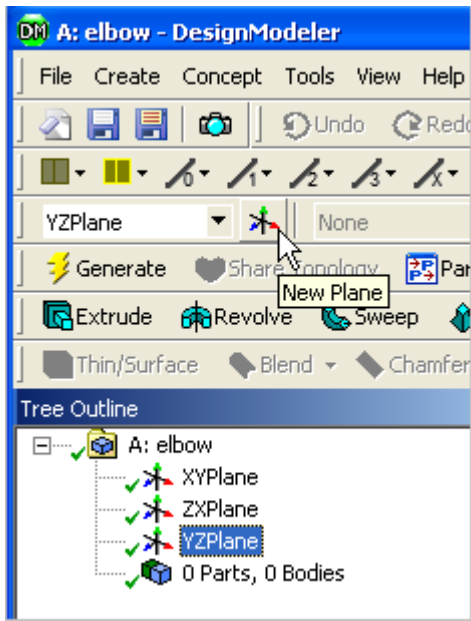


3. Create the geometry.

The geometry for this tutorial (*Figure 1.1 (p. 2)*) consists of a large curved pipe accompanied by a smaller side pipe. To create the larger main pipe, you will use the **Sweep** operation. Sweeping requires the use of two sketches; one that defines the profile to be swept and the other that defines the path through which the profile is to be swept. In this case, the profile is a half circle because the symmetry of the problem means that you do not have to generate the entire pipe geometry.

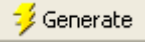

- a. Set up the plane:
  - i. Create a new plane by selecting **YZPlane** from the **Tree Outline**, then clicking the **New Plane** icon  in the **Active Plane/Sketch** toolbar near the top of the ANSYS Workbench window. Selecting **YZPlane** before you create the new plane ensures that the new plane is based on the **YZPlane**.

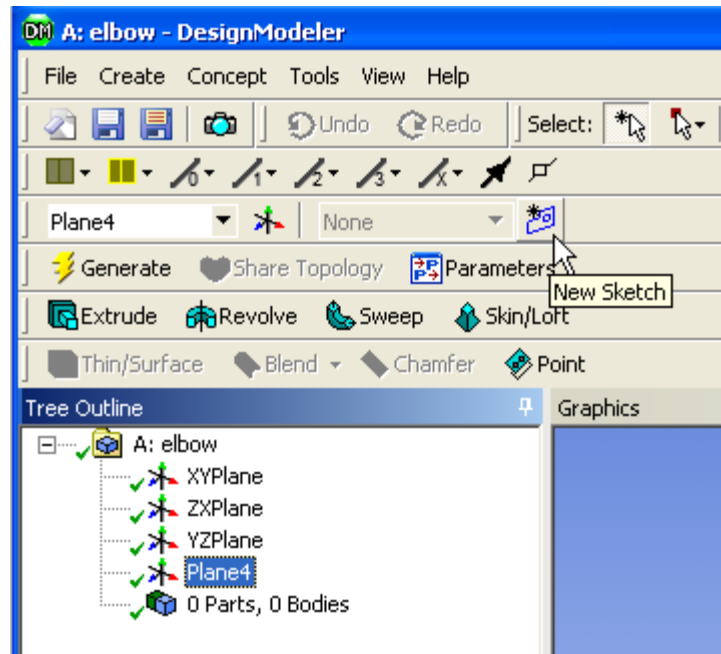




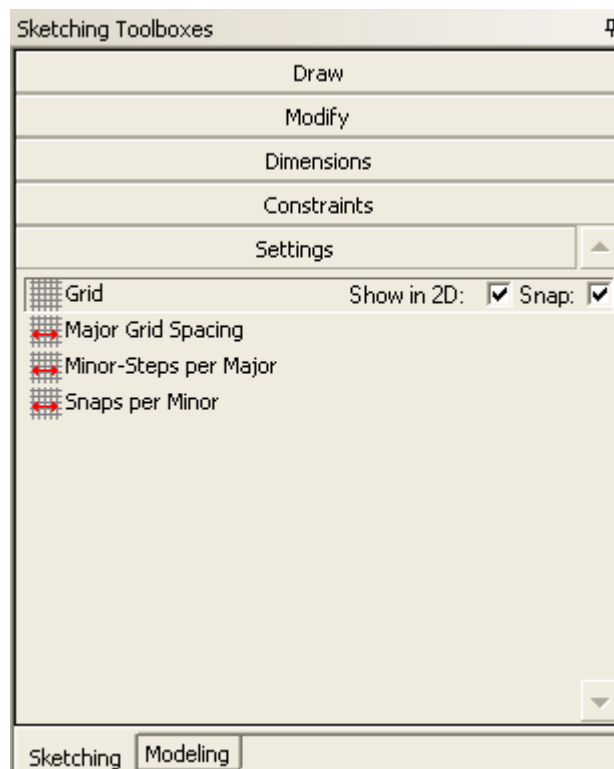
- ii. In the **Details View** for the new plane (**Plane4**), set **Transform1 (RMB)** to **Offset Global X** by clicking the word **None** to the right of **Transform1 (RMB)** and selecting **Offset Global X** from the drop-down menu. Set the **Value** of the offset to **-8** in by clicking the 0 to the right of **FD1, Value 1** and entering **-8**. Note that the **inches** measurement appears automatically when you click outside of this box.

Details View	
<b>Details of Plane4</b>	
Plane	Plane4
Type	From Plane
Base Plane	YZPlane
Transform 1 (RMB)	Offset Global X
<b>FD1, Value 1</b>	<b>-8 in</b>
Transform 2 (RMB)	None
Reverse Normal/Z-Axis?	No
Flip XY-Axes?	No
Export Coordinate System?	No

- iii. To create the plane, click the **Generate** button  which is located in the ANSYS DesignModeler toolbar.
- iv. Create a new sketch by selecting **Plane4** from the **Tree Outline** and then clicking the **New Sketch** icon  in the **Active Plane/Sketch** toolbar, near the top of the ANSYS Workbench window. Selecting the plane first ensures that the new sketch is based on **Plane4**.



- b. Set up the grid:
- i. On the **Sketching** tab of the **Tree Outline**, open the **Settings** toolbox, select **Grid**, and enable the **Show in 2D** and the **Snap** options.



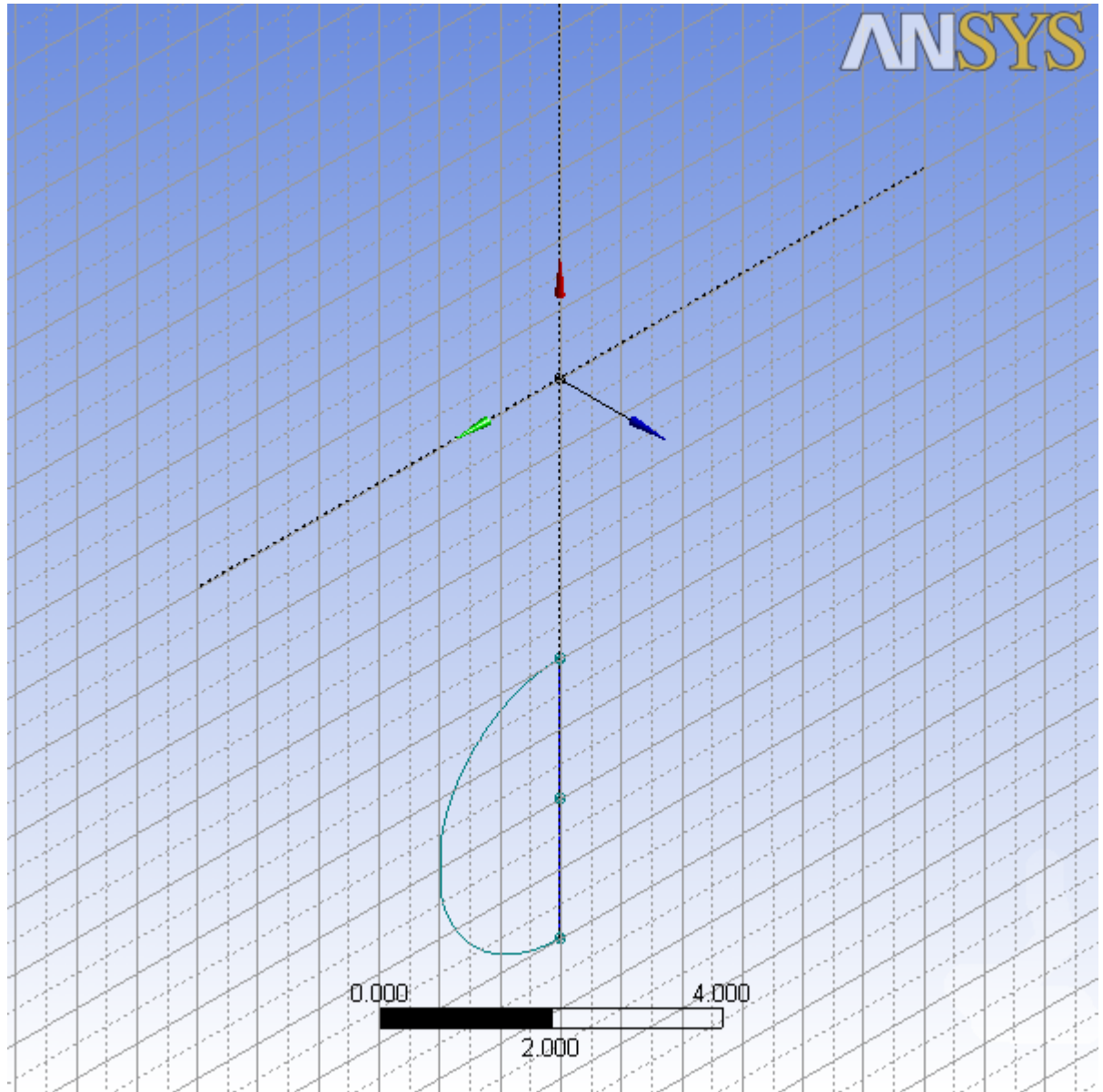
- ii. Set **Major Grid Spacing** to 1 inch and **Minor-Steps per Major** to 2.
- c. Create the profile:
- i. Zoom in on the center of the grid so that you can see the grid lines clearly. You can do this by holding down the right mouse button and dragging a box over the desired viewing area.

- ii. On the **Sketching** tab, open the **Draw** toolbox and select **Arc by Center** (you may need to use the arrows in the toolbox to scroll down to see the correct tool). You will be drawing an arc with a radius of 2 inches, centered on Y=- 6 inches, Z=0 inches (located below the origin of **Plane4**). The grid settings that you have just set up will help you to position the arc and set its radius correctly.

To create the arc, click the point where you want to create the center of the circle (in this case, Y=- 6 inches, Z=0 inches). Then click the point where you want to start drawing the arc (in this case Y=- 4 inches, Z=0 inches). At this stage, when you drag the unclicked mouse, DesignModeler will create an arc with a fixed radius. With the mouse button unclicked, drag through an arc of 180°. Click on Y=- 8 inches, Z=0 inches to stop the drawing process. Your drawing should now resemble [Figure 1.6 \(p. 13\)](#)

- iii. Next you will close the half-circle with a line that joins the two arc ends. On the **Sketching** tab, in the **Draw** toolbox, select **Line** (you may need to use the arrows in the toolbox to scroll up to see the correct tool). Draw a line from Y=- 4 inches, Z=0 inches to Y=- 8 inches, Z=0 inches. This closed half-circle will be used as the profile during the sweep.

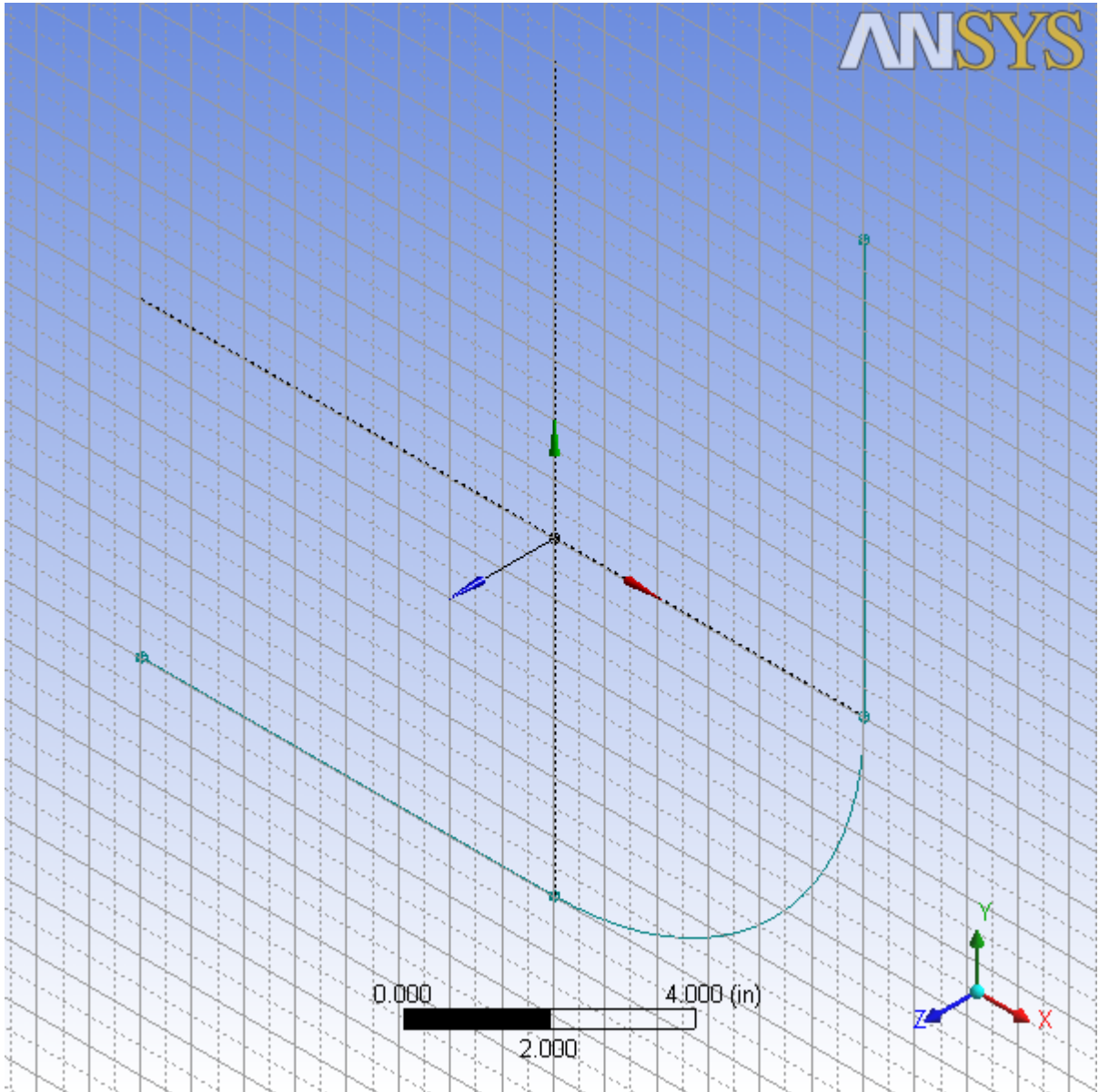
Figure 1.6 Creating the Arc Profile

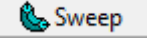


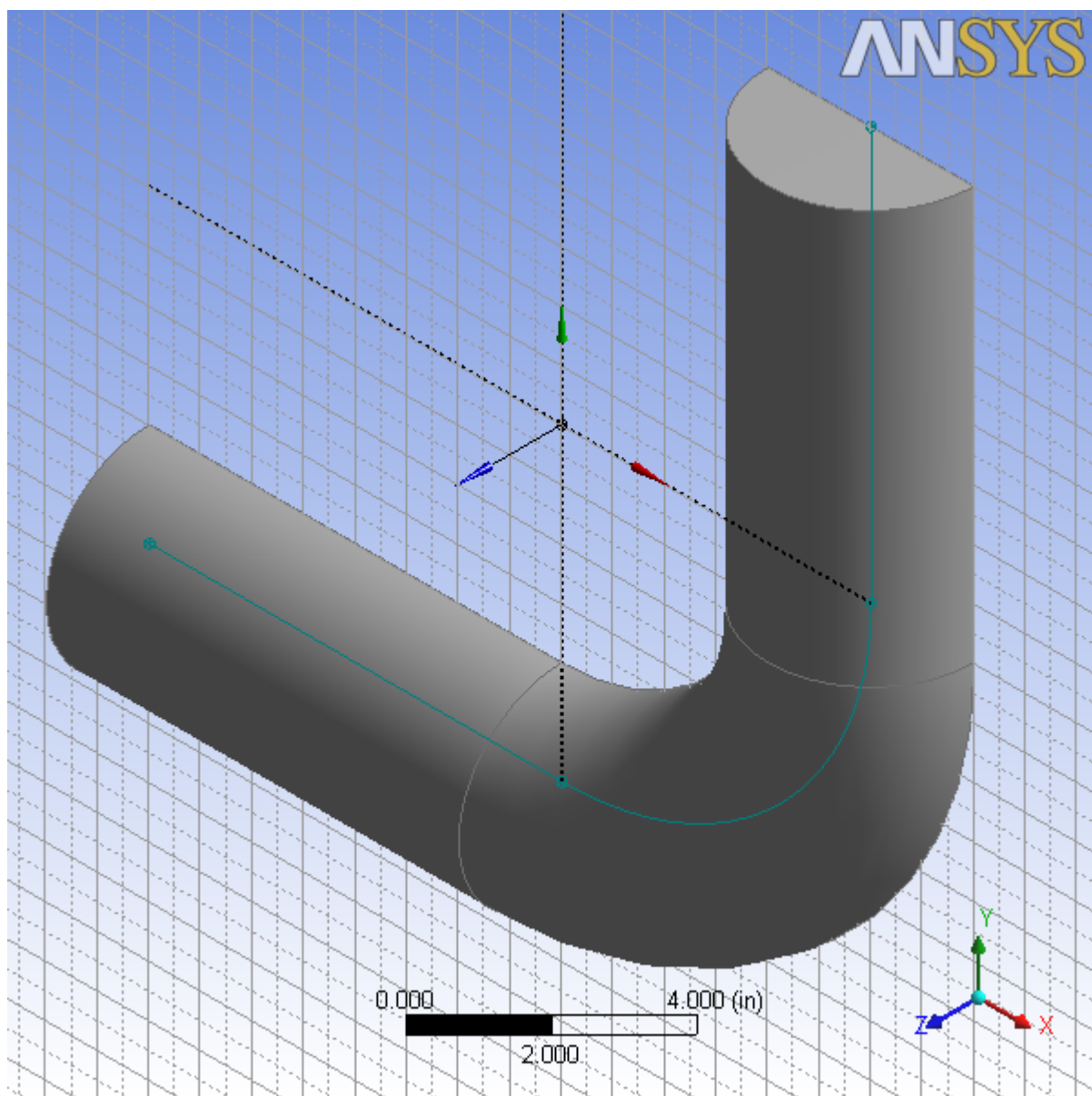
- d. Create the path:
- On the **Modeling** tab select **XYPlane**, then click the **New Sketch** icon and then the **Generate** button to create a new sketch based on the **XYPlane**.
  - On the **Sketching** tab, open the **Settings** toolbox, select **Grid**, and enable the **Show in 2D** and the **Snap** options.
  - Set **Major Grid Spacing** to 1 inch and **Minor-Steps per Major** to 2.
  - In the **Draw** toolbox, select **Line** to draw two straight lines on the sketch. For reference, the coordinates of the endpoints of the lines are  $(X = -8 \text{ inches}, Y = -6 \text{ inches})$ ,  $(X = 0 \text{ inches}, Y = -6 \text{ inches})$  for the horizontal line and  $(X = 6 \text{ inches}, Y = 0 \text{ inches})$ ,  $(X = 6 \text{ inches}, Y = 8 \text{ inches})$  for the vertical line.
  - In the **Draw** toolbox, select **Arc by Center** and click once on the origin (center of the arc). Now click once on the point  $X = 6 \text{ inches}, Y = 0 \text{ inches}$  to select one of the end points of the arc. With the mouse unclicked, move the mouse around to the other end point

( $X = 0$  inches,  $Y = -6$  inches) and click it to draw the quarter-circle. If the wrong part of the arc is drawn and it does not resemble *Figure 1.7* (p. 14), click **Undo** from the **Undo/Redo** toolbar and try again. Together, the arc and the two straight lines form a curving line that is the path through which the half-circle profile will be swept.

**Figure 1.7 Creating the Elbow Path**




- e. Create the pipe:
  - i. Select the **Sweep** button  from the **3D Features** toolbar.
  - ii. Set the **Profile** to be **Sketch1** by clicking **Sketch1** in the **Tree Outline** and then clicking **Apply** next to **Profile** in the **Details View** at the bottom-left of the screen.
  - iii. Set the **Path** to **Sketch2**: Click **Sketch2** in the **Tree Outline**, then click the **Not selected** text next to **Path** in the **Details View** and click **Apply**.
  - iv. Click the **Generate** button to create the pipe.




**Figure 1.8 Generating the Pipe**


- f. View and manipulate the image:

You can use the mouse buttons to change your view of this 3D image. The following table describes mouse actions that are available:

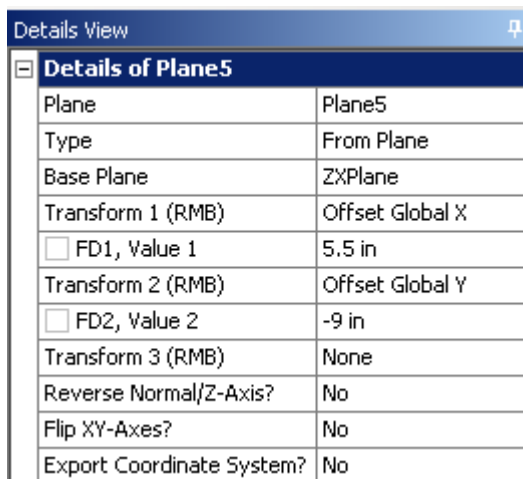
**Table 1.1 DesignModeler View Manipulation Instructions**

Action	Using Graphics Toolbar Buttons and the Mouse
Rotate view (vertical, horizontal)	After clicking the <b>Rotate</b> icon,  , press and hold the left mouse button and drag the mouse. Dragging side to side rotates the view about the vertical axis, and dragging up and down rotates the view about the horizontal axis.

Action	Using Graphics Toolbar Buttons and the Mouse
Translate or pan view	After clicking the <b>Pan</b> icon,  , press and hold the left mouse button and drag the object with the mouse until the view is satisfactory.
Zoom in and out of view	After clicking the <b>Zoom</b> icon,  , press and hold the left mouse button and drag the mouse up and down to zoom in and out of the view.
Box zoom	After clicking the <b>Box Zoom</b> icon,  , press and hold the left mouse button and drag the mouse diagonally across the screen. This action will cause a rectangle to appear in the display. When you release the mouse button, a new view will be displayed which consists entirely of the contents of the rectangle.

Clicking the **Zoom to Fit** icon, , will cause the object to fit exactly and be centered in the window.

- g. Create the side pipe:
  - i. Create a new plane based on the **ZXPlane**. As before, first make the **ZXPlane** active by clicking on it, then click the **New Plane** icon to create the plane based upon it.
  - ii. In the **Details View**, set **Transform 1 (RMB)** to **Offset Global X**, and set the **FD1, Value 1** of the offset to 5.5 inches.
  - iii. Set **Transform 2 (RMB)** to **Offset Global Y** and the **FD2, Value 2** of the offset to -9 inches.

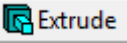


Details of Plane5	
Plane	Plane5
Type	From Plane
Base Plane	ZXPlane
Transform 1 (RMB)	Offset Global X
<input type="checkbox"/> FD1, Value 1	5.5 in
Transform 2 (RMB)	Offset Global Y
<input type="checkbox"/> FD2, Value 2	-9 in
Transform 3 (RMB)	None
Reverse Normal/Z-Axis?	No
Flip XY-Axes?	No
Export Coordinate System?	No

- iv. Click **Generate** to create the plane.
- v. With the new plane selected in the **Tree Outline**, click the **New Sketch** icon.
- vi. As with the main pipe, a profile will be created, but with the side pipe, the **Extrude** operation will be used to create the body of the pipe. On the **Sketching** tab, open the **Draw** toolbox and select **Arc by Center**. You will be drawing a 180° arc that is centered on the origin, with the two ends on the X axis. The radius for this arc will be adjusted later, so you can use any convenient radius for now. Click once on the origin, then once some distance along the X axis in either a positive or a negative direction. With the mouse button unclicked, draw an arc to the opposite side of the origin and click the X axis again.

- vii. To adjust the radius, open the **Dimensions** toolbox on the **Sketching** tab and select **Radius**. Click once on the curve of the arc you have just created then click once on the radius (anywhere between the arc and the origin). In the **Details View** under the **Dimensions: 1** section, set the radius, **R1** to 0.5 inches.

Details View	
[-] <b>Details of Sketch3</b>	
Sketch	Sketch3
Sketch Visibility	Show Sketch
Show Constraints?	No
[-] <b>Dimensions: 1</b>	
<input checked="" type="checkbox"/> R1	0.5 in
[-] <b>Edges: 1</b>	
Circular Arc	Cr16

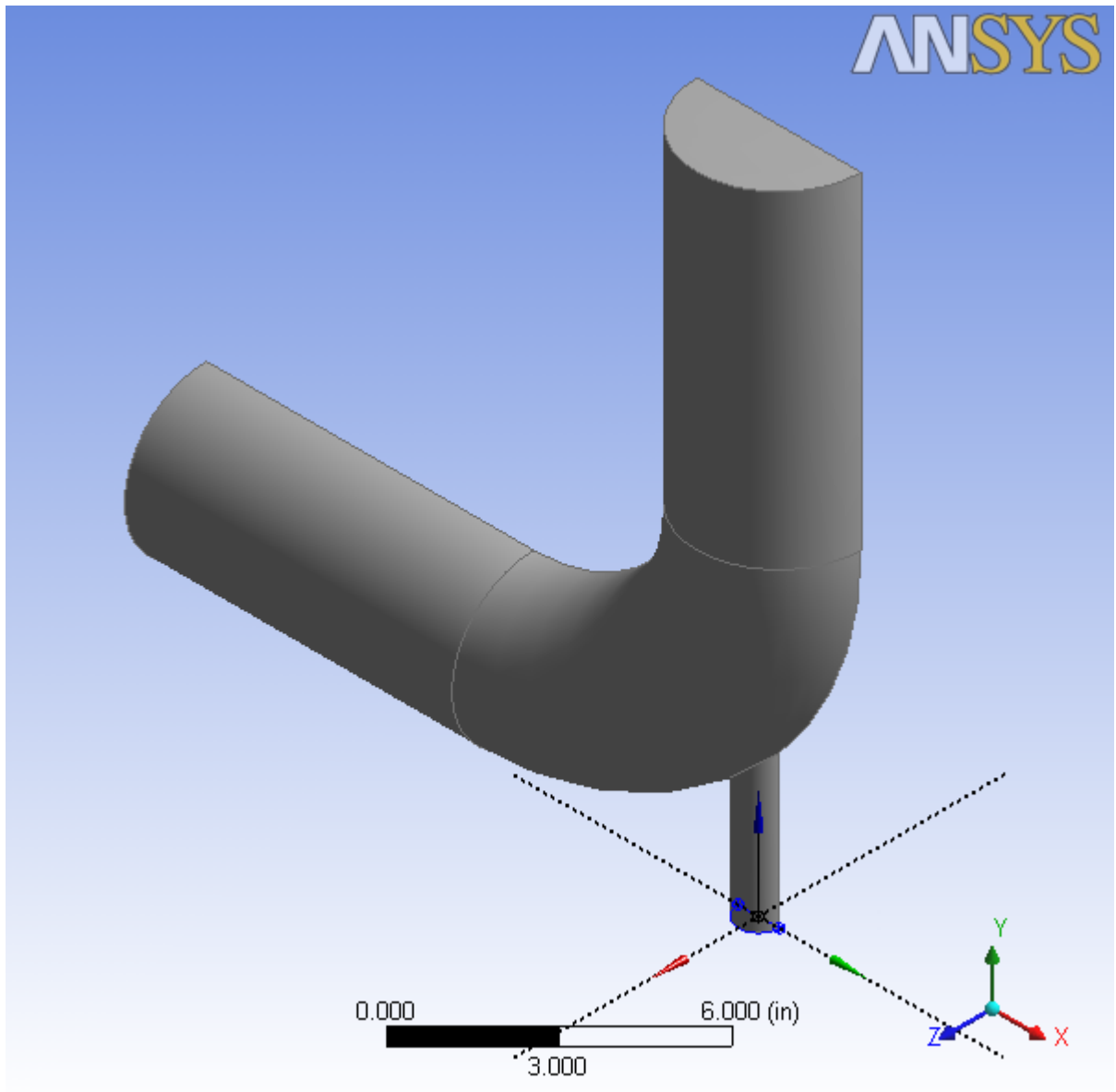
- viii. Next you will close the half-circle with a line that joins the two arc ends. On the **Sketching** tab, in the **Draw** toolbox, select **Line**. Draw a line connecting the open ends of the arc. This closed half-circle will be used as the profile for the extrude operation.
- ix. Click the **Extrude** button  from the **3D Features** toolbar.
- x. In the **Details View**, set **Geometry** to be the new sketch (**Sketch3**), and set **Operation** to **Add Material**.
- xi. Set **Direction** to **Normal** and **Extent Type** to **Fixed**. Set **FD1, Depth (>0)** to 4 inches.

Details View	
[-] <b>Details of Extrude1</b>	
Extrude	Extrude1
Geometry	Sketch3
Operation	Add Material
Direction Vector	None (Normal)
Direction	Normal
Extent Type	Fixed
<input checked="" type="checkbox"/> FD1, Depth (>0)	4 in
As Thin/Surface?	No
Merge Topology?	Yes
[-] <b>Geometry Selection: 1</b>	
Sketch	Sketch3

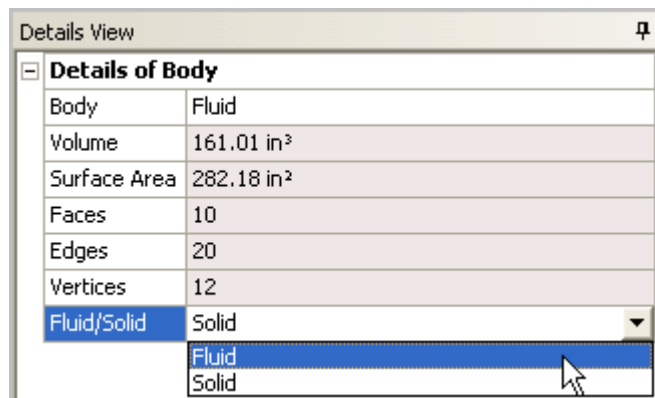
- xii. Click **Generate** to create the side pipe.



**Figure 1.9 Generating the Additional Pipe**



- h. Specify the geometry as a fluid body.
  - i. In the **Tree Outline**, open the **1 Part, 1 Body** branch and select **Solid**.
  - ii. In the **Details View** of the body, change the name of the **Body** from **Solid** to **Fluid**.
  - iii. In the **Fluid/Solid** section, select **Fluid**.



- iv. Click **Generate**.
4. Close ANSYS DesignModeler by selecting **File** → **Close DesignModeler** or by clicking the 'X' icon in the upper right-hand corner. ANSYS Workbench automatically saves the geometry and updates the **Project Schematic** accordingly. The question mark in the **Geometry** cell is replaced by a check mark, indicating that there is a geometry now associated with the fluid flow analysis system.
5. View the list of files generated by ANSYS Workbench by selecting **View** → **Files**.

**Figure 1.10 ANSYS Workbench Files View for the Project After Creating the Geometry**

Files						
	A	B	C	D	E	
1	Name	Cell ID	Size	Type	Date Modified	Location
2	elbow-workbench.wbpj		124 KB	ANSYS Project File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01
3	FFF.agdb	A2	2 MB	Geometry File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
4	designPoint.wbdp		26 KB	Design Point File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo

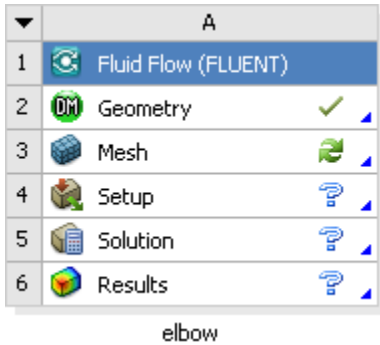
Note the addition of the geometry file (FFF.agdb, where FFF indicates a FLUENT-based fluid flow system) to the list of files. If you had imported the geometry file provided for this tutorial rather than creating the geometry yourself, the elbow\_geometry.agdb (or the elbow\_geometry.stp) file would be listed instead.

### 1.4.4. Step 3: Meshing the Geometry in the ANSYS Meshing Application

Now that you have created the mixing elbow geometry, you need to generate a computational mesh for throughout the flow volume. For this section of the tutorial, you will use the ANSYS Meshing application to create a mesh for your CFD analysis, then review the list of files generated by ANSYS Workbench.

## Important

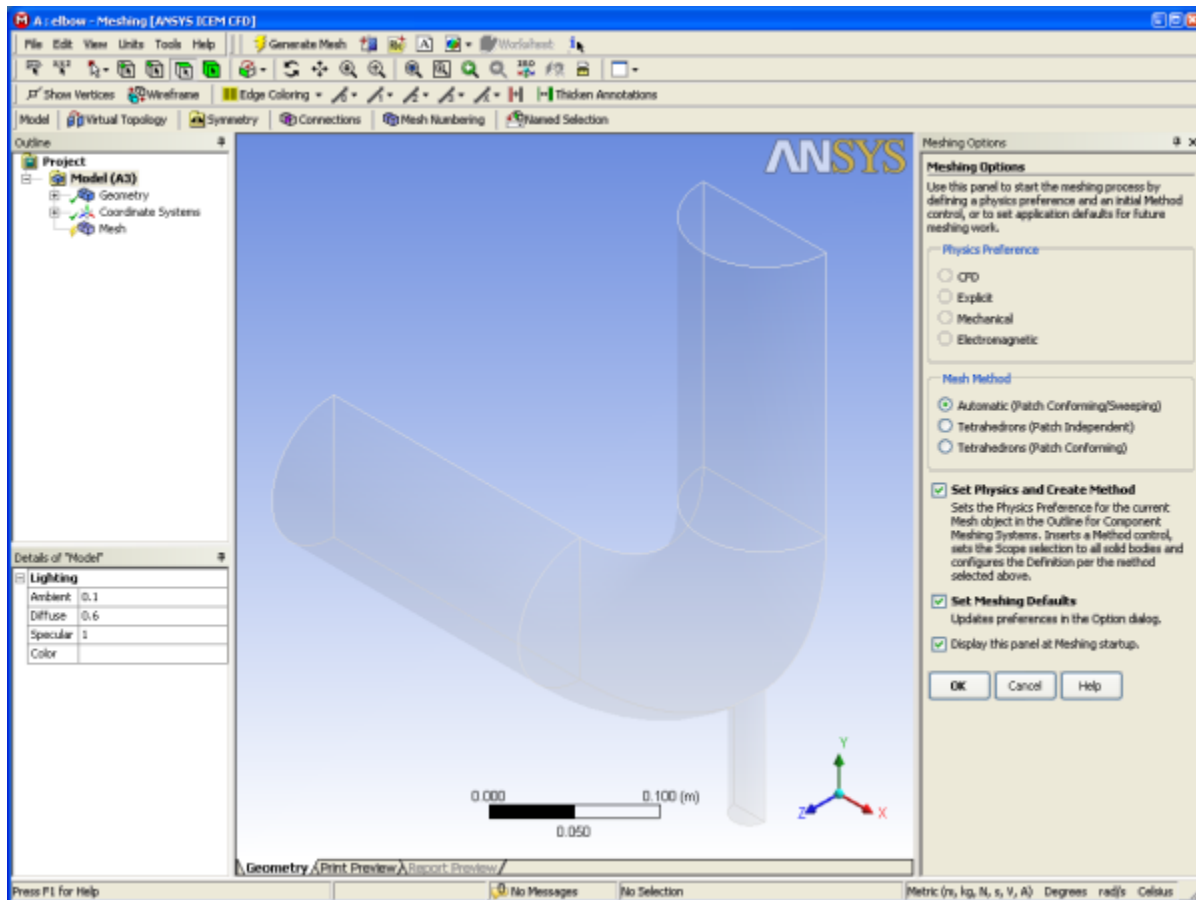
Note the Refresh Required icon (🔄) within the **Mesh** cell for the system. This indicates that the state of the cell requires a refresh and that upstream data has changed since the last refresh or update (such as an update to the geometry). Once the mesh is defined, the state of the **Mesh** cell will change accordingly, as will the state of the next cell in the system, in this case the **Setup** cell.



1. Open the ANSYS Meshing application.

In the ANSYS Workbench **Project Schematic**, double-click the **Mesh** cell in the `elbow` fluid flow analysis system (cell A3). This displays the ANSYS Meshing application with the elbow geometry already loaded. You can also right-click the **Mesh** cell to display the context menu where you can select the **Edit...** option.

**Figure 1.11 The ANSYS Meshing Application with the Elbow Geometry Loaded**



### Important

The first time you open the ANSYS Meshing application, the **Meshing Options** are displayed on the right-hand side of the application window. For this tutorial, no meshing options need to be set, so click **OK** to close the **Meshing Options** pane.

2. Create named selections for the geometry boundaries.


In order to simplify your work later on in ANSYS FLUENT, you should label each boundary in the geometry by creating named selections for the pipe inlets, the outlet, and the symmetry surface (the outer wall boundaries are automatically detected by ANSYS FLUENT).

- a. Select the large inlet in the geometry that is displayed in the ANSYS Meshing application.

### Tip

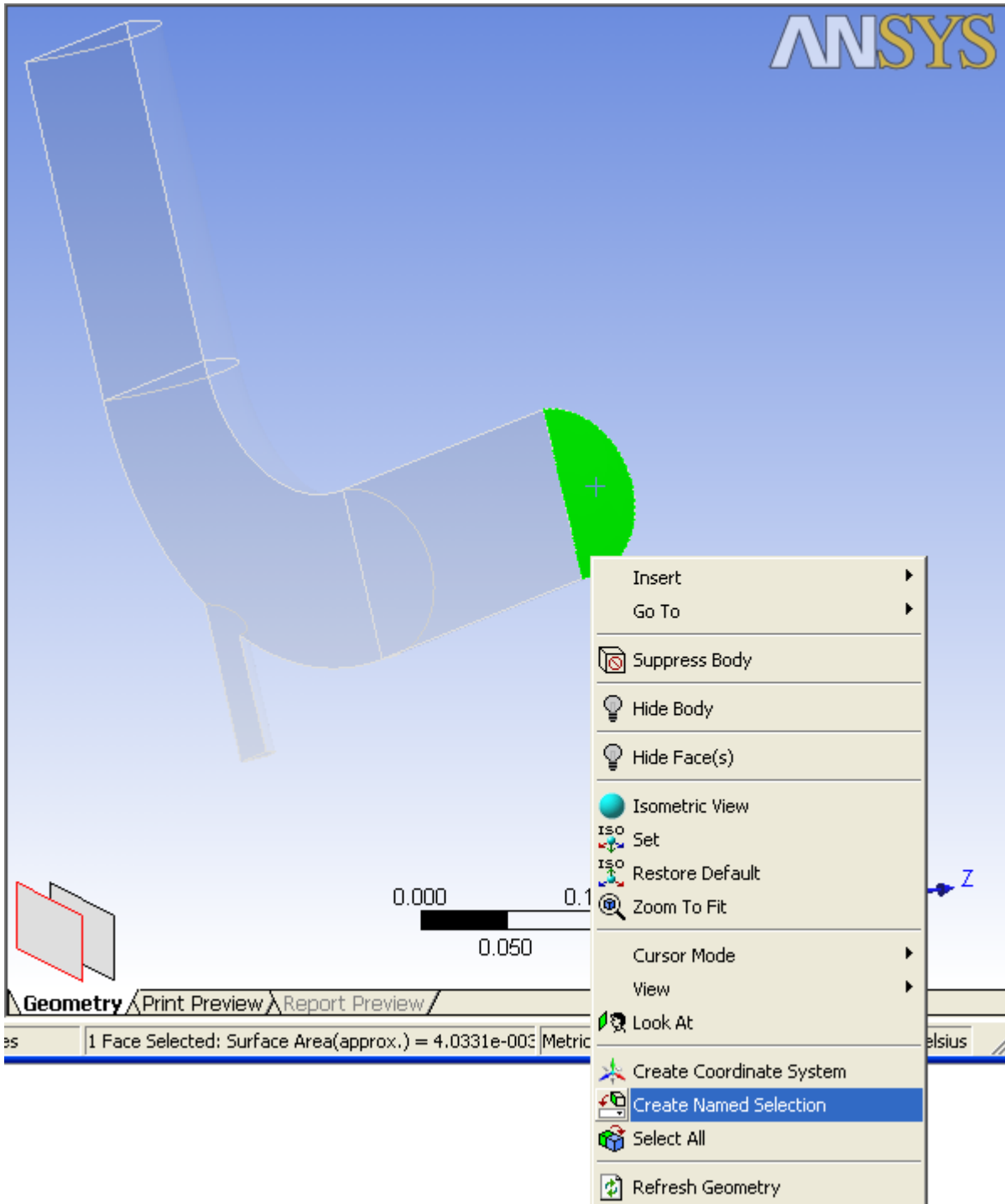
Use the Graphics Toolbar buttons and the mouse to manipulate the image until you can easily see the pipe openings and surfaces.

**Tip**

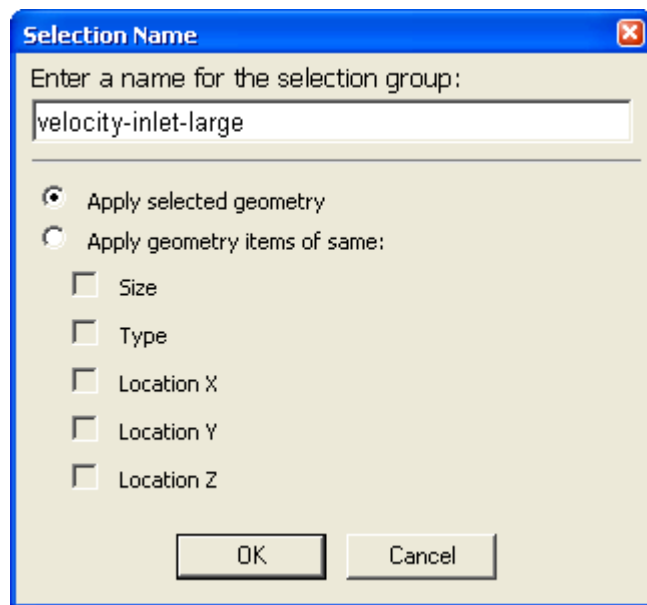
To select the inlet, the *Single select* (  ) mode must be active.

- b. Right-click and select the **Create Named Selection** option.

**Figure 1.12 Selecting a Face to Name**



This displays the **Selection Name** dialog box.

**Figure 1.13 Applying a Name to a Selected Face**

- c. In the **Selection Name** dialog box, enter `velocity-inlet-large` for the name and click **OK**.
- d. Perform the same operations for:
  - The small inlet (`velocity-inlet-small`)
  - The large outlet (`pressure-outlet`)
  - The symmetry planes (`symmetry`).

---

### Note

To select multiple planes, hold down the **Ctrl** key and select the individual planes in the graphics window.

---

### Important

It is important to note that by using the strings “velocity inlet” and “pressure outlet” in the named selections (with or without hyphens or underscore characters), ANSYS FLUENT automatically detects and assigns the corresponding boundary types accordingly.


3. Set some basic meshing parameters for the ANSYS Meshing application.

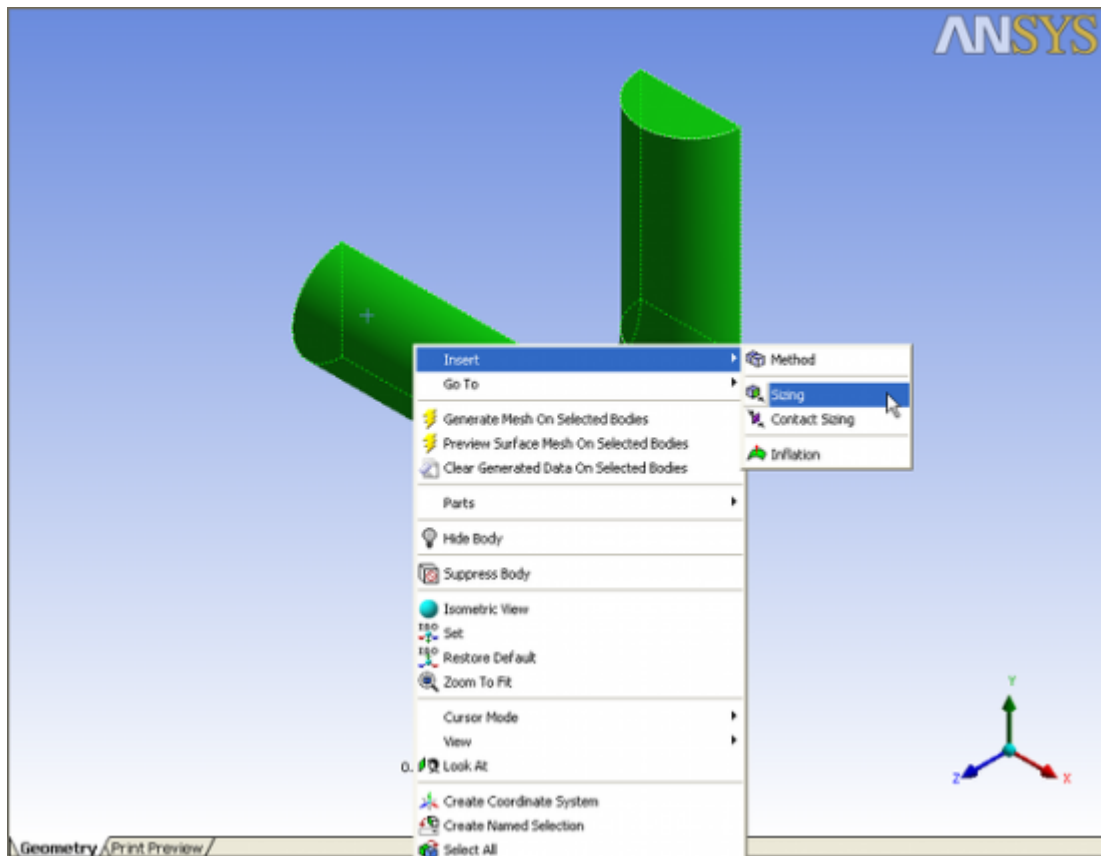
*For this analysis, you will adjust several meshing parameters to obtain a finer mesh.*

- a. In the **Outline** view, select **Mesh** under **Project/Model** to display the **Details of “Mesh”** view below the **Outline** view.

## Important

Note that because the ANSYS Meshing application automatically detects that you are going to perform a CFD fluid flow analysis using ANSYS FLUENT, the **Physics Preference** is already set to **CFD** and the **Solver Preference** is already set to **Fluent**.

- b. Expand the **Sizing** node by clicking the “+” sign to the left of the word **Sizing** to reveal additional sizing parameters.
  - i. Change **Relevance Center** to **Fine** by clicking the on the default value, **Coarse**, and selecting **Fine** from the drop down list.
  - ii. Change **Smoothing** to **High**
  - iii. Change **Curvature Normal Angle** to **10°**.
- c. Add a **Body Sizing** control.
  - i. With **Mesh** still selected in the **Outline** tree, change the selection filter to **Body** in the **Graphics Toolbar** (  ).
  - ii. Click the elbow in the graphics display to select it.
  - iii. Right click in the graphics area and select **Insert** → **Sizing** from the context menu.



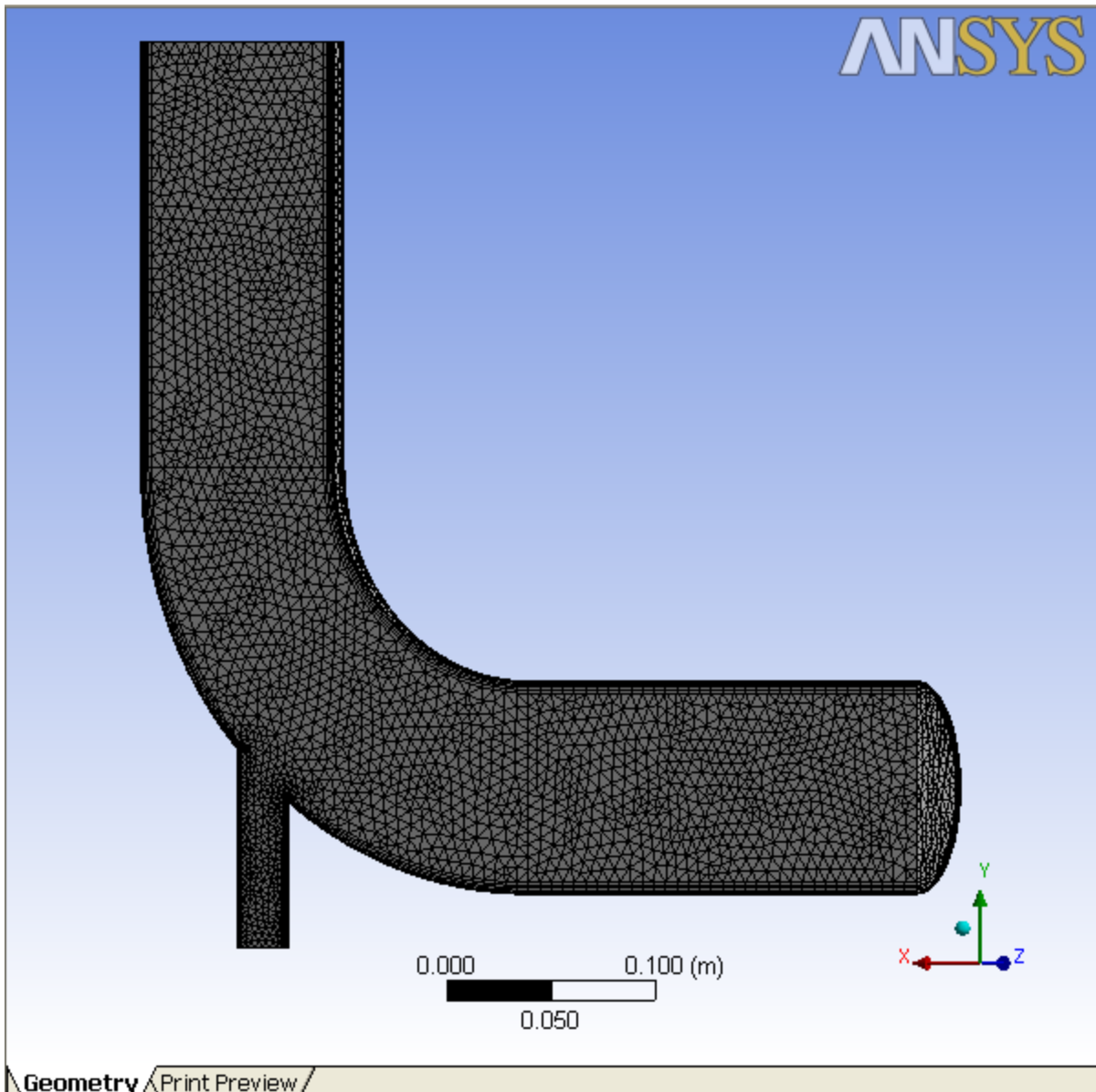
A new **Body Sizing** entry appears under **Mesh** in the project **Outline** tree

- iv. Click the new **Body Sizing** control in the **Outline** tree.
- v. Enter  $6e-3$  for **Element Size**.

- d. Click again on **Mesh** in the **Outline** view and expand the **Inflation** node to reveal additional inflation parameters. Change **Use Automatic Inflation** to **Program Controlled**.
4. Generate the mesh.

Right-click **Mesh** in the project **Outline** tree, and select **Update** in the context menu.

**Figure 1.14 The Computational Mesh for the Elbow Geometry in the ANSYS Meshing Application**



### Important

Using the **Generate Mesh** option creates the mesh, but does not actually create the relevant mesh files for the project and is optional if you already know that the mesh is acceptable. Using the **Update** option automatically generates the mesh, creates the relevant mesh files for your project, and updates the ANSYS Workbench cell that references this mesh.



**Note**

Once the mesh is generated, you can view the mesh statistics by opening the **Statistics** node in the **Details of “Mesh”** view. This will display information such as the number of nodes and the number of elements.






5. Close the ANSYS Meshing application.

You can close the ANSYS Meshing application without saving it because ANSYS Workbench automatically saves the mesh and updates the **Project Schematic** accordingly. The **Refresh Required** icon in the **Mesh** cell has been replaced by a check mark, indicating that there is a mesh now associated with the fluid flow analysis system.

6. View the list of files generated by ANSYS Workbench.

**View** → **Files**

**Figure 1.15 ANSYS Workbench Files View for the Project After Mesh Creation**

Files						
	A	B	C	D	E	
1	Name	Cell ID	Size	Type	Date Modified	Location
2	 elbow-workbench.wbpj		122 KB	ANSYS Project File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01
3	 FFF.agdb	A2	1 MB	Geometry File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
4	 FFF.msh	A3	8 MB	Fluent Mesh File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
5	 FFF.mshdb	A3	2 MB	Mesh Database Files	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
6	 designPoint.wbdp		25 KB	Design Point File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo

Note the addition of the mesh files (FFF.msh and FFF.mshdb) to the list of files. The FFF.msh file is created when you update the mesh, and the FFF.mshdb file is generated when you close the ANSYS Meshing application.

### 1.4.5. Step 4: Setting Up the CFD Simulation in ANSYS FLUENT

Now that you have created a computational mesh for the elbow geometry, in this step you will set up a CFD analysis using ANSYS FLUENT, then review the list of files generated by ANSYS Workbench.

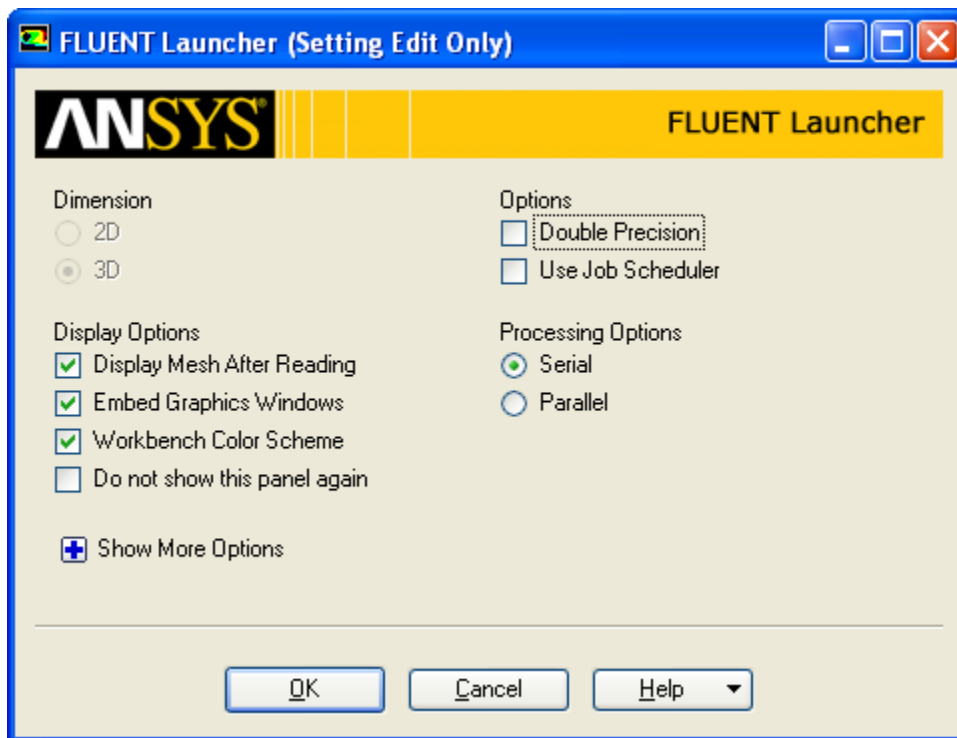
1. Start ANSYS FLUENT.

In the ANSYS Workbench **Project Schematic**, double-click the **Setup** cell in the elbow fluid flow analysis system. You can also right-click the **Setup** cell to display the context menu where you can select the **Edit...** option.

When ANSYS FLUENT is first started, the FLUENT Launcher is displayed, enabling you to view and/or set certain ANSYS FLUENT start-up options.

**Note**

The FLUENT Launcher allows you to decide which version of ANSYS FLUENT you will use, based on your geometry and on your processing capabilities.

**Figure 1.16 FLUENT Launcher**

- a. Ensure that the proper options are enabled.

**Important**

Note that the **Dimension** setting is already filled in and cannot be changed, since ANSYS FLUENT automatically sets it based on the mesh or geometry for the current system.

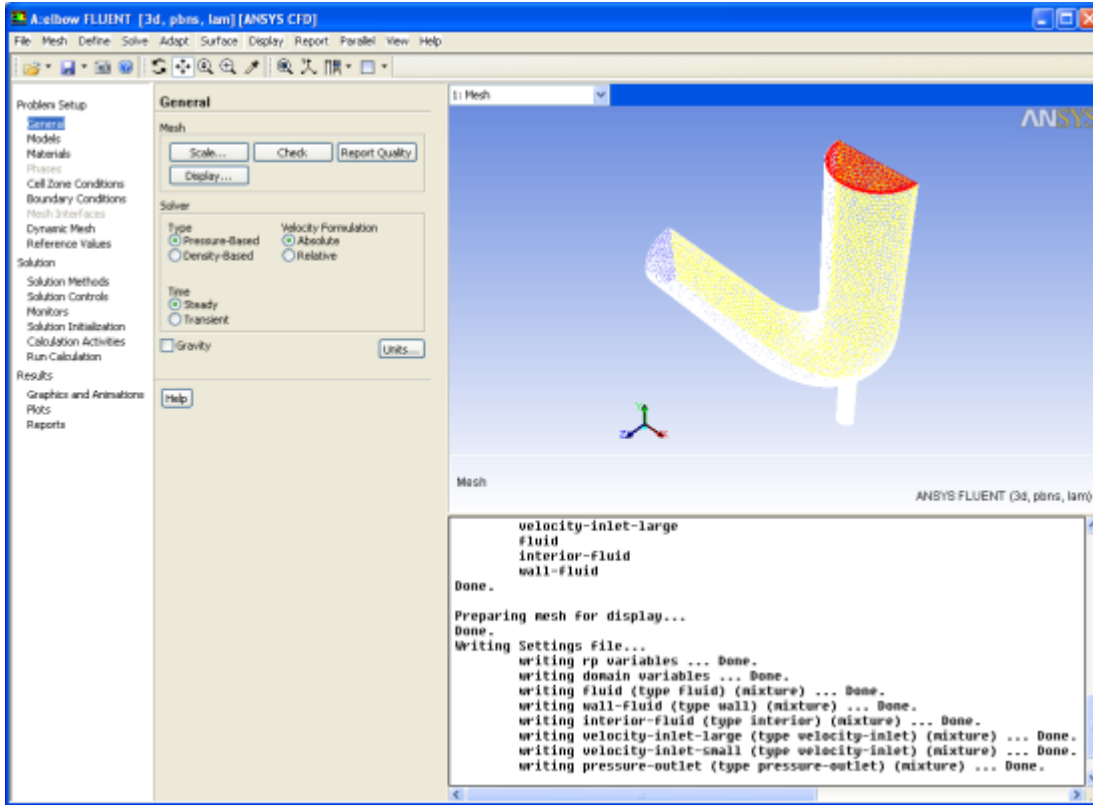
- i. Make sure that **Serial** from the **Processing Options** list is enabled.
- ii. Make sure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.

**Note**

An option is enabled when there is a check mark in the check box, and disabled when the check box is empty. To change an option from disabled to enabled (or vice versa), click the check box or the text.

- iii. Make sure that the **Double Precision** option is disabled.
- b. Click **OK** to launch ANSYS FLUENT.

**Figure 1.17 The ANSYS FLUENT Application**



**Note**

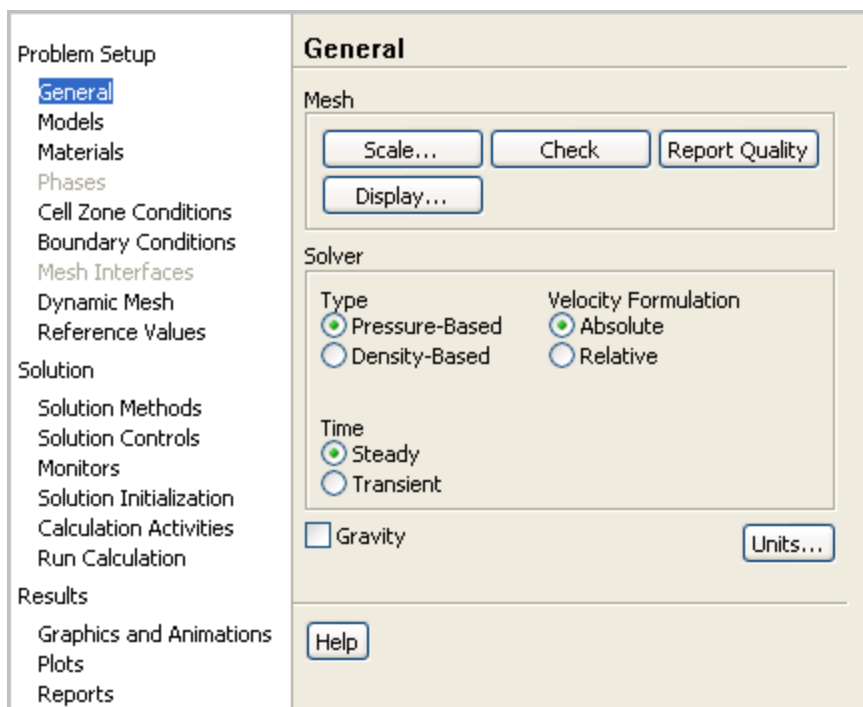
The mesh is automatically loaded and displayed in the graphics window by default.

2. Set some general settings for the CFD analysis.

**Note**

Select **General** in the navigation pane to perform the mesh-related activities and to choose a solver.





- a. Change the units for length.

Because you want to specify and view values based on a unit of length in inches from within ANSYS FLUENT, change the units of length within ANSYS FLUENT from meters (the default) to inches.

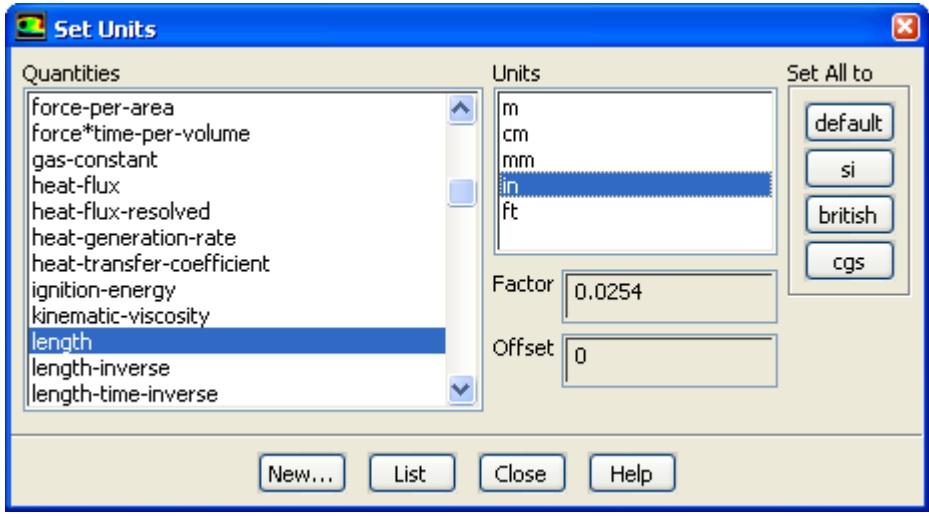
---

### Important

Note that the ANSYS Meshing application automatically converts and exports meshes for ANSYS FLUENT using meters (m) as the unit of length regardless of what units were used to create them. This is so you do not have to scale the mesh in ANSYS FLUENT under ANSYS Workbench.

◆ **General** → **Units...**

This displays the **Set Units** dialog box.



- i. Select **length** in the **Quantities** list.
- ii. Select **in** in the **Units** list.
- iii. Close the dialog box.

---

**Note**

Now, all subsequent inputs that require a value based on a unit of length can be specified in inches rather than meters.

- b. Check the mesh.

 **General** → **Check**

---

**Note**

ANSYS FLUENT will report the results of the mesh check in the console.

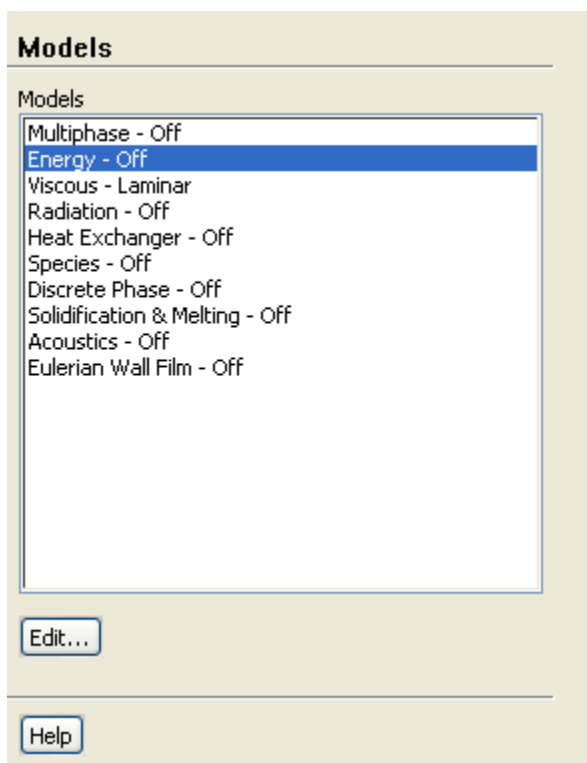
```
Domain Extents:
x-coordinate: min (m) = -2.032000e-01, max (m) = 2.032000e-01
y-coordinate: min (m) = -2.286000e-01, max (m) = 2.032000e-01
z-coordinate: min (m) = -2.332952e-18, max (m) = 5.080000e-02
Volume statistics:
minimum volume (m3): 1.148430e-10
maximum volume (m3): 5.741104e-08
total volume (m3): 2.633922e-03
Face area statistics:
minimum face area (m2): 2.147325e-07
maximum face area (m2): 3.444069e-05
Checking mesh.....
Done.
```

## Note

The minimum and maximum values may vary slightly when running on different platforms. The mesh check will list the minimum and maximum x and y values from the mesh in the default SI unit of meters. It will also report a number of other mesh features that are checked. Any errors in the mesh will be reported at this time. Ensure that the minimum volume is not negative as ANSYS FLUENT cannot begin a calculation when this is the case.

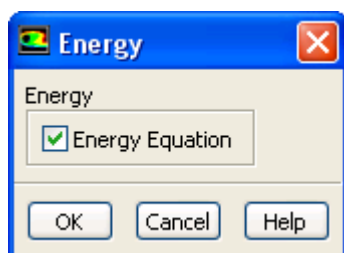
3. Set up your models for the CFD simulation.

## Models



- a. Enable heat transfer by activating the energy equation.

Models → Energy → Edit...

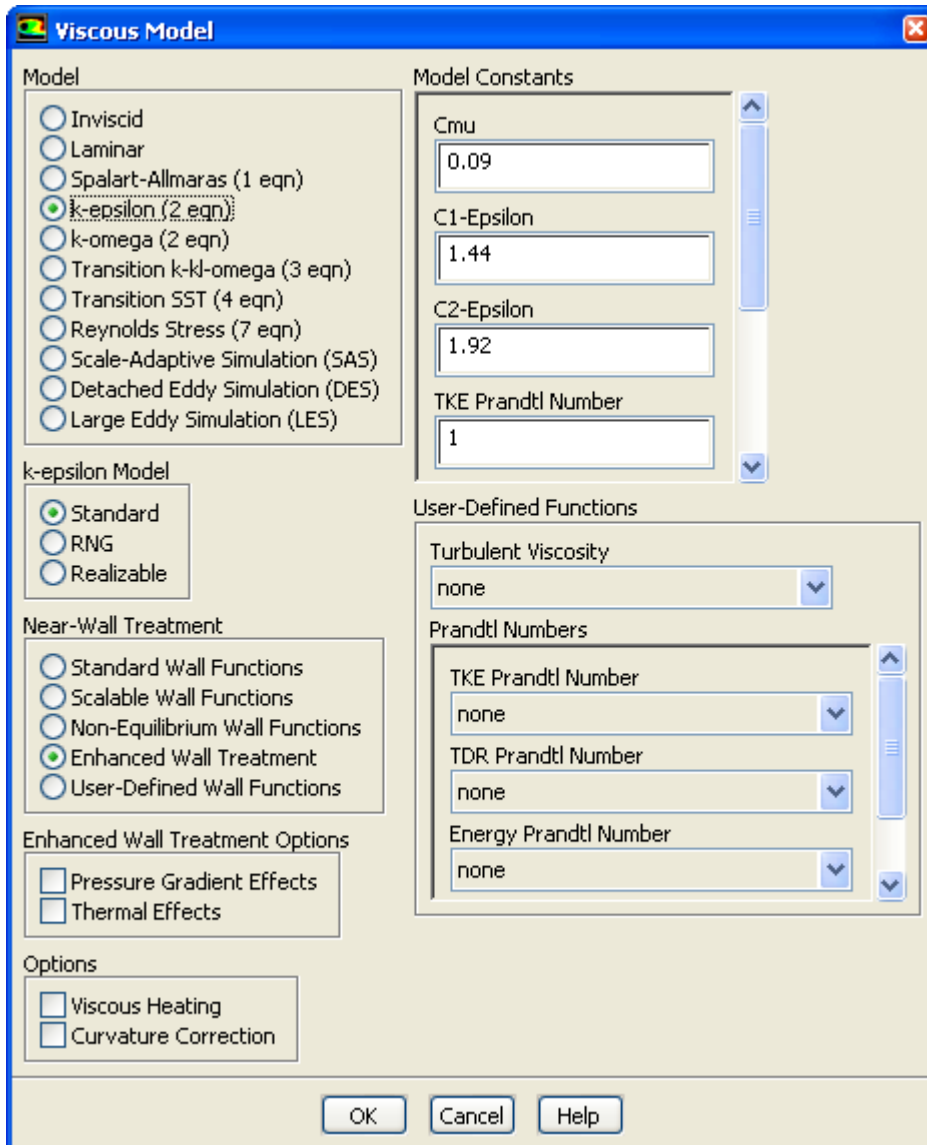


**Note**

You can also double-click a list item in order to open the corresponding dialog box.

- i. Enable the **Energy Equation** option.
  - ii. Click **OK** to close the **Energy** dialog box.
- b. Enable the  $k$ - $\epsilon$  turbulence model.

 **Models** →  **Viscous** → **Edit...**



- i. Select **k-epsilon** from the **Model** list.

**Note**

The **Viscous Model** dialog box will expand.

- ii. Use the default **Standard** from the **k-epsilon Model** list.
- iii. Select **Enhanced Wall Treatment** for the **Near-Wall Treatment**.

---

### Note

The default Standard Wall Functions are generally applicable if the first cell center adjacent to the wall has a  $y^+$  larger than 30. In contrast, the Enhanced Wall Treatment option provides consistent solutions for all  $y^+$  values. Enhanced Wall Treatment is recommended when using the k-epsilon model for general single-phase fluid flow problems. For more information about Near Wall Treatments in the k-epsilon model please refer to [Setting Up the k- \$\epsilon\$  Model](#) in the [User's Guide](#).

- iv. Click **OK** to accept the model and close the **Viscous Model** dialog box.
4. Set up your materials for the CFD simulation.

### Materials



- a. Create a new material called **water** using the **Create/Edit Materials** dialog box ([Figure 1.18 \(p. 34\)](#)).

 **Materials** →  **Fluid** → **Create/Edit...**

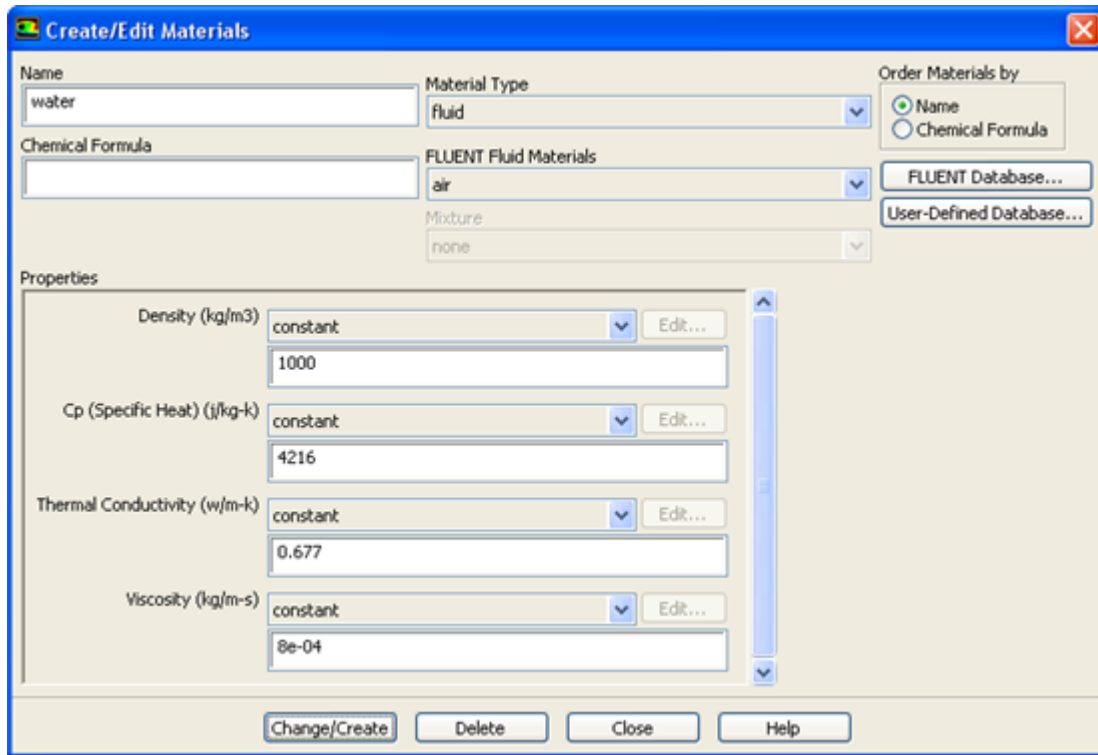
- i. Type **water** for **Name**.



- ii. Enter the following values in the **Properties** group box:

Property	Value
Density	$1000 \text{ kg/m}^3$
$c_p$ (Specific Heat)	$4216 \text{ J/kg} - K$
Thermal Conductivity	$0.677 \text{ W/m} - K$
Viscosity	$8e-04 \text{ kg/m} - s$

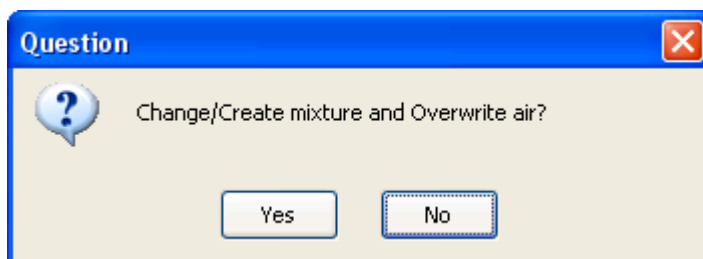
**Figure 1.18 The Create/Edit Materials Dialog Box**



- iii. Click **Change/Create**.

**Note**

A **Question** dialog box will open, asking if you want to overwrite air. Click **No** so that the new material **water** is added to the **Fluent Fluid Materials** list of materials that originally contained only **air**.



---

### Extra

You could have copied the material **water-liquid (h2o < I >)** from the materials database (accessed by clicking the **ANSYS FLUENT Database...** button). If the properties in the database are different from those you want to use, you can edit the values in the **Properties** group box in the **Create/Edit Materials** dialog box and click **Change/Create** to update your local copy. The original copy will not be affected.

- iv. Make sure that there are now two materials (water and air) defined locally by examining the **Fluent Fluid Materials** drop-down list.

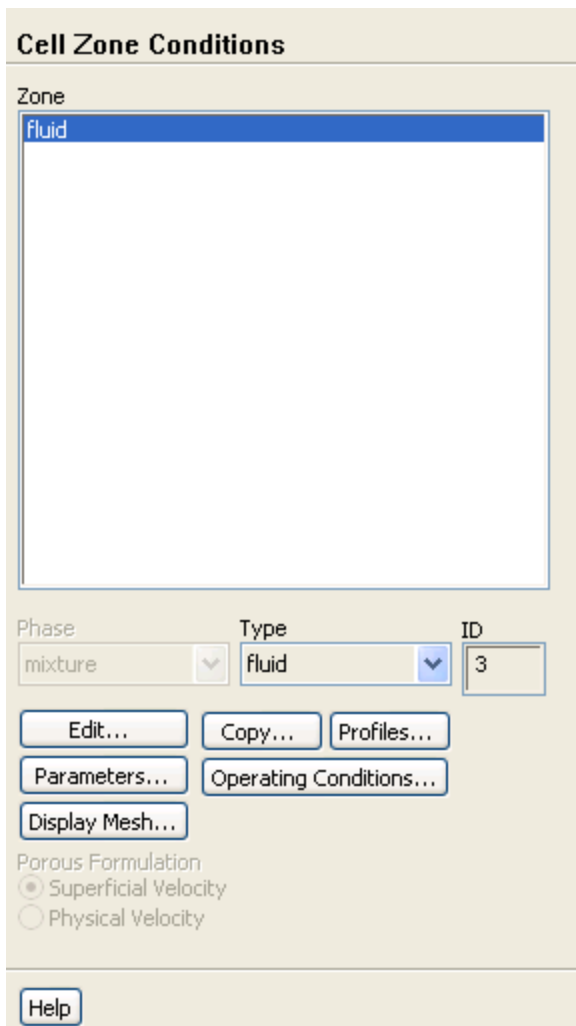
---

### Note

Both the materials will also be listed under **Fluid** in the **Materials** task page.

- v. Close the **Create/Edit Materials** dialog box.
5. Set up the cell zone conditions for the CFD simulation.

### Cell Zone Conditions

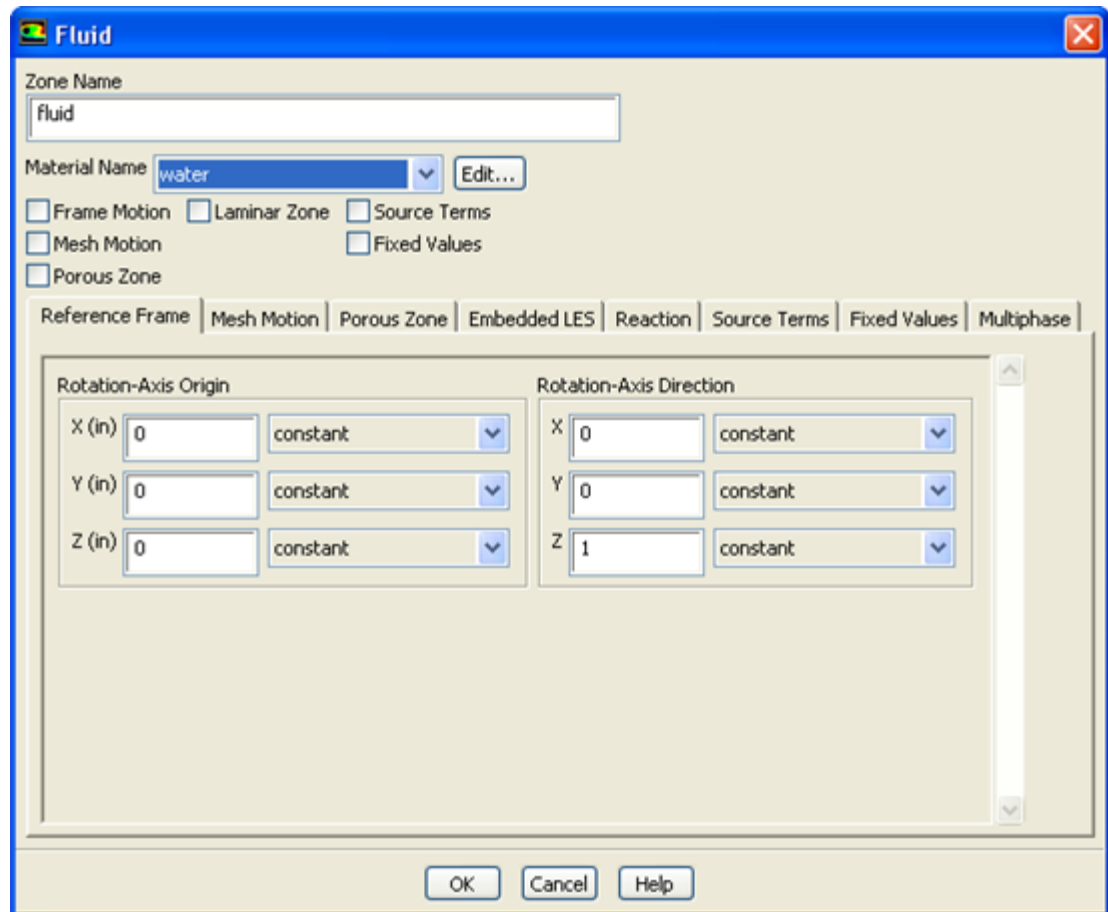


- a. Set the cell zone conditions for the fluid zone.
  - i. Select **fluid** in the **Zone** list in the **Cell Zone Conditions** task page, then click **Edit...** to open the **Fluid** dialog box.

---

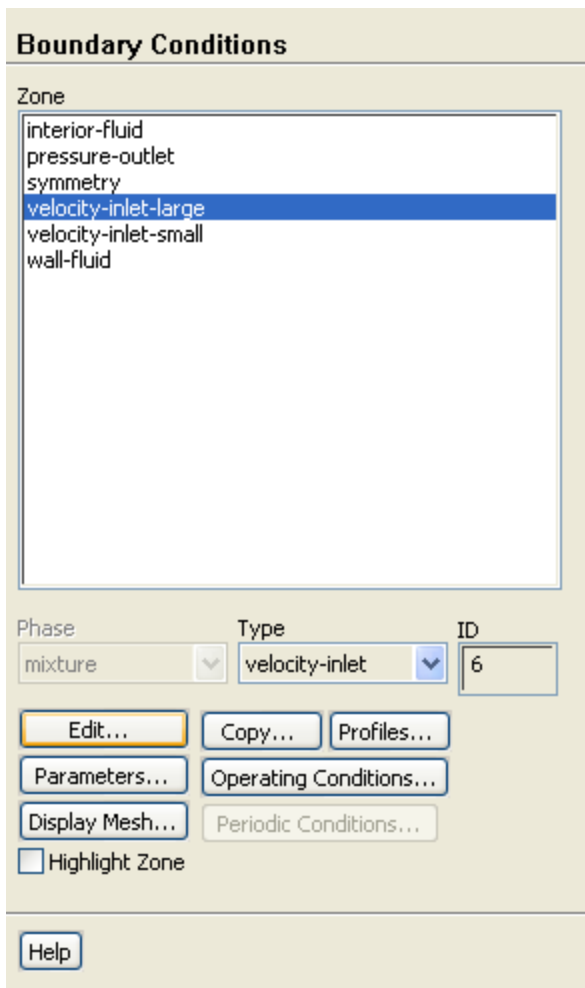
### Note

You can also double-click a list item in order to open the corresponding dialog box.



- ii. In the **Fluid** dialog box, select **water** from the **Material Name** drop-down list.
  - iii. Click **OK** to close the **Fluid** dialog box.
6. Set up the boundary conditions for the CFD analysis.

## **Boundary Conditions**



- a. Set the boundary conditions at the cold inlet (**velocity-inlet-large**).


 **Boundary Conditions** →  **velocity-inlet-large** → **Edit...**

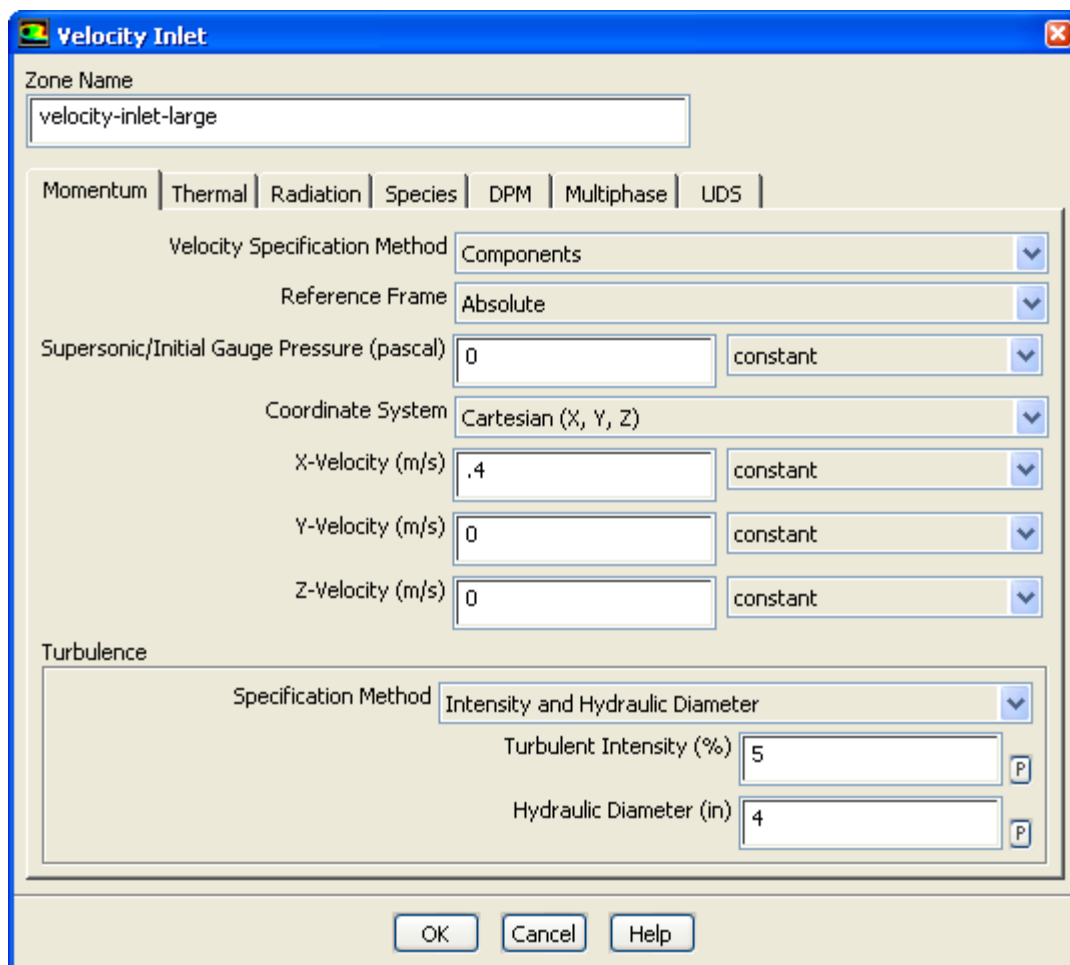
This opens the **Velocity Inlet** dialog box.

---

### Tip

If you are unsure of which inlet zone corresponds to the cold inlet, you can use the mouse to probe for mesh information in the graphics window. If you click the right mouse button with the pointer on any node in the mesh, information about the associated zone will be displayed in the ANSYS FLUENT console, including the name of the zone. The zone you probed will be automatically selected from the **Zone** selection list in the **Boundary Conditions** task page.

Alternatively, you can click the probe button () in the graphics toolbar and click the left mouse button on any node. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly. The information will be displayed in the console.



- i. Select **Components** from the **Velocity Specification Method** drop-down list.

---

### Note

The **Velocity Inlet** dialog box will expand.

- ii. Enter  $0.4 \text{ m/s}$  for **X-Velocity**.
- iii. Retain the default value of  $0 \text{ m/s}$  for both **Y-Velocity** and **Z-Velocity**.
- iv. Select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list in the **Turbulence** group box.
- v. Enter  $5 \%$  for **Turbulent Intensity**.
- vi. Enter  $4 \text{ inches}$  for **Hydraulic Diameter**.

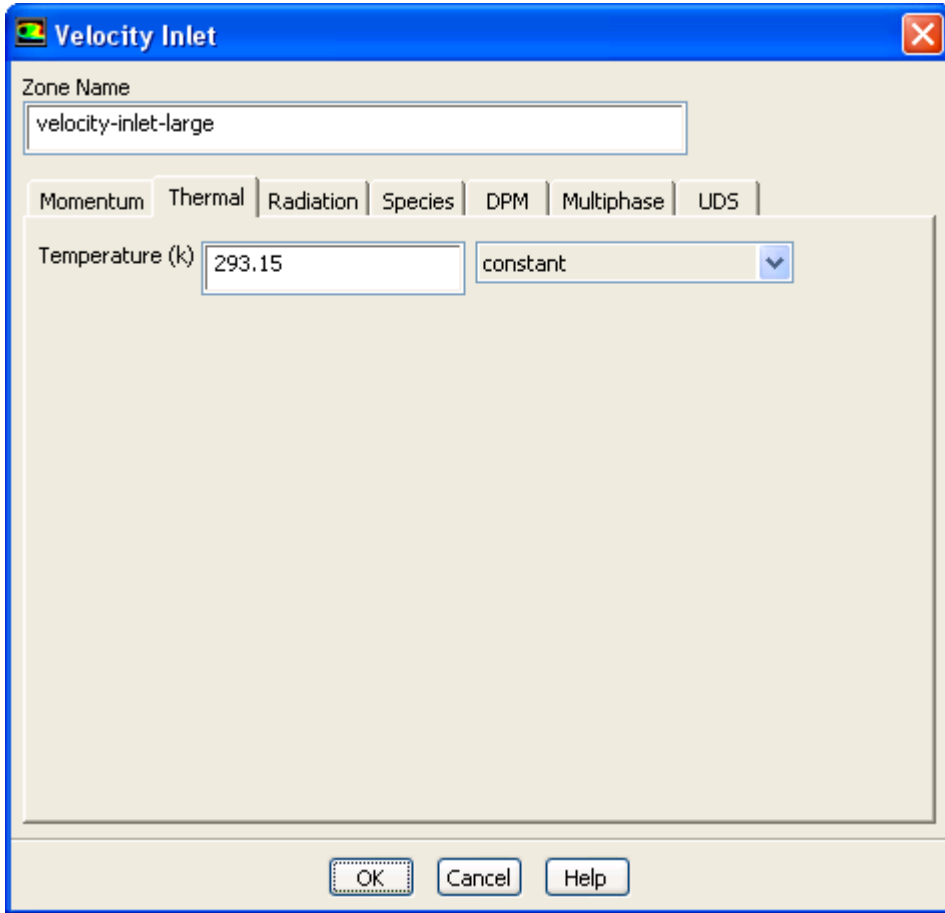
**Note**

The hydraulic diameter  $D_h$  is defined as:

$$D_h = \frac{4A}{P_w}$$

where  $A$  is the cross-sectional area and  $P_w$  is the wetted perimeter.

- vii. Click the **Thermal** tab.



- viii. Enter 293.15 K for **Temperature**.

- ix. Click **OK** to close the **Velocity Inlet** dialog box.

- b. In a similar manner, set the boundary conditions at the hot inlet (**velocity-inlet-small**), using the values in the following table:

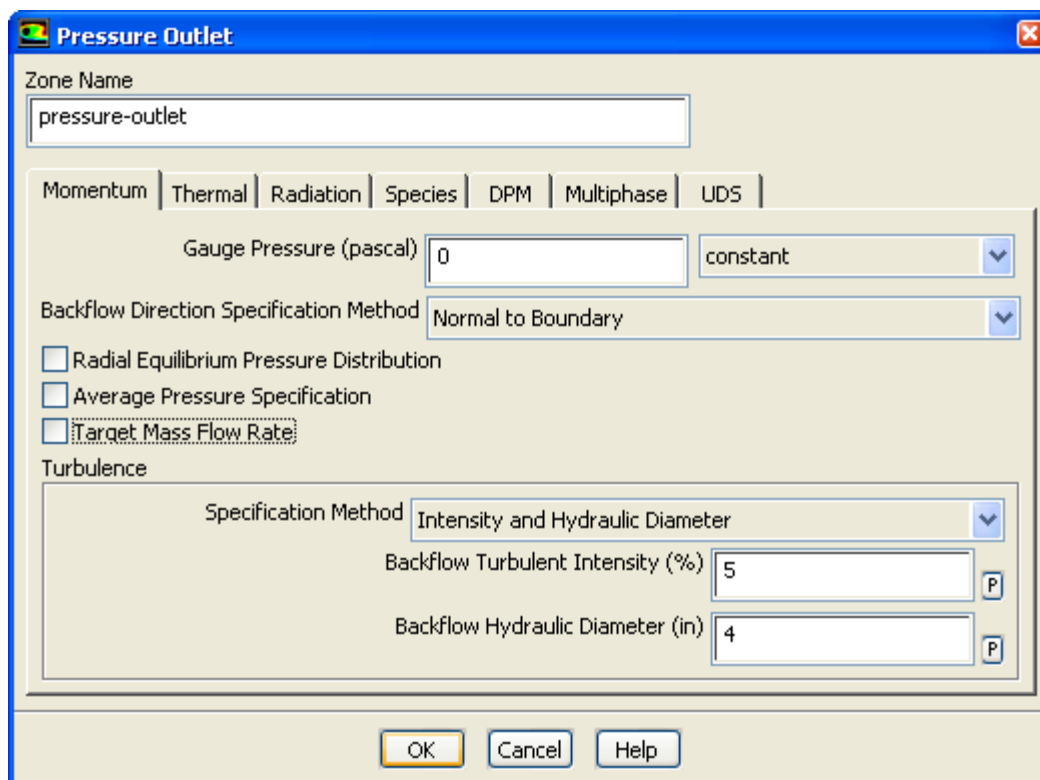
Boundary Conditions → velocity-inlet-small → Edit...

Velocity Specification Method	Components
X-Velocity	0 m/s
Y-Velocity	1.2 m/s

<b>Z-Velocity</b>	0 m/s
<b>Specification Method</b>	<b>Intensity &amp; Hydraulic Diameter</b>
<b>Turbulent Intensity</b>	5%
<b>Hydraulic Diameter</b>	1 in
<b>Temperature</b>	313.15 K

- c. Set the boundary conditions at the outlet (**pressure-outlet**), as shown in the **Pressure Outlet** dialog box.

◆ **Boundary Conditions** →  **pressure-outlet** → **Edit...**



### Note

ANSYS FLUENT will use the backflow conditions only if the fluid is flowing into the computational domain through the outlet. Since backflow might occur at some point during the solution procedure, you should set reasonable backflow conditions to prevent convergence from being adversely affected.

7. Set up solution parameters for the CFD simulation.

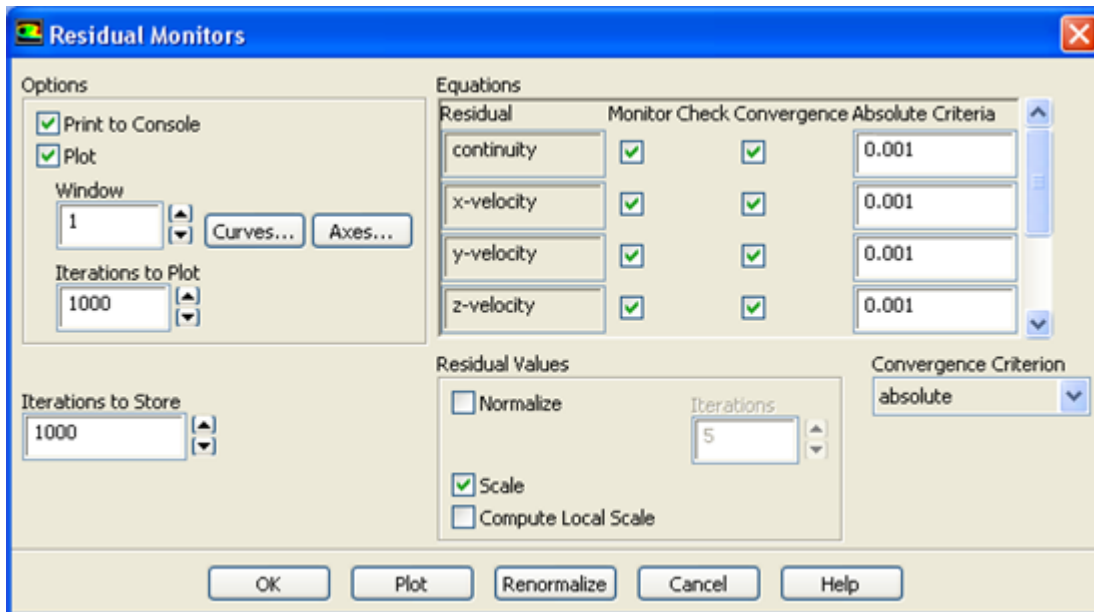
### Note

In the steps that follow, you will set up and run the calculation using the task pages listed under the **Solution** heading in the navigation pane.



- a. Examine the convergence criteria for the equation residuals.

 **Monitors** →  **Residuals** → **Edit...**



- i. Make sure that **Plot** is enabled in the **Options** group box.
- ii. Keep the default values for the **Absolute Criteria** of the **Residuals**, as shown in the **Residual Monitors** dialog box.
- iii. Click **OK** to close the **Residual Monitors** dialog box.

---

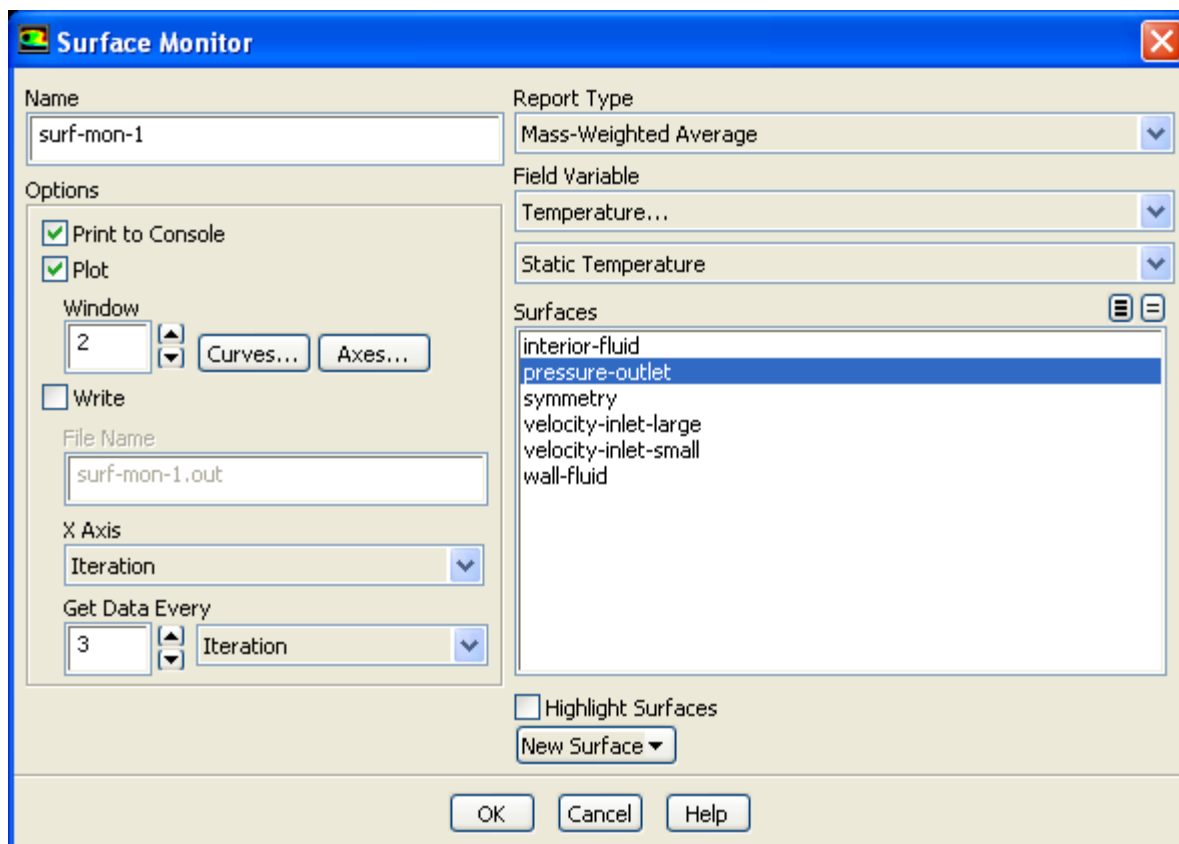
### Note

By default, all variables will be monitored and checked by ANSYS FLUENT as a means to determine the convergence of the solution.

- b. Create a surface monitor at the outlet (**pressure-outlet**)

*It is good practice to monitor physical solution quantities in addition to equation residuals when assessing convergence.*

 **Monitors** → **Create... (Surface Monitors)**



- i. Retain the default entry of **surf-mon-1** for the **Name** of the surface monitor.
- ii. Enable the **Plot** option for **surf-mon-1**.
- iii. Set **Get Data Every** to 3 by clicking the up-arrow button.

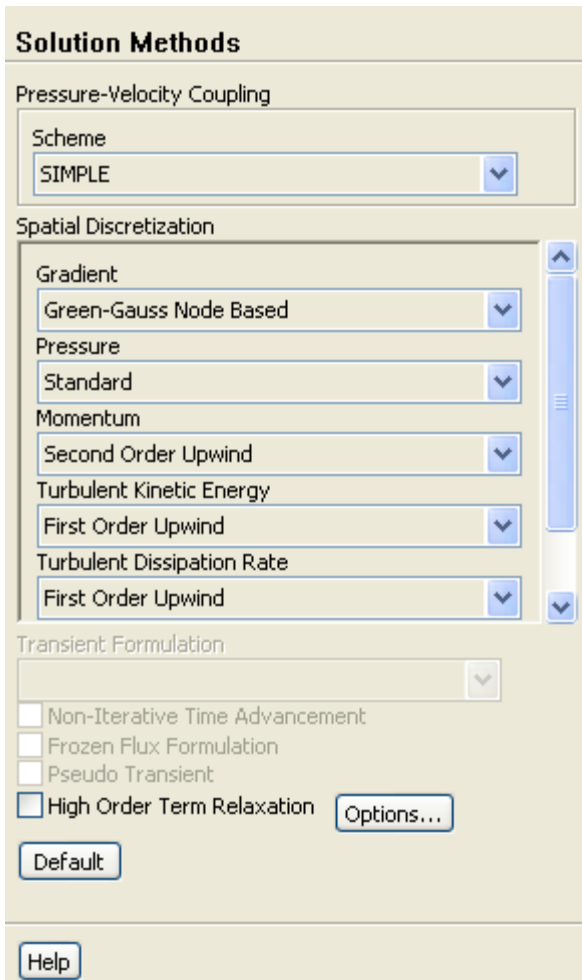
*This setting instructs ANSYS FLUENT to update the plot of the surface monitor and write data to a file after every 3 iterations during the solution.*

- iv. Select **Mass-Weighted Average** from the **Report Type** drop-down list.
- v. Select **Temperature...** and **Static Temperature** from the **Field Variable** drop-down lists.
- vi. Select **pressure-outlet** from the **Surfaces** selection list.
- vii. Click **OK** to save the surface monitor settings and close the **Surface Monitor** dialog box.

*The name and report type of the surface monitor you created will be displayed in the **Surface Monitors** selection list in the **Monitors** task page.*

- c. Change the Gradient method.

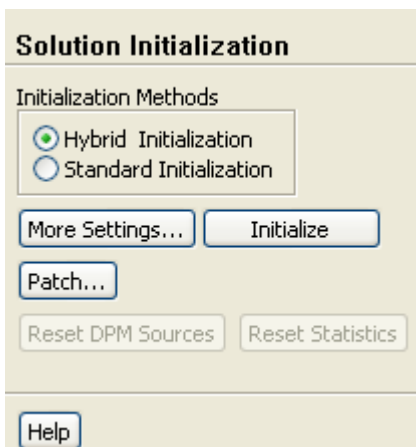
## Solution Methods



In the **Spatial Discretization** section of the **Solution Methods** pane, change the **Gradient** to **Green-Gauss Node Based**. This gradient method is suggested for tetrahedral meshes.

- d. Initialize the flow field.

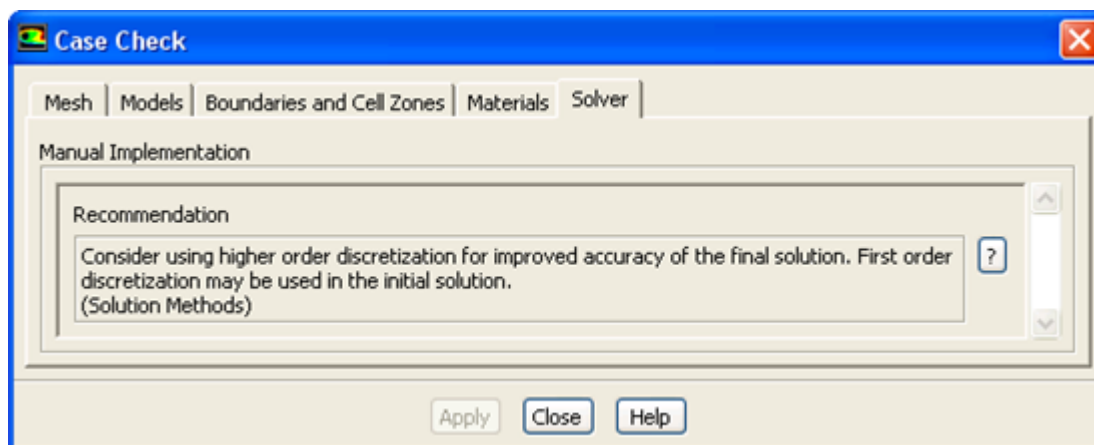
### Solution Initialization



- i. Keep the default of **Hybrid Initialization** from the **Initialization Methods** group box.
- ii. Click **Initialize**.

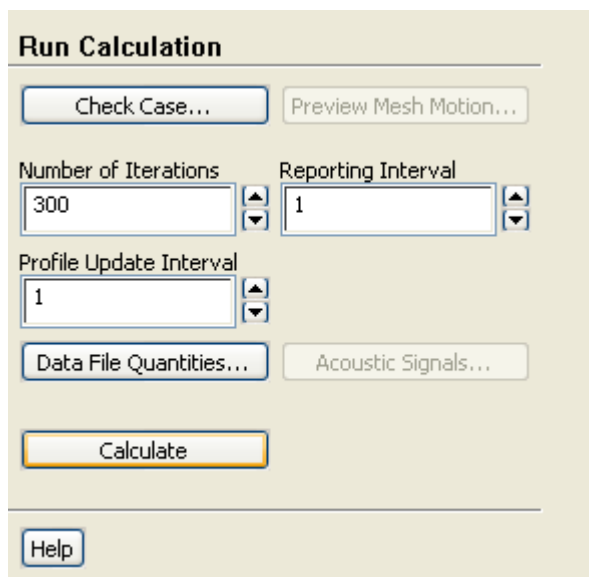
- e. Check to see if the case conforms to best practices.

◆ **Run Calculation** → **Check Case**



- i. Click the **Mesh** and **Solver** tabs and examine the **Recommendation** in each. These recommendations can be ignored for this tutorial. The issues they raise will be addressed in later tutorials.
  - ii. Close the **Case Check** dialog box.
8. Calculate a solution.
- a. Start the calculation by requesting 300 iterations.

◆ **Run Calculation**



- i. Enter 300 for **Number of Iterations**.
- ii. Click **Calculate**.

### Important

Note that the ANSYS FLUENT settings file (`FFF.set`) is written before the calculation begins.

---

### Important

Note that while the program is calculating the solution, the states of the **Setup** and **Solution** cells in the fluid flow ANSYS FLUENT analysis system in ANSYS Workbench are changing. For example:

- The state of the **Setup** cell becomes **Up-to-Date** and the state of the **Solution** cell becomes **Refresh Required** after the **Run Calculation** task page is visited and the number of iterations is specified.
  - The state of the **Solution** cell is **Update Required** while iterations are taking place.
  - The state of the **Solution** cell is **Up-to-Date** when the specified number of iterations are complete (or if convergence is reached).
- 

### Note

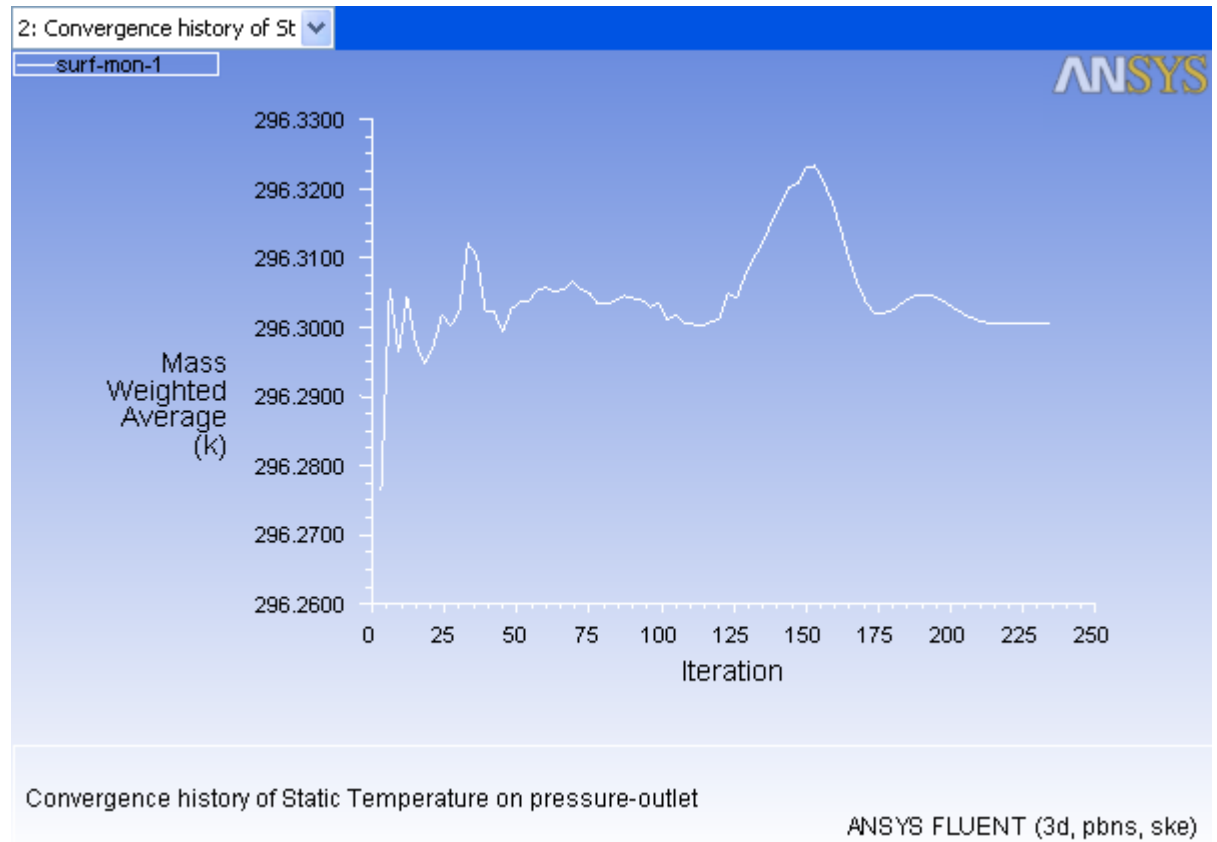
As the calculation progresses, the surface monitor history will be plotted in the graphics window (*Figure 1.19 (p. 47)*).

---

### Note

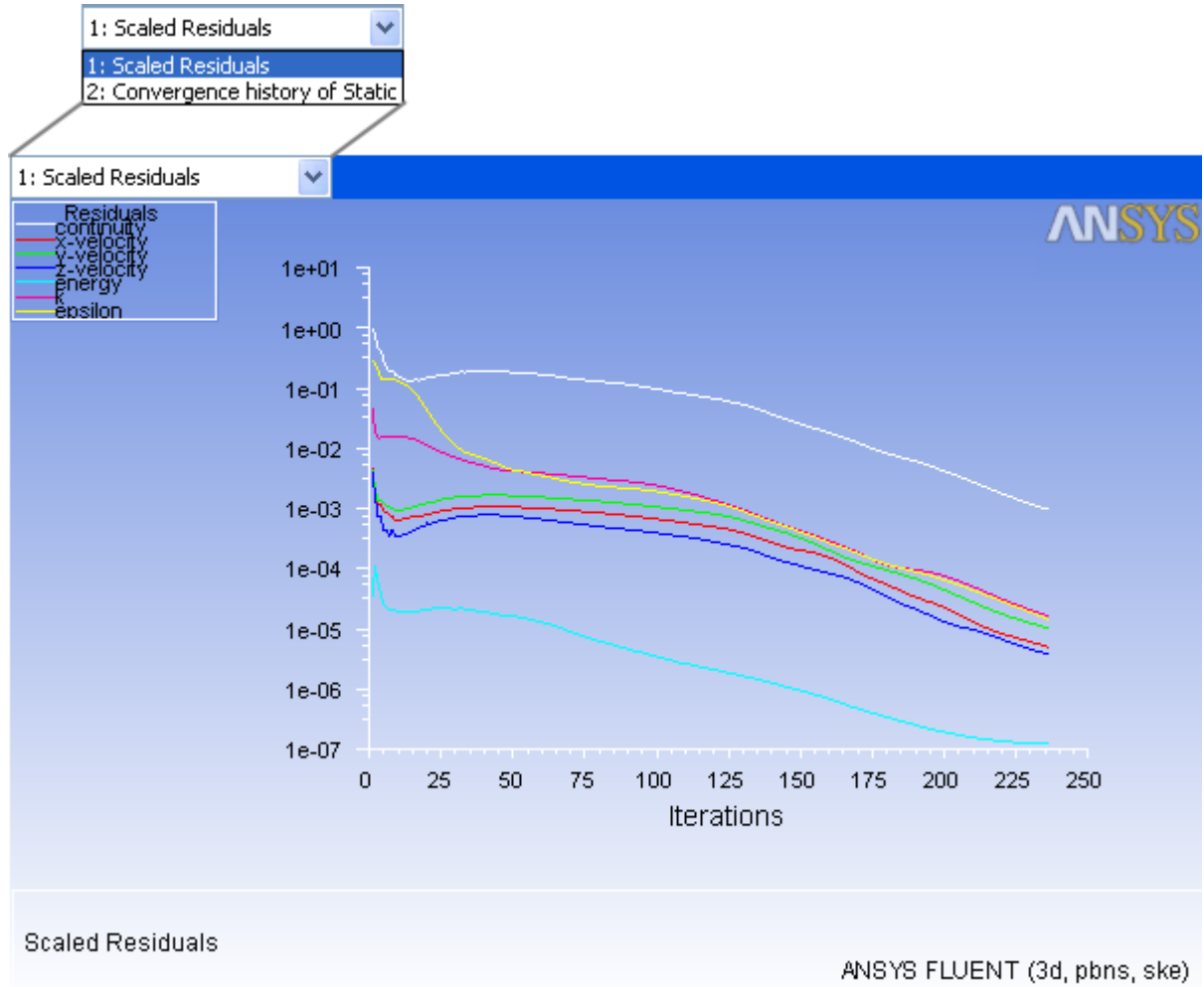
The solution will be stopped by ANSYS FLUENT when the residuals reach their specified values or after 300 iterations. The exact number of iterations will vary depending on the platform being used. An **Information** dialog box will open to alert you that the calculation is complete. Click **OK** in the **Information** dialog box to proceed.

Because the residual values vary slightly by platform, the plot that appears on your screen may not be exactly the same as the one shown here.

**Figure 1.19 Convergence History of the Mass-Weighted Average Temperature**

You can display the residuals history (Figure 1.20 (p. 48)), by selecting it from the graphics window drop-down list.

**Figure 1.20 Residuals for the Converged Solution**



- b. Examine the plots for convergence (*Figure 1.20* (p. 48) and *Figure 1.19* (p. 47)).

---

**Note**

There are no universal metrics for judging convergence. Residual definitions that are useful for one class of problem are sometimes misleading for other classes of problems. Therefore it is a good idea to judge convergence not only by examining residual levels, but also by monitoring relevant integrated quantities and checking for mass and energy balances.

There are three indicators that convergence has been reached:

- The residuals have decreased to a sufficient degree.

The solution has converged when the **Convergence Criterion** for each variable has been reached. The default criterion is that each residual will be reduced to a value of less than  $10^{-3}$ , except the **energy** residual, for which the default criterion is  $10^{-6}$ .

- The solution no longer changes with more iterations.

Sometimes the residuals may not fall below the convergence criterion set in the case setup. However, monitoring the representative flow variables through iterations may show that the residuals have stagnated and do not change with further iterations. This could also be considered as convergence.

- The overall mass, momentum, energy, and scalar balances are obtained.

You can examine the overall mass, momentum, energy and scalar balances in the **Flux Reports** dialog box. The net imbalance should be less than 0.2 % of the net flux through the domain when the solution has converged. In the next step you will check to see if the mass balance indicates convergence.

9. View the list of files generated by ANSYS Workbench.

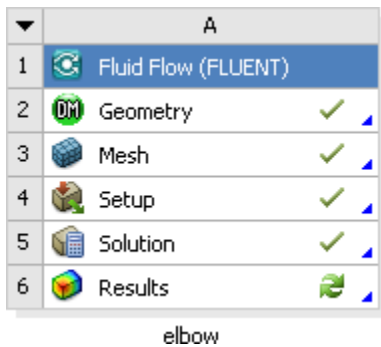
**View** → **Files**



**Figure 1.21 ANSYS Workbench Files View for the Project After Generating a Solution**

Files						
	A	B	C	D	E	
1	Name	Cell ID	Size	Type	Date Modified	Location
2	elbow-workbench.wbpj		125 KB	ANSYS Project File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01
3	FFF.agdb	A2	1 MB	Geometry File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
4	FFF.msh	A3,A4	8 MB	Fluent Mesh File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
5	FFF.mshdb	A3	1 MB	Mesh Database Files	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
6	FFF.set	A4	15 KB	FLUENT Model File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
7	designPoint.wbdp		33 KB	Design Point File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo

Note that the status of the **Solution** cell is now up-to-date.



### 1.4.6. Step 5: Displaying Results in ANSYS FLUENT and CFD-Post

In this step, you will display the results of the simulation in ANSYS FLUENT, display the results in CFD-Post, then review the list of files generated by ANSYS Workbench.

1. Display results in ANSYS FLUENT.

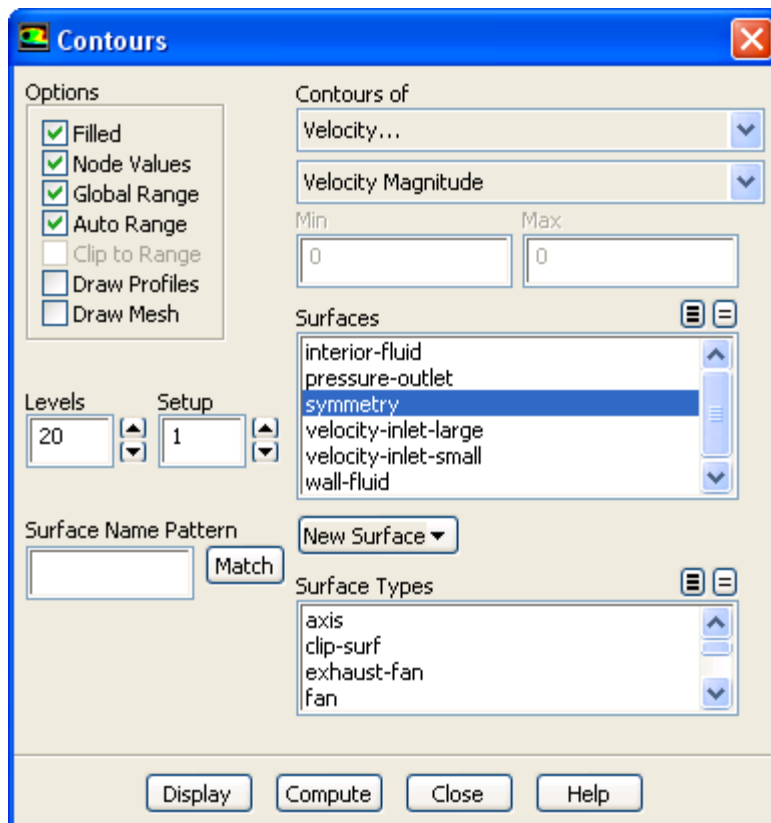
With ANSYS FLUENT still running, you can perform a simple evaluation of the velocity and temperature contours on the symmetry plane. Later, you will use CFD-Post (from within ANSYS Workbench) to perform the same evaluation.

- a. Display filled contours of velocity magnitude on the symmetry plane (*Figure 1.22 (p. 52)*).

**Graphics and Animations** → **Contours** → **Set Up...**

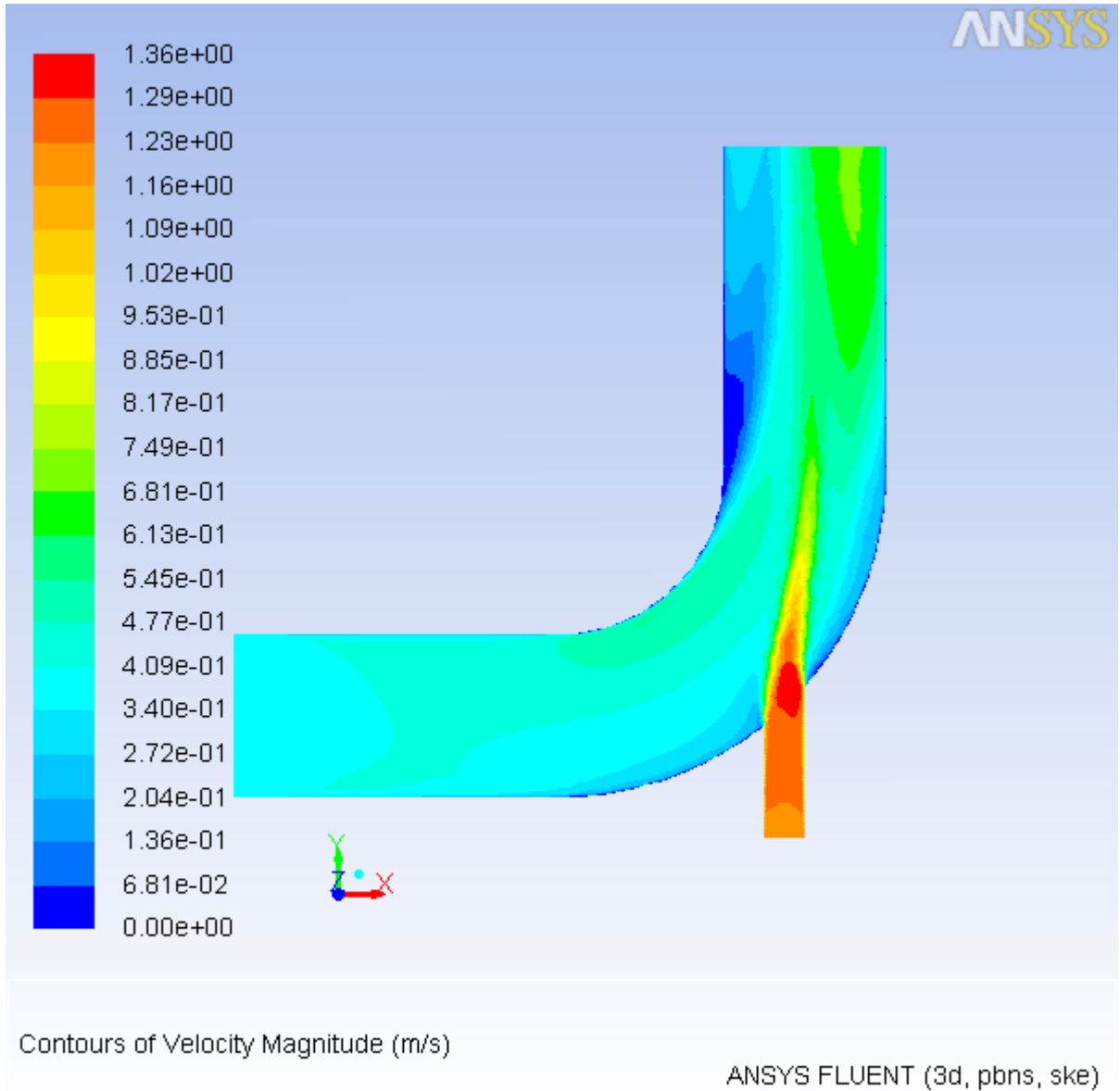
#### Note

You can also double-click a list item in order to open the corresponding dialog box.



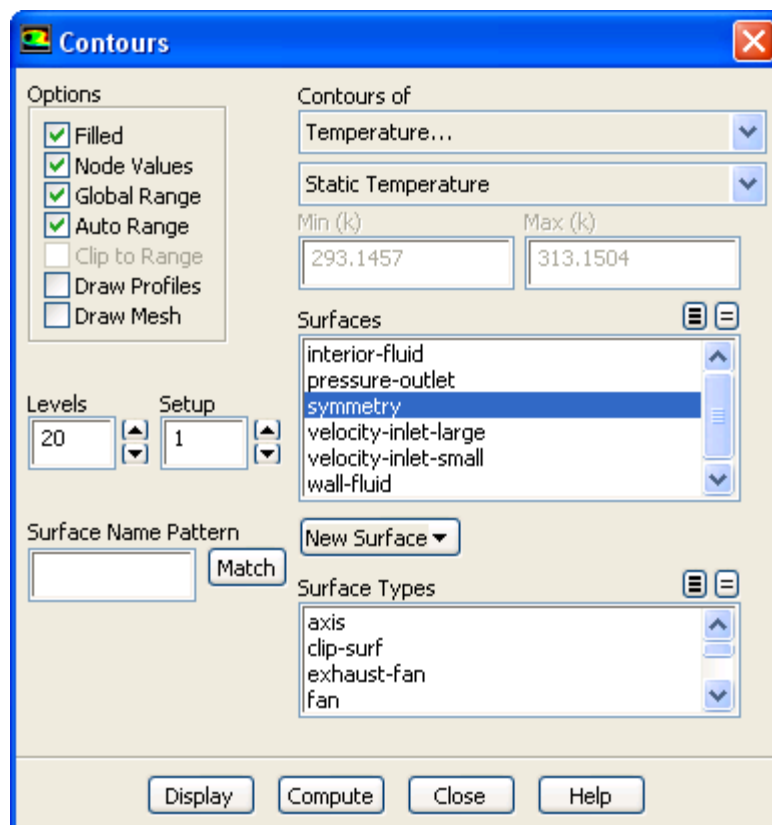
- i. In the **Contours** dialog box, enable **Filled** in the **Options** group box.
- ii. Make sure that **Node Values** is enabled in the **Options** group box.
- iii. Select **Velocity...** and **Velocity Magnitude** from the **Contours of** drop-down lists.
- iv. Select **symmetry** from the **Surfaces** selection list.
- v. Click **Display** to display the contours in the active graphics window.

**Figure 1.22 Velocity Distribution Along Symmetry Plane**



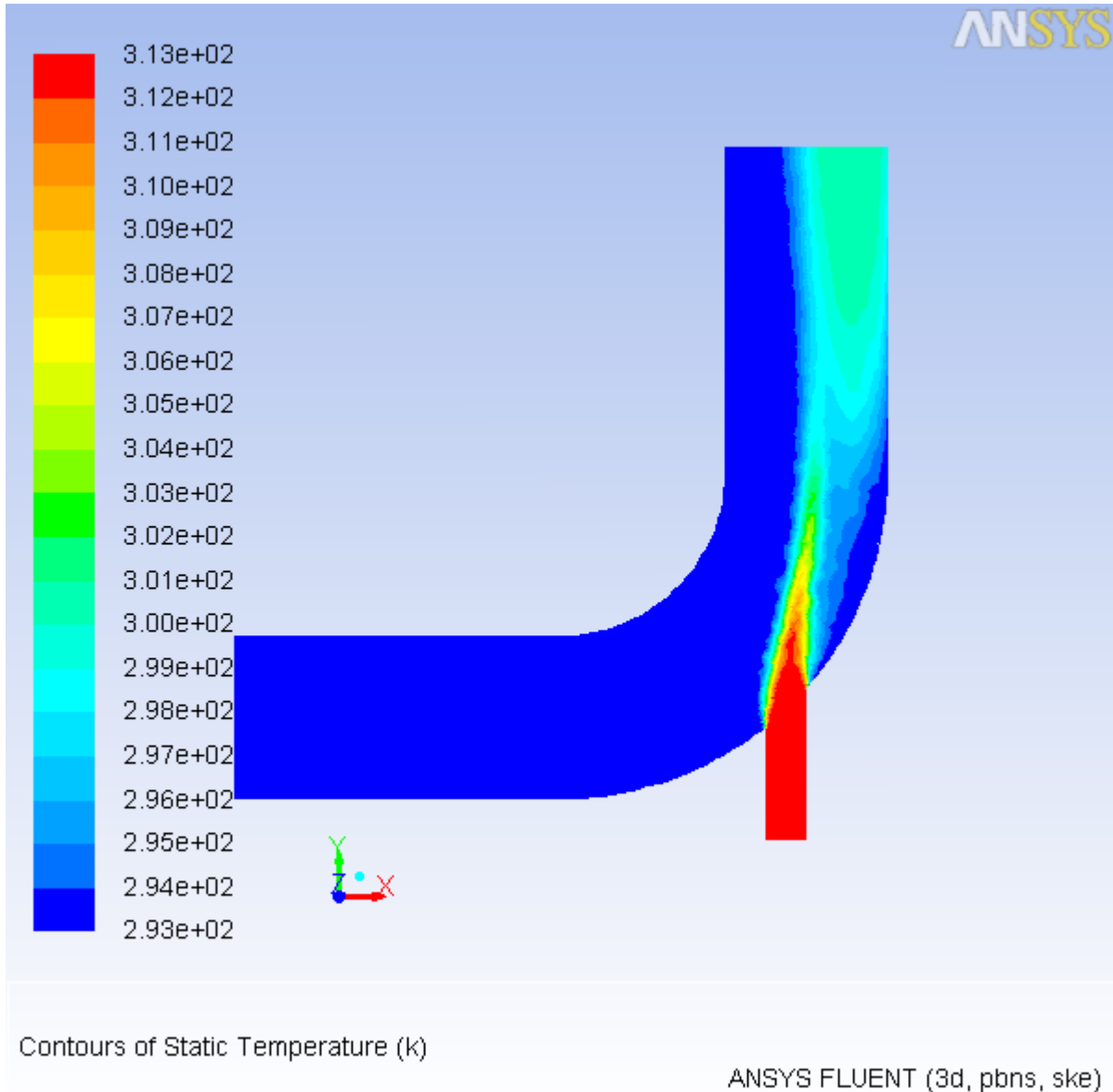
- b. Display filled contours of temperature on the symmetry plane (*Figure 1.23* (p. 54)).

 **Graphics and Animations** →  **Contours** → **Set Up...**



- i. Select **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.
- ii. Click **Display** and close the **Contours** dialog box.

**Figure 1.23 Temperature Distribution Along Symmetry Plane**



- c. Close the ANSYS FLUENT application.

**File → Close FLUENT**

---









**Important**

Note that the ANSYS FLUENT case and data files are automatically saved when you exit ANSYS FLUENT and return to ANSYS Workbench.

- d. View the list of files generated by ANSYS Workbench.

**View → Files**

**Figure 1.24 ANSYS Workbench Files View for the Project After Exiting ANSYS FLUENT**

Files						
	A	B	C	D	E	
1	Name	Cell ID	Size	Type	Date Modified	Location
2	 elbow-workbench.wbpj		128 KB	ANSYS Project File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01
3	 FFF.agdb	A2	1 MB	Geometry File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
4	 FFF.msh	A3,A4	8 MB	Fluent Mesh File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
5	 FFF.mshdb	A3	1 MB	Mesh Database Files	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
6	 FFF.set	A4	16 KB	FLUENT Model File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
7	 FFF-1.cas.gz	A5	3 MB	FLUENT Case File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
8	 FFF-1-00236.dat.gz	A5	6 MB	FLUENT Data File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
9	 designPoint.wbdp		38 KB	Design Point File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo

Note the addition of the compressed ANSYS FLUENT case file (`FFF-1.cas.gz`) and corresponding data file (`FFF-1-00236.dat.gz`) to the list of files. Also note that the name of the data file is based on the number of iterations, so your data file name may be different.

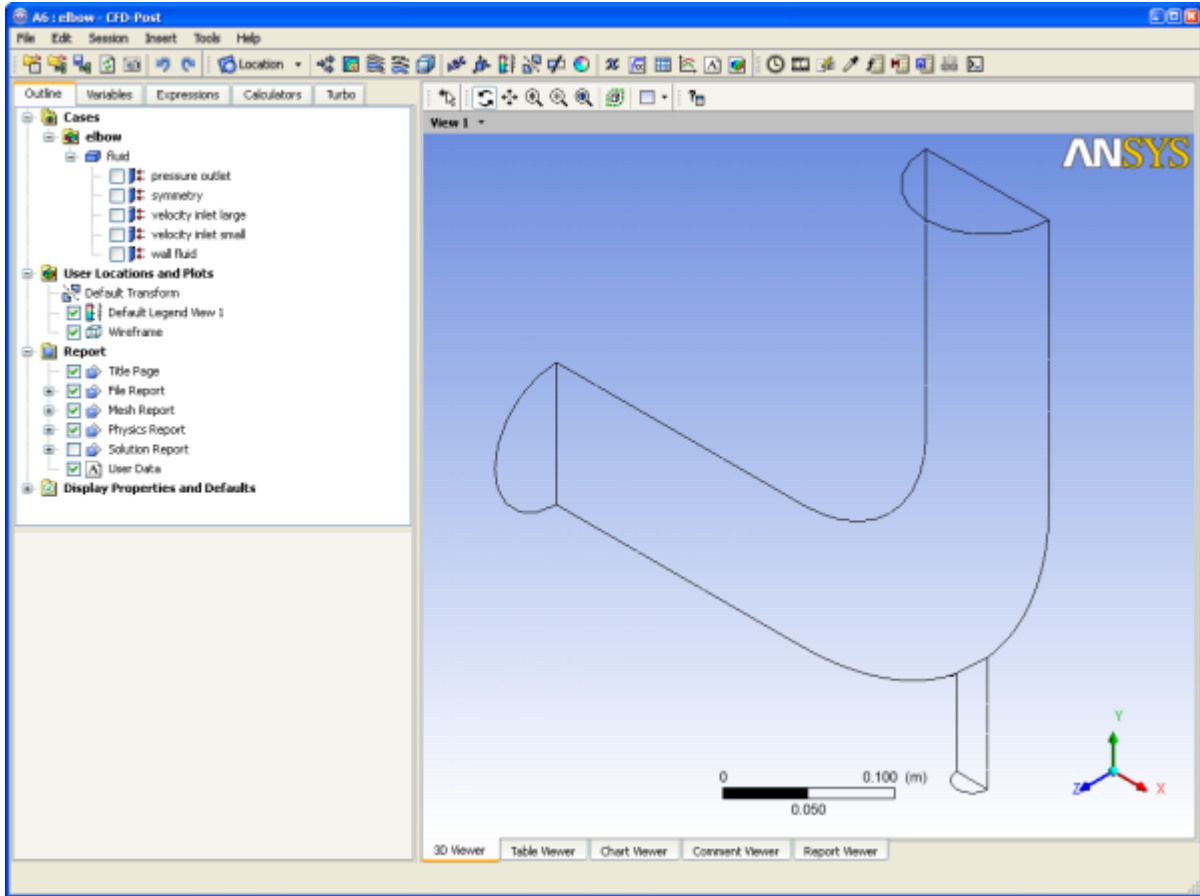
2. Display results in CFD-Post.
  - a. Start CFD-Post.


In the ANSYS Workbench **Project Schematic**, double-click the **Results** cell in the `elbow` fluid flow analysis system (cell `A6`). This displays the CFD-Post application. You can also right-click the **Results** cell to display the context menu where you can select the **Edit...** option.

### Note

The elbow geometry is already loaded and is displayed in outline mode. ANSYS FLUENT case and data files are also automatically loaded into CFD-Post.

**Figure 1.25 The Elbow Geometry Loaded into CFD-Post**



- b. Reorient the display.  
  
Click the blue Z axis on the axis triad in the bottom right hand corner of the graphics display to orient the display so that the view is of the front of the elbow geometry.
- c. Ensure that *Highlighting* () is disabled.
- d. Display filled contours of velocity magnitude on the symmetry plane (*Figure 1.26* (p. 58)).
  - i. Insert a contour object using the **Insert** menu item at the top of the CFD-Post window.

**Insert** → **Contour**

This displays the **Insert Contour** dialog box.

- ii. Keep the default name of the contour (Contour\_1) and click **OK** to close the dialog box. This displays the **Details of Contour 1** view below the **Outline** view in CFD-Post. This view contains all of the settings for a contour object.
- iii. In the **Geometry** tab, select **fluid** in the **Domains** list.
- iv. Select **symmetry** in the **Locations** list.
- v. Select **Velocity** in the **Variable** list.
- vi. Click **Apply**.

Details of **Contour 1**

Geometry Labels Render View

Domains fluid

Locations symmetry

Variable Velocity

Range Global

Min 0 [m s<sup>-1</sup>]

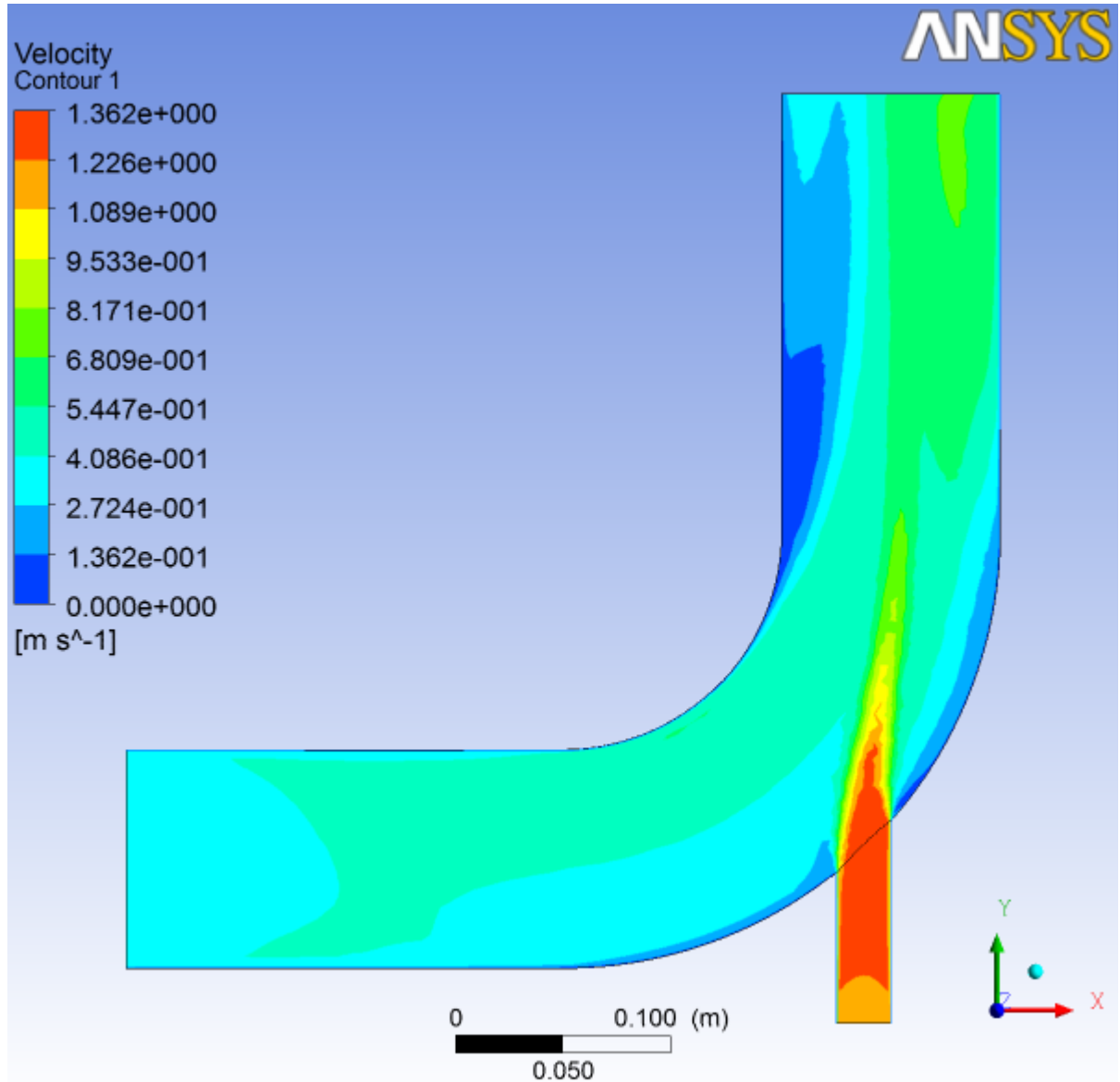
Max 1.36185 [m s<sup>-1</sup>]

Boundary Data  Hybrid  Conservative

Apply Reset Defaults



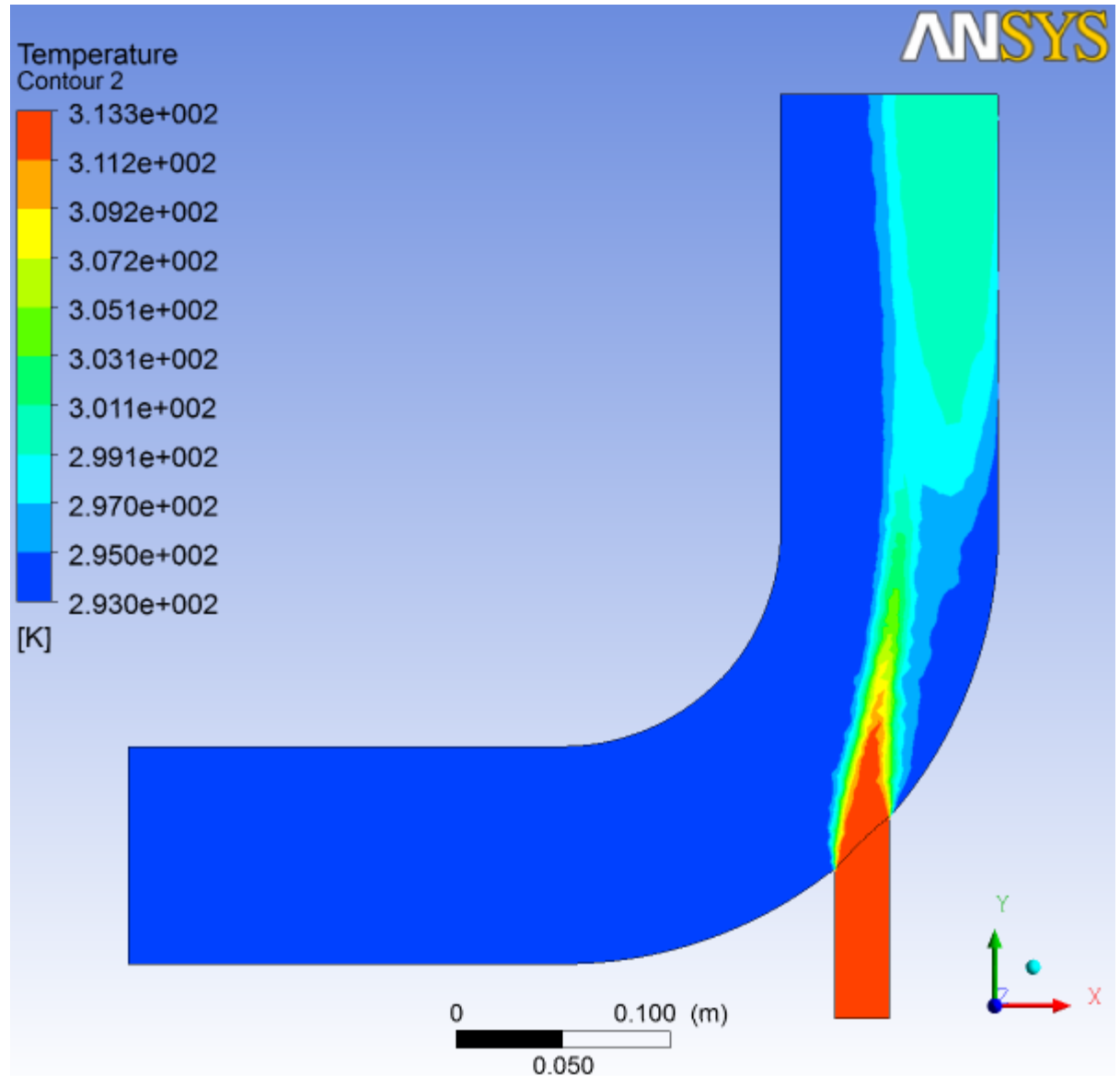
**Figure 1.26 Velocity Distribution Along Symmetry Plane**



- e. Display filled contours of temperature on the symmetry plane (*Figure 1.27* (p. 59)).
  - i. Click the check-marked box beside the **Contour 1** object under **User Locations and Plots** to deselect the **Contour 1** object and hide the first contour display.
  - ii. Insert a contour object.  
**Insert** → **Contour**  
This displays the **Insert Contour** dialog box.
  - iii. Keep the default name of the contour (Contour 2) and click **OK** to close the dialog box. This displays the **Details of Contour 2** view below the **Outline** view.
  - iv. In the **Geometry** tab, select **fluid** from the **Domains** list.
  - v. Select **symmetry** in the **Locations** list.
  - vi. Select **Temperature** in the **Variable** list.

vii. Click **Apply**.

**Figure 1.27 Temperature Distribution Along Symmetry Plane**



3. Close the CFD-Post application by selecting **File** → **Close CFD-Post** or by clicking the 'X' in the top right corner of the window.










### Important

Note that the CFD-Post state files are automatically saved when you exit CFD-Post and return to ANSYS Workbench.

4. Save the elbow-workbench project in ANSYS Workbench.
5. View the list of files generated by ANSYS Workbench.

**View** → **Files**

**Figure 1.28 ANSYS Workbench Files View for the Project After Viewing Results in CFD-Post**

Files						
	A	B	C	D	E	
1	Name	Cell ID	Size	Type	Date Modified	Location
2	 elbow-workbench.wbpj		128 KB	ANSYS Project File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01
3	 FFF.agdb	A2	1 MB	Geometry File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
4	 FFF.msh	A3,A4	8 MB	Fluent Mesh File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
5	 FFF.mshdb	A3	1 MB	Mesh Database Files	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
6	 FFF.set	A4	16 KB	FLUENT Model File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
7	 FFF-1.cas.gz	A5	3 MB	FLUENT Case File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
8	 FFF-1-00236.dat.gz	A5	6 MB	FLUENT Data File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
9	 elbow.cst	A6	29 KB	CFD-Post State File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
10	 designPoint.wbdp		39 KB	Design Point File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo

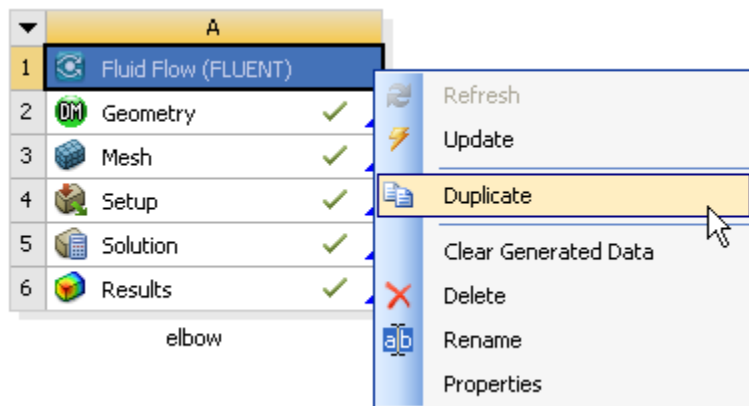
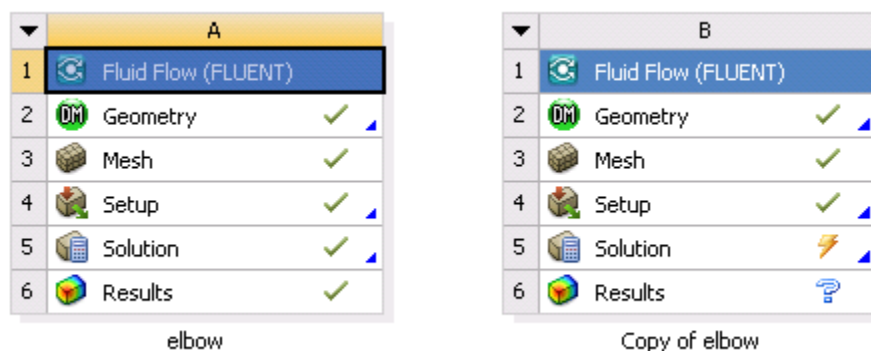
Note the addition of the CFD-Post state file (elbow.cst) to the list of files. For more information about CFD-Post (and the files associated with it), see the CFD-Post documentation.

### 1.4.7. Step 6: Duplicating the FLUENT-Based Fluid Flow Analysis System

At this point, you have a completely defined fluid flow system that is comprised of a geometry, a computational mesh, a CFD setup and solution, and corresponding results. In order to study the effects upon the flow field that may occur if you were to alter the geometry, another fluid flow analysis is required. One approach would be to use the current system and change the geometry, however you would overwrite the data from your previous simulation. A more suitable and effective approach would be to create a copy, or duplicate, of the current system, and then make the appropriate changes to the duplicate system.

In this step, you will create a duplicate of the original FLUENT-based fluid flow system, then review the list of files generated by ANSYS Workbench.

1. In the **Project Schematic**, right-click the title cell of the **Fluid Flow (FLUENT)** system and select **Duplicate** from the context menu.

**Figure 1.29 Duplicating the Fluid Flow System****Figure 1.30 The Original Fluid Flow System and Its Duplicate****Note**

Notice that in the duplicated system, the state of the **Solution** cell indicates that the cell requires an update while the state of the **Results** cell indicates that the cell requires attention. This is because when a system is duplicated, the case and data files are not copied to the new system, therefore, the new system does not yet have solution data associated with it.

2. Rename the duplicated system to `new-elbow`.
3. Save the `elbow-workbench` project in ANSYS Workbench.

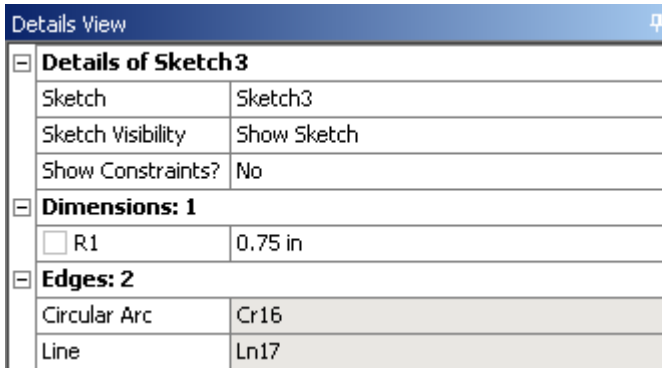
**1.4.8. Step 7: Changing the Geometry in ANSYS DesignModeler**

Now that you have two separate, but equivalent, FLUENT-based fluid flow systems to work from, you can make changes to the second system without impacting the original system. In this step, you will make a slight alteration to the elbow geometry in ANSYS DesignModeler by changing the diameter of the smaller inlet, then review the list of files generated by ANSYS Workbench.

1. Open ANSYS DesignModeler.

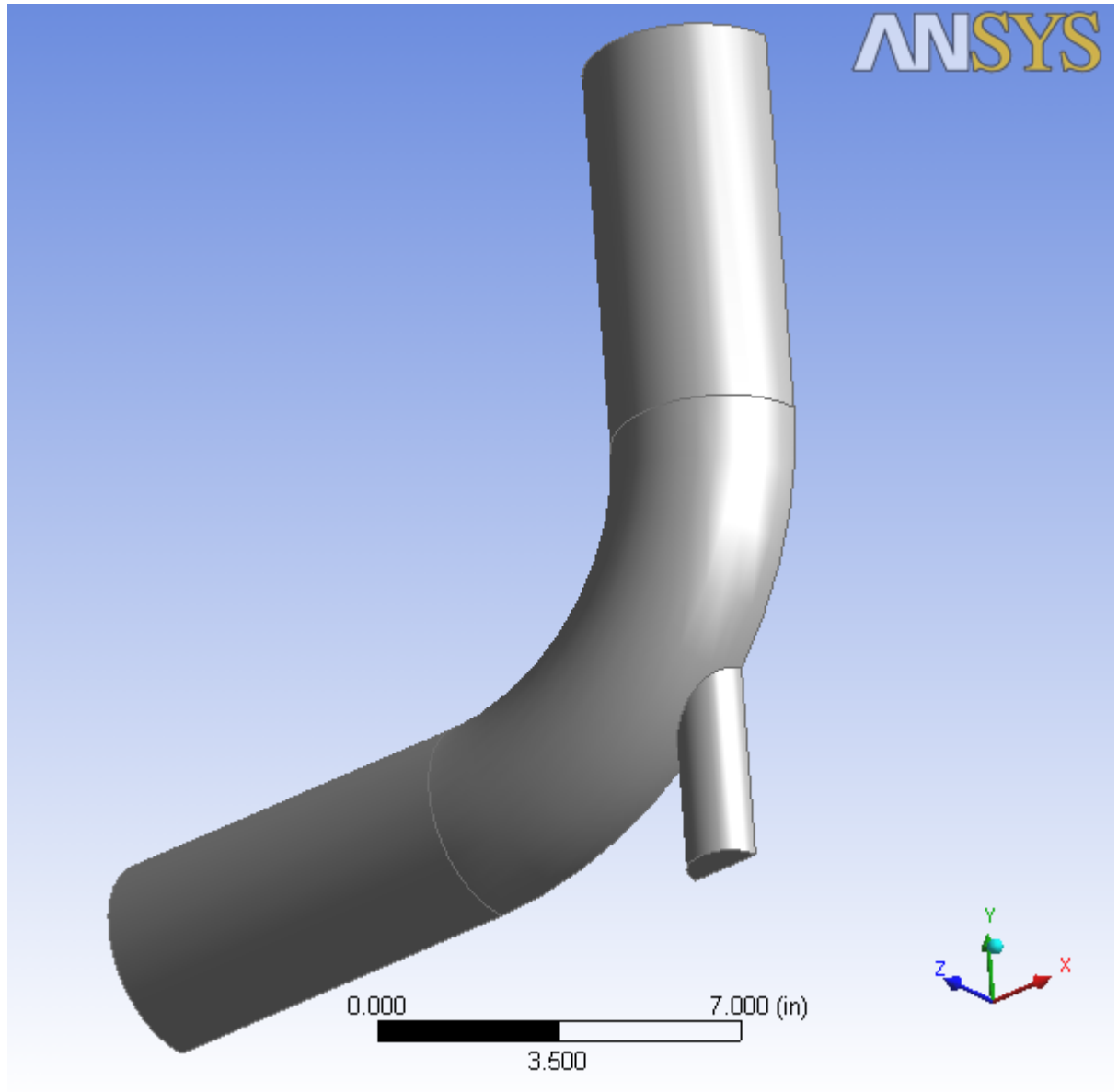
Double-click the **Geometry** cell of the `new-elbow` system (cell B2) to display the geometry in ANSYS DesignModeler.

2. Change the diameter of the small inlet (`velocity-inlet-small`).
  - a. Select **Sketch3**, under **Extrude1** to open the **Details View** of the small inlet extrusion.
  - b. In the **Details View**, under **Dimensions:1**, change the **R1** (radius) value from 0.5 inches to 0.75 inches.



Details View	
[-] <b>Details of Sketch3</b>	
Sketch	Sketch3
Sketch Visibility	Show Sketch
Show Constraints?	No
[-] <b>Dimensions: 1</b>	
<input type="checkbox"/> R1	0.75 in
[-] <b>Edges: 2</b>	
Circular Arc	Cr16
Line	Ln17

- c. Click the **Generate** button to generate the geometry with your new values.

**Figure 1.31 Changing the Diameter of the Small Inlet in ANSYS DesignModeler**

3. Close ANSYS DesignModeler.
4. View the list of files generated by ANSYS Workbench.

**View** → **Files**

**Figure 1.32 ANSYS Workbench Files View for the Project After Duplicating the System and Changing the Geometry**

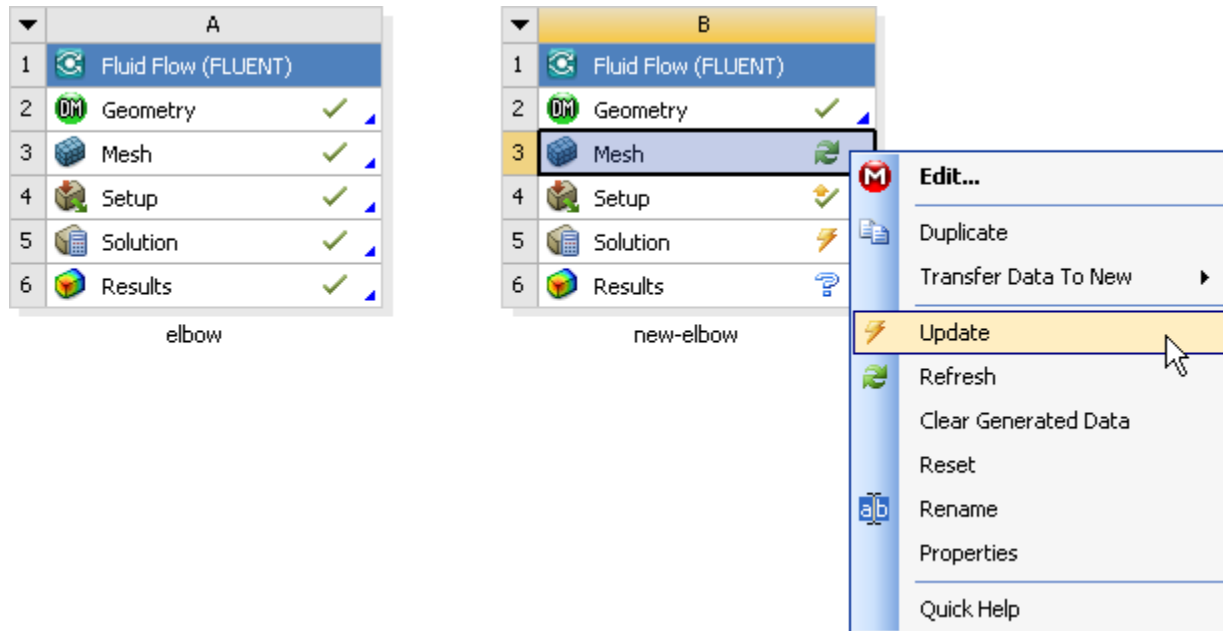
Files						
	A	B	C	D	E	
1	Name	Cell ID	Size	Type	Date Modified	Location
2	elbow-workbench.wbpj		141 KB	ANSYS Project File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01
3	FFF.agdb	A2	1 MB	Geometry File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
4	FFF.msh	A3,A4	8 MB	Fluent Mesh File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
5	FFF.mshdb	A3	1 MB	Mesh Database Files	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
6	FFF.set	A4	16 KB	FLUENT Model File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
7	FFF-1.cas.gz	A5	3 MB	FLUENT Case File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
8	FFF-1-00236.dat.gz	A5	6 MB	FLUENT Data File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
9	elbow.cst	A6	29 KB	CFD-Post State File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
10	FFF-1.agdb	B2	2 MB	Geometry File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
11	FFF-1.msh	B3,B4	8 MB	Fluent Mesh File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
12	FFF-1.mshdb	B3	1 MB	Mesh Database Files	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
13	FFF.set	B4	16 KB	FLUENT Model File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
14	elbow.cst	B6	29 KB	CFD-Post State File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
15	designPoint.wbdp		74 KB	Design Point File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo

Note the addition of the geometry, mesh, and ANSYS FLUENT settings files now associated with the new, duplicated system.

### 1.4.9. Step 8: Updating the Mesh in the ANSYS Meshing Application

The modified geometry now requires a new computational mesh. The mesh settings for the duplicated system are retained in the duplicated system. In this step, you will update the mesh based on the mesh settings from the original system, then review the list of files generated by ANSYS Workbench.

In the **Project Schematic**, right-click the **Mesh** cell of the new-elbow system (cell B3) and select **Update** from the context menu. This will update the mesh for the new geometry based on the mesh settings you specified earlier in the ANSYS Meshing application without having to open the editor to regenerate the mesh.

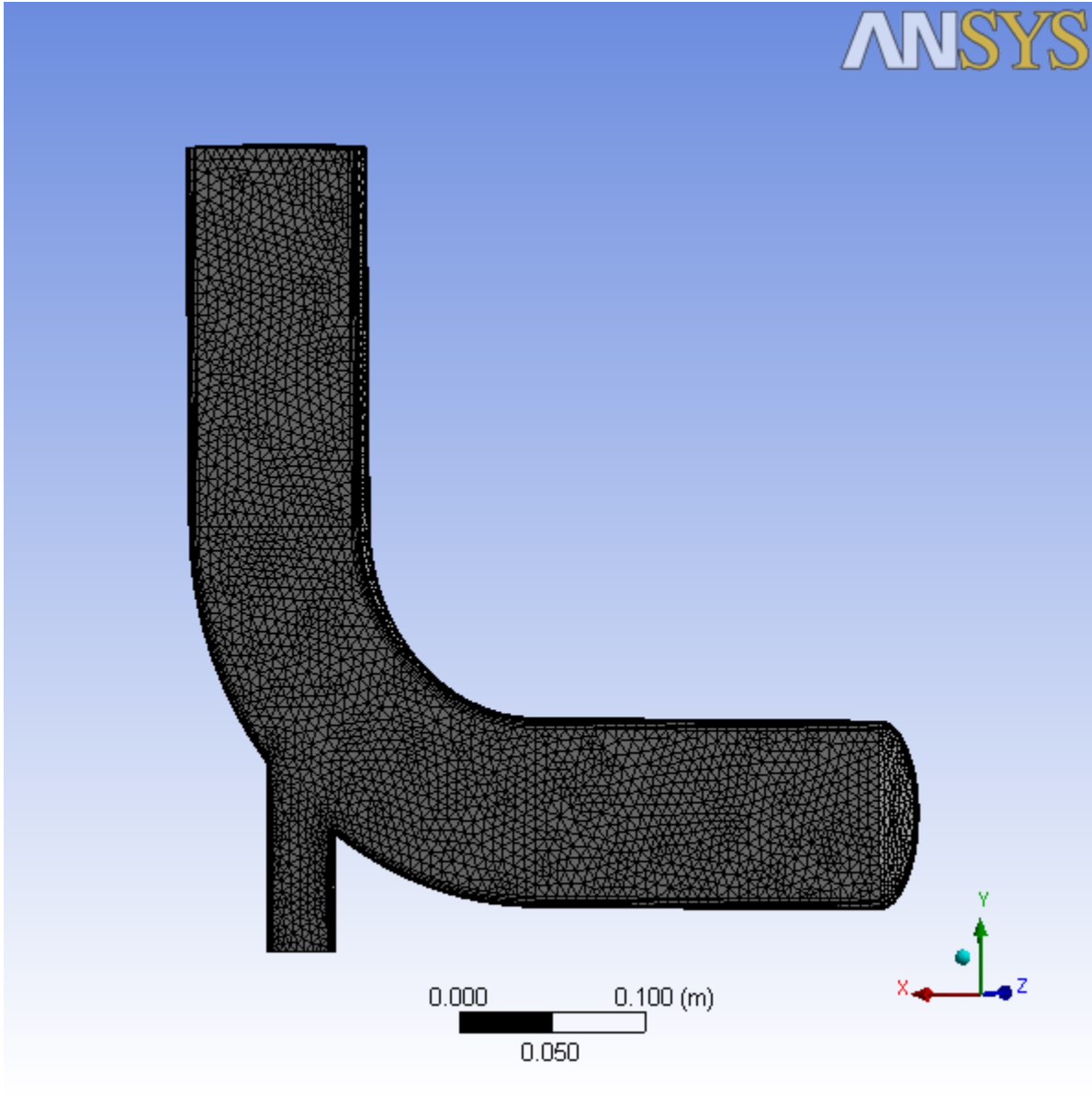
**Figure 1.33 Updating the Mesh for the Changed Geometry**

It will take a few moments to update the mesh. Once the update is complete, the state of the **Mesh** cell is changed to up-to-date, symbolized by a green check mark.

For illustrative purposes of the tutorial, the new geometry and the new mesh are displayed below.


















**Figure 1.34 The Updated Geometry and Mesh in the ANSYS Meshing Application**



Inspecting the files generated by ANSYS Workbench reveals the updated mesh file for the duplicated system.

**View → Files**

**Figure 1.35 ANSYS Workbench Files View for the Project After Updating the Mesh for the Altered Geometry**

Files						
	A	B	C	D	E	
1	Name	Cell ID	Size	Type	Date Modified	Location
2	 elbow-workbench.wbpj		141 KB	ANSYS Project File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01
3	 FFF.agdb	A2	1 MB	Geometry File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
4	 FFF.msh	A3,A4	8 MB	Fluent Mesh File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
5	 FFF.mshdb	A3	1 MB	Mesh Database Files	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
6	 FFF.set	A4	16 KB	FLUENT Model File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
7	 FFF-1.cas.gz	A5	3 MB	FLUENT Case File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
8	 FFF-1-00236.dat.gz	A5	6 MB	FLUENT Data File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
9	 elbow.cst	A6	29 KB	CFD-Post State File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
10	 FFF-1.agdb	B2	1 MB	Geometry File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
11	 FFF-1.msh	B3,B4	8 MB	Fluent Mesh File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
12	 FFF-1.mshdb	B3	1 MB	Mesh Database Files	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
13	 FFF.set	B4	16 KB	FLUENT Model File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
14	 elbow.cst	B6	29 KB	CFD-Post State File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
15	 FFF-1.1.msh	B3	8 MB	Fluent Mesh File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
16	 designPoint.wbdp		74 KB	Design Point File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo

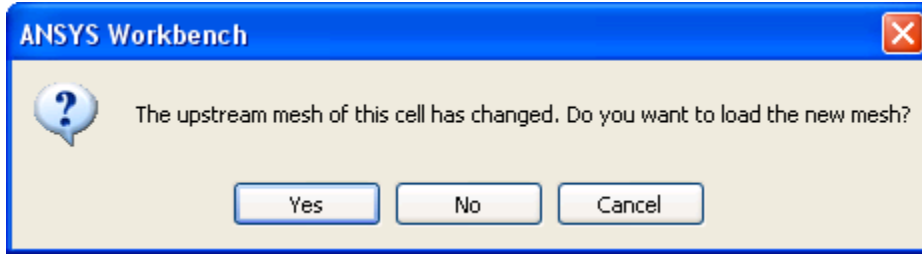
### 1.4.10. Step 9: Calculating a New Solution in ANSYS FLUENT

Now that there is an updated computational mesh for the modified geometry in the duplicated system, a new solution needs to be generated using ANSYS FLUENT. In this step, you will revisit the settings within ANSYS FLUENT, calculate another solution, view the new results, then review the list of files generated by ANSYS Workbench.

1. Open ANSYS FLUENT.

In the **Project Schematic**, right-click the **Setup** cell of the `new-elbow` system (cell B4) and select **Edit...** from the context menu. Since the mesh has been changed, you are prompted as to whether you want to load the new mesh into ANSYS FLUENT or not. Select **Yes** to continue, and click **OK** when FLUENT Launcher is displayed in order to open ANSYS FLUENT.

**Figure 1.36 ANSYS Workbench Prompt When the Upstream Mesh Has Changed**



2. Make sure that the unit of length is set to inches.

◆ **General** → **Units...**

3. Check the mesh (optional).

◆ **General** → **Check**

4. Revisit the boundary conditions for the small inlet.

◆ **Boundary Conditions** →  **velocity-inlet-small** → **Edit...**

Here, you need to set the hydraulic diameter to 1.5 in based on the new dimensions of the small inlet.

5. Reinitialize the solution.

◆ **Solution Initialization**

Keep the default **Hybrid Initialization** and click **Initialize**.

6. Recalculate the solution.

◆ **Run Calculation**

Keep the **Number of Iterations** set to 300 and click **Calculate**.

7. Close ANSYS FLUENT.
8. Revisit the results of the calculations in CFD-Post.

Double-click the **Results** cell of the `new-elbow` fluid flow system to re-open CFD-Post where you can review the results of the new solution.

9. Close CFD-Post.
10. Save the `elbow-workbench` project in ANSYS Workbench.
11. View the list of files generated by ANSYS Workbench.

**View** → **Files**

**Figure 1.37 ANSYS Workbench Files View for the Project After Viewing the New Results in CFD-Post**

Files						
	A	B	C	D	E	
1	Name	Cell ID	Size	Type	Date Modified	Location
2	 elbow-workbench.wbpj		141 KB	ANSYS Project File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01
3	 FFF.agdb	A2	1 MB	Geometry File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
4	 FFF.msh	A3,A4	8 MB	Fluent Mesh File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
5	 FFF.mshdb	A3	1 MB	Mesh Database Files	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
6	 FFF.set	A4	16 KB	FLUENT Model File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
7	 FFF-1.cas.gz	A5	3 MB	FLUENT Case File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
8	 FFF-1-00236.dat.gz	A5	6 MB	FLUENT Data File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
9	 elbow.cst	A6	29 KB	CFD-Post State File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
10	 FFF-1.agdb	B2	1 MB	Geometry File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
11	 FFF-1.1.set	B4	16 KB	FLUENT Model File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
12	 FFF-1.mshdb	B3	1 MB	Mesh Database Files	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
13	 FFF-1.1-1.cas.gz	B5	393 KB	FLUENT Case File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
14	 elbow.cst	B6	29 KB	CFD-Post State File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
15	 FFF-1.1.msh	B3,B4	8 MB	Fluent Mesh File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
16	 FFF-1.1-1-00242.dat.gz	B5	6 MB	FLUENT Data File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
17	 designPoint.wbdp		74 KB	Design Point File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo

Note the addition of the solution and state files now associated with new duplicated system.

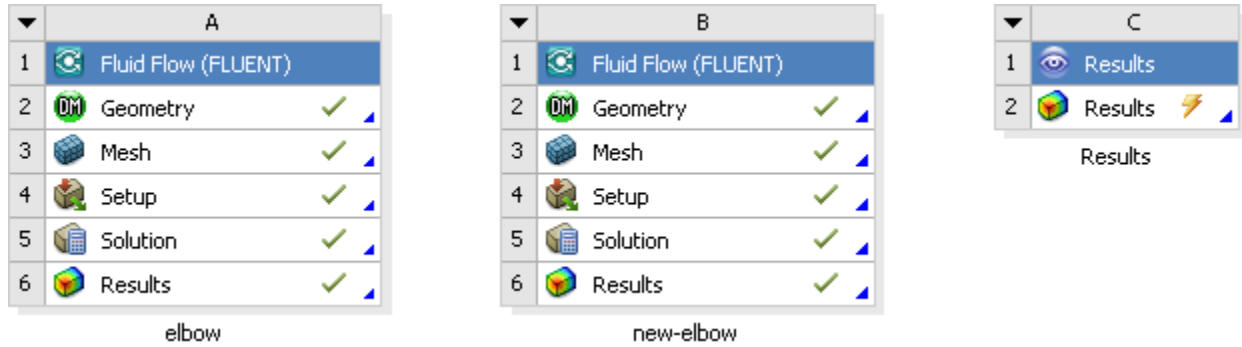
### 1.4.11. Step 10: Comparing the Results of Both Systems in CFD-Post

In this step, you will create a new **Results** system in ANSYS Workbench, use that system to compare the solutions from each of the two FLUENT-based fluid flow analysis systems in CFD-Post at the same time, then review the list of files generated by ANSYS Workbench.

1. Create a **Results** system.

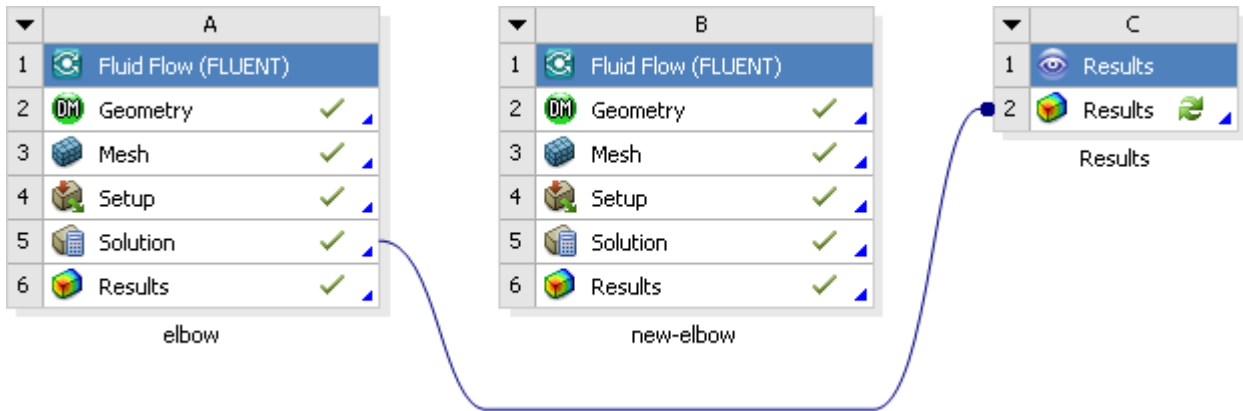
In ANSYS Workbench, drag a **Results** system from the **Component Systems** section of the **Toolbox** and drop it into the **Project Schematic**, next to the fluid flow systems.

**Figure 1.38 The NewResults System in the Project Schematic**



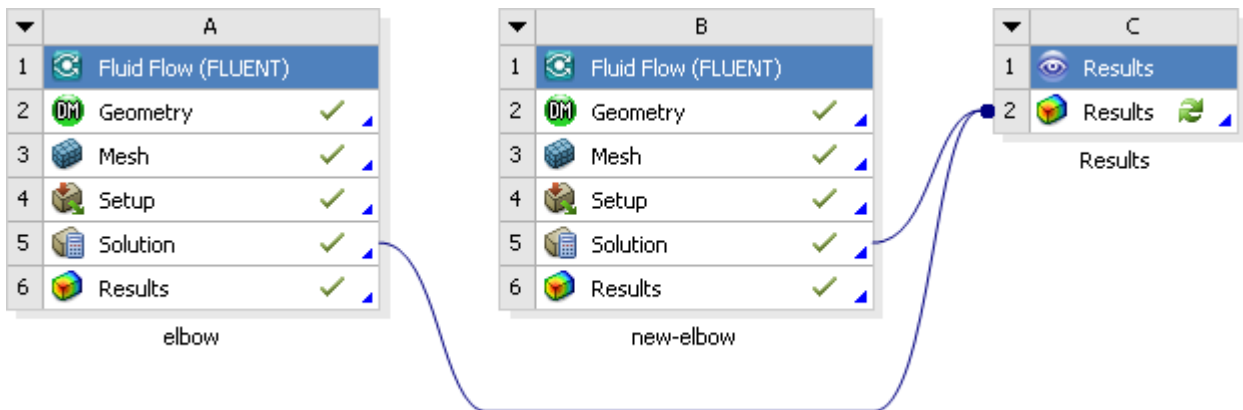
2. Add the solutions of each of the systems to the **Results** system.
  - a. Select the **Solution** cell in the first Fluid Flow analysis system (cell A5) and drag it over the **Results** cell in the **Results** system (cell C2). This creates a transfer data connection between the two systems.

**Figure 1.39 Connecting the First Fluid Flow System to the New Results System**



- b. Select the **Solution** cell in the second Fluid Flow analysis system (cell B5) and drag it over the **Results** cell in the **Results** system (cell C2). This creates a transfer data connection between the two systems.

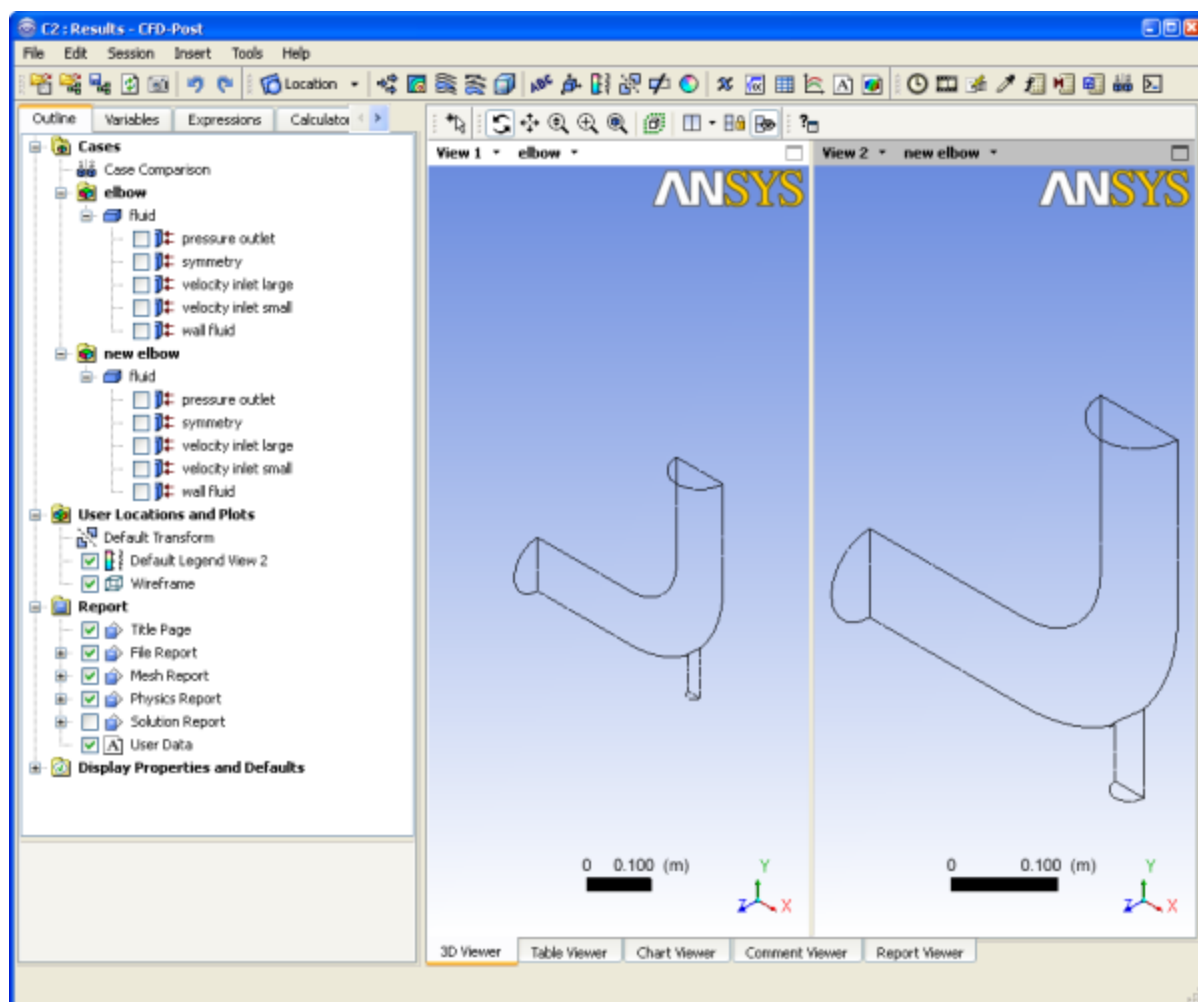
**Figure 1.40 Connecting the Second Fluid Flow System to the New Results System**



3. Open CFD-Post to compare the results of the two fluid flow systems.

Now that the two fluid flow systems are connected to the **Results** system, double-click the **Results** cell in the **Results** system (cell C2) to open CFD-Post. Within CFD-Post, both geometries are displayed side by side.


**Figure 1.41 CFD-Post with Both Fluid Flow Systems Displayed**



- a. Re-orient the display.

In each view, click the blue Z axis on the axis triad in the bottom right hand corner of the graphics display to orient the display so that the view is of the front of the elbow geometry.

### Important

Alternatively, you can select the synchronization tool () in the **3D Viewer Toolbar** to synchronize the views, so that when you re-orient one view, the other view is automatically updated.

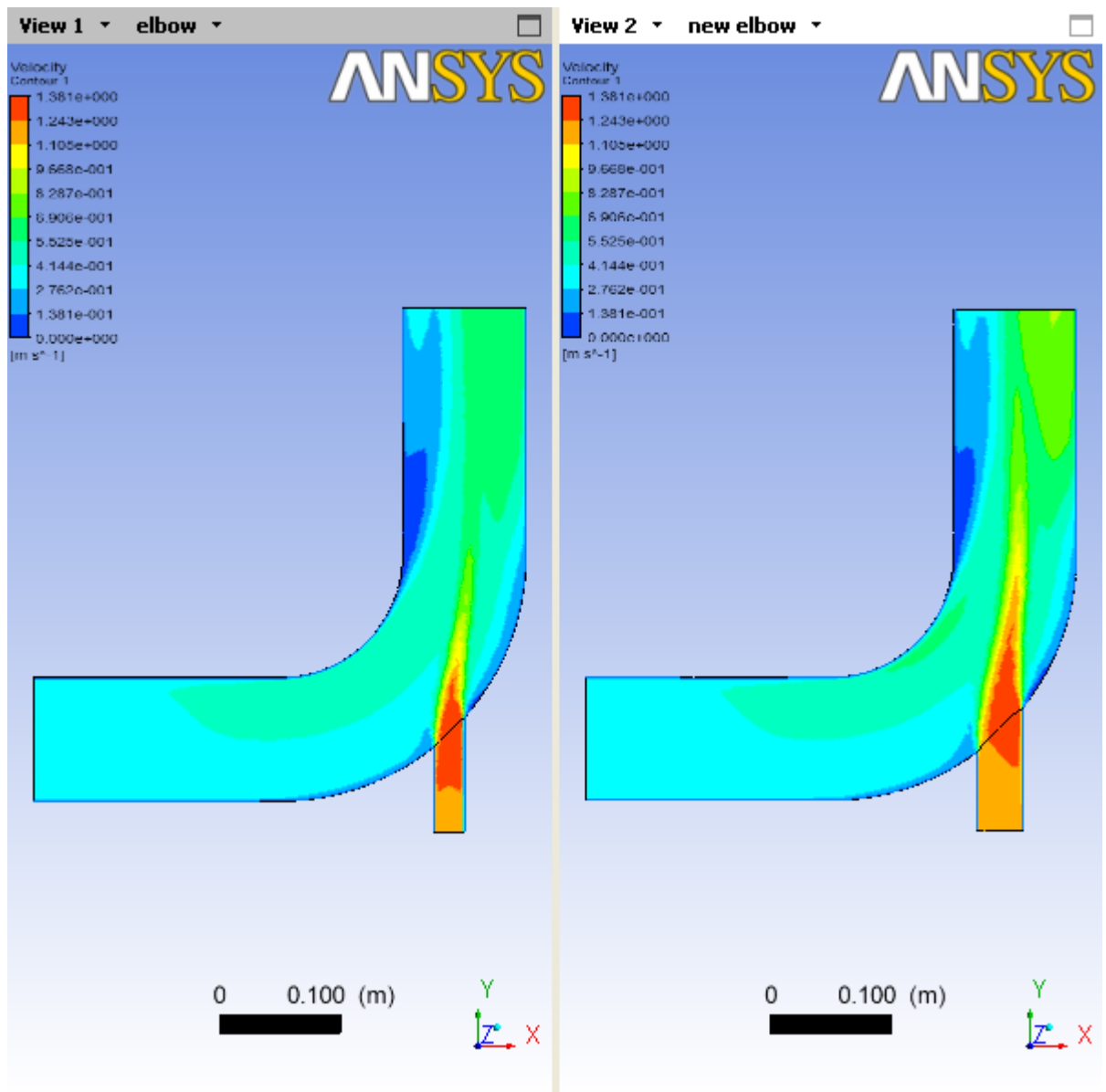
- b. Display filled contours of velocity magnitude on the symmetry plane.
  - i. Insert a contour object.

**Insert** → **Contour**

This displays the **Insert Contour** dialog box.

- ii. Keep the default name of the contour (Contour 1) and click **OK** to close the dialog box. This displays the **Details of Contour 1** view below the **Outline** view in CFD-Post. This view contains all of the settings for a contour object.
- iii. In the **Geometry** tab, select **fluid** in the **Domains** list.
- iv. Select **symmetry** in the **Locations** list.
- v. Select **Velocity** in the **Variable** list.
- vi. Click **Apply**. The velocity contours are displayed in each view.

**Figure 1.42** CFD-Post Displaying Velocity Contours for Both Geometries



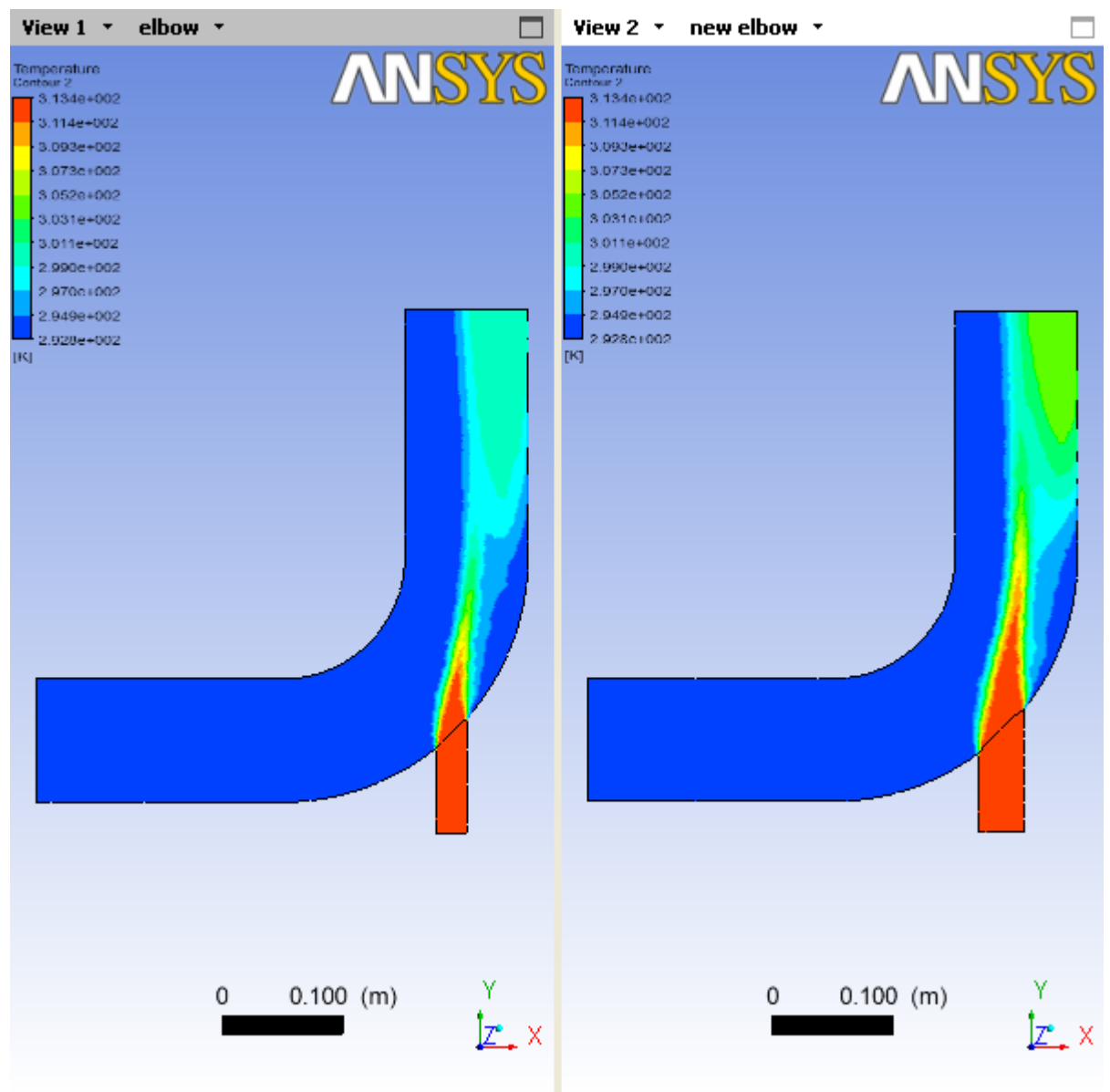
- c. Display filled contours of temperature on the symmetry plane.
  - i. Deselect the **Contour 1** object under **User Locations and Plots** in CFD-Post to hide the first contour display.
  - ii. Insert another contour object.

**Insert** → **Contour**

This displays the **Insert Contour** dialog box.

- iii. Keep the default name of the contour (Contour 2) and click **OK** to close the dialog box. This displays the **Details of Contour 2** view below the **Outline** view in CFD-Post.
- iv. In the **Geometry** tab, select **fluid** in the **Domains** list.
- v. Select **symmetry** in the **Locations** list.
- vi. Select **Temperature** in the **Variable** list.
- vii. Click **Apply**. The temperature contours are displayed in each view.

**Figure 1.43** CFD-Post Displaying Temperature Contours for Both Geometries



4. Close the CFD-Post application.
5. Save the elbow-workbench project in ANSYS Workbench.
6. View the list of files associated with your project using the **Files** view.



View → Files

**Figure 1.44 ANSYS Workbench Files View for the Project After Viewing the Combined Results in CFD-Post**

Files						
	A	B	C	D	E	
1	Name	Cell ID	Size	Type	Date Modified	Location
2	elbow-workbench.wbpj		146 KB	ANSYS Project File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01
3	FFF.agdb	A2	1 MB	Geometry File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
4	FFF.msh	A3,A4	8 MB	Fluent Mesh File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
5	FFF.mshdb	A3	1 MB	Mesh Database Files	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
6	FFF.set	A4	16 KB	FLUENT Model File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
7	FFF-1.cas.gz	A5	3 MB	FLUENT Case File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
8	FFF-1-00236.dat.gz	A5	6 MB	FLUENT Data File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
9	elbow.cst	A6	29 KB	CFD-Post State File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
10	FFF-1.agdb	B2	1 MB	Geometry File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
11	FFF-1.1.set	B4	16 KB	FLUENT Model File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
12	FFF-1.mshdb	B3	1 MB	Mesh Database Files	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
13	FFF-1.1-1.cas.gz	B5	3 MB	FLUENT Case File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
14	elbow.cst	B6	29 KB	CFD-Post State File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
15	FFF-1.1.msh	B3,B4	8 MB	Fluent Mesh File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
16	FFF-1.1-1-00242.dat.gz	B5	6 MB	FLUENT Data File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
17	Results.cst	C2	36 KB	CFD-Post State File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
18	designPoint.wbdp		83 KB	Design Point File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo

Note the addition of the **Results** system and its corresponding files.

### 1.4.12. Step 11: Summary

In this tutorial, portions of ANSYS Workbench were used to compare the fluid flow through two slightly different geometries. ANSYS DesignModeler was used to create a mixing elbow geometry, ANSYS Meshing was used to create a computational mesh, ANSYS FLUENT was used to calculate the fluid flow throughout the geometry using the computational mesh, and CFD-Post was used to analyze the results. In addition, the geometry was altered, a new mesh was generated, and a new solution was calculated.

Finally, ANSYS Workbench was set up so that CFD-Post could directly compare the results of both calculations at the same time.