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

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.

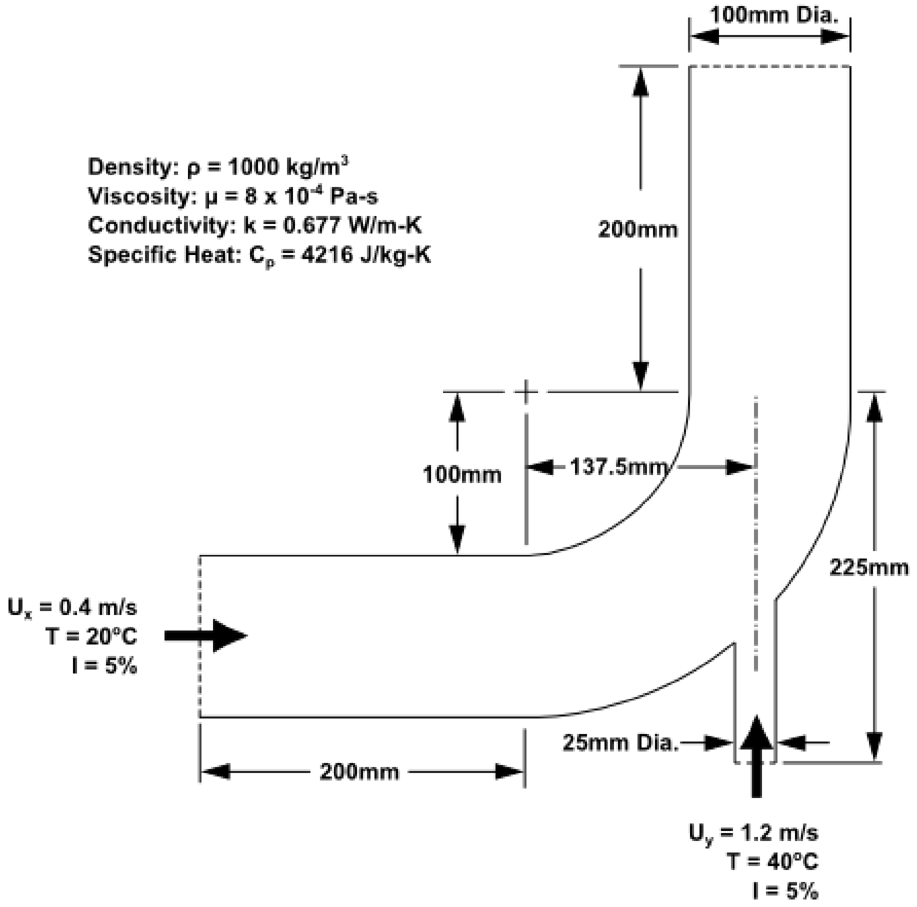
1.3. Problem Description

The problem to be considered is shown schematically in Figure 1.1: Problem Specification (p. 3). 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 must be modeled.

Figure 1.1: Problem Specification



Note

The functionality to create named selections exists in both ANSYS DesignModeler and ANSYS Meshing. For the purposes of this tutorial, named selections are created in ANSYS Meshing since the meshing application provides more comprehensive and extensive named selection functionality.

1.4. Setup and Solution

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

1.4.1. Preparation

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

   Note

   If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

   a. Click **ANSYS Fluent** under **Product**.

   b. Click **16.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **elbow-workbench_R160.zip** link to download the input and solution files.

7. Unzip `elbow-workbench_R160.zip` to your working folder. This file contains a folder, `elbow-workbench`, that holds the following items:

   - two geometry files, `elbow_geometry.agdb` and `elbow_geometry.stp`
   - an ANSYS Workbench project archive, `elbow-workbench.wbpz`

   Tip

   The Workbench project archive contains the project as it will be once you have completed all of the steps of the tutorial and is included for reference. If you want to extract the project archive, start Workbench and select the **File** → **Restore Archive...** menu item. You will be prompted with a dialog box to specify a location in which to extract the project and its supporting files. You may choose any convenient location.

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. 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. From the Windows **Start** menu, select **Start > All Programs > ANSYS 16.0** > **Workbench 16.0** to start a new ANSYS Workbench session.

   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

   Depending on which other products you have installed, the analysis systems that appear may differ from those in the figures that follow in this tutorial.

   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**.

   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 1.2: Selecting the Fluid Flow (Fluent) Analysis System in ANSYS Workbench

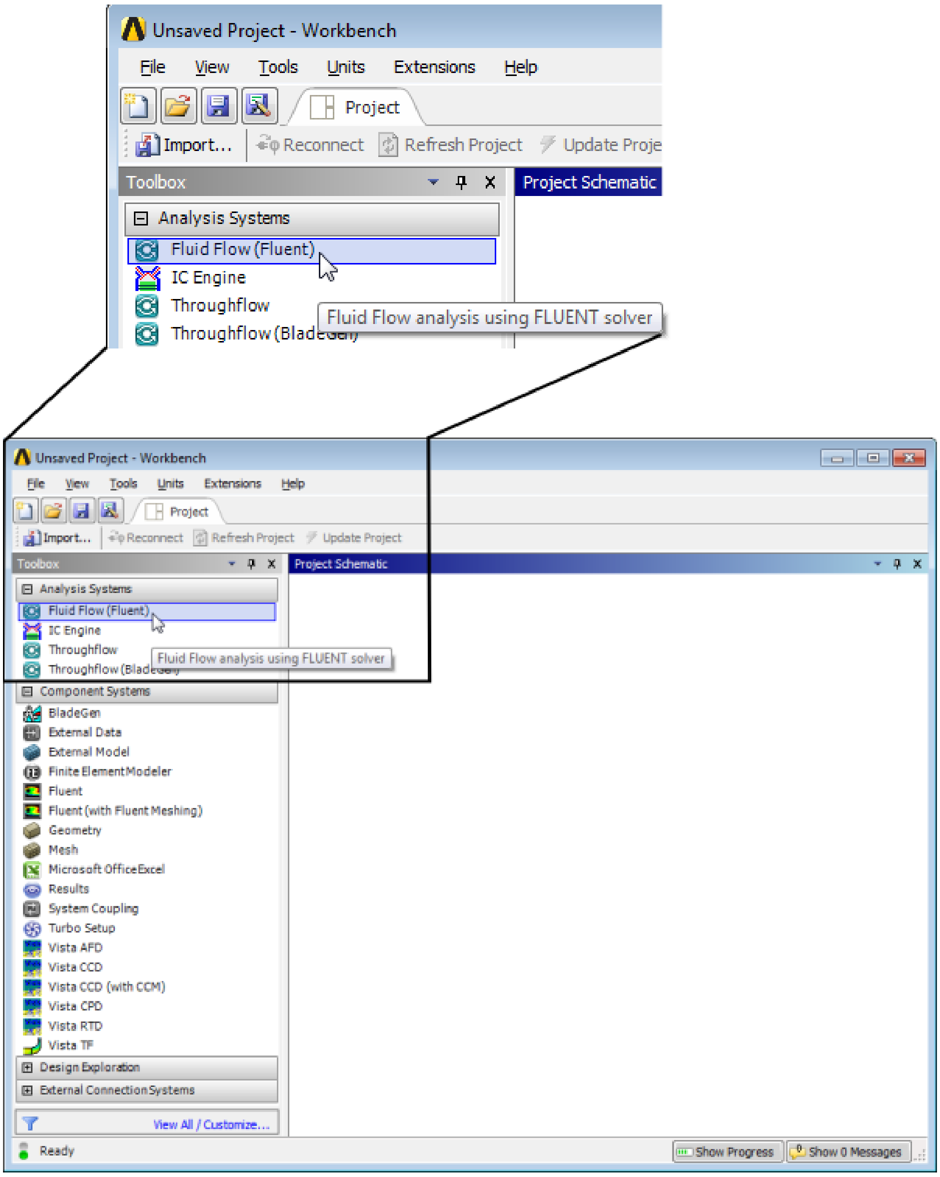
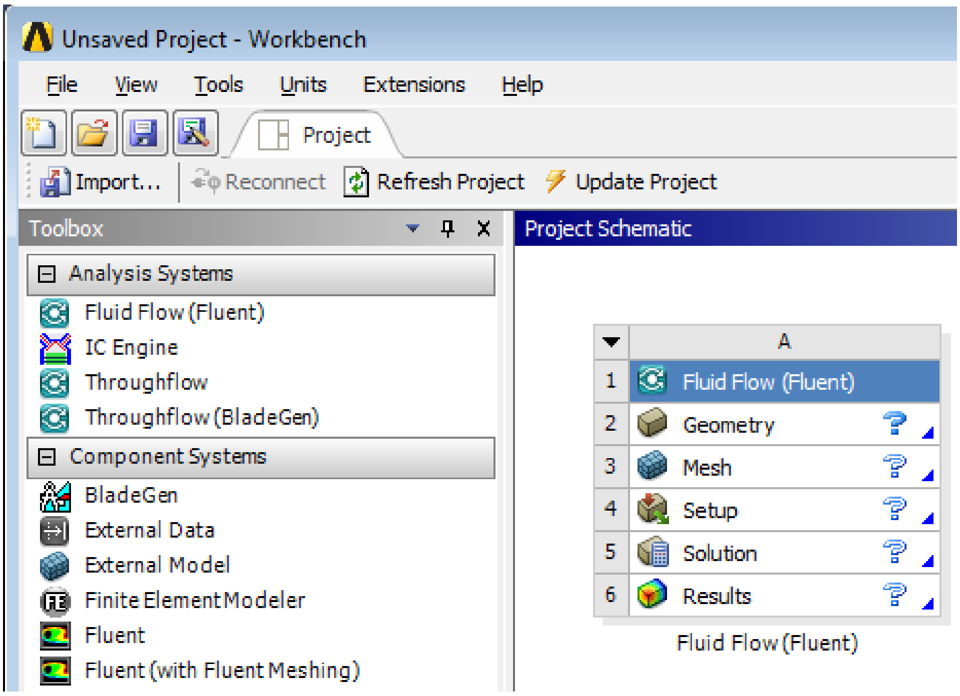


Figure 1.3: ANSYS Workbench with a New Fluent-Based Fluid Flow Analysis System



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 `.wbpj` 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 must be re-generated. 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 must 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, Update Required** ( ) 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, Update Required**.

- **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.

Note

The sizes of the files listed may differ slightly from those portrayed in Figure 1.4: ANSYS Workbench Files View for the Project After Adding a Fluent-Based Fluid Flow Analysis System (p. 9).

From here, you will create the geometry described in Figure 1.1: Problem Specification (p. 3), and later create a mesh and set up a fluid flow analysis for the geometry.

1.4.3. 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 Meshing the Geometry in the ANSYS Meshing Application (p. 20).

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 should select desired system of length units to work from. For the purposes of this tutorial (where you will create the geometry in millimeters and perform the CFD analysis using SI units) set the units to **Millimeter**.

   Units → **Millimeter**

3. Create the geometry.

   The geometry for this tutorial (Figure 1.1: Problem Specification (p. 3)) consists of a large curved pipe accompanied by a smaller side pipe. ANSYS DesignModeler provides various geometry primitives that can be combined to rapidly create geometries such as this one. You will perform the following tasks to create the geometry:

   - Create the bend in the main pipe by defining a segment of a torus.
   - Extrude the faces of the torus segment to form the straight inlet and outlet lengths.
   - Create the side pipe by adding a cylinder primitive.
   - Use the symmetry tool to reduce the model to half of the pipe assembly, thus reducing computational cost.

   a. Create the main pipe:

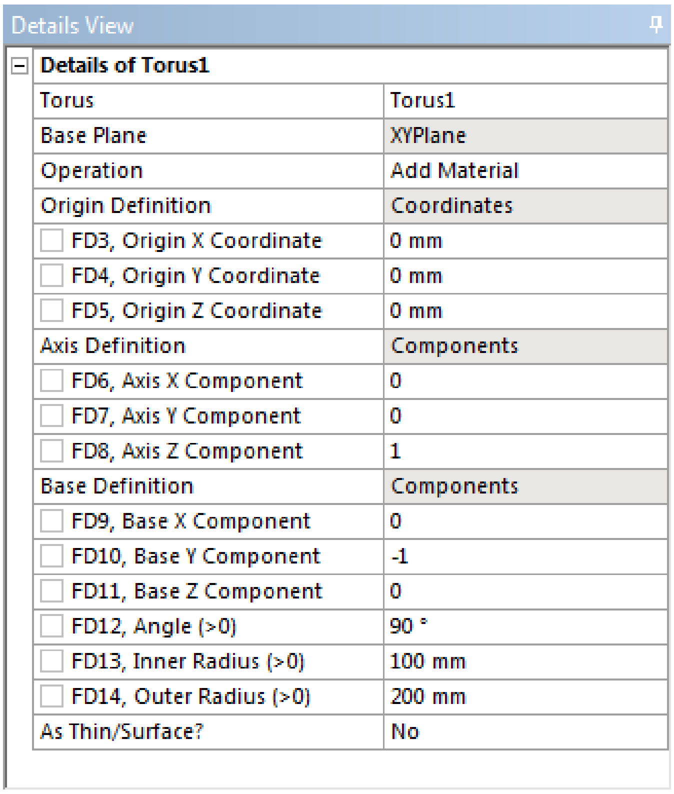
      i. Create a new torus for the pipe bend by choosing the **Create** → **Primitives** → **Torus** menu item from the menubar.

         A preview of the torus geometry will appear in the graphics window. Note that this is a preview and the geometry has not been created yet. First you must specify the parameters of the torus primitive in the next step.

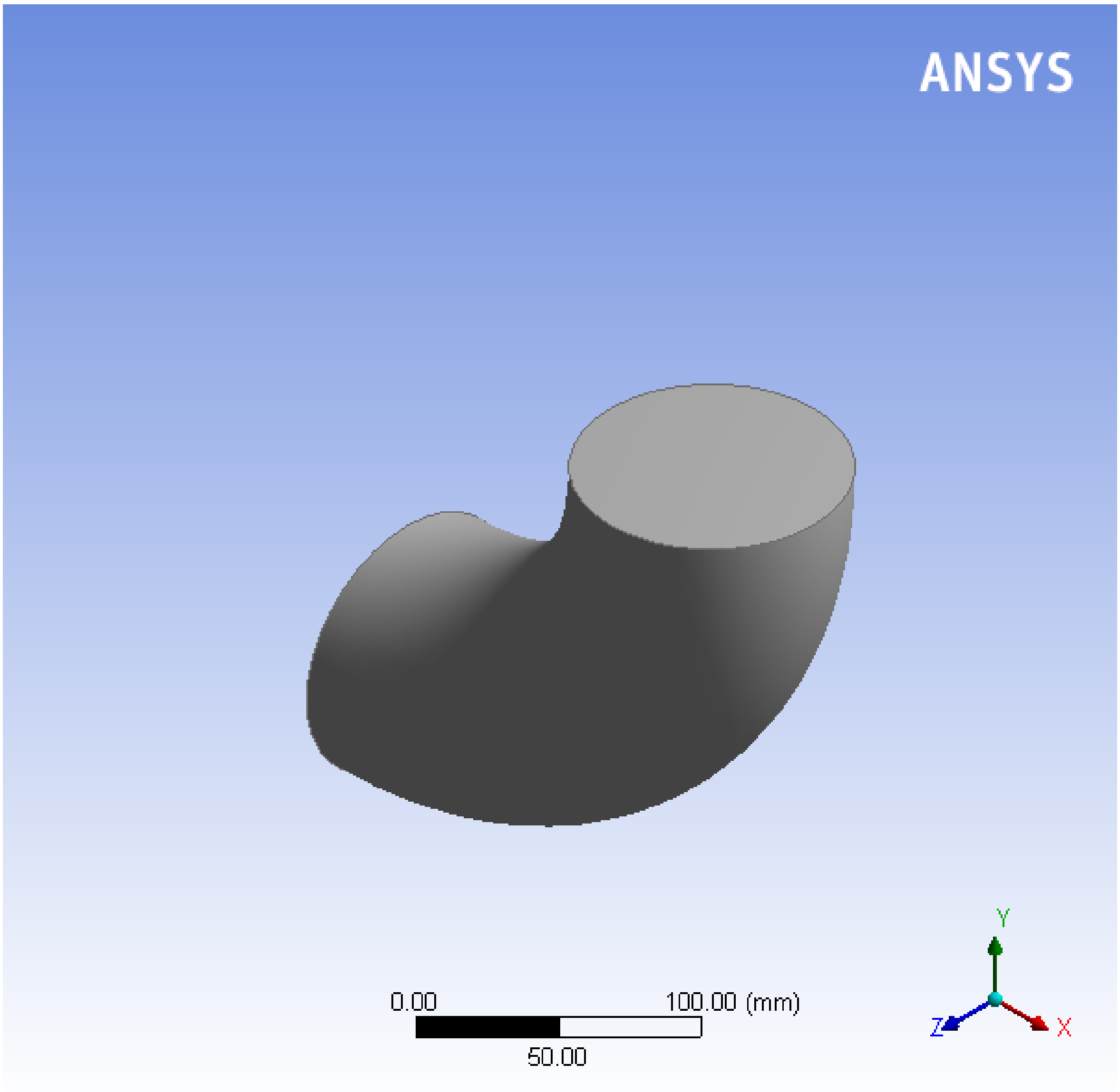
      ii. In the **Details View** for the new torus (**Torus1**), set **Base Y Component** to `-1` by clicking the **1** to the right of **FD10, Base Y Component**, entering `-1`, and pressing **Enter**. This specifies the direction vector from the origin to the center of the circular cross-section at the start of the torus. In the same manner, specify **Angle**; **Inner Radius**; and **Outer Radius** as shown below.

         Note

         Enter only the value without the units of `mm`. They will be appended automatically because you specified the units previously.



iii. To create the torus segment, click the **Generate** button Generate that is located in the ANSYS DesignModeler toolbar.

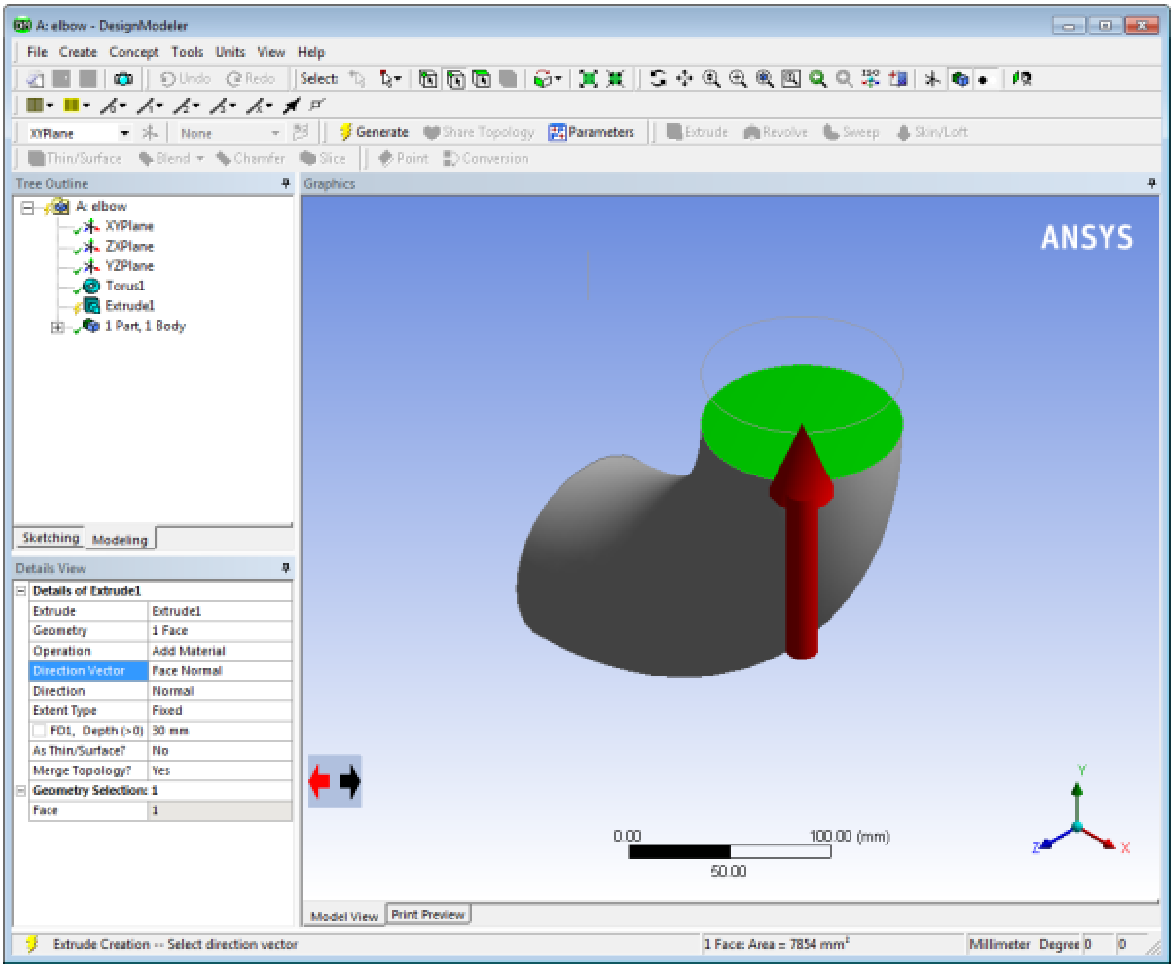


iv. Ensure that the selection filter is set to **Faces**. This is indicated by the **Faces** button  appearing depressed in the toolbar and the appearance of the Face selection cursor,  when you mouse over the geometry.

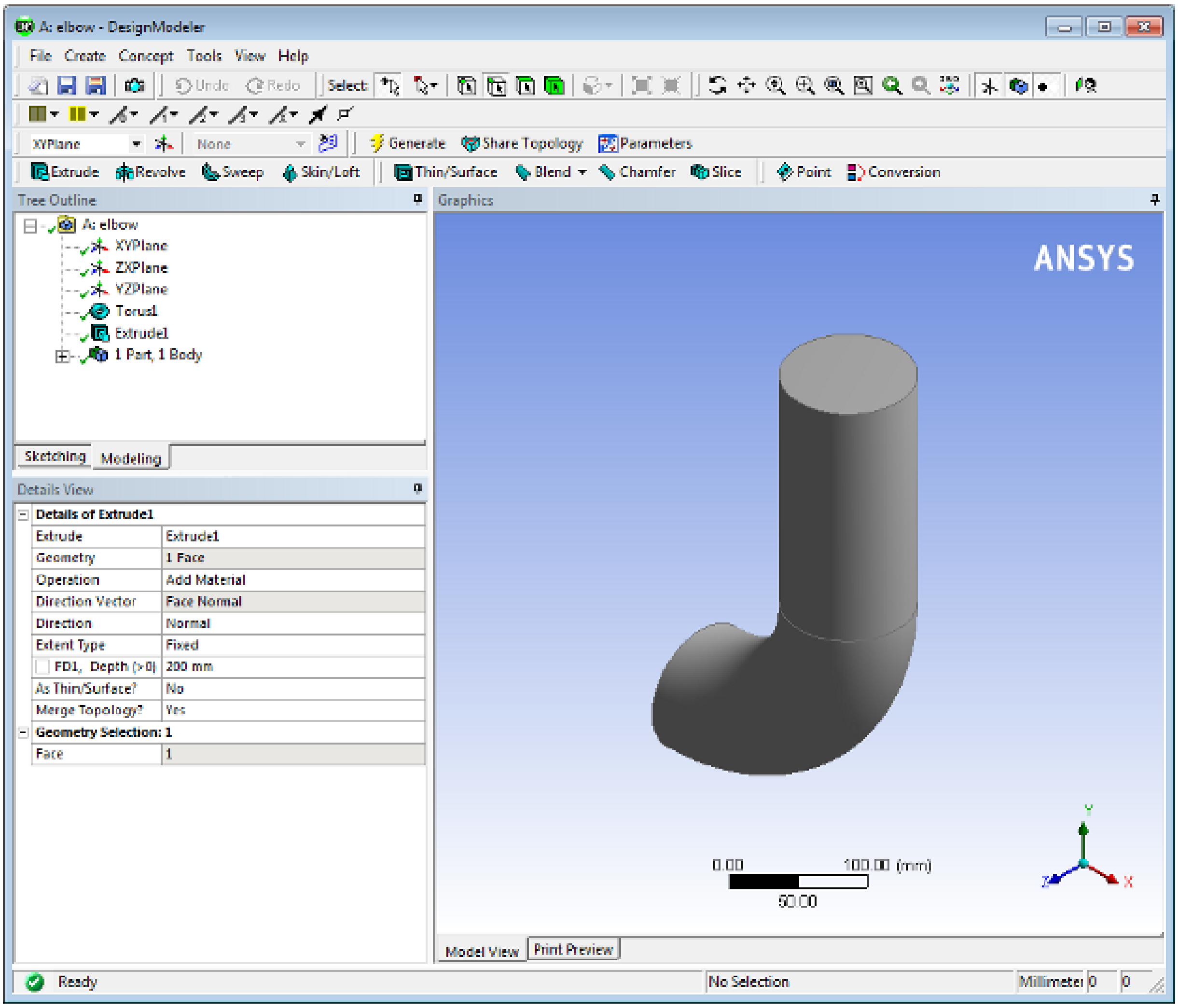
v. Select the top face (in the positive Y direction) of the elbow and click the **Extrude** button  from the **3D Features** toolbar.

vi. In the **Details View** for the new extrusion (**Extrude1**), click **Apply** to the right of **Geometry**. This accepts the face you selected as the base geometry of the extrusion.

vii. Click **None (Normal)** to the right of **Direction Vector**. Again, ensure that the selection filter is set to **Faces**, select the same face on the elbow to specify that the extrusion will be normal to the face and click **Apply**.



viii. Enter 200 for **FD1, Depth (>0)** and click **Generate**.



ix. In a similar manner, create an extrusion of the other face of the torus segment to create the 200 mm inlet extension. You will probably find it helpful to rotate the view so that you can easily select the other face of the bend.

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

Table 1.1: DesignModeler View Manipulation Instructions

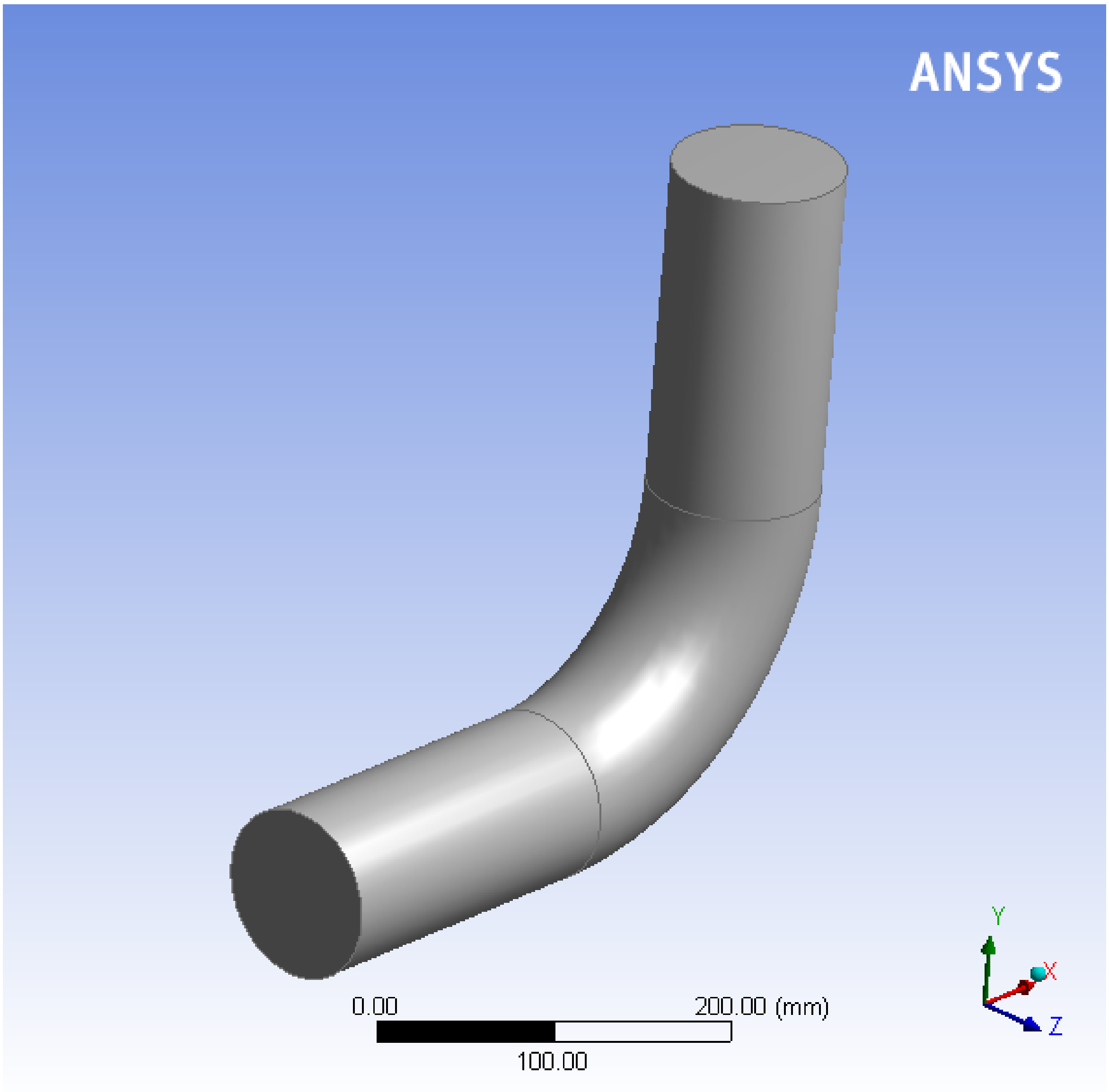
Action	Using Graphics Toolbar Buttons and the Mouse
Rotate view (vertical, horizontal)	After clicking the **Rotate** icon, , press and hold the left mouse button and drag the mouse. Dragging side to side rotates the view about the vertical axis, and dragging up and down rotates the view about the horizontal axis.
Translate or pan view	After clicking the **Pan** 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 **Zoom** icon, , press and hold the left mouse button and drag the mouse up and down to zoom in and out of the view.

Action	Using Graphics Toolbar Buttons and the Mouse
Box zoom	After clicking the **Box Zoom** 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 that 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.

After entering the extrusion parameters and clicking **Generate**, the geometry should appear as in Figure 1.5: Elbow Main Pipe Geometry (p. 15).

Figure 1.5: Elbow Main Pipe Geometry



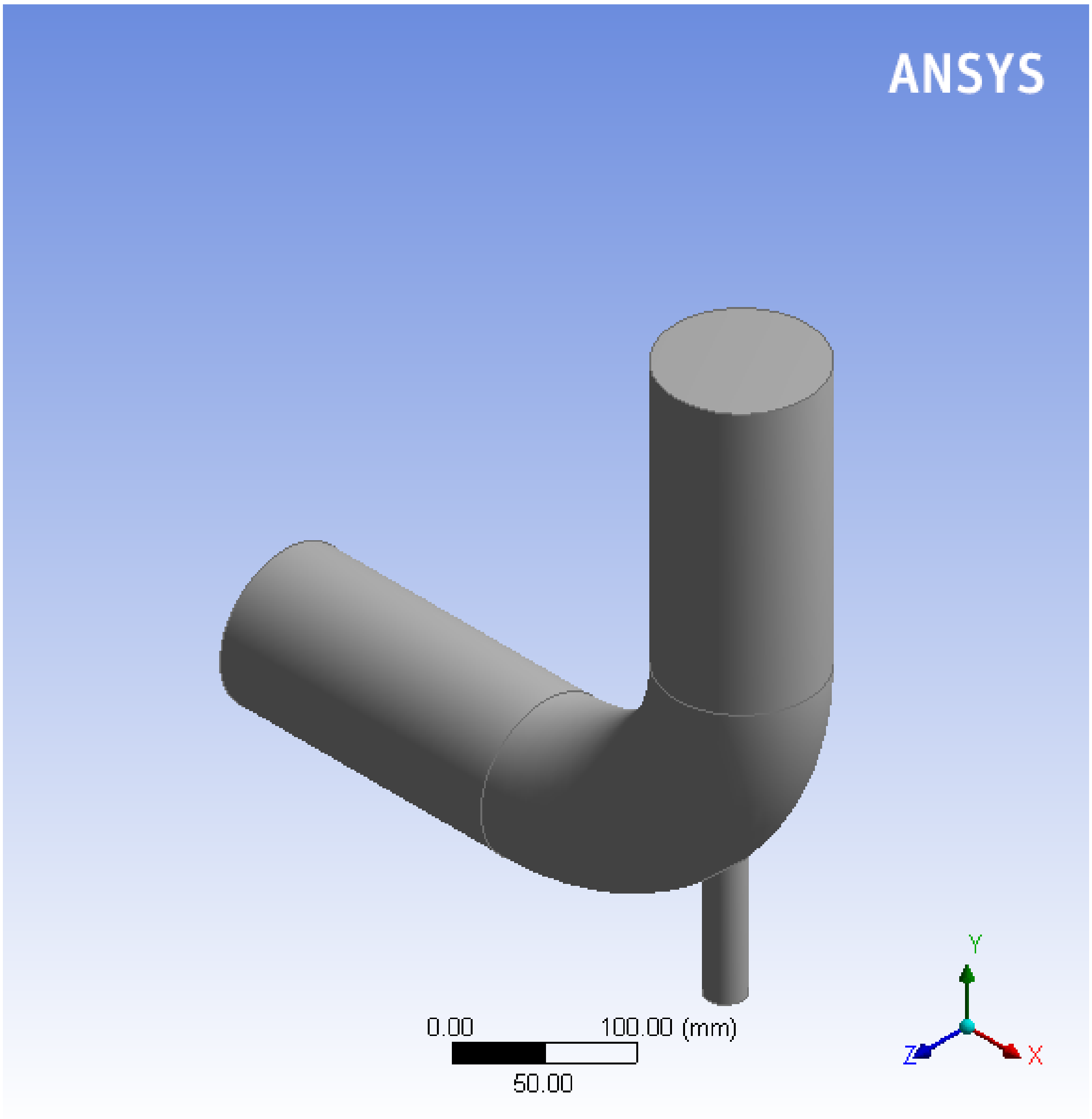
b. Next you will use a cylinder primitive to create the side pipe.

i. Choose **Create** → **Primitives** → **Cylinder** from the menubar.

ii. In the **Details View**, set the parameters for the cylinder as follows and click **Generate**:

Tab	Setting	Value
Details of Cylinder1	**BasePlane**	XYPlane
	FD3, Origin X Coordinate	137.5
	FD4, Origin Y Coordinate	-225
	FD5, Origin Z Coordinate	0
	FD6, Axis X Component	0
	FD7, Axis Y Component	125
	FD8, Axis Z Component	0
	FD10, Radius (>0)	12.5

The Origin Coordinates determine the starting point for the cylinder and the Axis Components determine the length and orientation of the cylinder body.



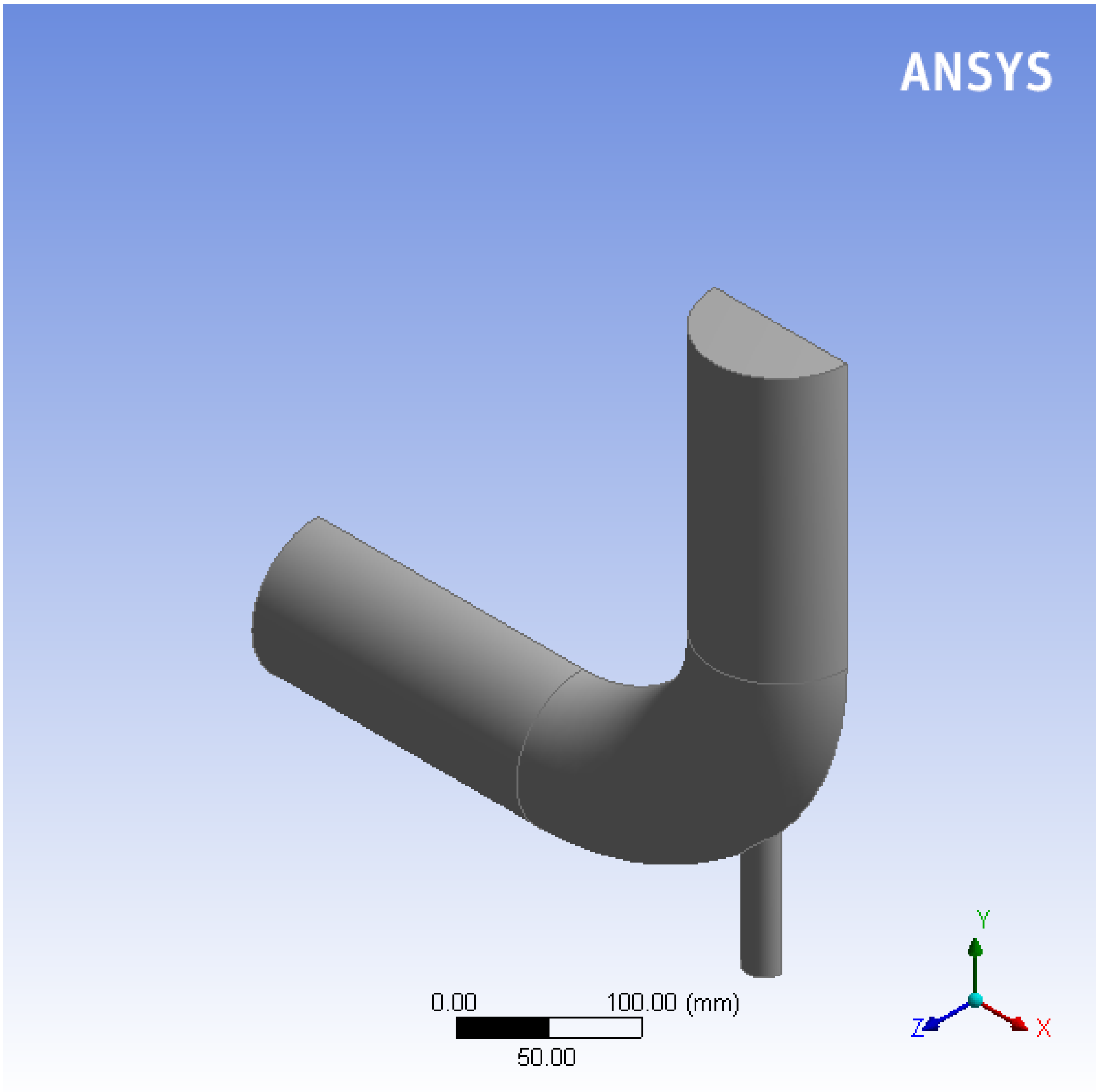
c. The final step in creating the geometry is to split the body on its symmetry plane which will halve the computational domain.

   i. Choose **Tools** → **Symmetry** from the menu bar.

   ii. Select the **XYPlane** in the **Tree Outline**.

   iii. Click **Apply** next to **Symmetry Plane 1** in the **Details** view.

   iv. Click **Generate**.



The new surface created with this operation will be assigned a symmetry boundary condition in Fluent so that the model will accurately reflect the physics of the complete elbow geometry even though only half of it is meshed.

d. 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**.

Details View

Details of Body	
Body	Fluid
Volume	2.5159e+006 mm³
Surface Area	1.7636e+005 mm²
Faces	8
Edges	18
Vertices	12
Fluid/Solid	Fluid
Shared Topology Method	Automatic
Geometry Type	DesignModeler

iv. Click **Generate**.

Tip

In addition to the primitives you used in this tutorial, ANSYS DesignModeler offers a full suite of 2D sketching and 3D solid modeling tools for creating arbitrary geometry. Refer to ANSYS DesignModeler User's Guide for more information.

---

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 1.6: 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

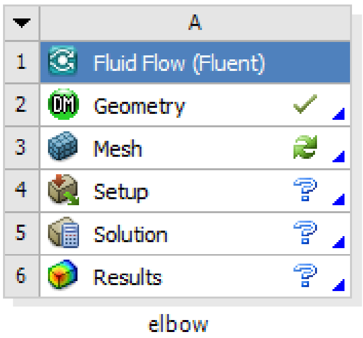
Note the addition of the geometry file (`FFF.agdb`, where `FFF` indicates a Fluent-based fluid flow system) to the list of files. If you had imported the geometry file provided for this tutorial rather than creating the geometry yourself, the `elbow_geometry.agdb` (or the `elbow_geometry.stp`) file would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

1.4.4. Meshing the Geometry in the ANSYS Meshing Application

Now that you have created the mixing elbow geometry, you must generate a computational mesh 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

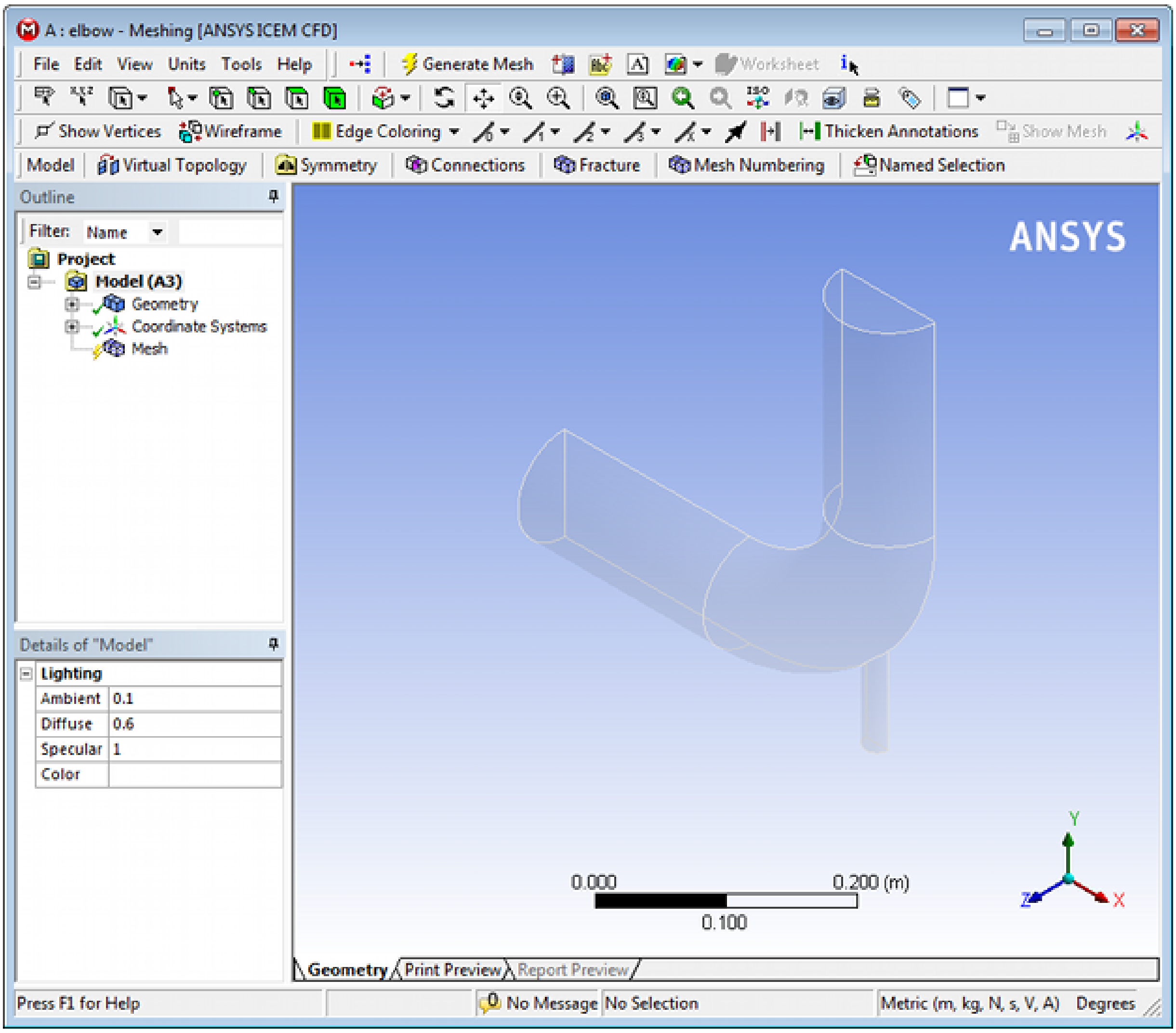
Note the Refresh Required icon ( ) within the **Mesh** cell for the system. This indicates that the state of the cell requires a refresh and that upstream data has changed since the last refresh or update (such as an update to the geometry). Once the mesh is defined, the state of the **Mesh** cell will 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 1.7: The ANSYS Meshing Application with the Elbow Geometry Loaded



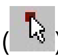
2. Create named selections for the geometry boundaries.

   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

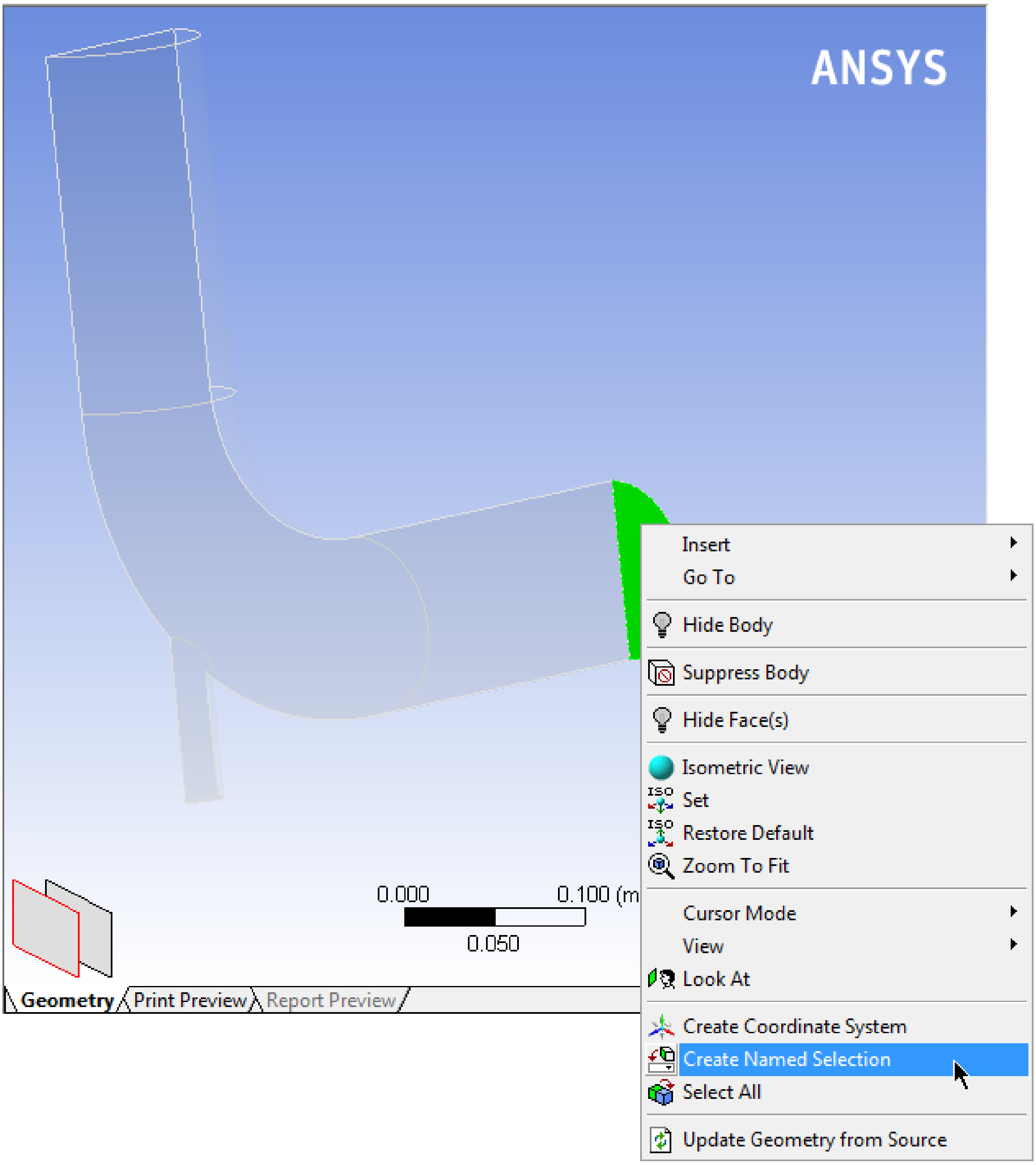
   - Use the Graphics Toolbar buttons and the mouse to manipulate the image until you can easily see the pipe openings and surfaces.

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

   ---

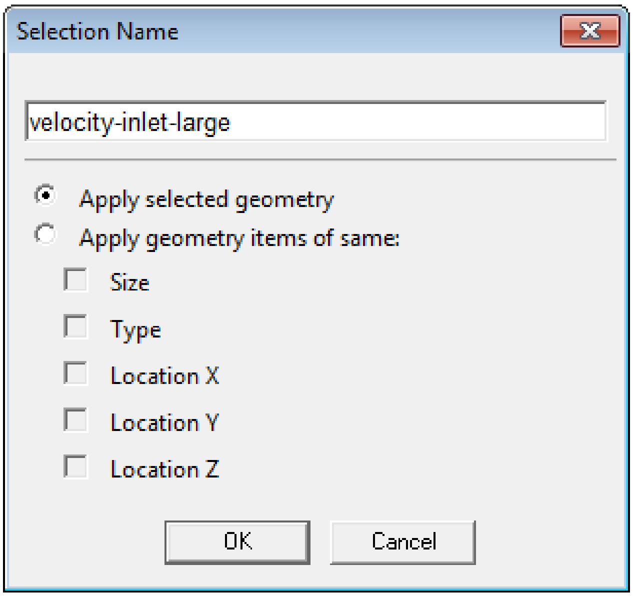
   b. Right-click and select the **Create Named Selection** option.

Figure 1.8: Selecting a Face to Name



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

Figure 1.9: 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 plane (`symmetry`).

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. Create a named selection for the fluid body.

   a. Change the selection filter to **Body** in the **Graphics Toolbar** ()

   b. Click the elbow in the graphics display to select it.

   c. Right-click, select the **Create Named Selection** option and name the body `Fluid`.

   By creating a named selection called `Fluid` *for the fluid body you will ensure that ANSYS Fluent automatically detects that the volume is a fluid zone and treats it accordingly.*

4. Set 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.

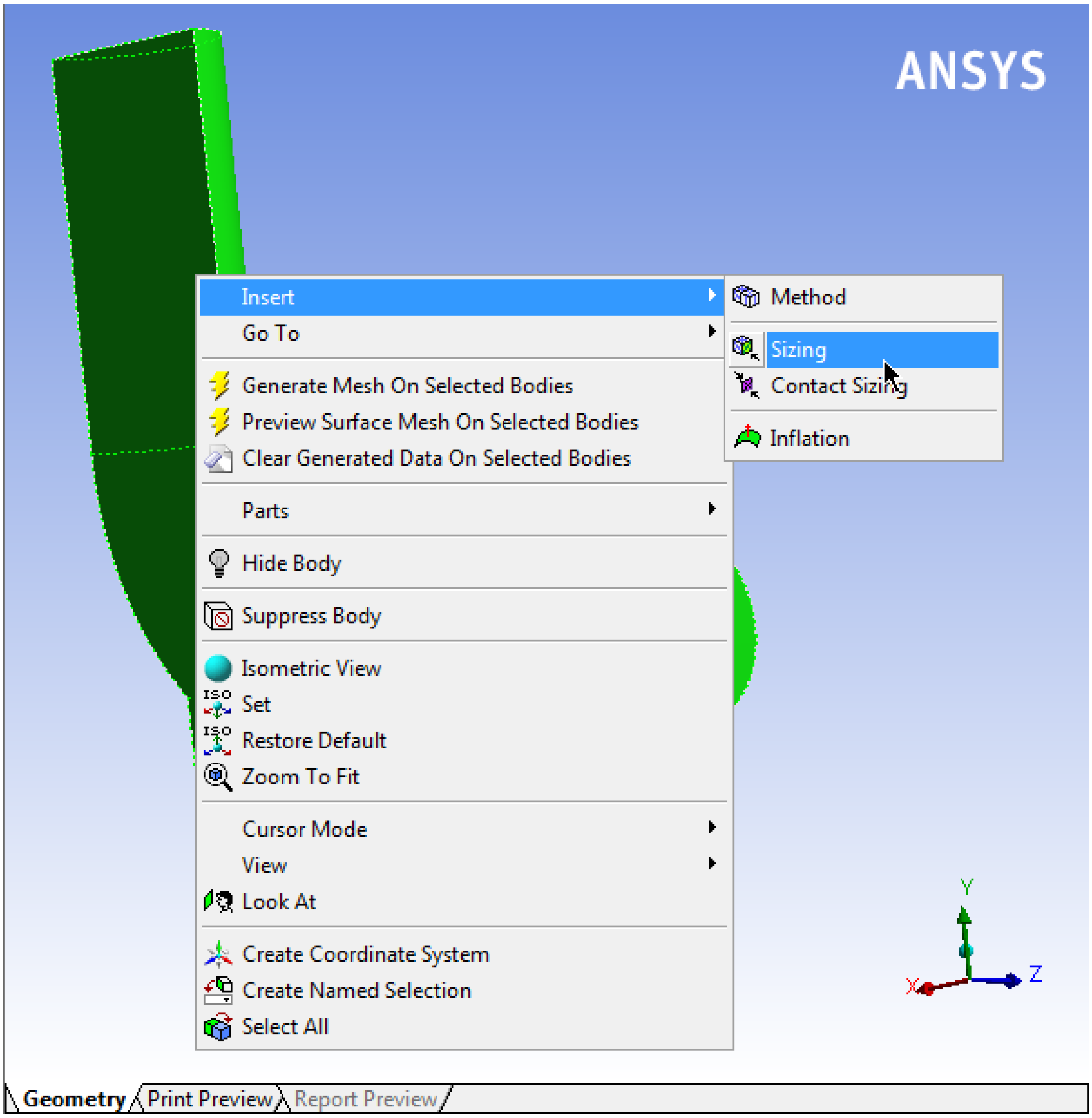
   i. Change **Relevance Center** to **Fine** by clicking on the default value, **Coarse**, and selecting **Fine** from the drop-down list.

   ii. Change **Smoothing** to **High**

c. Add a **Body Sizing** control.

   i. With **Mesh** still selected in the **Outline** tree, click the elbow in the graphics display to select it.

   ii. 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*

iii. Click the new **Body Sizing** control in the **Outline** tree.

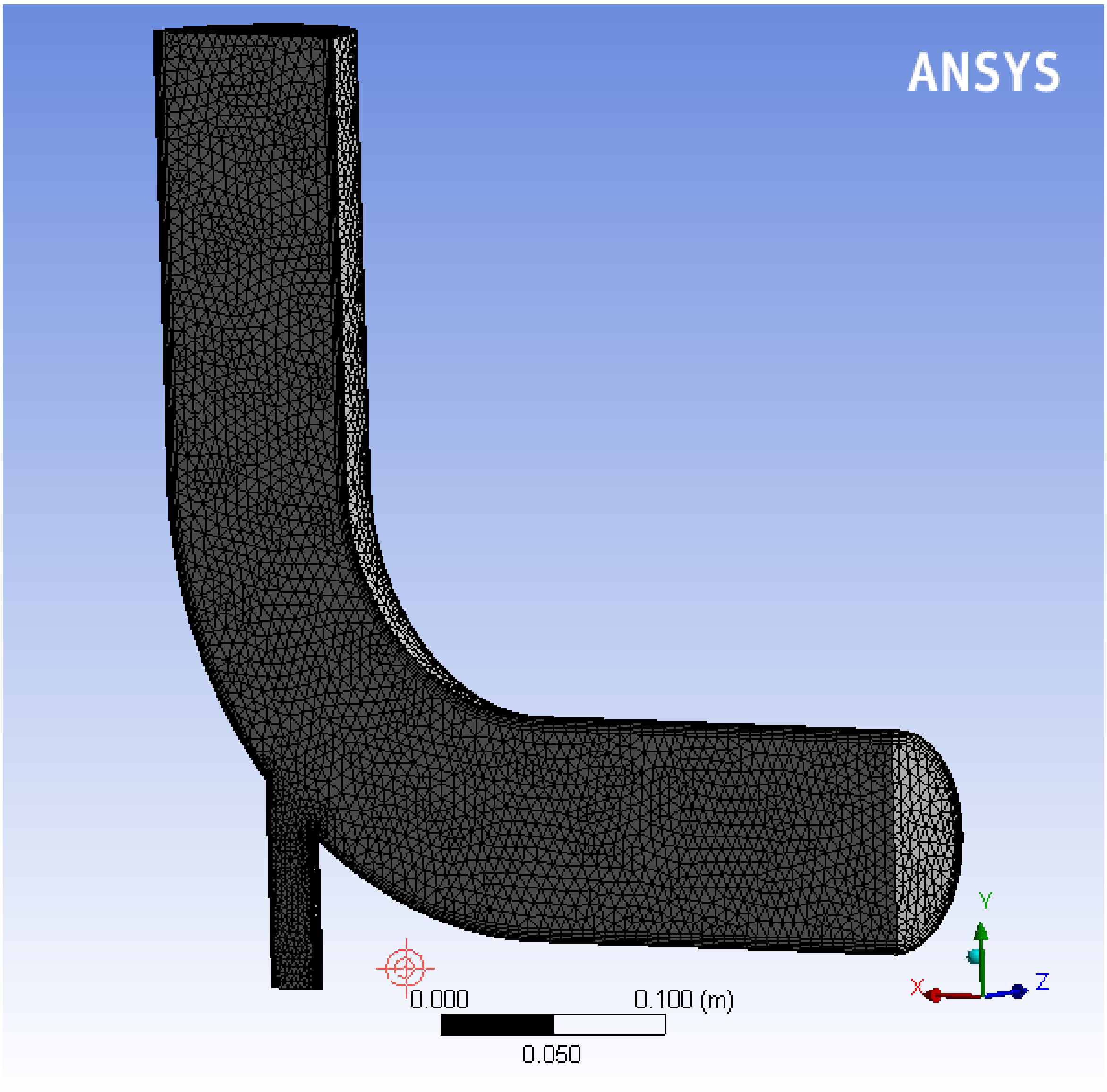
iv. Enter `6e-3` for **Element Size** and press **Enter**.

d. Click again on **Mesh** in the **Outline** view and expand the **Inflation** node in the **Details of "Mesh"** view to reveal additional inflation parameters. Change **Use Automatic Inflation** to **Program Controlled**.

5. Generate the mesh.

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

Figure 1.10: 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.

6. 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.

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

   View → **Files**

   Figure 1.11: 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. 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.

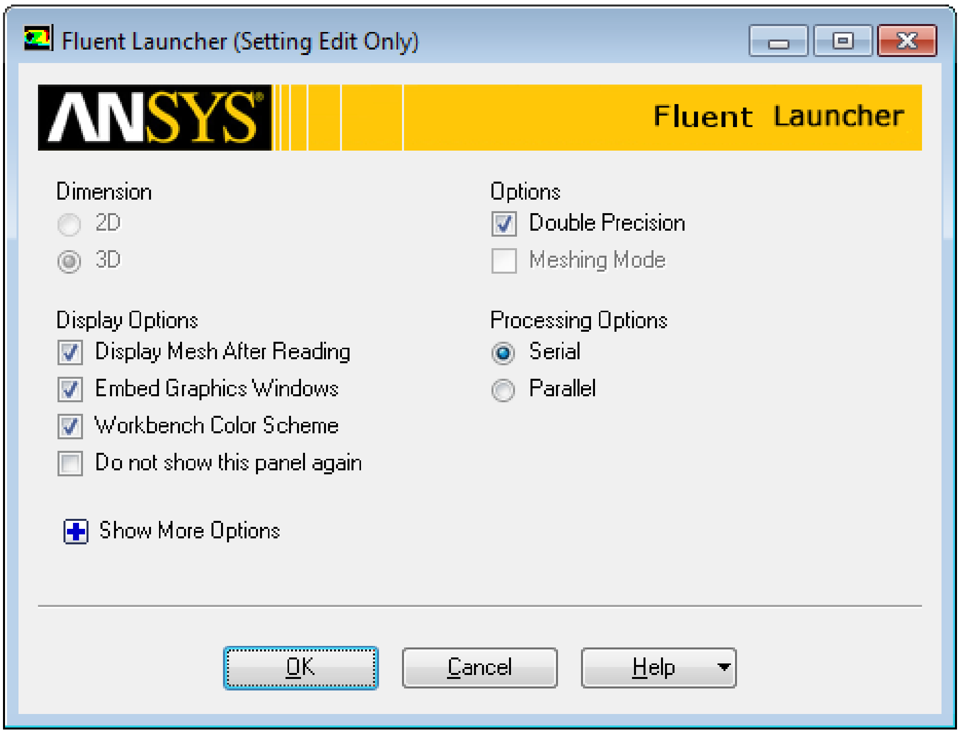
   In the ANSYS Workbench **Project Schematic**, double-click the **Setup** cell in the `elbow` fluid flow analysis system. You can also right-click the **Setup** cell to display the context menu where you can select the **Edit...** option.

   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 1.12: 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. Ensure that **Serial** from the **Processing Options** list is enabled.

ii. Select **Double Precision** under **Options**.

iii. Ensure 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.

Note

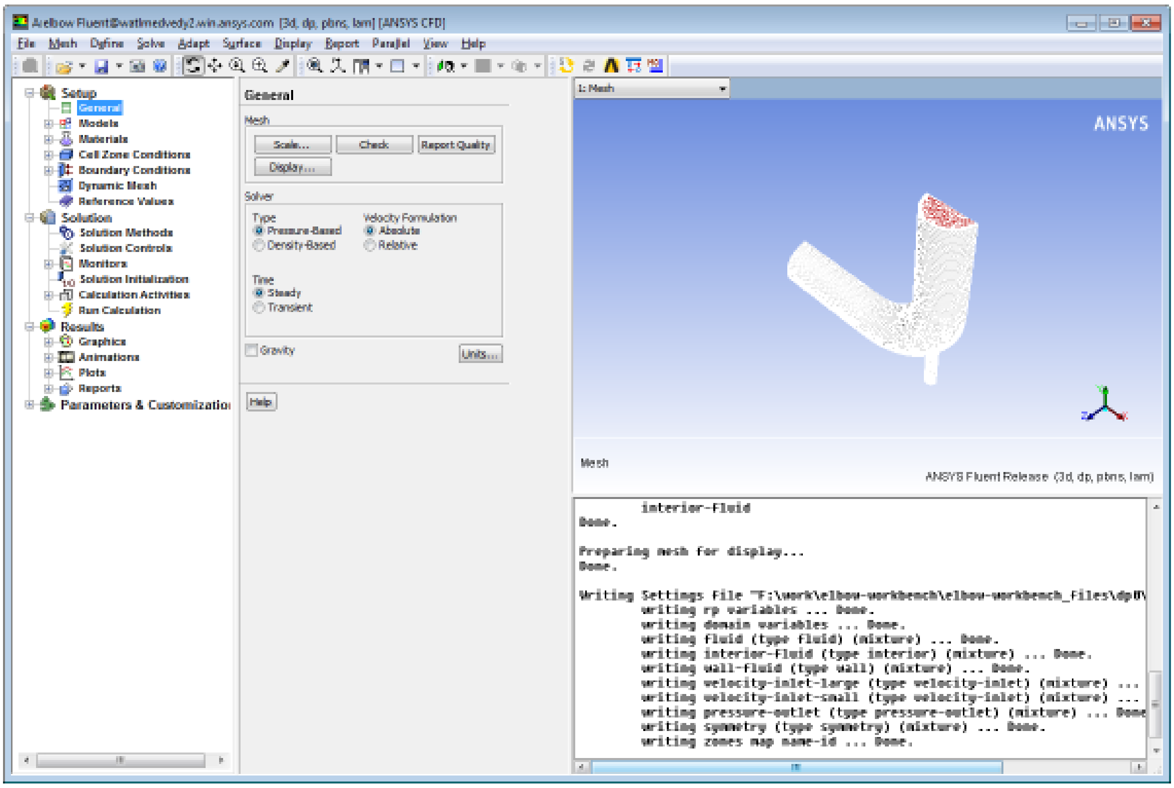
Fluent will retain your preferences for future sessions.

---

b. Click **OK** to launch ANSYS Fluent.

> **Note**
>
> The ANSYS Fluent settings file (`FFF.set`) is written as soon as ANSYS Fluent opens.

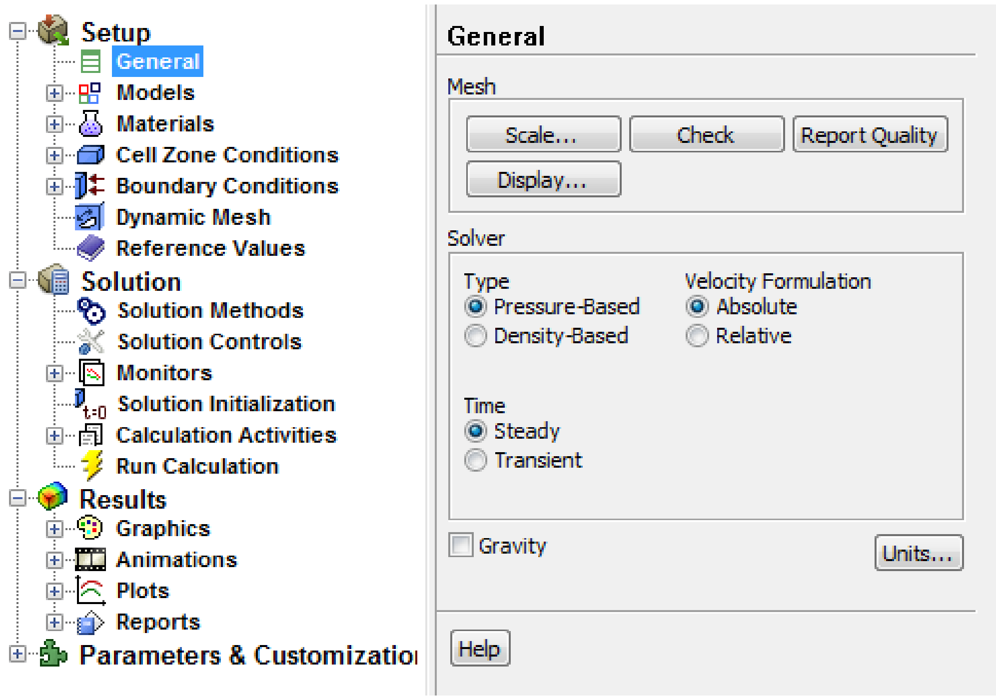
Figure 1.13: The ANSYS Fluent Application



2. Set general settings for the CFD analysis.

   Click **General** in the tree to open the **General** task page where you can perform the mesh-related activities and to choose a solver.

   Setup → **General**



Note

Many of the common setup, solution and results commands are also accessible through the submenus that open by right-clicking the tree items under the relevant tree branches.

a. Change the units for length.

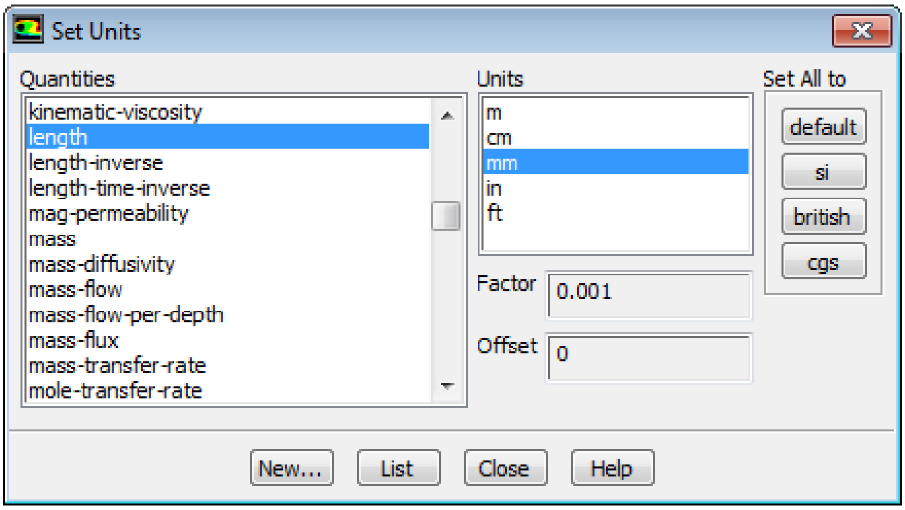
Because you want to specify and view values based on a unit of length in millimeters from within ANSYS Fluent, change the units of length within ANSYS Fluent from meters (the default) to millimeters.

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.

Setup → **General** → **Units...**

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



i. Select **length** in the **Quantities** list.

ii. Select **mm** 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 millimeters rather than meters.

---

b. Check the mesh.

Setup → **General** → **Check**

Note

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

```
 Domain Extents:
   x-coordinate: min (m) = -2.000000e-01, max (m) = 2.000000e-01
   y-coordinate: min (m) = -2.250000e-01, max (m) = 2.000000e-01
   z-coordinate: min (m) = 0.000000e+00, max (m) = 5.000000e-02
 Volume statistics:
   minimum volume (m3): 1.144763e-10
   maximum volume (m3): 5.871098e-08
     total volume (m3): 2.511309e-03
 Face area statistics:
   minimum face area (m2): 2.051494e-07
   maximum face area (m2): 3.429518e-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.

c. Review the mesh quality.

Setup → **General** → **Report Quality**

Note

ANSYS Fluent will report the results of the mesh quality below the results of the mesh check in the console.

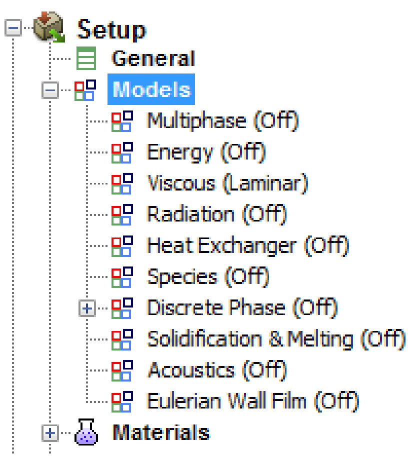
```
Mesh Quality:
Minimum Orthogonal Quality = 1.77384e-01
(Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.)
Maximum Ortho Skew = 7.60448e-01
(Ortho Skew ranges from 0 to 1, where values close to 1 correspond to low quality.)
Maximum Aspect Ratio = 1.92111e+01
```

Note

The quality of the mesh plays a significant role in the accuracy and stability of the numerical computation. Checking the quality of your mesh is, therefore, an important step in performing a robust simulation. Minimum cell orthogonality is an important indicator of mesh quality. Values for orthogonality can vary between 0 and 1 with lower values indicating poorer quality cells. In general, the minimum orthogonality should not be below 0.01 with the average value significantly larger. The high aspect ratio cells in this mesh are near the walls and are a result of the boundary layer inflation applied in the meshing step. For more information about the importance of mesh quality refer to Mesh Quality in the User's Guide.

3. Set up your models for the CFD simulation.

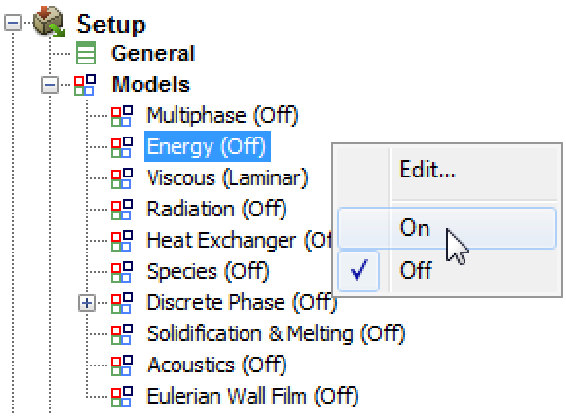
Setup → **Models**



a. Enable heat transfer by activating the energy equation.

In the tree, right-click **Energy** under **Models** and from the submenu that opens, click **On**.

Setup → **Models** → **Energy** **On**

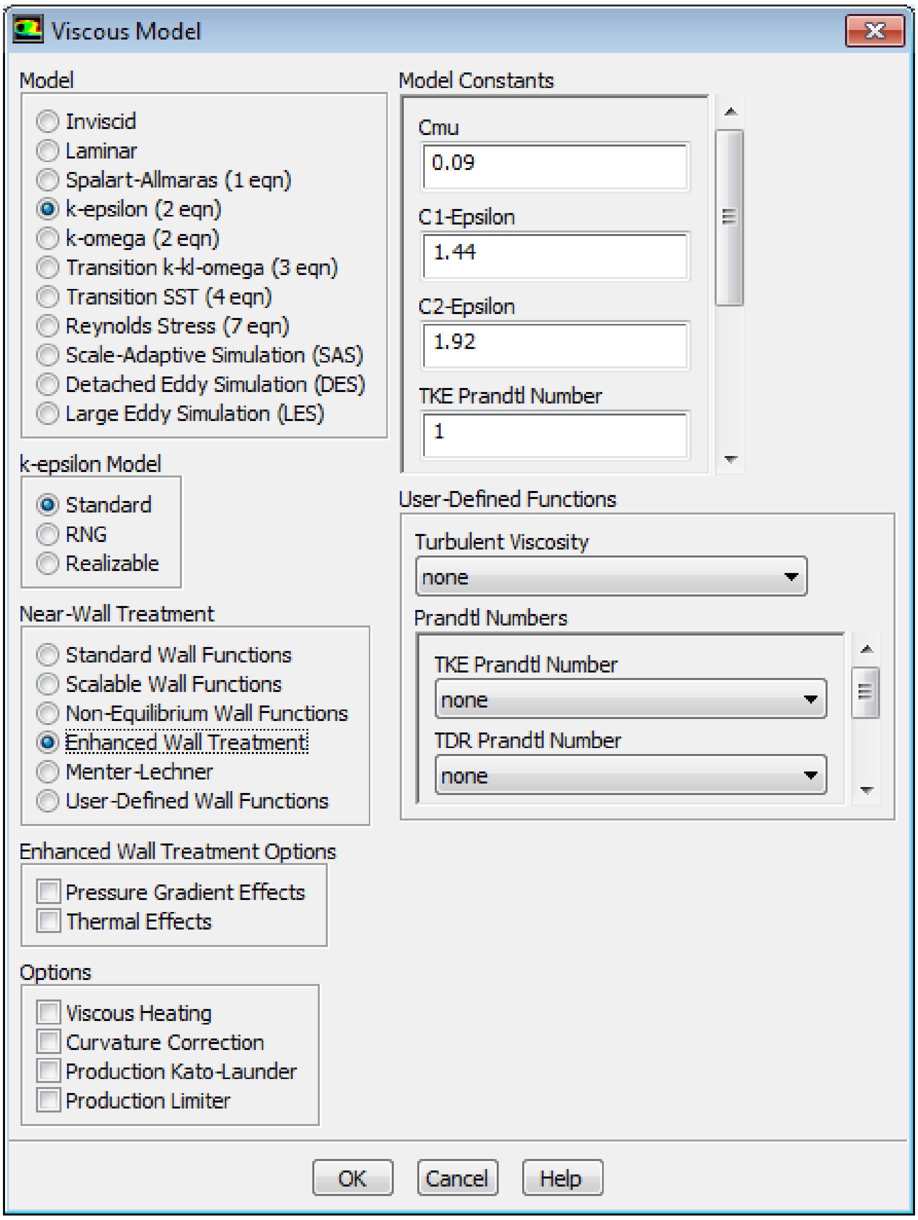


Note

You can also double-click a tree item and enable the energy equation in the **Energy** dialog box.

b. Enable the k- ε turbulence model.

Setup → **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

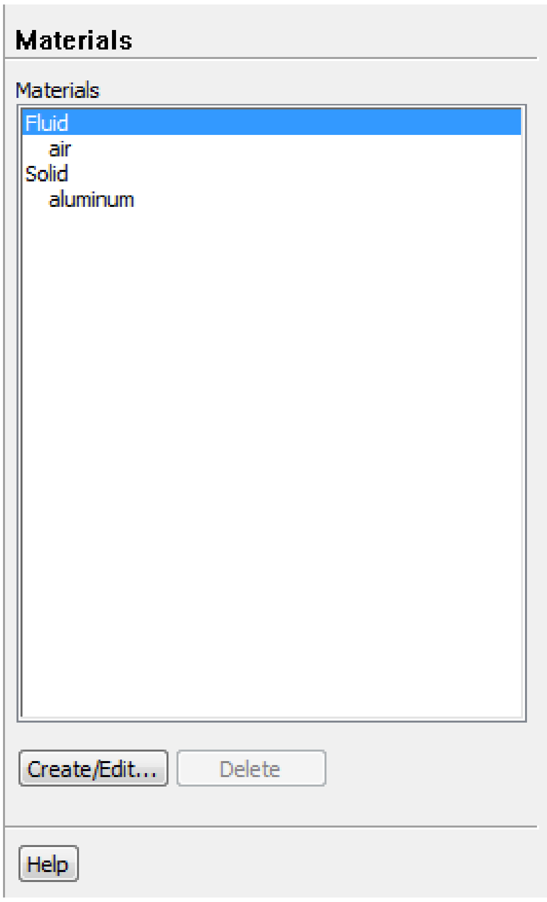
The default Standard Wall Functions are generally applicable if the first cell center adjacent to the wall has a y+ larger than 30. In contrast, the Enhanced Wall

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 refer to Setting Up the k-ε 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.

Setup → **Materials**



a. Create a new material called **water** using the **Create/Edit Materials** dialog box (Figure 1.14: The Create/Edit Materials Dialog Box (p. 36)).

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

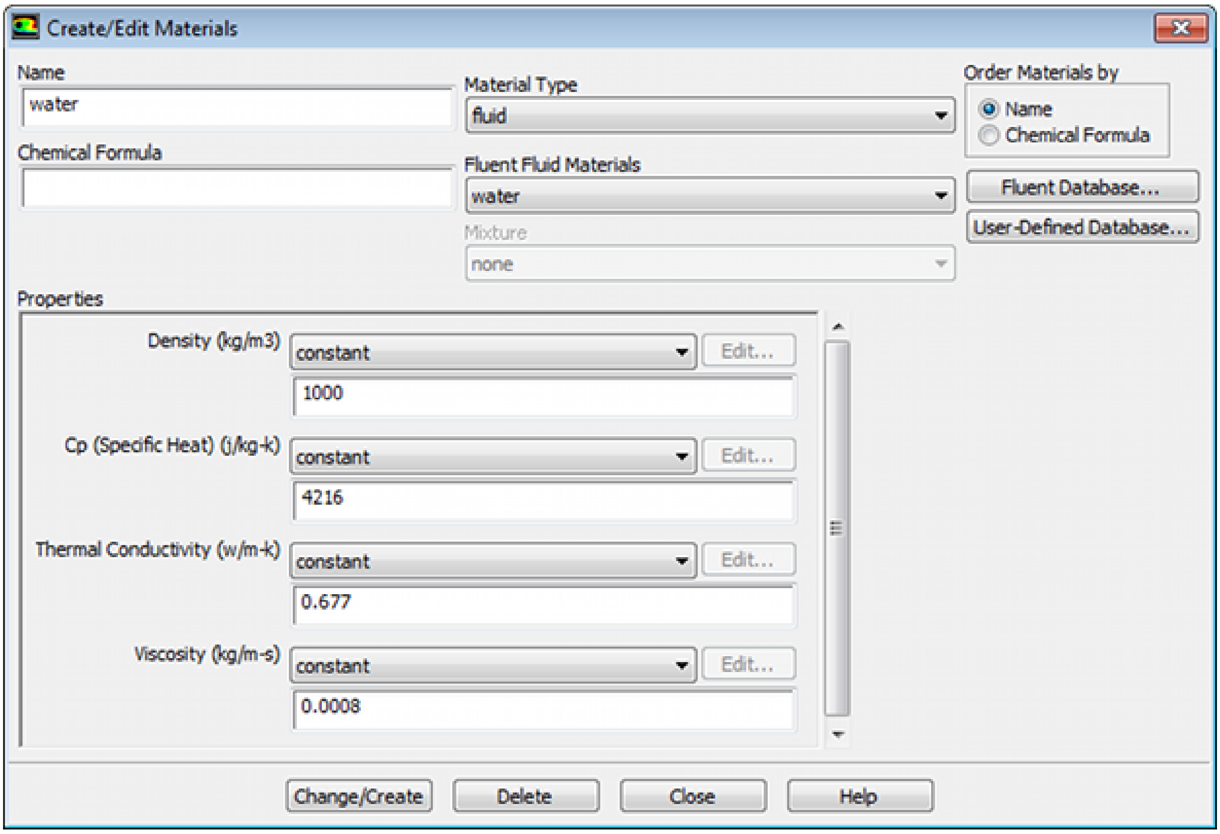
i. Type `water` for **Name**.

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

Property	**Value**
Density	1000 kg/m^3

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

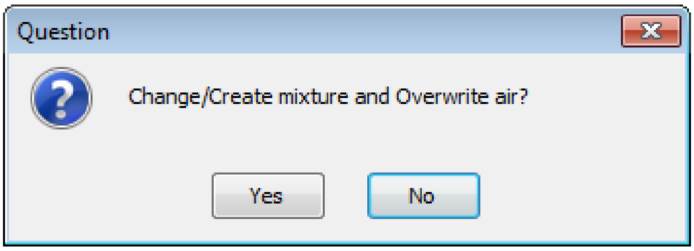
Figure 1.14: The Create/Edit Materials Dialog Box



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 < l >)** 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. Ensure 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.

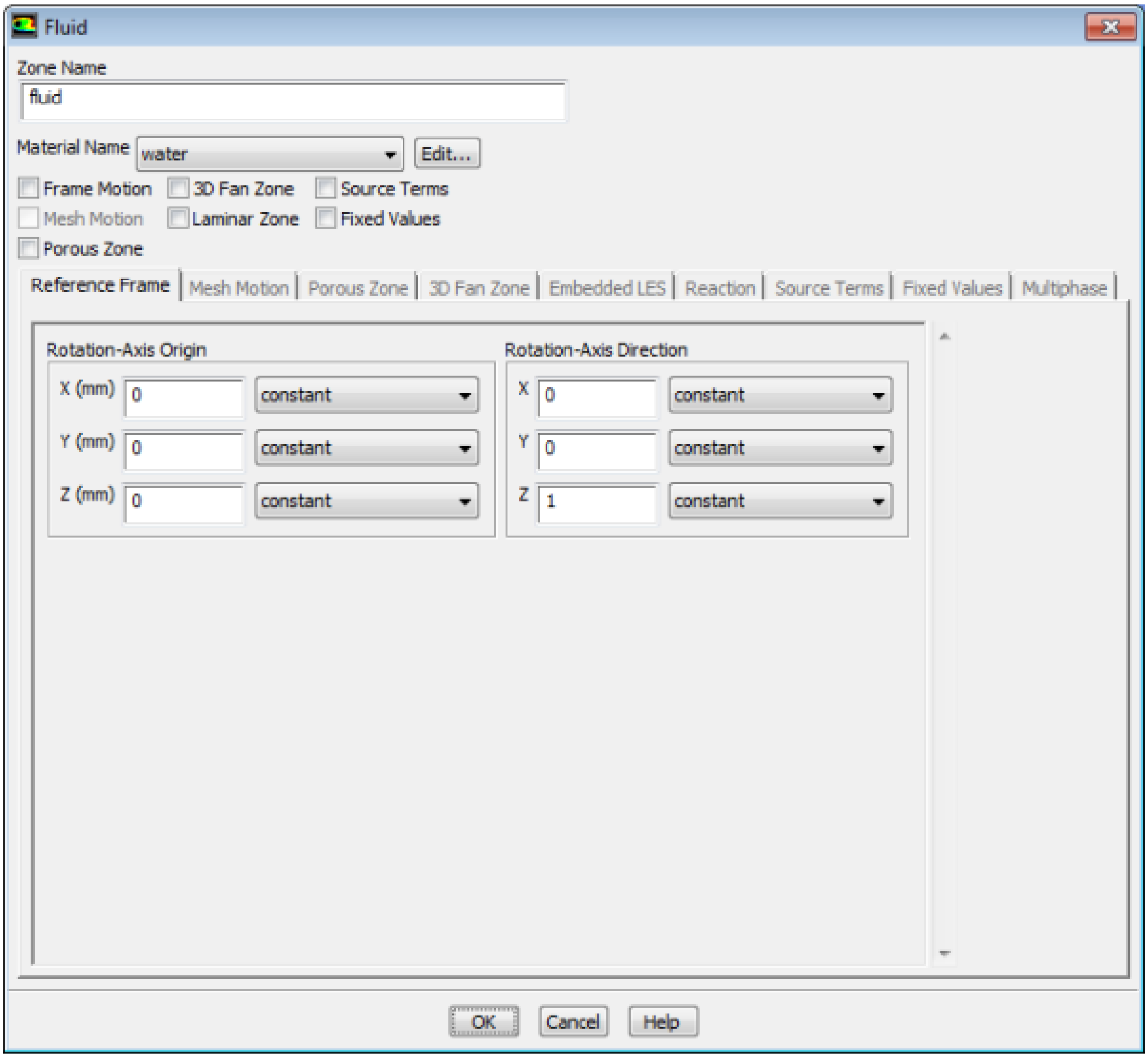
Setup → **Cell Zone Conditions**

a. Set the cell zone conditions for the fluid zone.

i. In the **Cell Zone Conditions** task page, in the **Zone** list, select **fluid** and click **Edit...** to open the **Fluid** dialog box.

Note

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

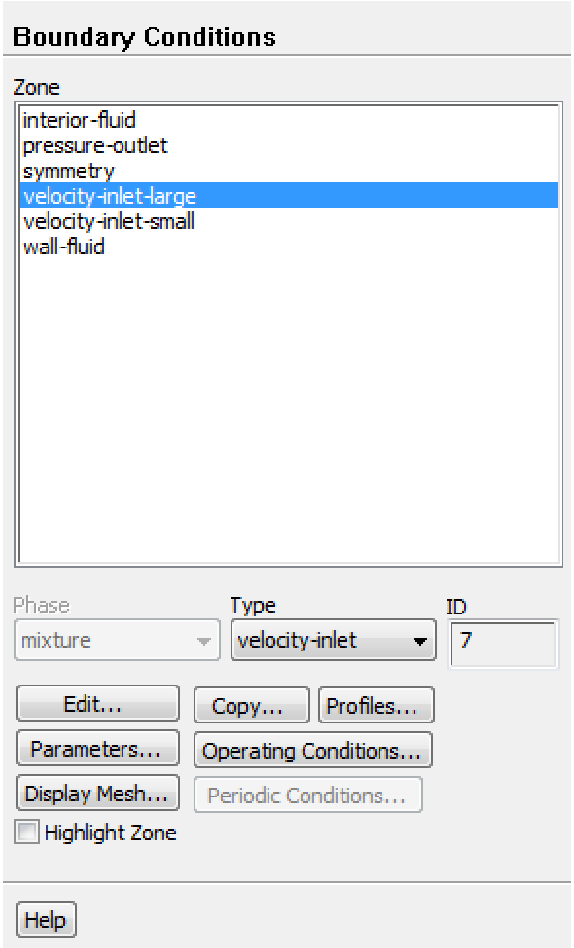


ii. In the **Fluid** dialog box, select **water** from the **Material Name** drop-down list.

iii. Click **OK** to close the **Fluid** dialog box.

6. Set up the boundary conditions for the CFD analysis.

Setup → **Boundary Conditions**



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

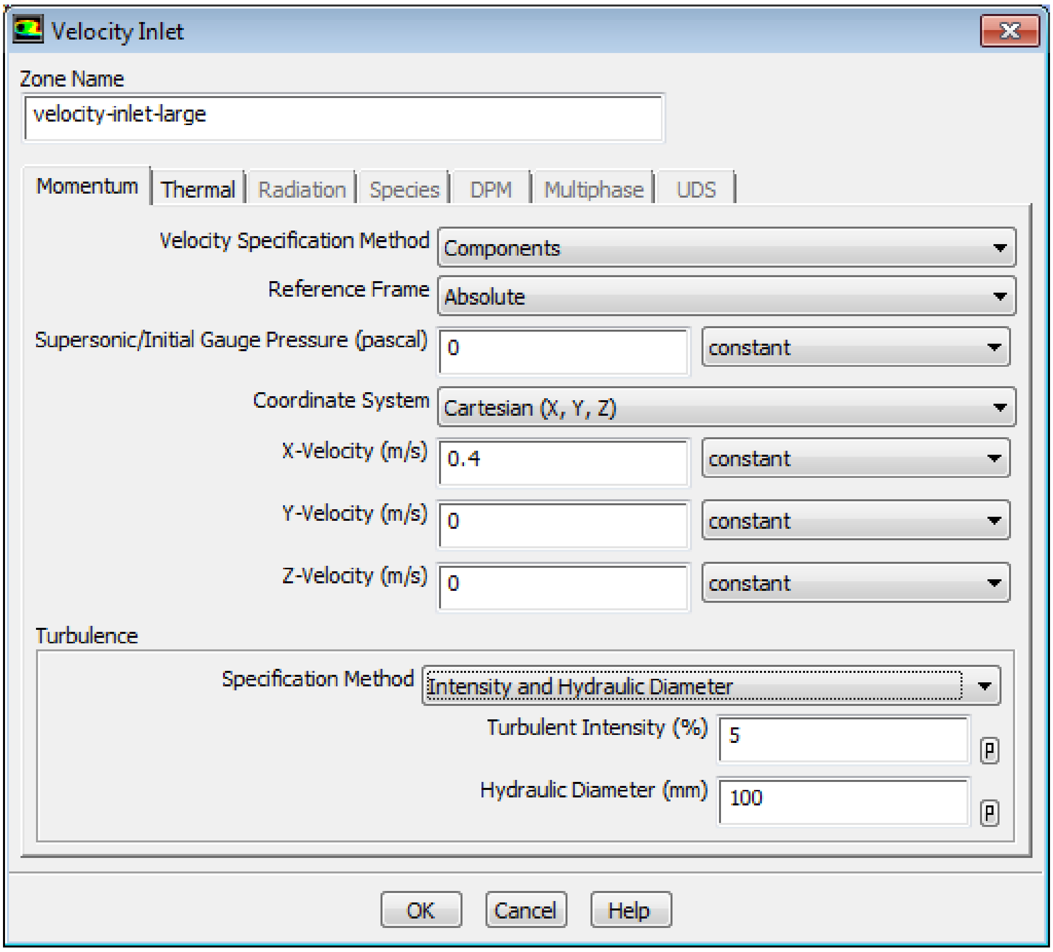
Setup → **Boundary Conditions** → **velocity-inlet-large** → **Edit...**

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 m/s for **X-Velocity**.

iii. Retain the default value of 0 m/s for both **Y-Velocity** and **Z-Velocity**.

iv. In the **Turbulence** group box, from the **Specification Method** drop-down list, select **Intensity and Hydraulic Diameter**.

v. Retain the default of 5 $\%$ for **Turbulent Intensity**.

vi. Enter 100 mm for **Hydraulic Diameter**.

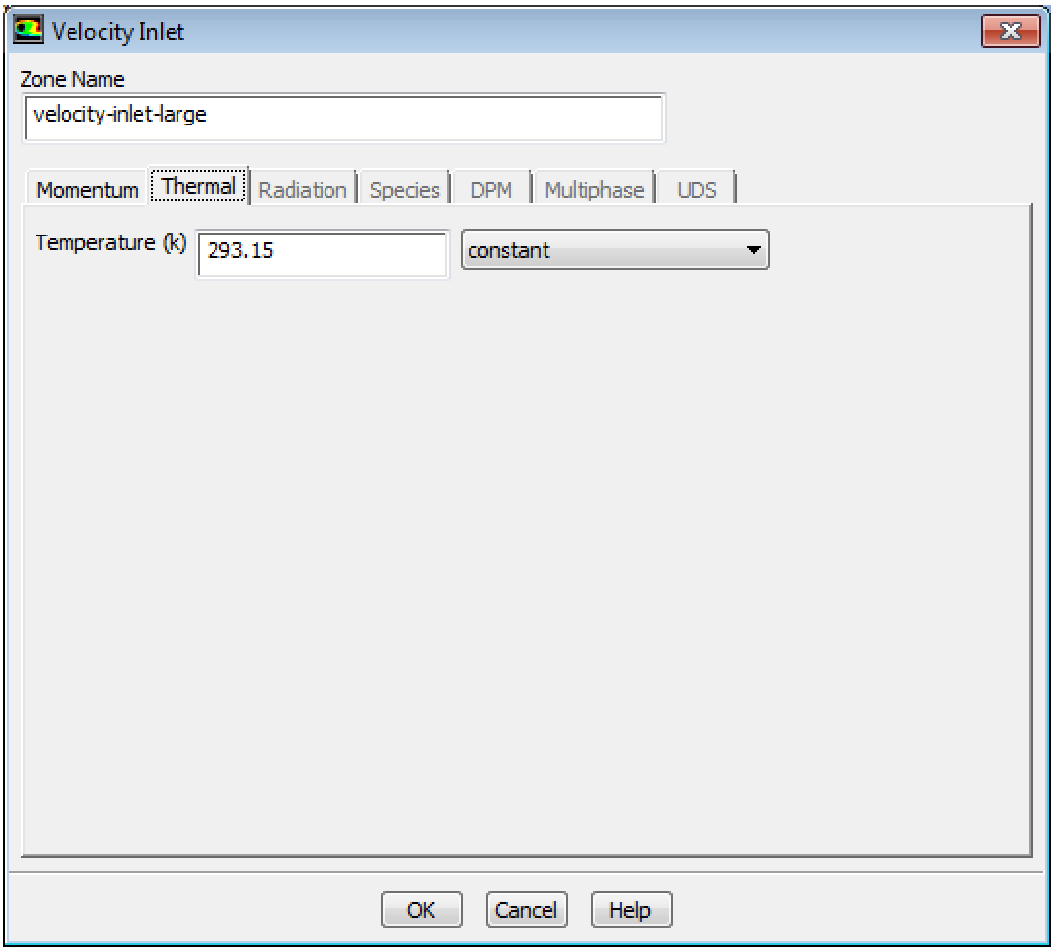
Note

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:

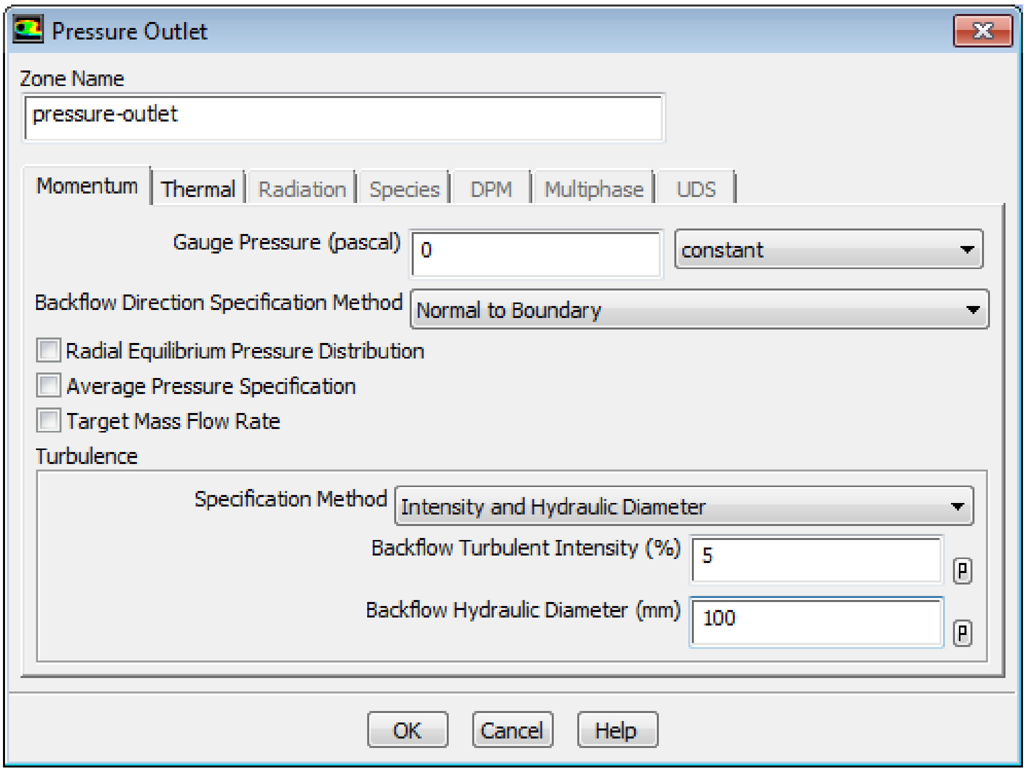
Setup → **Boundary Conditions** → **velocity-inlet-small** → **Edit...**

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

Velocity Specification Method	Components
Specification Method	Intensity & Hydraulic Diameter
Turbulent Intensity	5%
Hydraulic Diameter	25 mm
Temperature	313.15 K

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

Setup → **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.

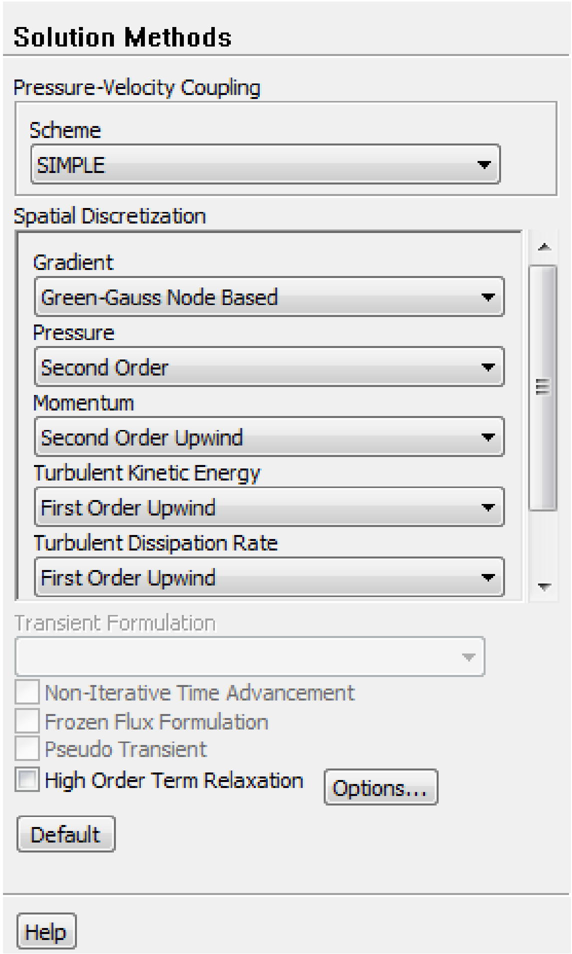
7. Set up solution parameters for the CFD simulation.

Note

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

a. Change the Gradient method.

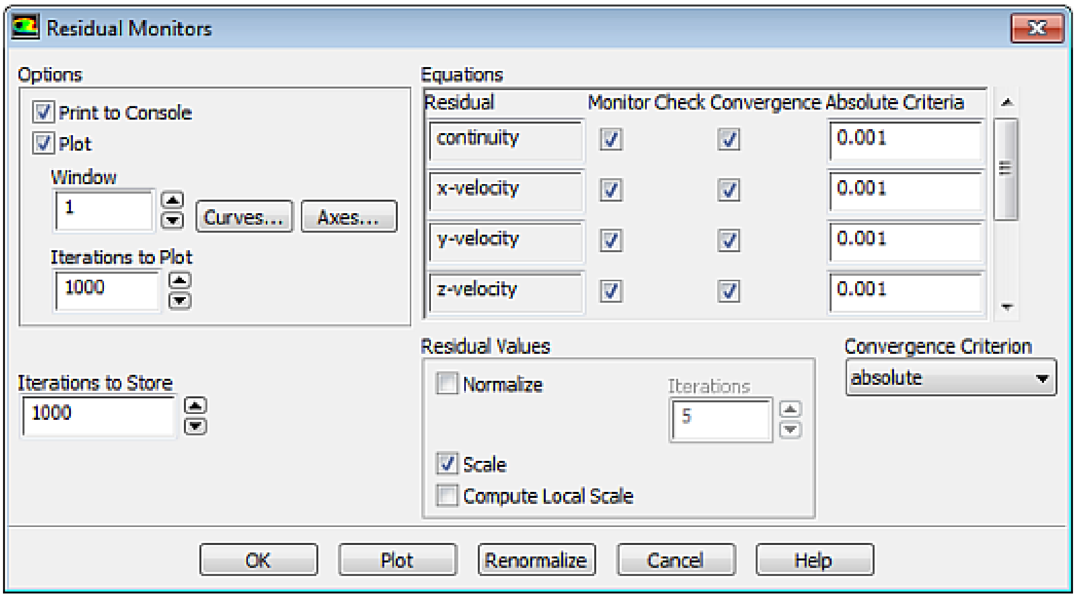
Solution → **Solution Methods**



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.

b. Examine the convergence criteria for the equation residuals.

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



i. Ensure 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

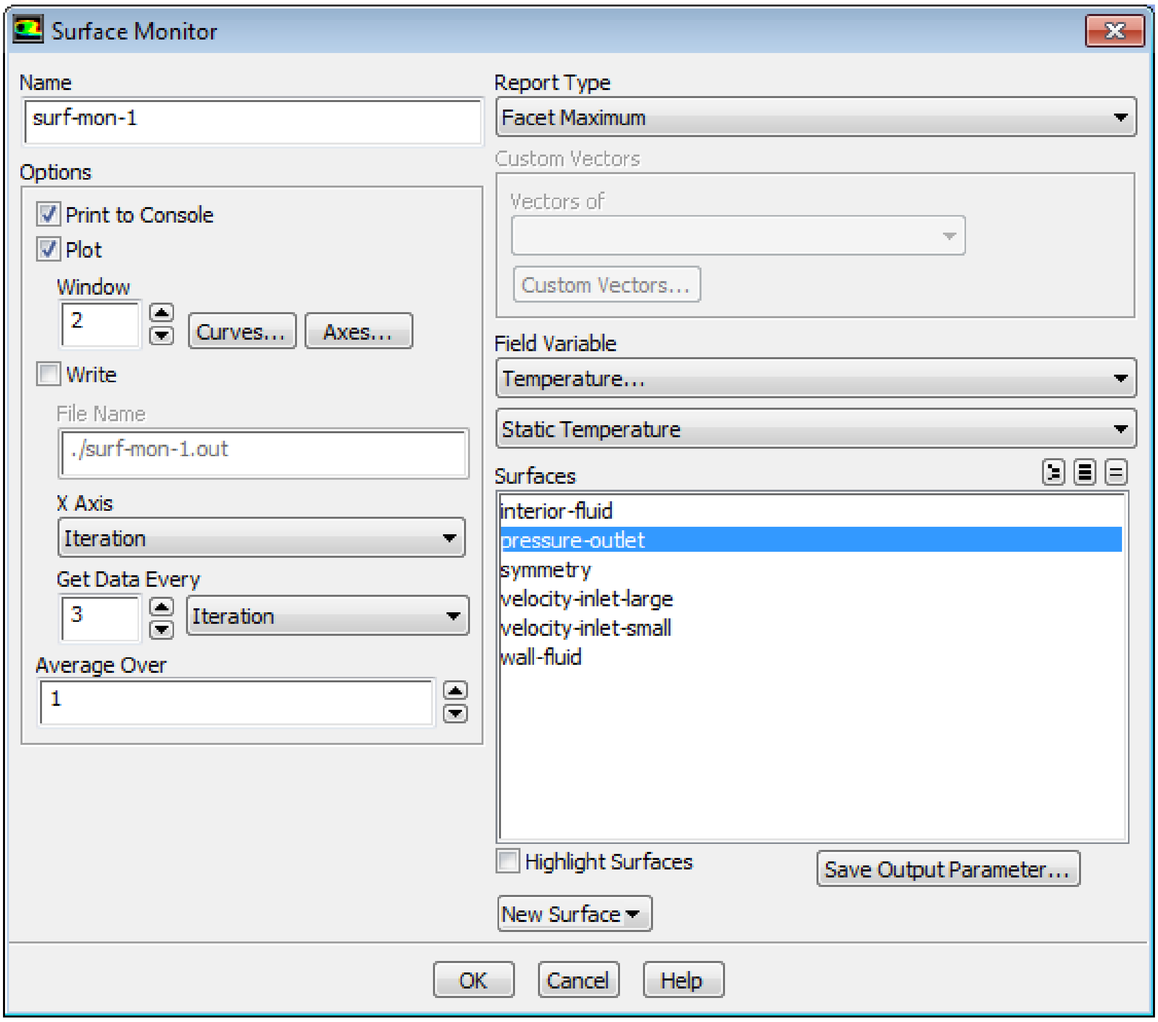
By default, all variables will be monitored and checked by ANSYS Fluent as a means to determine the convergence of the solution.

---

c. 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.

Solution → **Monitors** → **Surface** **New...**



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 **Facet Maximum** from the **Report Type** drop-down list.

v. Select **Temperature...** and **Static Temperature** from the **Field Variable** drop-down lists.

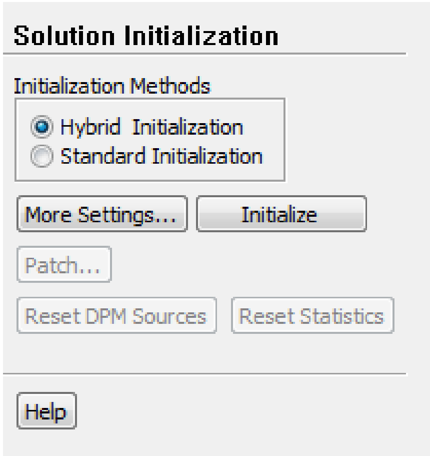
vi. Select **pressure-outlet** from the **Surfaces** selection list.

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 under the **Surface** tree item.*

d. Initialize the flow field.

Solution → **Solution Initialization**

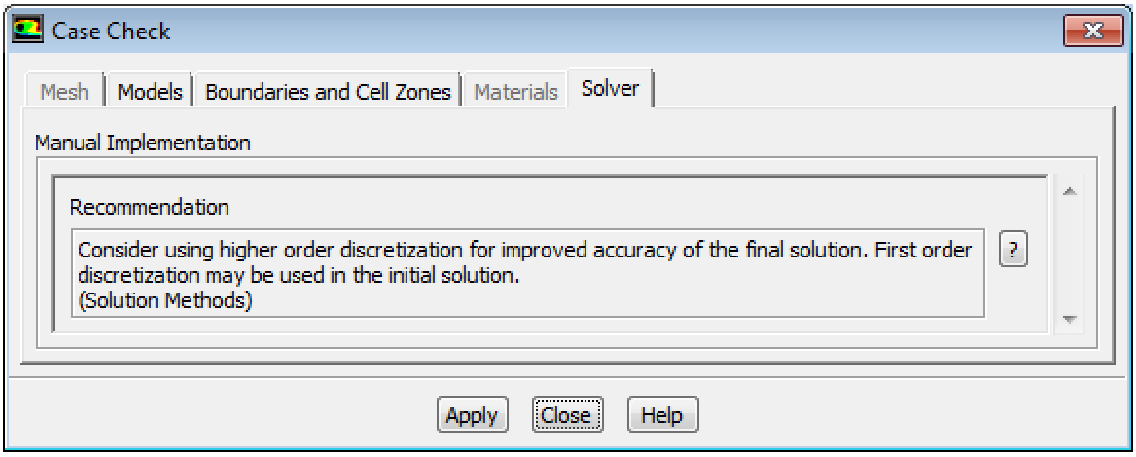


i. Keep the default of **Hybrid Initialization** from the **Initialization Methods** group box.

ii. Click **Initialize**.

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

Solution → **Run Calculation** → **Check Case**



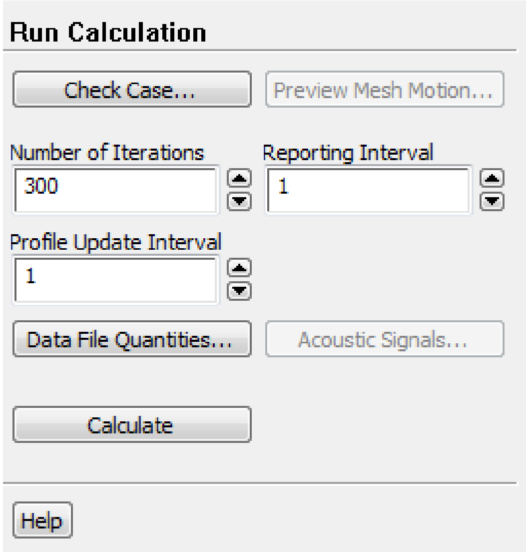
i. Click the **Models** and **Solver** tabs and examine the **Recommendation** in each. These recommendations can be ignored for this tutorial. The issues they raise will be addressed in later tutorials.

ii. Close the **Case Check** dialog box.

8. Calculate a solution.

a. Start the calculation by requesting 300 iterations.

Solution → **Run Calculation**



i. Enter `300` for **Number of Iterations**.

ii. Click **Calculate**.

Important

Note that the ANSYS Fluent settings file (`FFF.set`) is updated in the **Files** view of the ANSYS Workbench 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

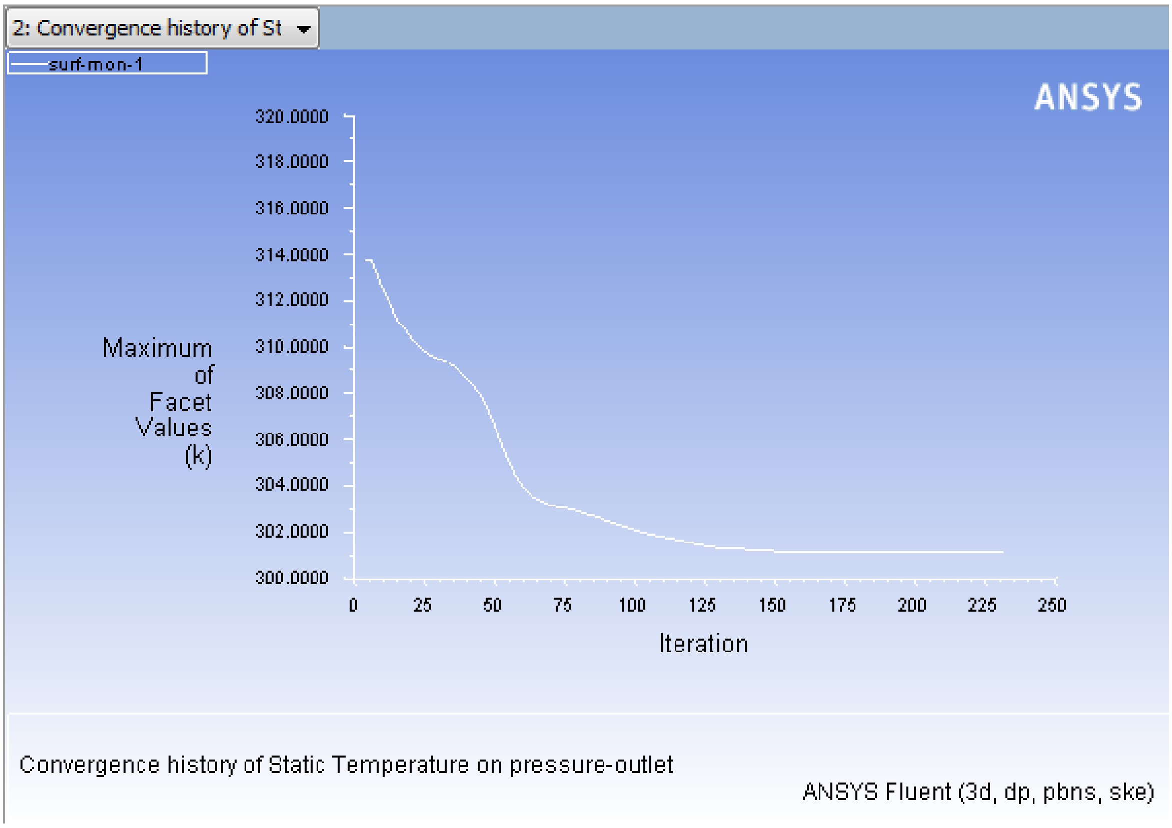
As the calculation progresses, the surface monitor history will be plotted in the graphics window (Figure 1.15: Convergence History of the Maximum Temperature at Pressure Outlet (p. 49)).

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.

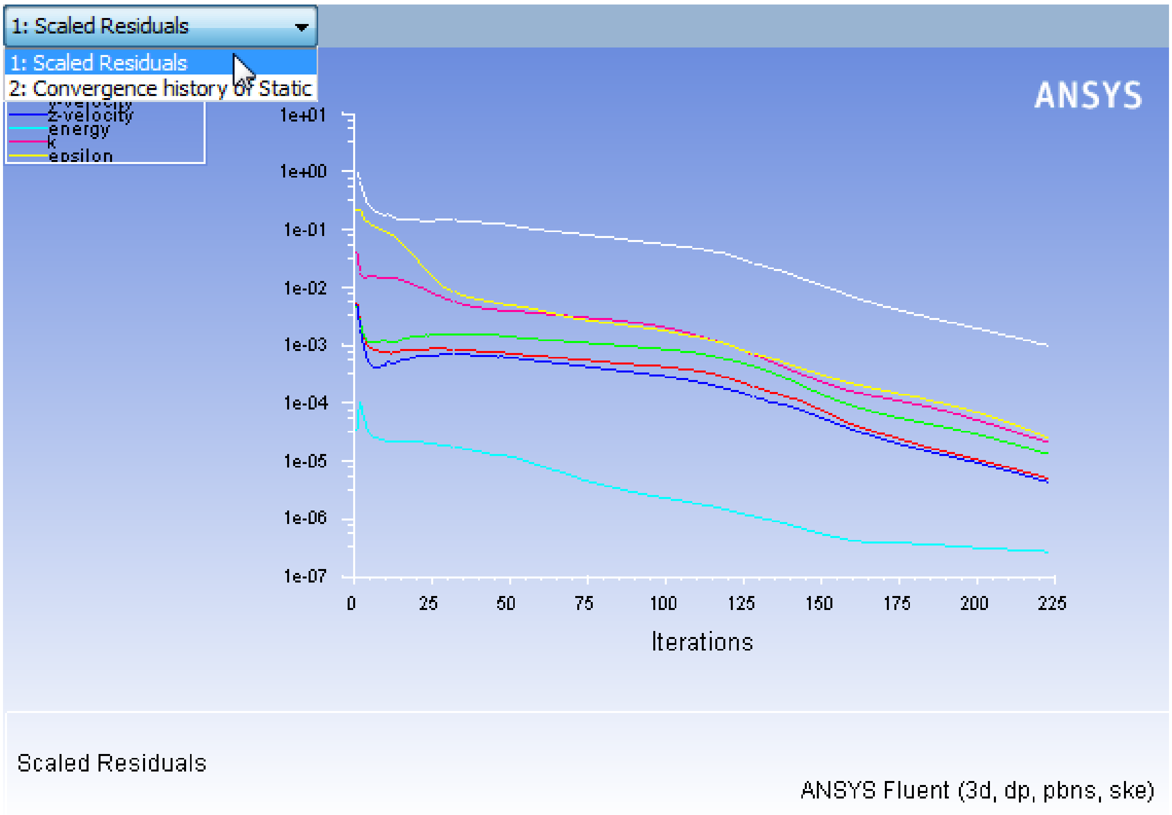
Because the residual values vary slightly by platform, the plot that appears on your screen may not be exactly the same as the one shown here.

Figure 1.15: Convergence History of the Maximum Temperature at Pressure Outlet



You can display the residuals history (Figure 1.16: Residuals for the Converged Solution (p. 50)), by selecting it from the graphics window drop-down list.

Figure 1.16: Residuals for the Converged Solution



b. Examine the plots for convergence (Figure 1.16: Residuals for the Converged Solution (p. 50) and Figure 1.15: Convergence History of the Maximum Temperature at Pressure Outlet (p. 49)).

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.

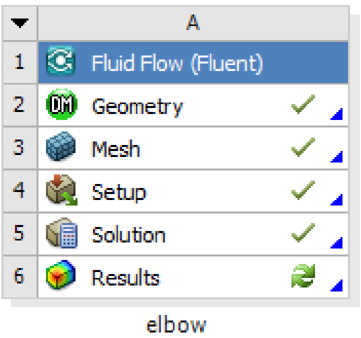
- The overall mass, momentum, energy, and scalar balances are obtained.

  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**

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



1.4.6. 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.

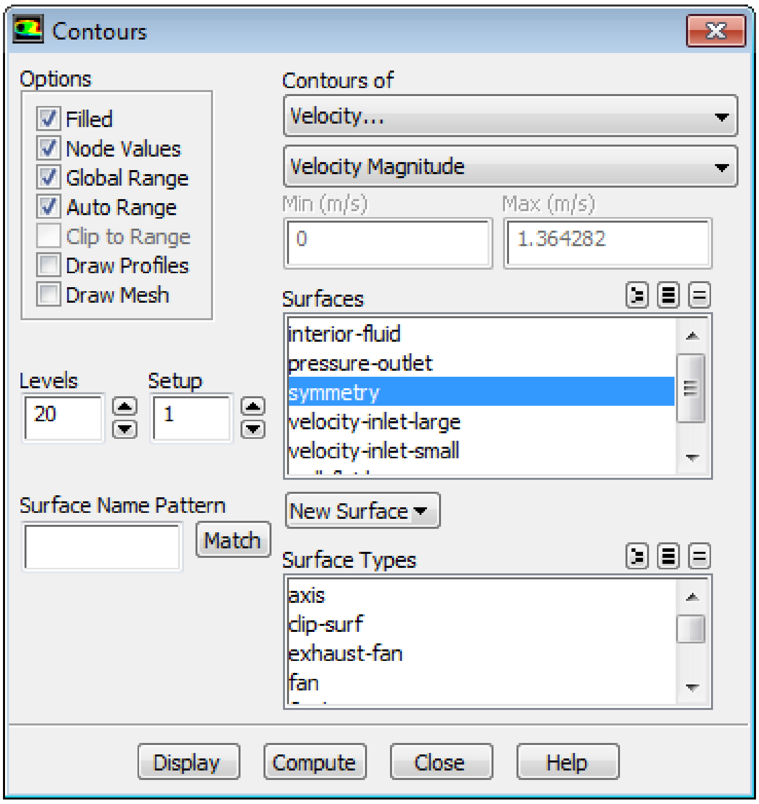
   With ANSYS Fluent still running, you can perform a simple evaluation of the velocity and temperature contours on the symmetry plane. Later, you will use CFD-Post (from within ANSYS Workbench) to perform the same evaluation.

   a. Display filled contours of velocity magnitude on the symmetry plane (Figure 1.17: Velocity Distribution Along Symmetry Plane (p. 53)).

      Results → **Graphics** → **Contours** **Edit...**

      Note

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



i. In the **Contours** dialog box, enable **Filled** in the **Options** group box.

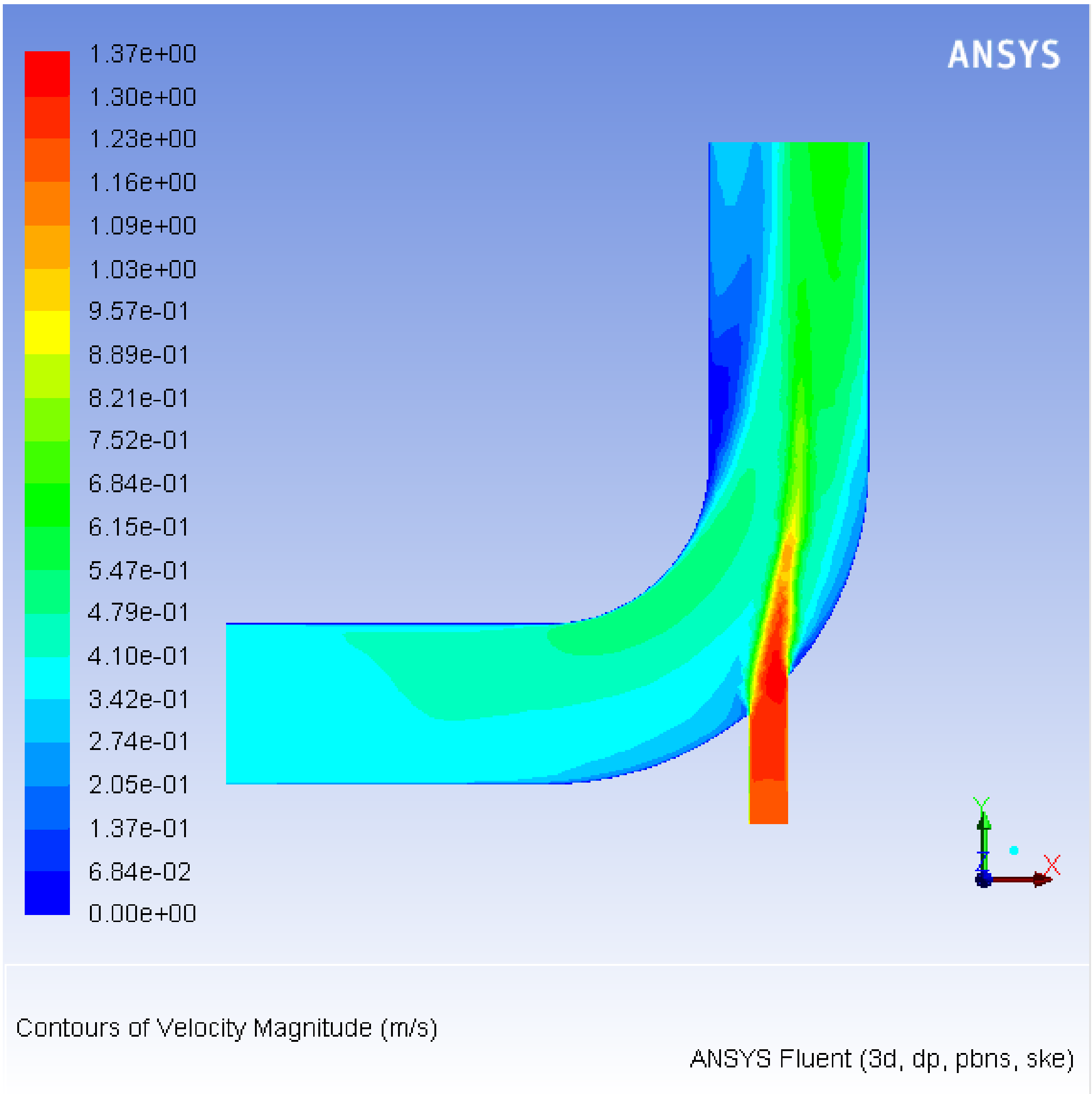
ii. Ensure that **Node Values** is enabled in the **Options** group box.

iii. Select **Velocity...** and **Velocity Magnitude** from the **Contours of** drop-down lists.

iv. Select **symmetry** from the **Surfaces** selection list.

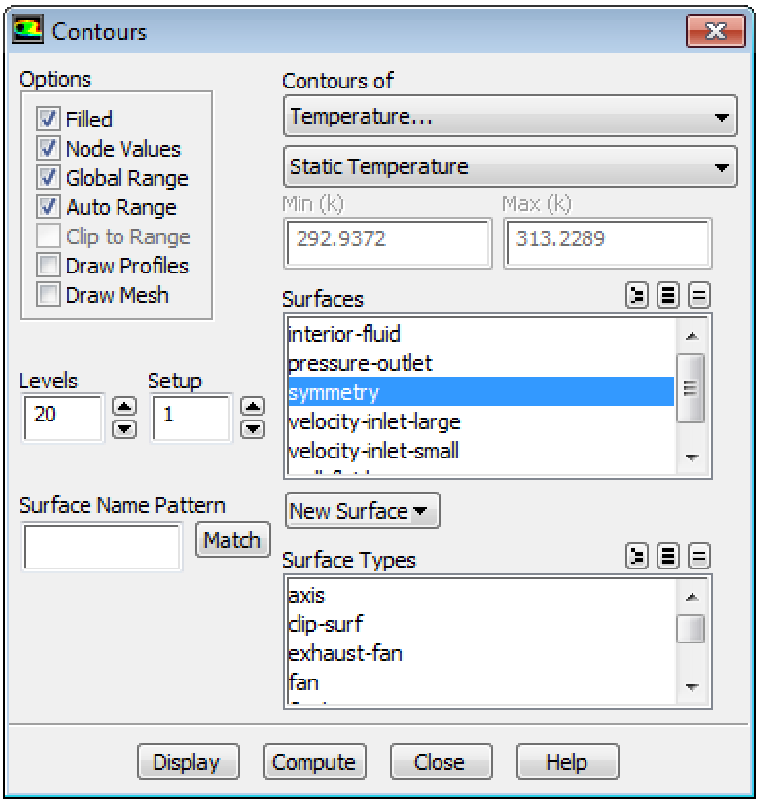
v. Click **Display** to display the contours in the active graphics window.

Figure 1.17: Velocity Distribution Along Symmetry Plane



b. Display filled contours of temperature on the symmetry plane (Figure 1.18: Temperature Distribution Along Symmetry Plane (p. 55)).

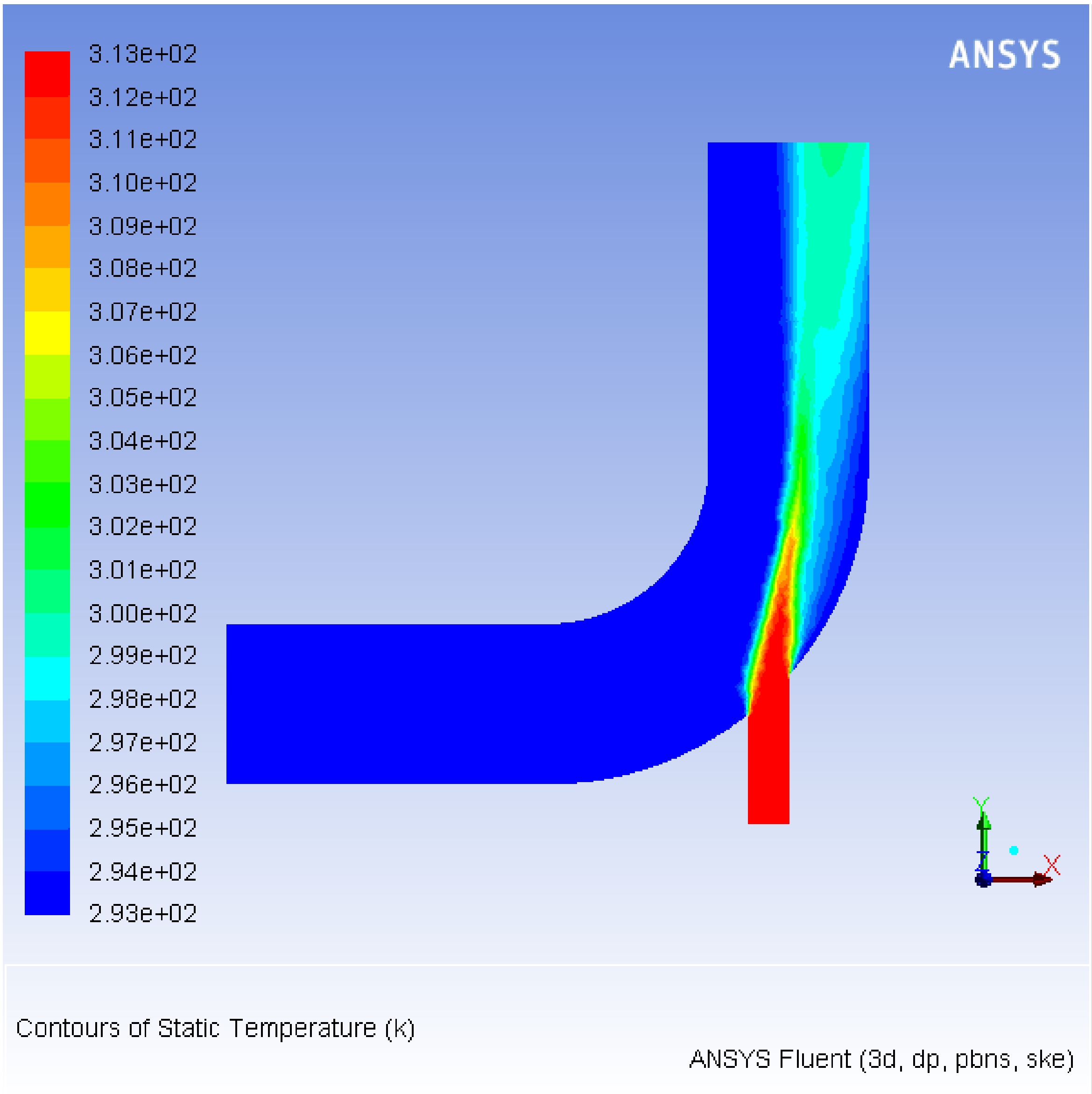
Results → **Graphics** → **Contours** **Edit...**



i. Select **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.

ii. Click **Display** and close the **Contours** dialog box.

Figure 1.18: 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

Note the addition of the compressed ANSYS Fluent case and data files to the list of files. These will have names like `FFF-1.cas.gz` and `FFF-1-00222.dat.gz`. Note that the digit(s) following `FFF` may be different if you have had to restart the meshing or calculation steps for any reason and that the name of the data file is based on the number of iterations. Thus your file names may be slightly different than those shown here.

2. Display results in CFD-Post.

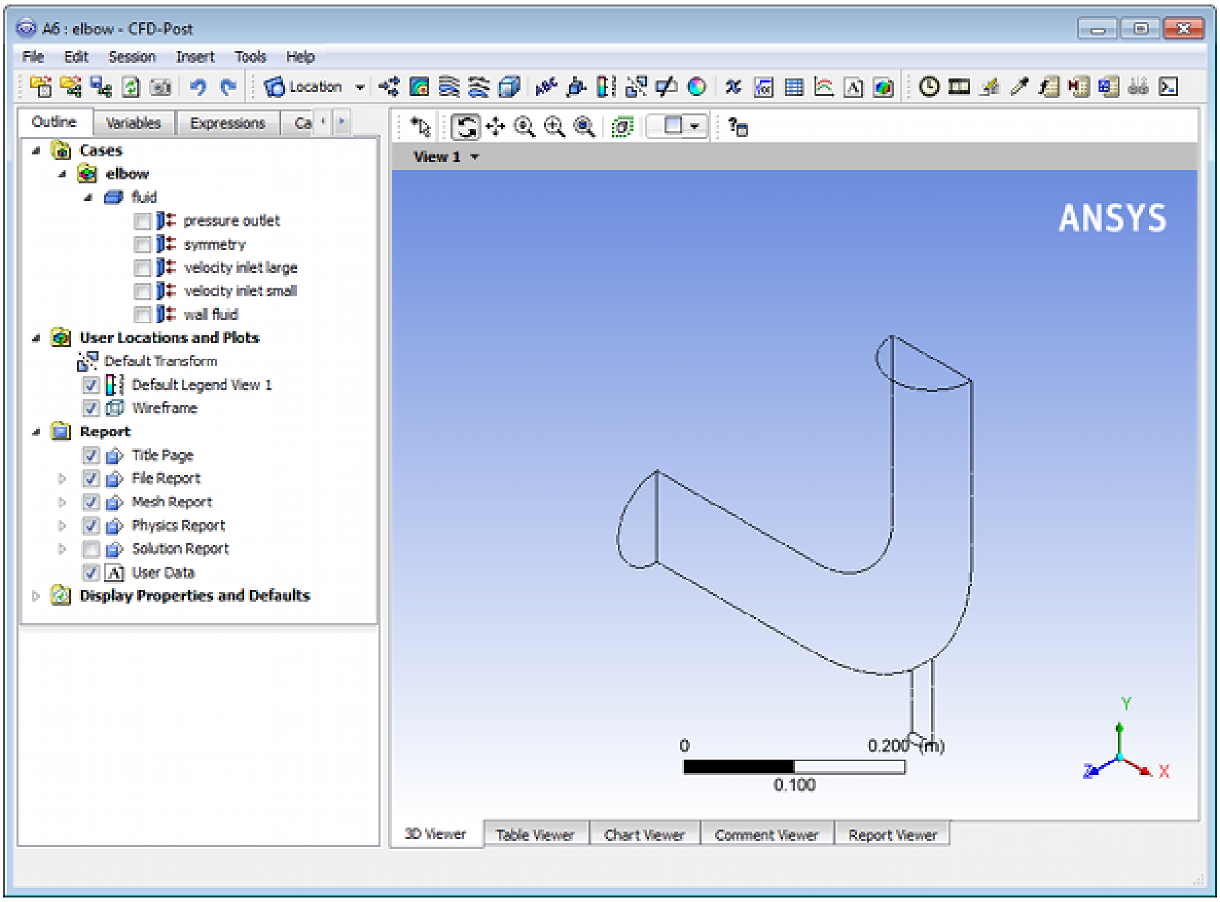
   a. Start CFD-Post.

   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

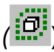
   The elbow geometry is already loaded and is displayed in outline mode. ANSYS Fluent case and data files are also automatically loaded into CFD-Post.

   Figure 1.19: 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 1.20: Velocity Distribution Along Symmetry Plane (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, from the **Domains** drop-down list, select **fluid**.

   iv. Select **symmetry** in the **Locations** list.

   v. Select **Velocity** in the **Variable** list.

   vi. Click **Apply**.

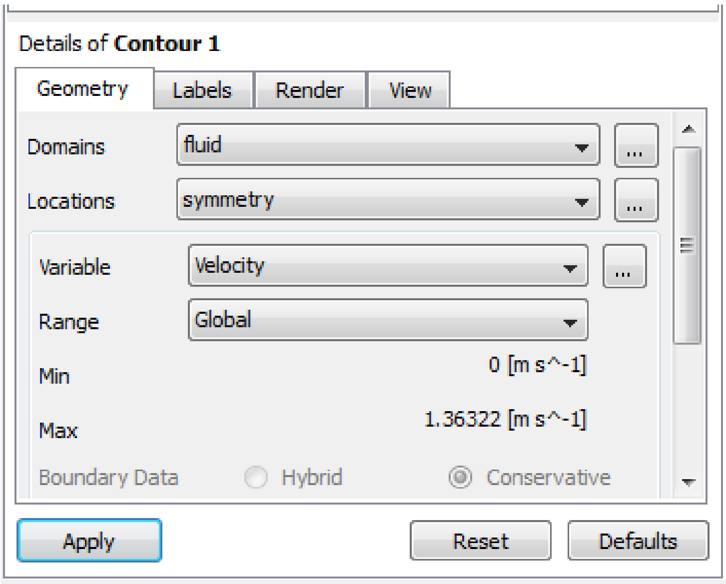
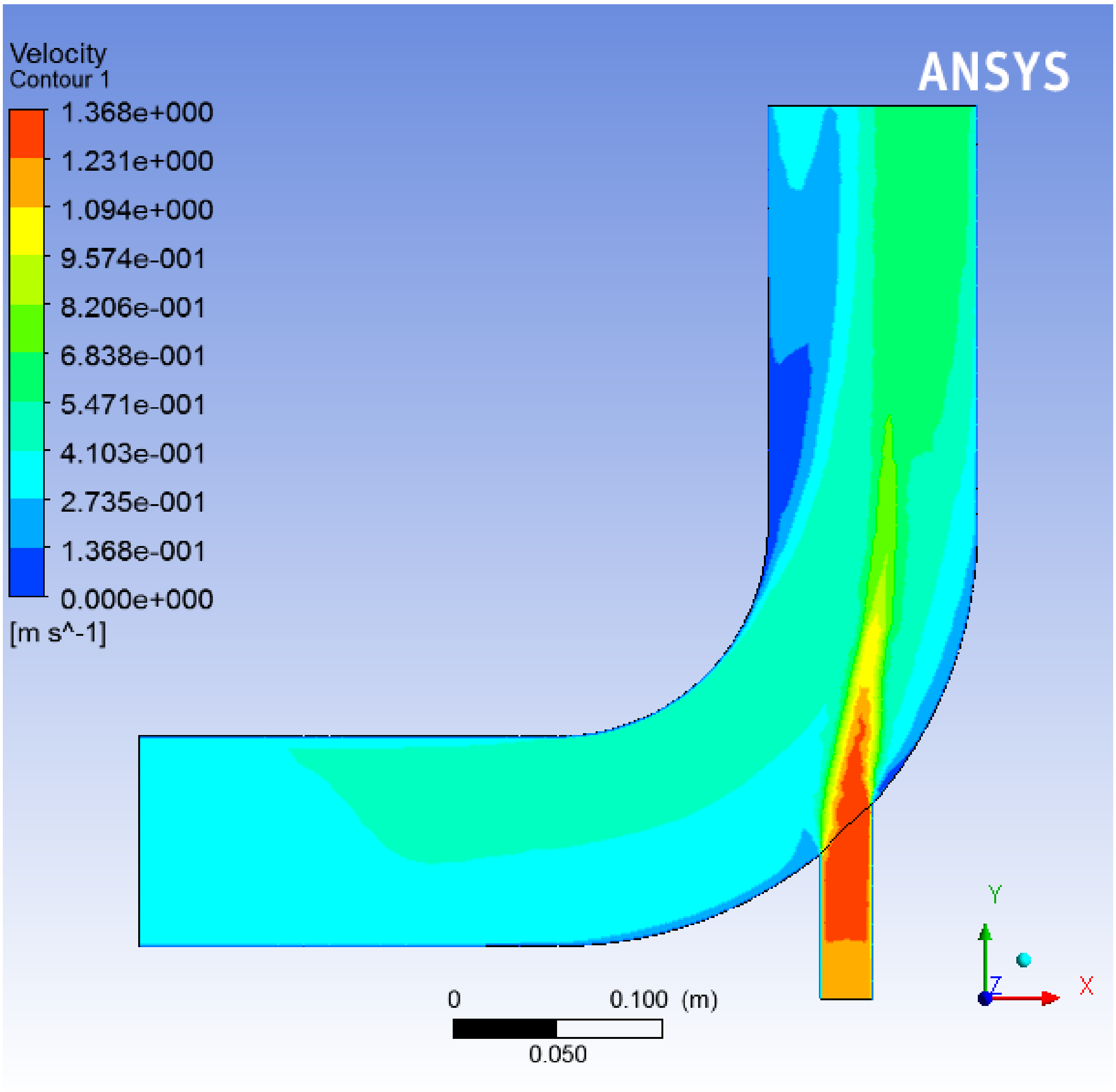


Figure 1.20: Velocity Distribution Along Symmetry Plane



e. Display filled contours of temperature on the symmetry plane (Figure 1.21: Temperature Distribution Along Symmetry Plane (p. 59)).

   i. Under **User Locations and Plots**, clear the check box beside the **Contour 1** object to disable 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.

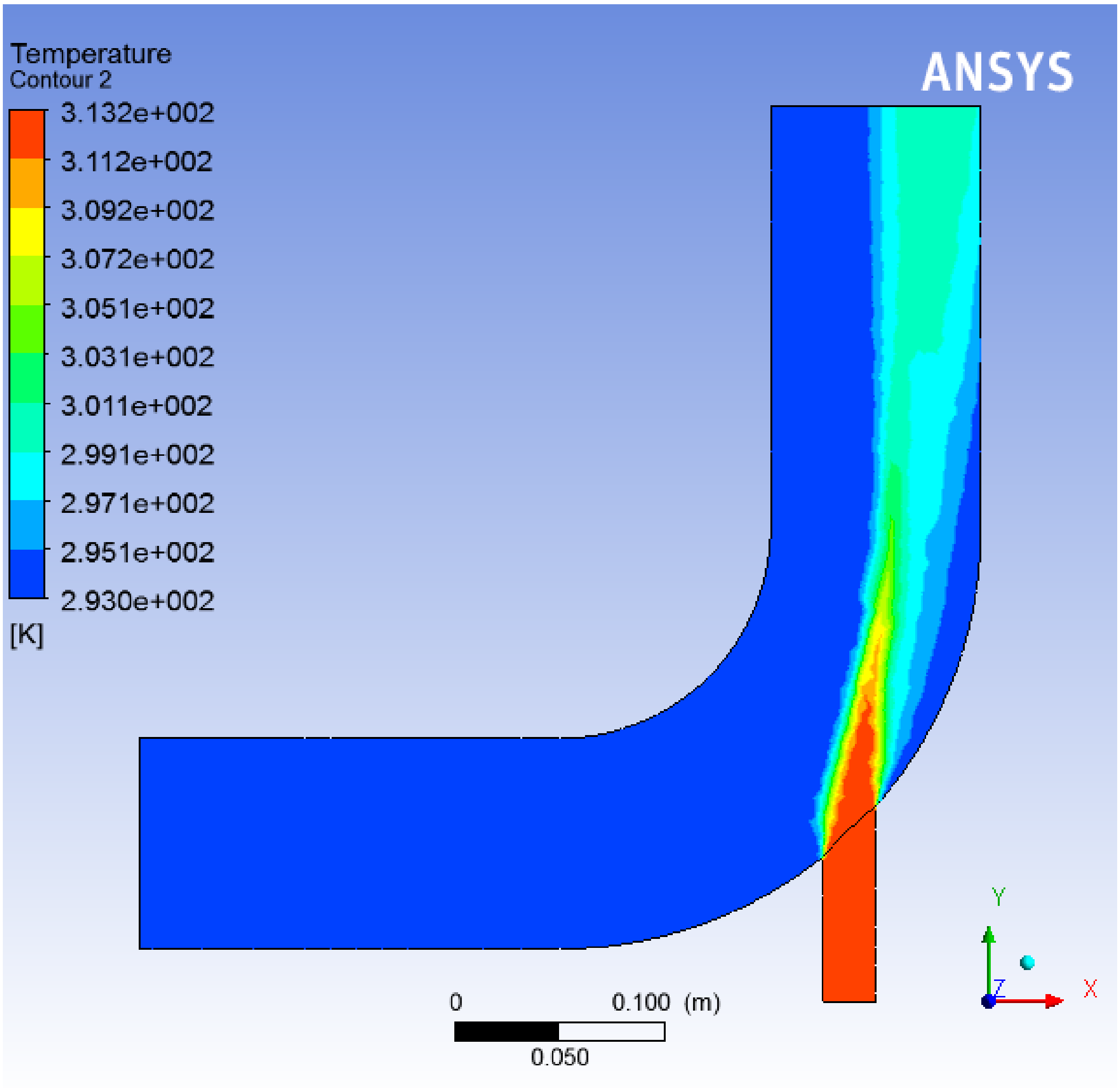
   iv. In the **Geometry** tab, from the **Domains** drop-down list, select **fluid**.

v. Select **symmetry** in the **Locations** list.

vi. Select **Temperature** in the **Variable** list.

vii. Click **Apply**.

Figure 1.21: Temperature Distribution Along Symmetry Plane



3. Close the CFD-Post application by selecting **File** → **Close ANSYS 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

   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. 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 1.22: Duplicating the Fluid Flow System

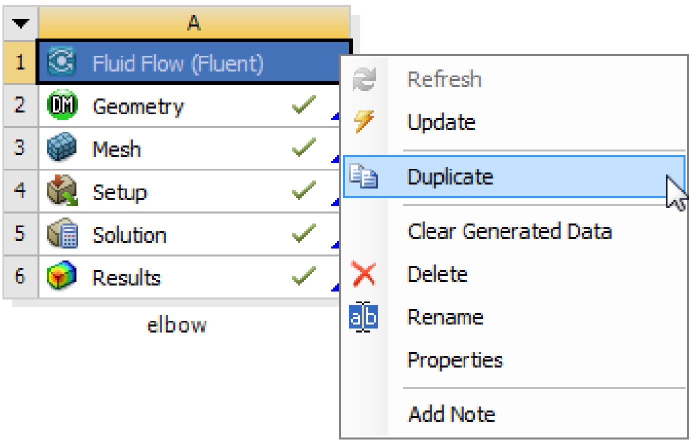
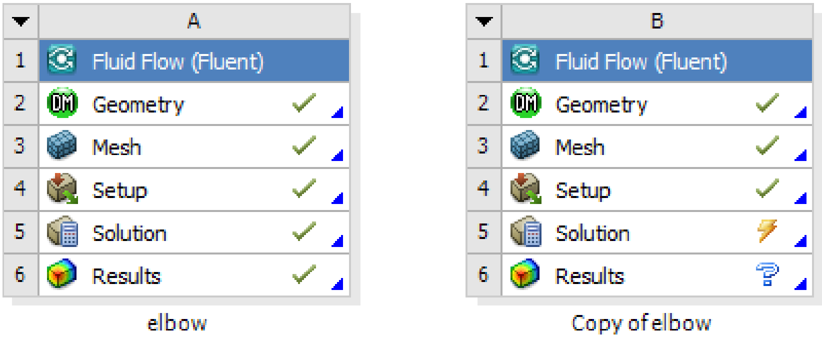


Figure 1.23: The Original Fluid Flow System and Its Duplicate



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. 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.

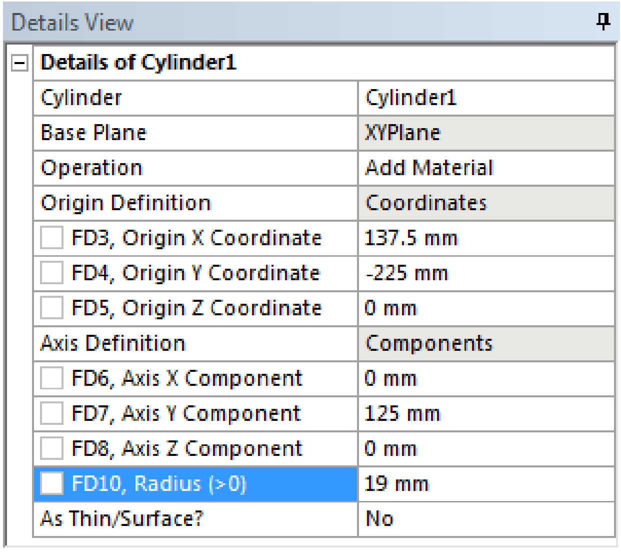
1. Open ANSYS DesignModeler.

   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`).

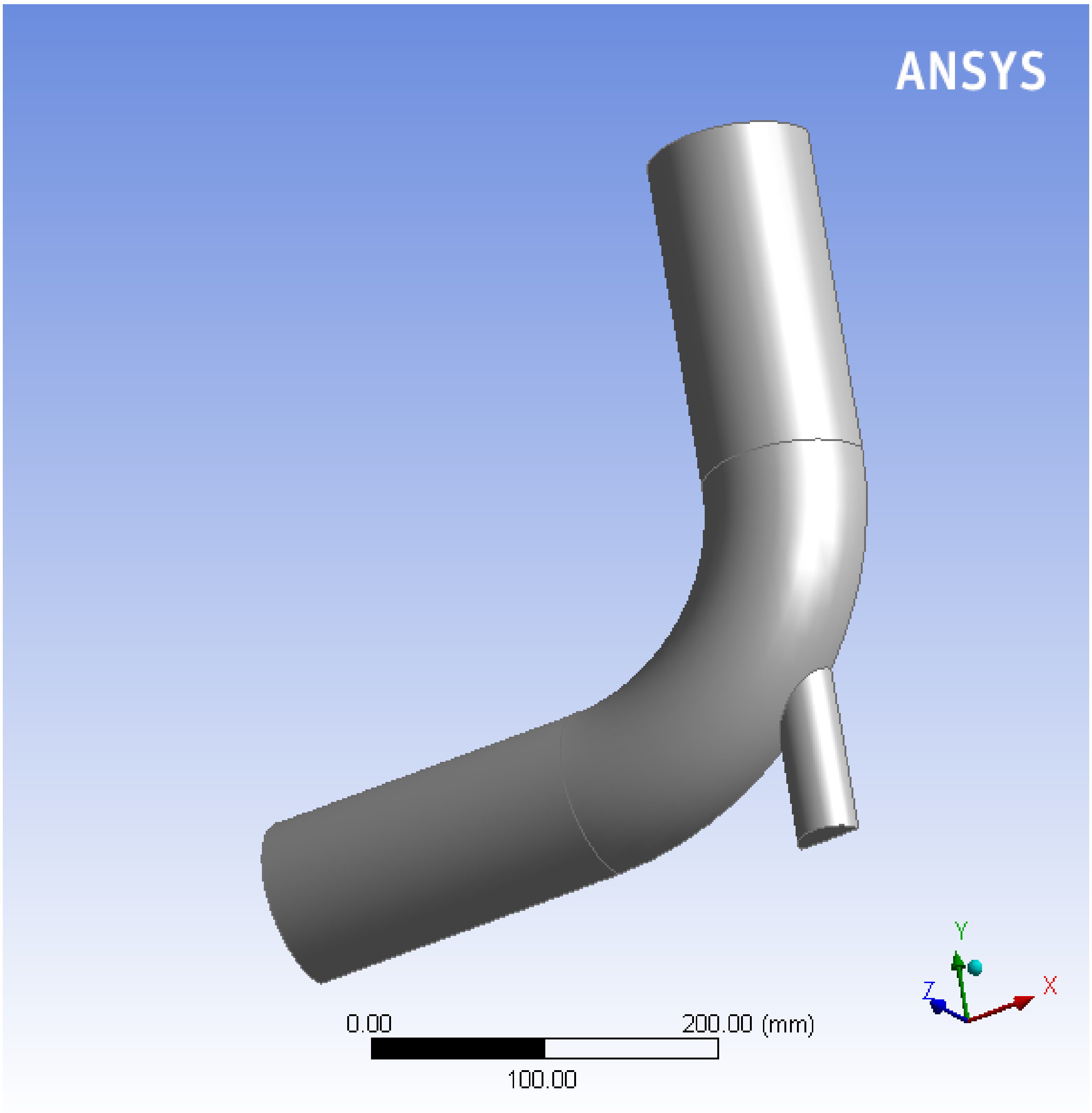
   a. Select **Cylinder1** to open the **Details View** of the small inlet pipe.

   b. In the **Details View**, change the **FD10, Radius (>0)** value from `12.5` millimeters to `19` millimeters.



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

Figure 1.24: 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**

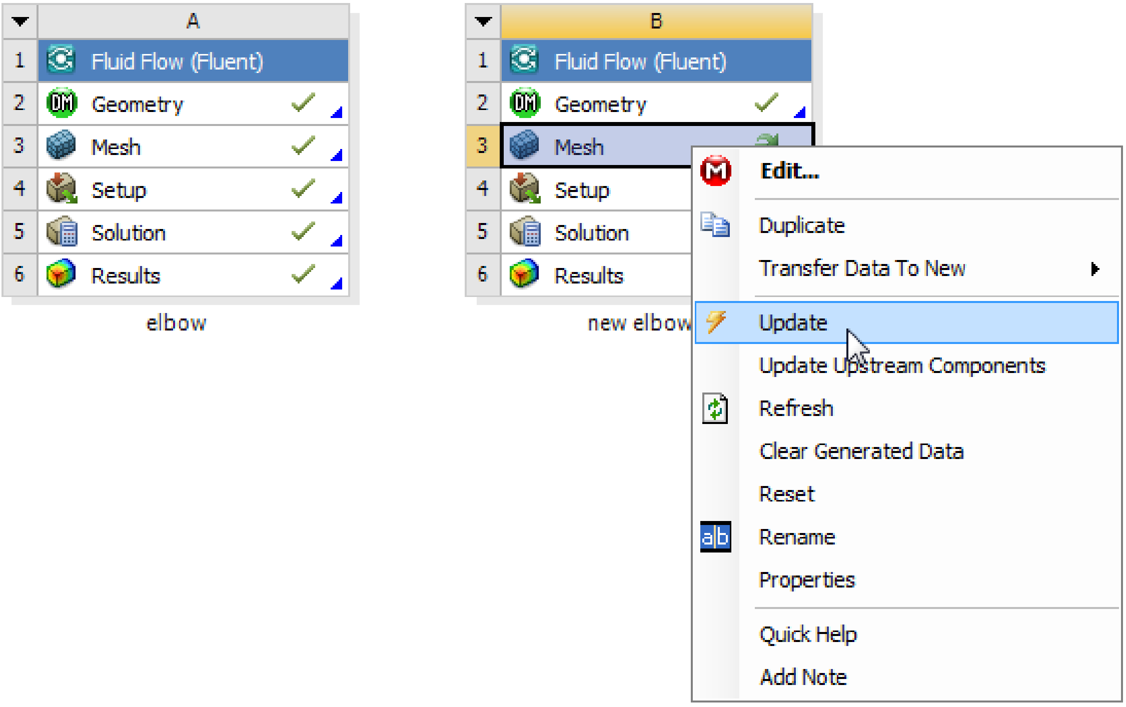
   Note the addition of the geometry, mesh, and ANSYS Fluent settings files now associated with the new, duplicated system.

1.4.9. 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 ANSYS Meshing editor to regenerate the mesh.

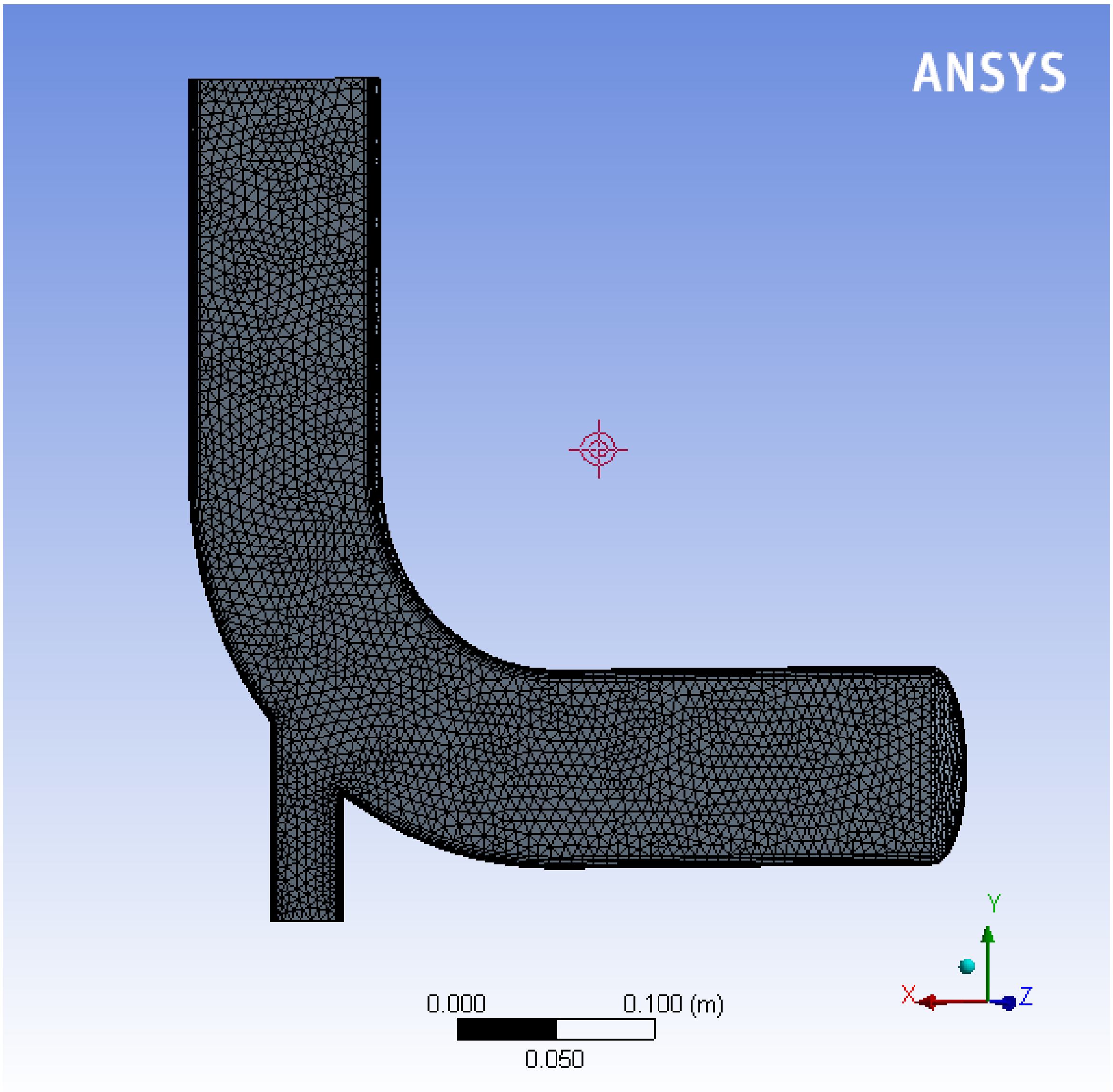
Figure 1.25: Updating the Mesh for the Changed Geometry



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 1.26: 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**

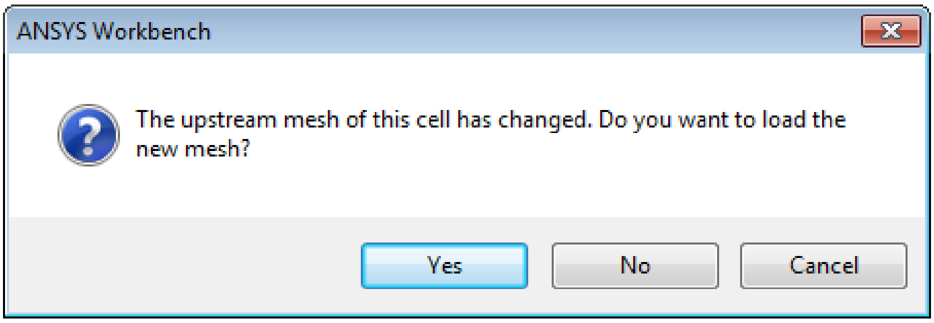
1.4.10. Calculating a New Solution in ANSYS Fluent

Now that there is an updated computational mesh for the modified geometry in the duplicated system, a new solution must 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. Click **Yes** to continue, and click **OK** when Fluent Launcher is displayed in order to open ANSYS Fluent.

Figure 1.27: ANSYS Workbench Prompt When the Upstream Mesh Has Changed



2. Ensure that the unit of length is set to millimeters.

   Setup → **General** → **Units...**

3. Check the mesh (optional).

   Setup → **General** → **Check**

4. Revisit the boundary conditions for the small inlet.

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

   Here, you must set the hydraulic diameter to `38` mm based on the new dimensions of the small inlet.

5. Re-initialize the solution.

   Solution → **Solution Initialization**

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

6. Recalculate the solution.

   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**

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

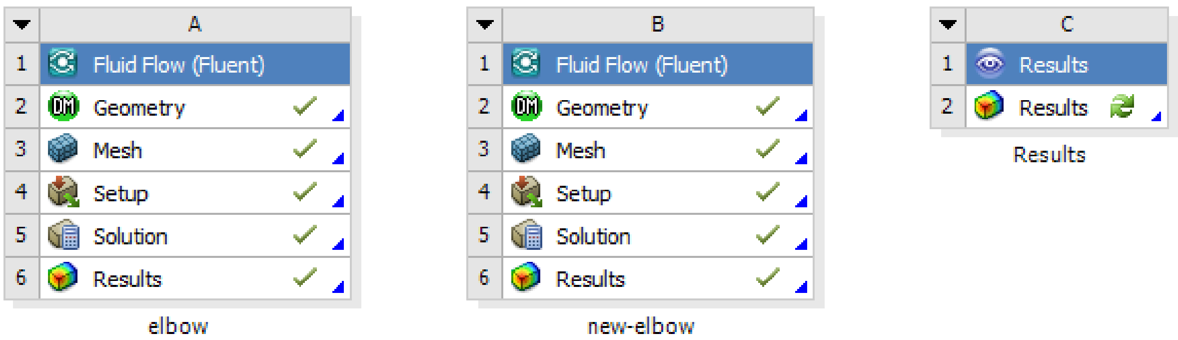
1.4.11. Comparing the Results of Both Systems in CFD-Post

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.

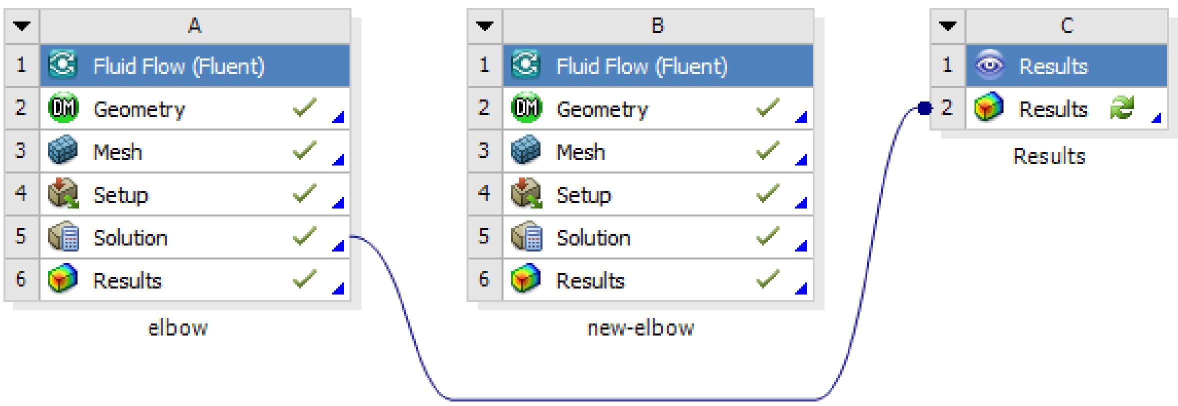
   Figure 1.28: The NewResults System in the Project Schematic



2. Add the solutions of each of the systems to the **Results** system.

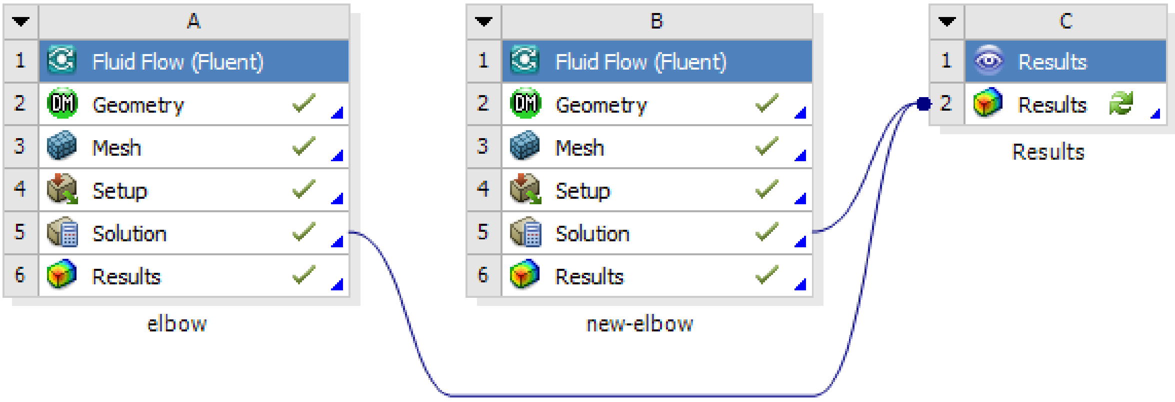
   a. Select the **Solution** cell in the first Fluid Flow analysis system (cell `A5`) and drag it over the **Results** cell in the **Results** system (cell `C2`). This creates a transfer data connection between the two systems.

   Figure 1.29: 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.

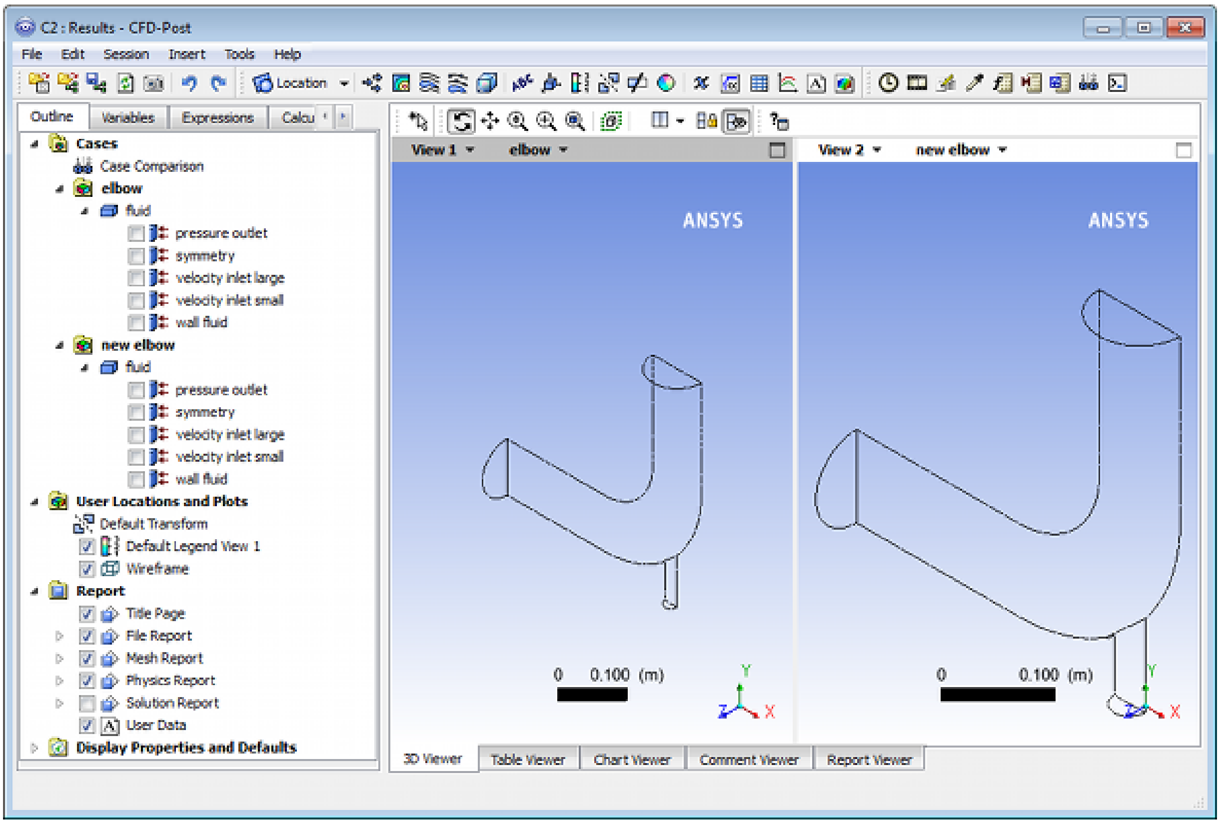
Figure 1.30: Connecting the Second Fluid Flow System to the New Results System



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 1.31: 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.

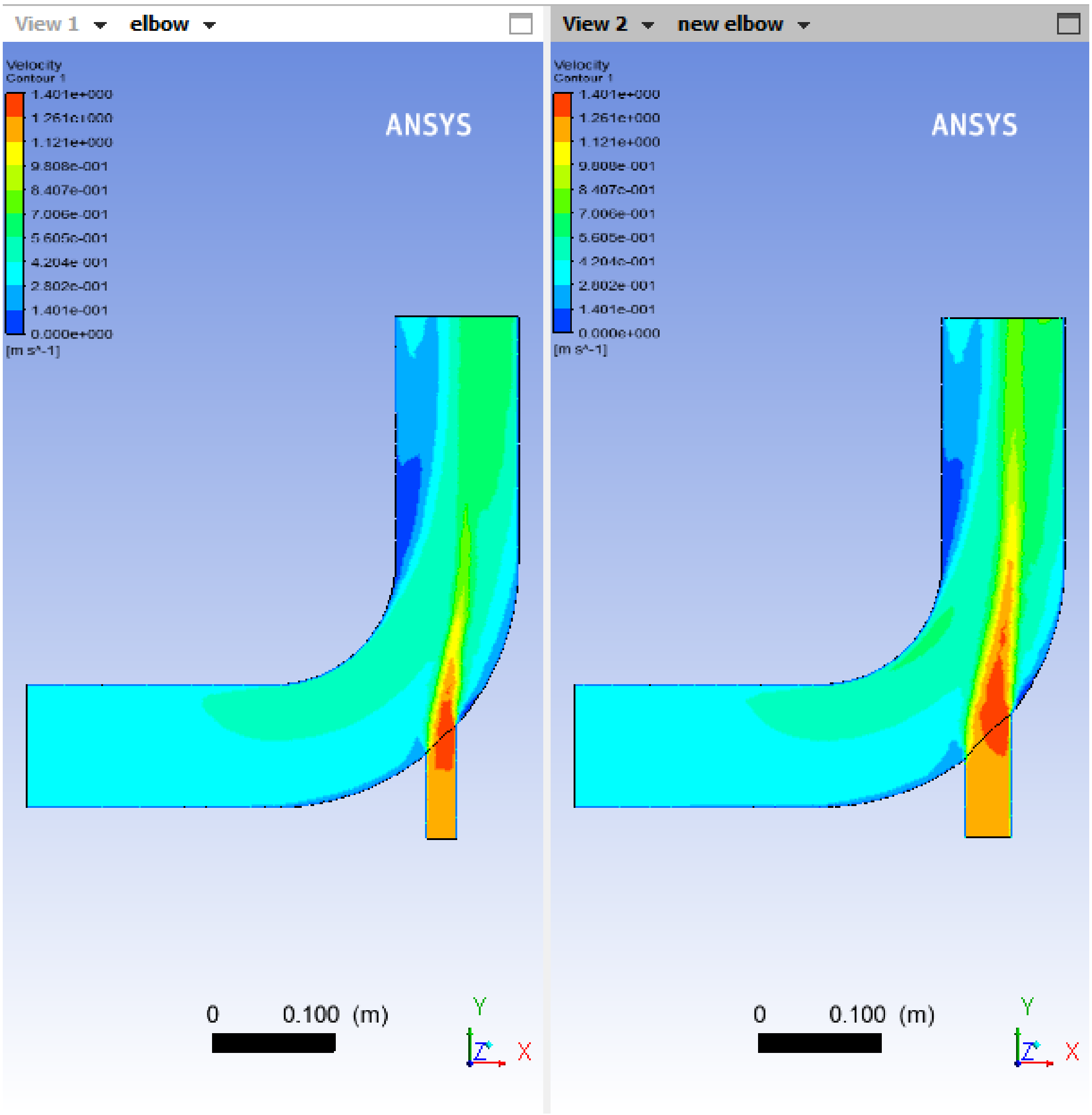
   v. Select **Velocity** in the **Variable** list.

   vi. Click **Apply**. The velocity contours are displayed in each view.

Note

To better visualize the velocity display, you can clear the **Wireframe** view option under **User Locations and Plots** in the **Outline** tree view.

Figure 1.32: 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.

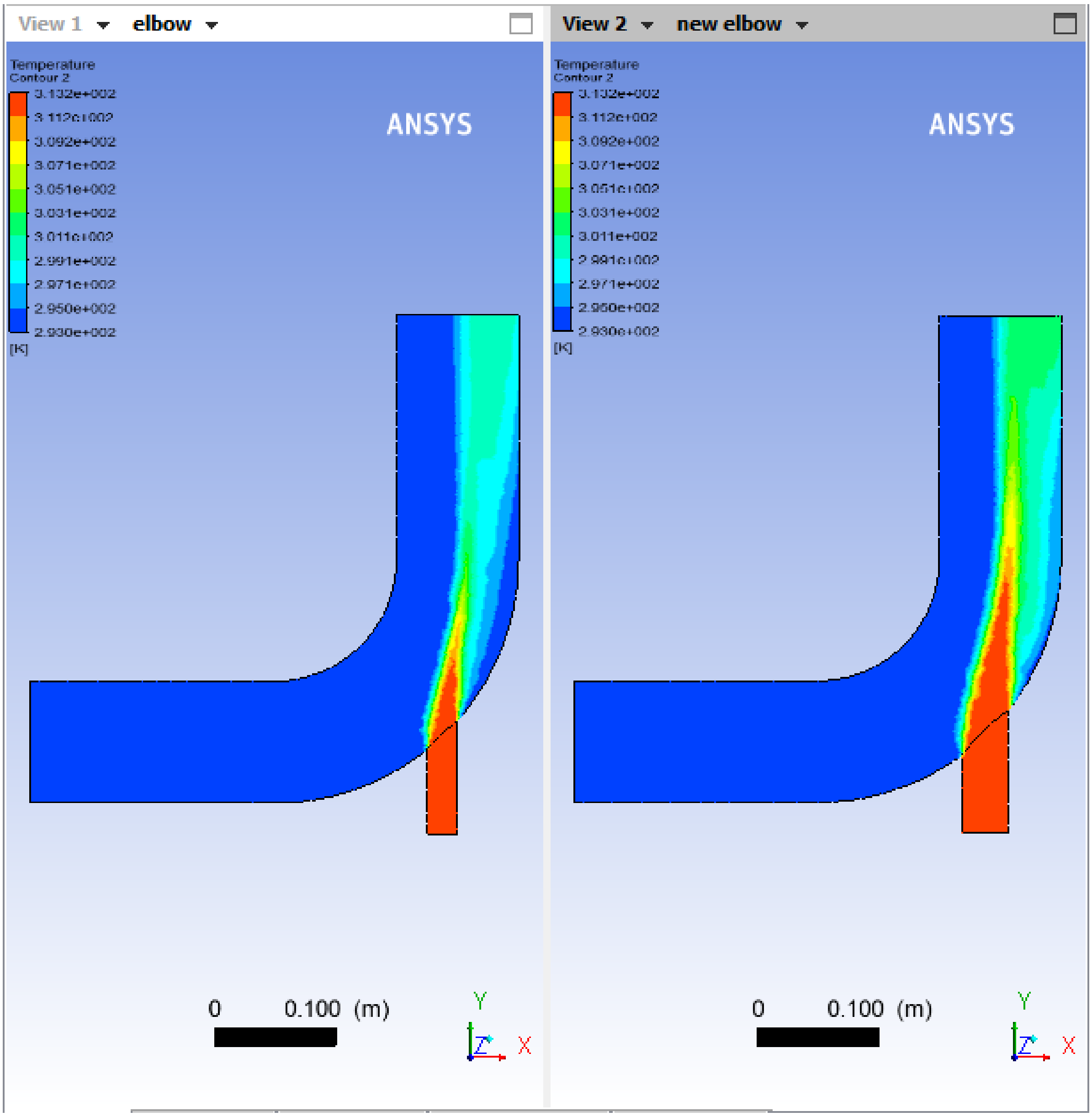
iv. In the **Geometry** tab, select **fluid** in the **Domains** list.

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 1.33: 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**

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

1.5. 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.