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

Introduction

This tutorial illustrates using ANSYS Workbench to set up and solve a three-dimensional turbulent fluid flow and heat transfer problem in a mixing elbow using ANSYS FLUENT fluid flow systems. This tutorial is designed to introduce you to the ANSYS Workbench tool set using a familiar geometry (the first tutorial in the separate Tutorial Guide). Within this tutorial, 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 the ANSYS CFD-Post postprocessing tool. Some capabilities of ANSYS Workbench (e.g., 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 an ANSYS 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:
    - Set material properties and boundary conditions for a turbulent forced convection problem.
    - Initiate the calculation with residual plotting.
    - Calculate a solution using the pressure-based solver.
    - Visually examine the flow and temperature fields using ANSYS FLUENT (and ANSYS CFD-Post).

- Create a copy of the original ANSYS 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 ANSYS CFD-Post.

Prerequisites

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

Problem Description

The problem to be considered is shown schematically in Figure 1.1. 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 pipe dimensions, the fluid properties and boundary conditions are given in SI units.

Note: *Since the geometry of the mixing elbow is symmetric, only half of the elbow needs to be modeled.*
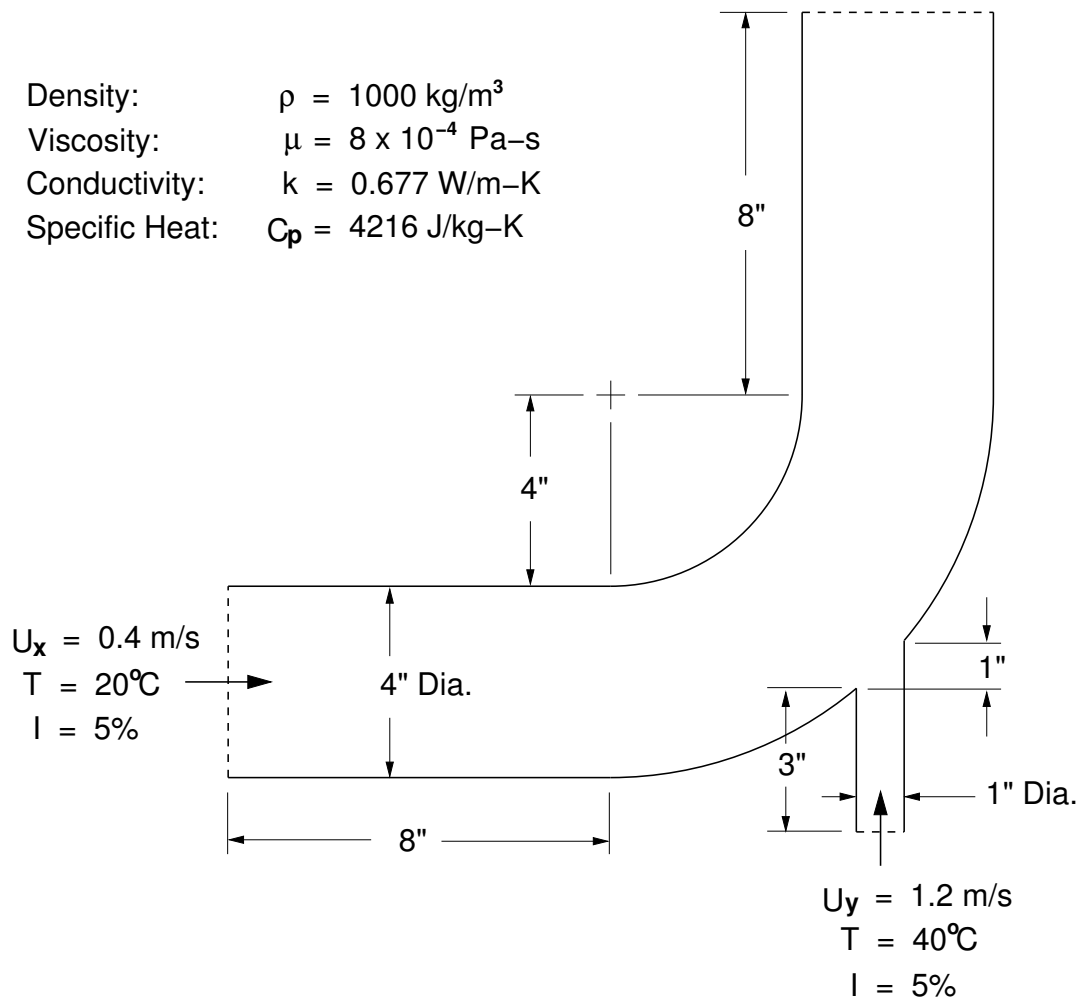
Density: ρ = 1000 kg/m^3
Viscosity: μ = 8 x 10^{-4} Pa–s
Conductivity: k = 0.677 W/m–K
Specific Heat: C_p = 4216 J/kg–K

8"

4"

U_x = 0.4 m/s
T = 20℃
I = 5%

4" Dia.

8"

1"

3"

1" Dia.

U_y = 1.2 m/s
T = 40℃
I = 5%

Figure 1.1: Problem Specification

Setup and Solution

Preparation

1. Download `elbow-workbench.zip` from the User Services Center to your working folder. This file can be found by using the Documentation link on the ANSYS FLUENT product page.

2. Unzip `elbow-workbench.zip`.

   The `.zip` file contains a folder (`elbow-workbench`) that in turn contains two geometry files (`elbow-geometry.agdb` and `elbow-geometry.stp`) along with an ANSYS Workbench project file (`elbow-workbench.wbpj`) and a corresponding project folder (`elbow-geometry_files`) that contains supporting files for the project (e.g., 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.*

Step 1: Creating a Fluid Flow Analysis System in ANSYS Workbench

In this step, you will start ANSYS Workbench and create a new fluid flow analysis system.

1. Start ANSYS Workbench by choosing the Start menu, then select the Workbench option in the ANSYS 12.1 program group.

   Start ⟶ All Programs ⟶ ANSYS 12.1 ⟶ Workbench

   This displays the ANSYS Workbench application window, containing the Toolbox on the left and the Project Schematic on the right. Various supported analyses and applications are listed in the Toolbox, while you visualize the components of the analysis in the Project Schematic.

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

2. Create a new fluid flow analysis system by double-clicking the Fluid Flow (FLUENT) option under Analysis Systems in the Toolbox. You will initially see a green dotted line outline in the Project Schematic, indicating potential locations for the new system.
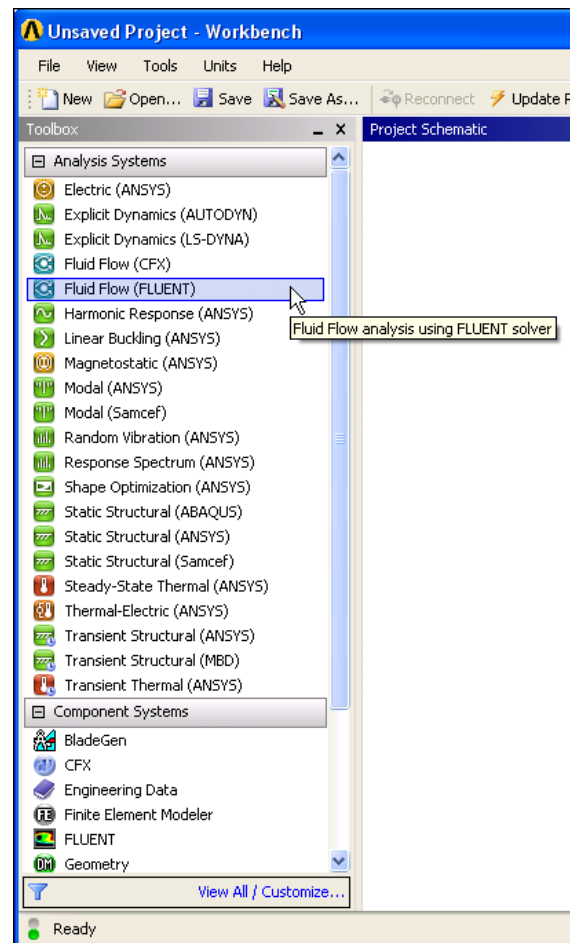

Figure 1.2: Selecting the Fluid Flow (FLUENT) Analysis System in ANSYS Workbench

You can also drag-and-drop the analysis system into the Project Schematic: the green dotted outline will turn into a red box, indicating the location of the new system.

This creates a new ANSYS FLUENT*-based fluid flow analysis system in the* Project Schematic*.*
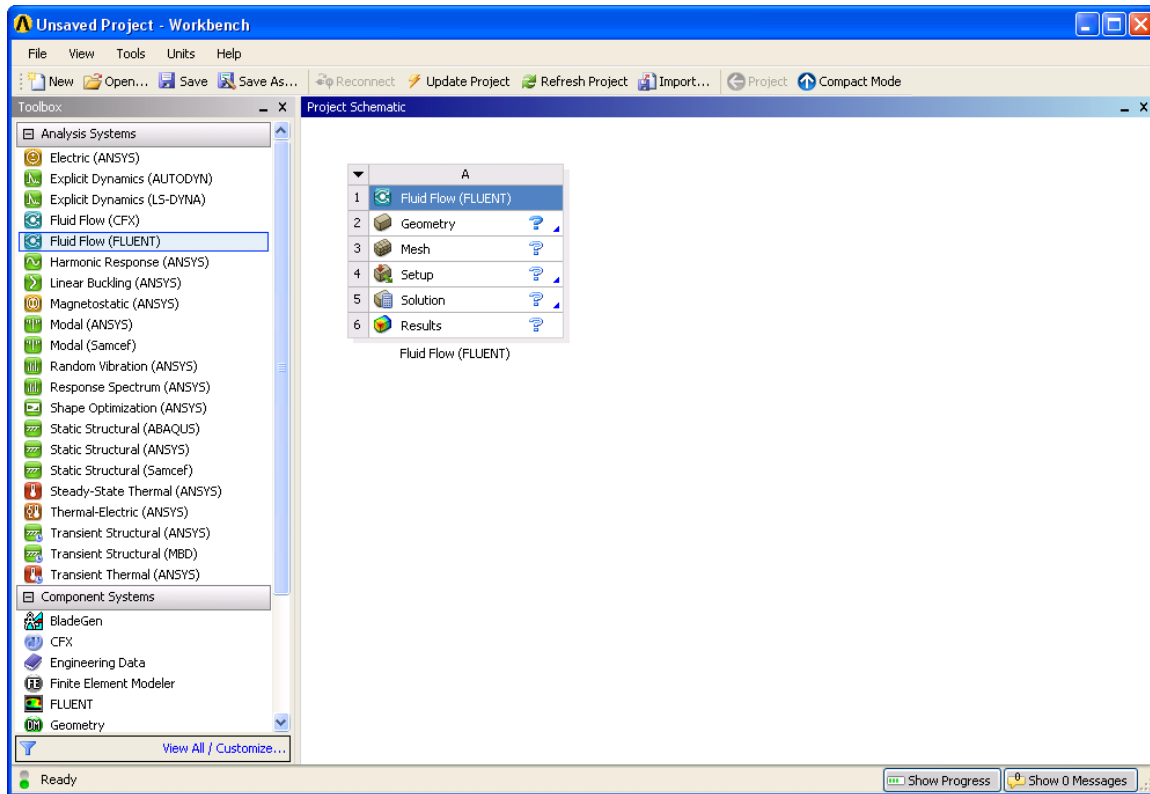
Figure 1.3: ANSYS Workbench with a New ANSYS FLUENT-Based Fluid Flow Analysis System

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, where you can browse to a specific folder and enter a specific name for the* ANSYS Workbench *project.*

   (b) In your working folder, 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 as well as supporting files for the project.

5. View the 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.
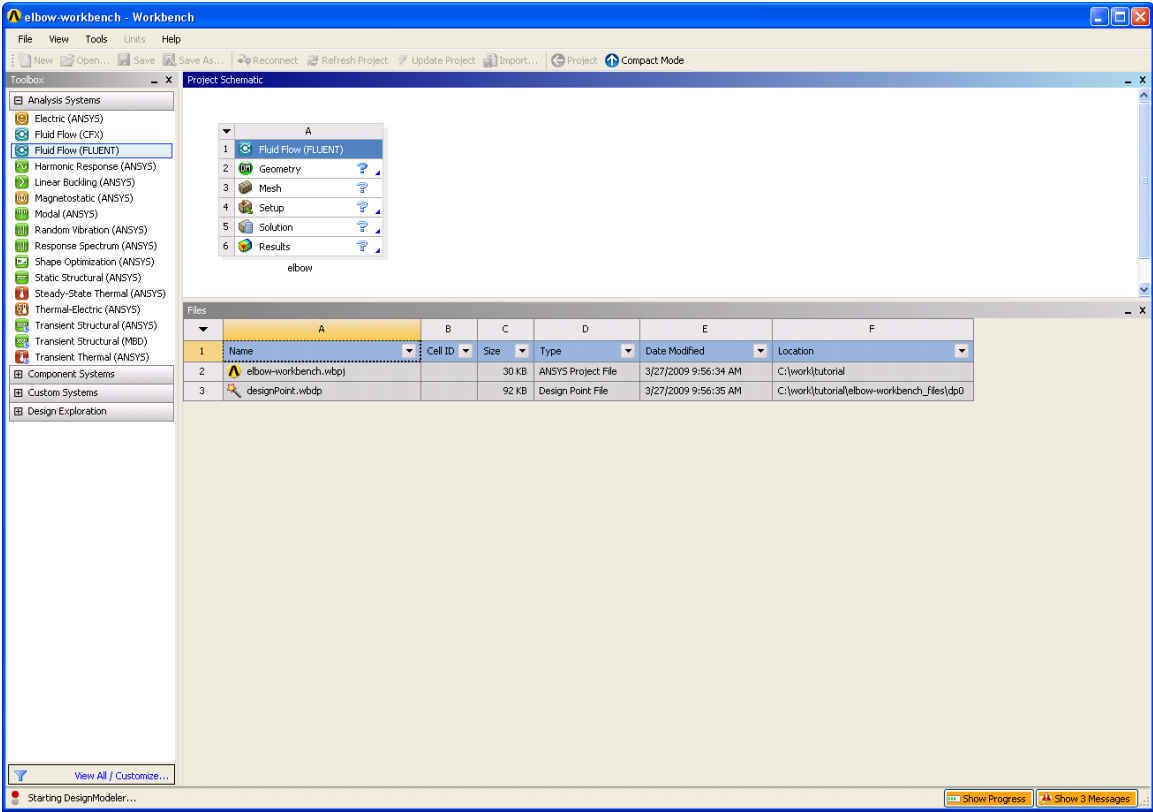
View ⟶Files

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

In the Files view, you will be able to see the name and type of file, the ID of the cell 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 the separate FLUENT in Workbench User's Guide and the ANSYS Workbench on-line help.

From here, you will move on to create the geometry described in Figure 1.1, and later set up a fluid flow analysis for the geometry.

You will note that the ANSYS FLUENT- based fluid flow analysis system, for example, is composed of various *cells* (Geometry, Mesh, etc.) that represent the work flow for performing the analysis. ANSYS Workbench is composed of multiple data-integrated (e.g., ANSYS FLUENT) and native applications into a single, seamless project flow, where individual cells can obtain data from 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 visual indications 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. For more information about cell states, see the ANSYS Workbench on-line help:

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

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

- **Attention Required** ( ) indicates that the current upstream data has been passed to the cell, however, you must take some action to proceed. For example, after you launch 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 a 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 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.

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 tutorial, we will create the geometry from scratch in ANSYS DesignModeler.

i   Note the Attention Required icon ( ) within the Geometry cell for the system. This indicates that the cell is requires data (e.g., a geometry). Once the geometry is defined, the state of the cell will change accordingly. Likewise for the state of the remaining cells in the system. For more information about system cell states, see the separate FLUENT in Workbench User's Guide and the on-line documentation for ANSYS Workbench.

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.*

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. You can also right-click on the Geometry cell to display the context menu where you can select the New Geometry... option.

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.
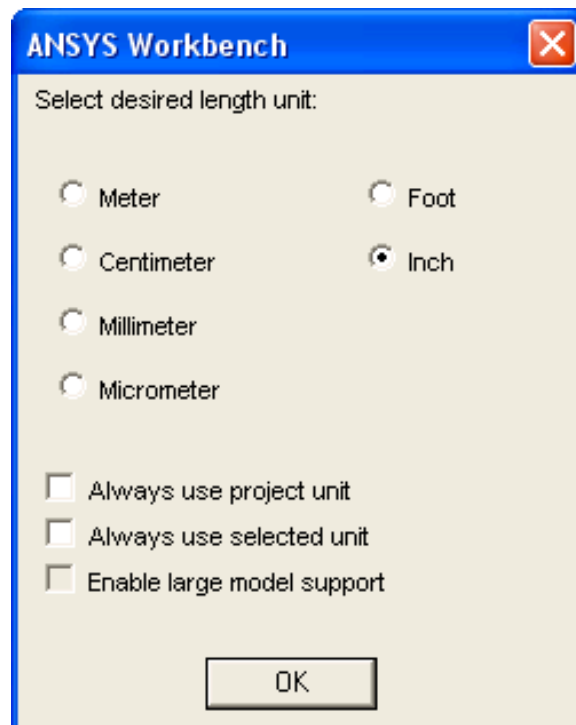
Figure 1.5: Setting the Units in ANSYS DesignModeler

3. Create the geometry.

The geometry for this tutorial (Figure 1.1) 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 (in this case, a half circle since the symmetry of the problem allows you to not have to generate the entire pipe geometry) and the other that defines the path through which the profile is swept.

(a) Create the profile.

i. Create a new plane by selecting YZPlane from the Tree Outline and click on New Plane from the Active Plane/Sketch toolbar, near the top of the ANSYS Workbench window. Clicking YZPlane first ensures that the new plane is based on the YZPlane.
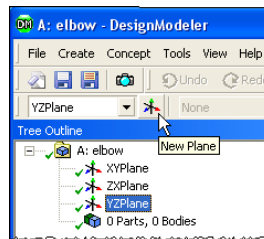
ii. In the Details View for the new plane (Plane 4), set Transform 1 (RMB) to Offset Global X, and set the Value of the offset to -8 in.
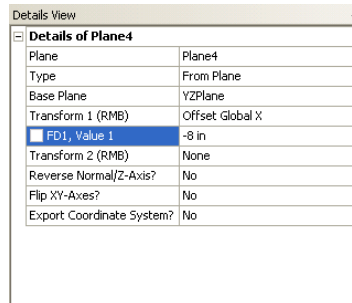
iii. Click on Generate (Generate located in the ANSYS DesignModeler toolbar) to create the plane.

iv. Create a new sketch by selecting Plane4 from the Tree Outline and then click New Sketch from the Active Plane/Sketch toolbar, near the top of the ANSYS Workbench window. Clicking the plane first ensures that the new sketch is based on Plane4.
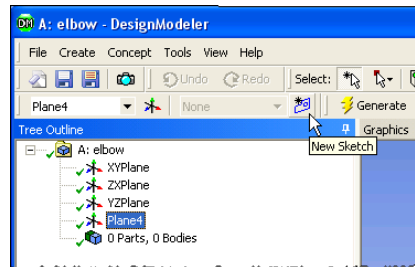
v. On the Sketching tab, open the Settings toolbox, select Grid, and enable the Show in 2D and the Snap options.
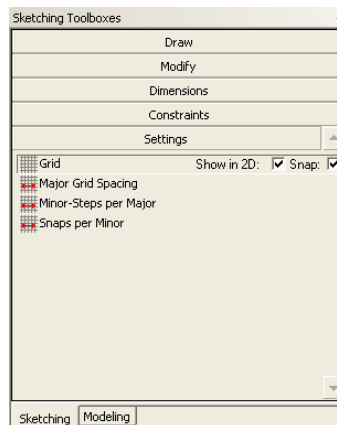
vi. Set Major Grid Spacing to 1 in and Minor-Steps per Major to 2.

vii. 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.

viii. 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). Draw an arc with a radius of 2 in, centered on $X = -6$ in, $Y = 0$ in (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.

ix. On the **Sketching** tab open the **Draw** toolbox and select **Line** (you may need to use the arrows in the toolbox to scroll up to see the correct tool). Draw a line from $X = -4$ in, $Y = 0$ in to $X = -8$ in, $Y = 0$ in.
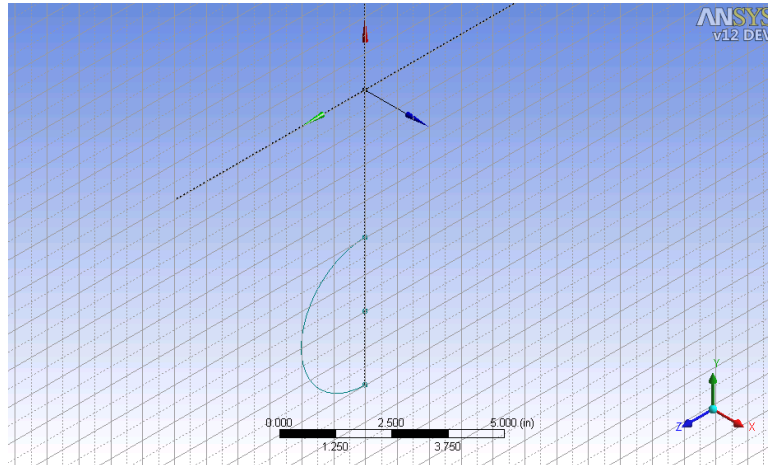

Figure 1.6: Creating the Arc Profile

(b) Create the path.

i. On the **Modeling** tab select **XYPlane**, then click **New Sketch** and then **Generate** to create a new sketch based on the **XYPlane**.

ii. On the **Sketching** tab, open the **Settings** toolbox, select **Grid**, and enable the **Show in 2D** and the **Snap** options.

iii. Set **Major Grid Spacing** to 1 in and **Minor-Steps per Major** to 2.

iv. 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 ($X = -8$ in, $Y = -6$ in), ($X = 0$ in, $Y = -6$ in) for the horizontal line and ($X = 6$ in, $Y = 0$ in), ($X = 6$ in, $Y = 8$ in) for the vertical line.

v. In the **Draw** toolbox, select **Arc by Center** and click once on the origin (center of the arc). Now select one of the end points of the arc, and then move the mouse around to the other end point and click on it to draw the quarter-circle. If the wrong part of the arc is drawn (that is, a 270 degree segment instead of a 90 segment), click **Undo** from the **Undo/Redo**

toolbar and try again, making sure that after you click on the first end point, that you move the mouse in the correct direction for the arc that is to be drawn.
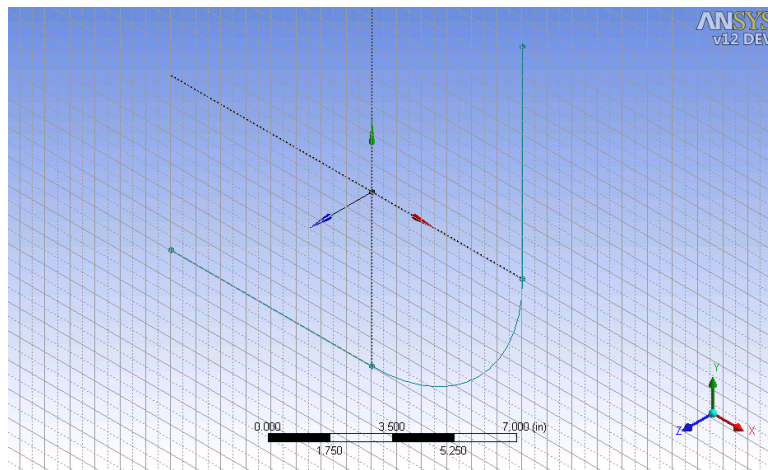

Figure 1.7: Creating the Elbow Path

(c) Create the pipe.

i. Select Sweep from the 3D Features toolbar.

ii. Set the Profile to be Sketch1: click on Sketch1 in the Tree Outline and then click on Apply in the Details View at the bottom-left of the screen.

iii. Set the Path to Sketch2: click on the Not selected text next to Path, click on Sketch2 in the Tree Outline, and then click Apply.

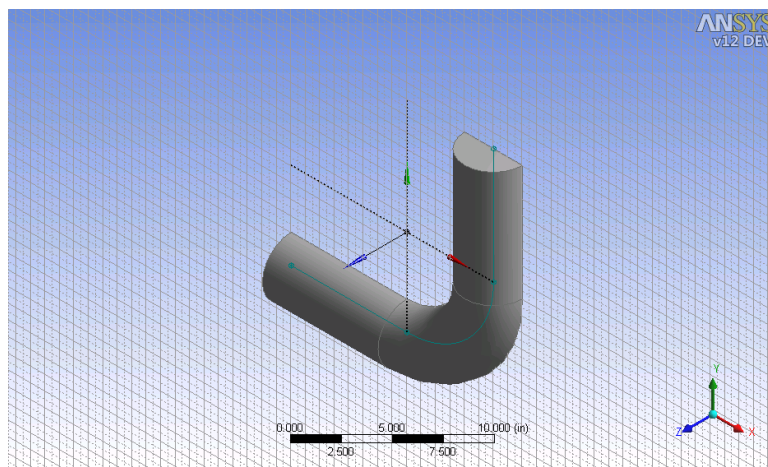iv. Click on Generate to create the pipe.

Figure 1.8: Generating the Pipe

(d) Create the side pipe.

    i. Create a new plane based on the **ZXPlane**: as before, first make **ZXPlane** active by clicking on it, then use **New Plane** to create the plane based upon it.

    ii. In the **Details View**, set **Transform 1 (RMB)** to **Offset Global X**, and set the **Value** of the offset to 5.5 in.

    iii. Set **Transform 2** to **Offset Global Y** and the **Value** of the offset to -9 in.

    iv. Click on **Generate** to create the plane.

    v. With the new plane selected in the **Tree Outline**, create a **New Sketch**.

    vi. On the **Sketching** tab, open the **Draw** toolbox and select **Arc by Center** to create the arc centered on the origin with a radius of 0.5 in. Create the arc initially with any convenient radius, and then open the **Dimensions** toolbox and select `Radius` to specify the radius more precisely.

    vii. On the **Sketching** tab, open the **Draw** toolbox and select **Line**. Draw a line connecting the open ends of the arc.

    viii. Select **Extrude** from the **3D Features** toolbar.

    ix. Set **Base Object** to be the new sketch (**Sketch3**), and set **Operation** to **Add Material**.

    x. Set **Direction** to **Normal** and **Extent Type** to **Fixed**. Set **Depth** to 4 in.

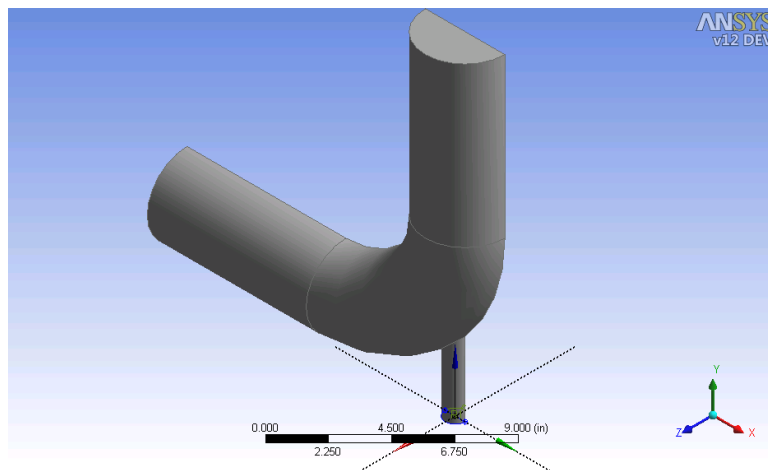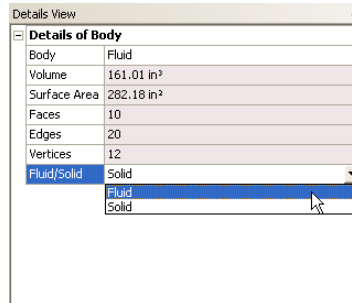   xi. Click on **Generate** to create the side-pipe.

Figure 1.9: Generating the Additional Pipe

(e) Specify the geometry as a fluid body.

    i. In the Tree Outline, open the 1 Part, 1 Body branch and select Solid branch.

    ii. In the Details View of the body, change the name of the Body from Solid to Fluid.

    iii. Change the Fluid/Solid property from Solid to Fluid.

Details View		
Details of Body		
Body	Fluid	
Volume	161.01 in³	
Surface Area	282.18 in²	
Faces	10	
Edges	20	
Vertices	12	
Fluid/Solid	Solid	
	Fluid	
	Solid	

    iv. Click on Generate.

4. Close ANSYS DesignModeler.

You can simply close the ANSYS DesignModeler application. 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).
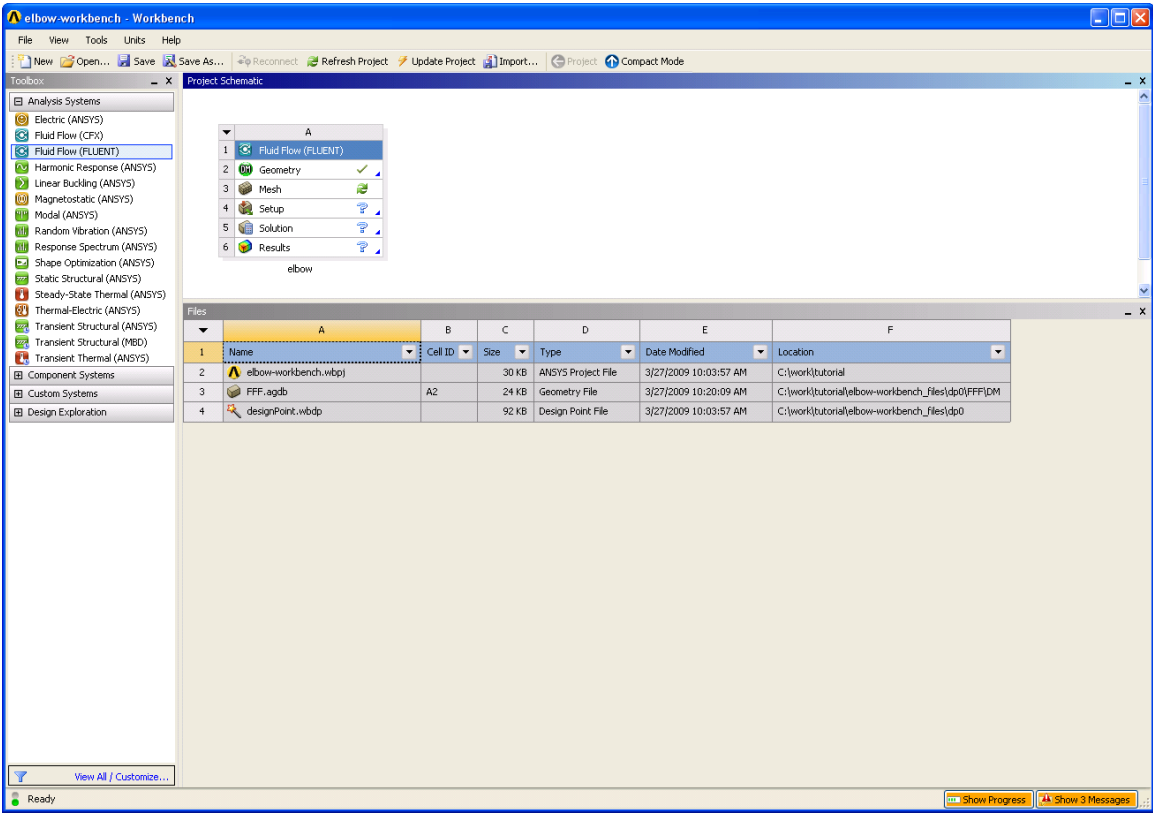
5. View the files generated by ANSYS Workbench.

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

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 from scratch), the `elbow-geometry.agdb` (or the `elbow-geometry.stp`) file would be listed instead.

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 throughout the flow volume. For this tutorial, you will use the ANSYS Meshing application to create a mesh for your CFD analysis.

i 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 (e.g., the geometry). Once the mesh is defined, the state of the cell will change accordingly. Likewise for the state of the remaining cells in the system. For more information about system cell states, see the separate FLUENT in Workbench User's Guide and the on-line documentation for ANSYS Workbench.

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 on the Mesh cell to display the context menu where you can select the Edit... option.
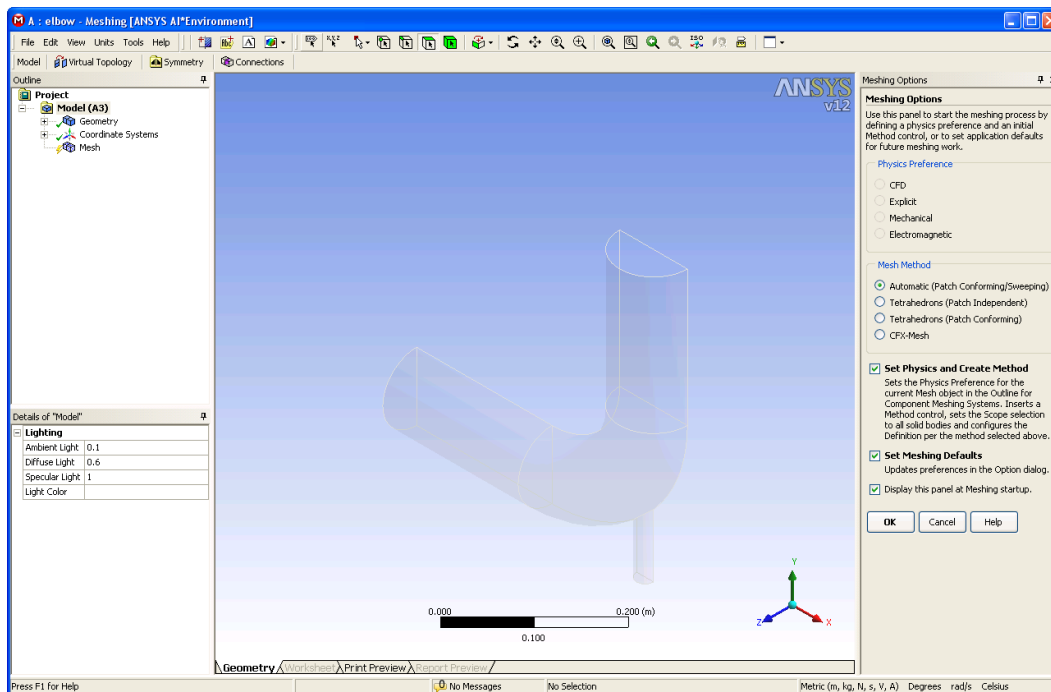

Figure 1.11: The ANSYS Meshing Application with the Elbow Geometry Loaded

> **_i_** The first time you open the ANSYS Meshing application, the Meshing Options are displayed on the right-hand side of the application window where you can set various meshing options. For the purposes of this tutorial, nothing needs be set here, so click OK to close the Meshing Options window.

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.

   (b) Right-click and select the Create Named Selection option.
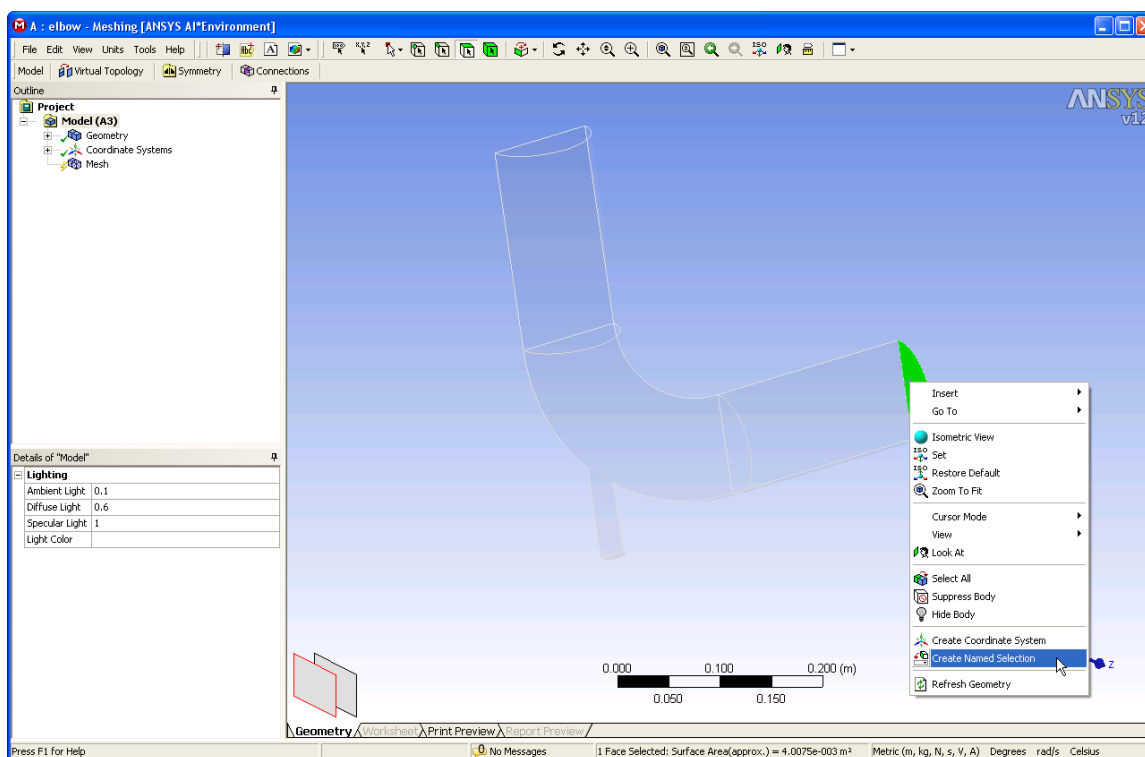
Figure 1.12: Selecting a Face to Name
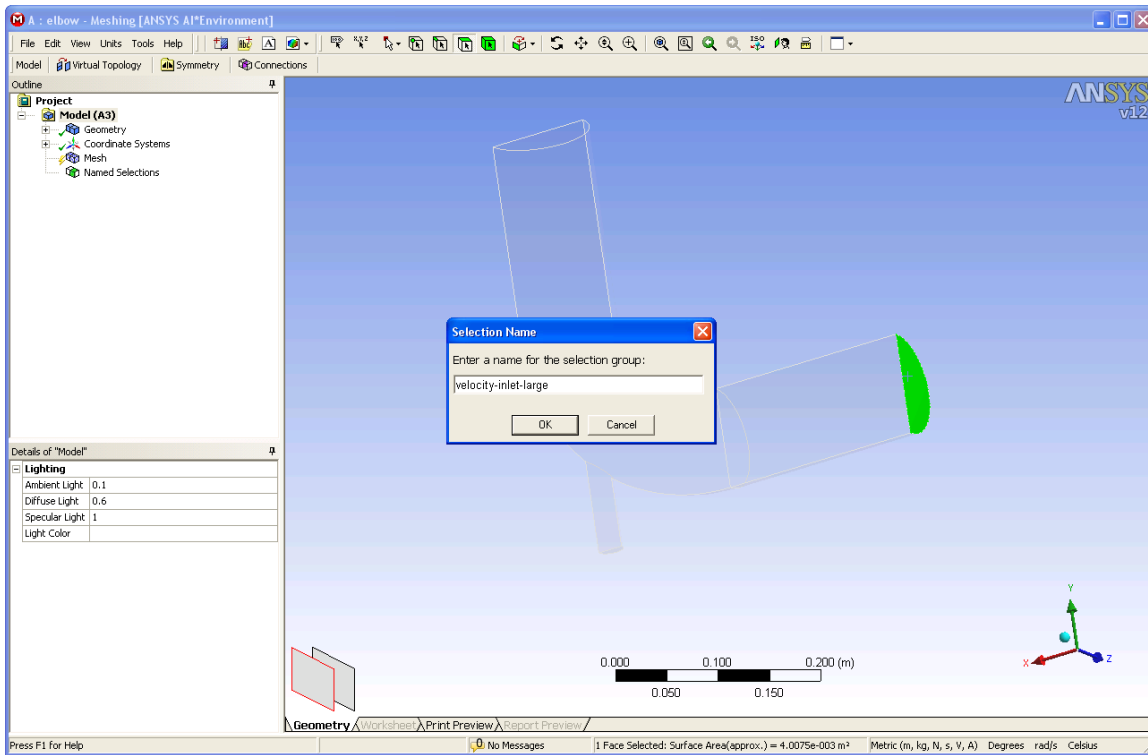
This displays the Selection Name dialog box.

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 smaller inlet `velocity-inlet-small`, the large outlet (`pressure-outlet`), and the symmetry planes (`symmetry`).

To select multiple planes, hold down the $< CTRL >$ key and select the individual planes in the graphics window.

> **i** 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.

(a) In the Outline view, select Mesh under Project/Model to display the Details view below the Outline view.

> **i** Note that since 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 to reveal additional sizing parameters, and make sure that **Use Advanced Size Function** is set to **On:Curvature**.

(c) Expand the **Inflation** node to reveal additional inflation parameters, and change **Use Automatic Tet Inflation** to **Program Controlled**.

4. Generate the mesh.

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

   Note: *Once the mesh is generated, to view the mesh statistics, open the* **Statistics** *node in the* **Details** *view to reveal information about the number of nodes, the number of elements, and other details.*
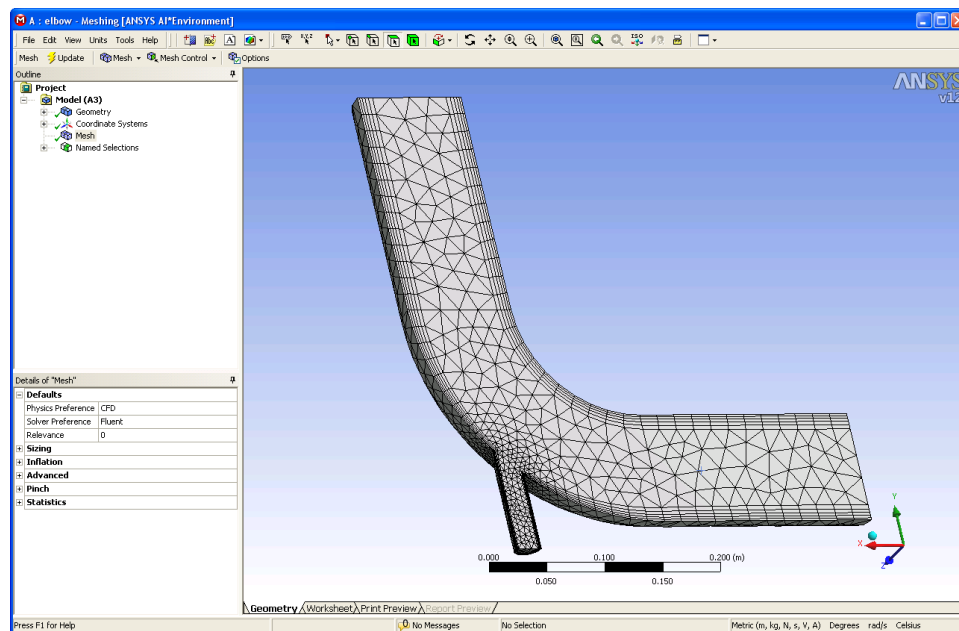
Figure 1.14: The Computational Mesh for the Elbow Geometry in the AN-SYS Meshing Application

> **i** 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 and creates the relevant mesh files for your project and updates the **ANSYS Workbench** cell that references this mesh.

5. Close the **ANSYS Meshing** application.

You can simply close the **ANSYS Meshing** application. **ANSYS Workbench** automatically saves the mesh and updates the **Project Schematic** accordingly (the Refresh Required icon in the **Mesh** cell is replaced by a check mark, indicating that there is a mesh now associated with the fluid flow analysis system).
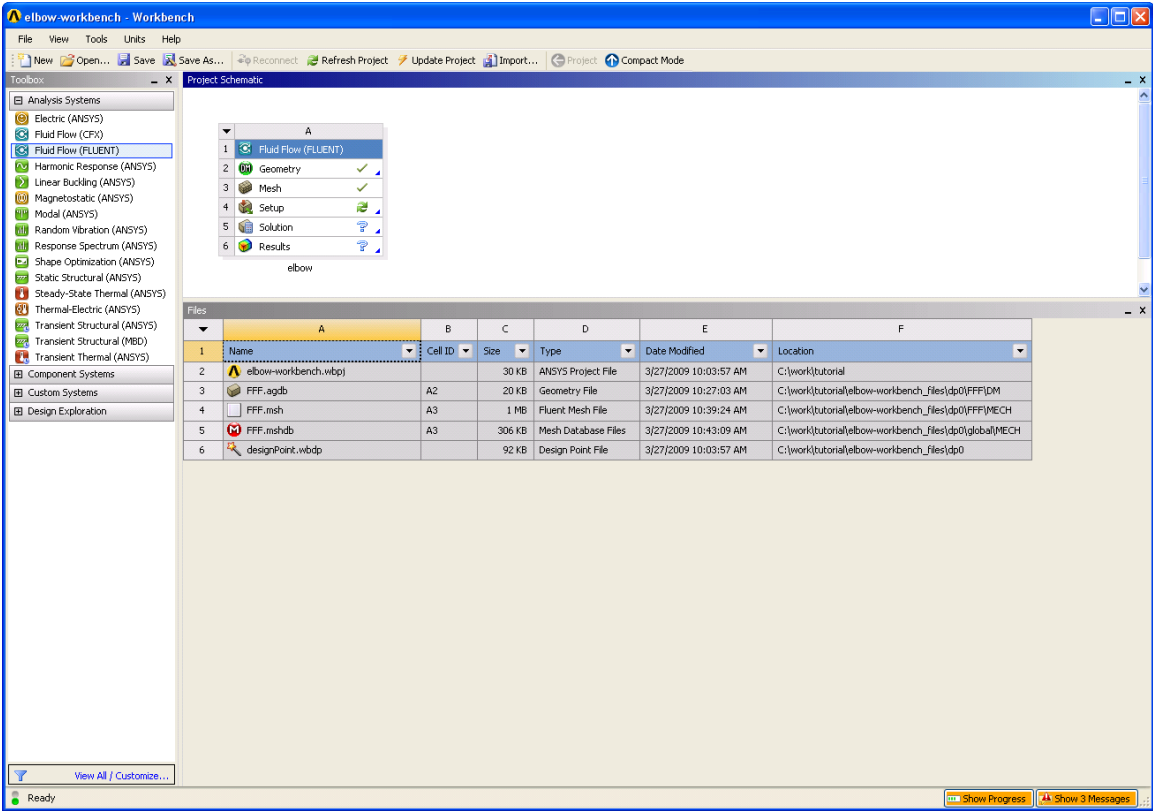
6. View the files generated by **ANSYS Workbench**.

$\boxed{\text{View}}\longrightarrow\text{Files}$

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

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 updated the mesh, and the `FFF.mshdb` file is generated when you close the **ANSYS Meshing** application.

Step 4: Setting Up the CFD Simulation in ANSYS FLUENT

Now that you have created a computational mesh for the elbow geometry, you can proceed to setting up a CFD analysis using ANSYS FLUENT.

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 on the Setup cell to display the context menu where you can select the Edit option.

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

   ANSYS FLUENT *Launcher allows you to decide which version of* ANSYS FLUENT *you will use, based on your geometry and on your processing capabilities.*
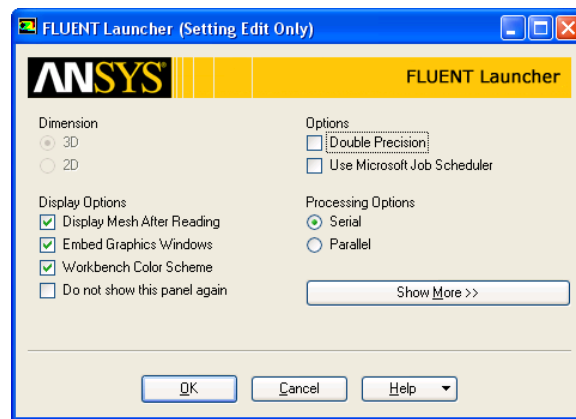
Figure 1.16: ANSYS FLUENT Launcher

   (a) Ensure that the proper options are enabled.

   i 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.
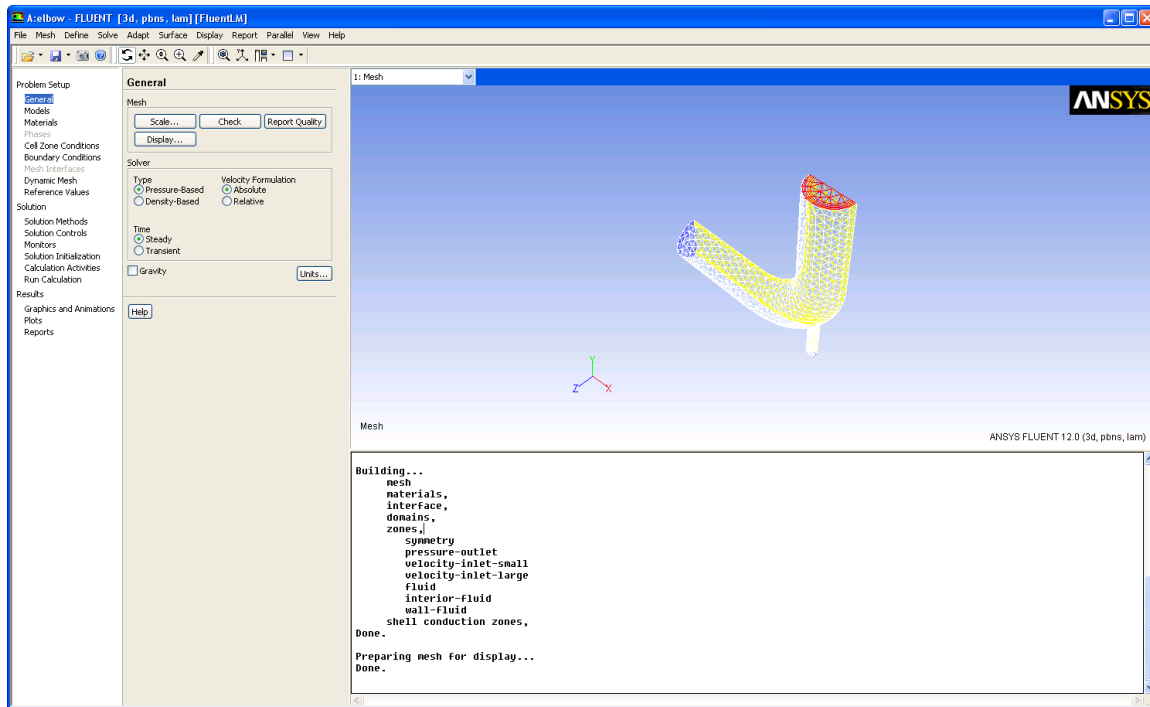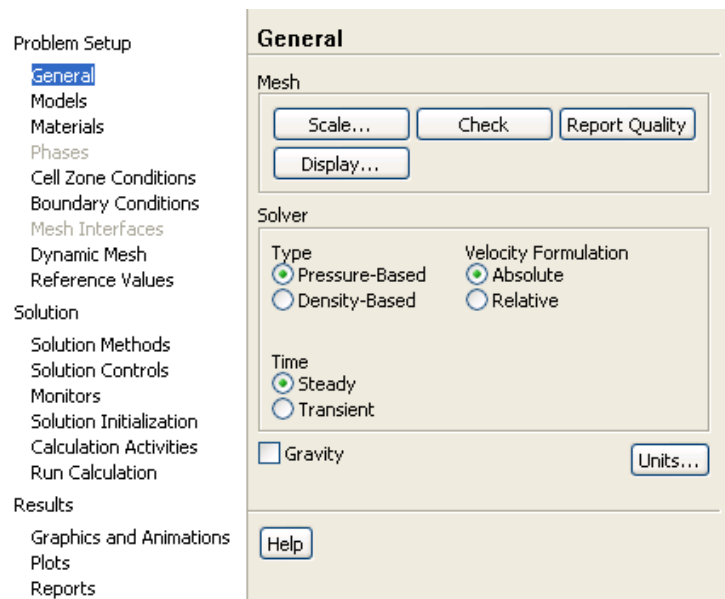
 (b) Click OK to launch ANSYS FLUENT.

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.

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

   ⬦ General

(a) Change the units for length.

Since we 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.

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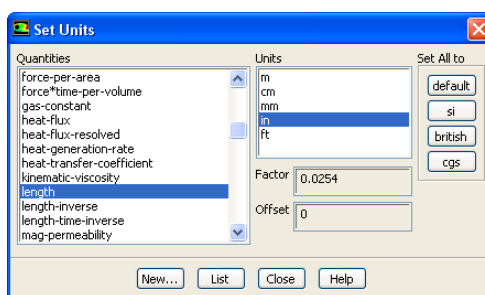
i. Select length in the Quantities list.

ii. Select in in the Units list.

iii. Close the dialog box.

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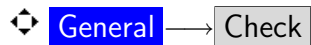
ANSYS FLUENT *will report the results of the mesh check in the console.*

```
  Domain Extents:
    x-coordinate: min (m) = -2.032000e-001, max (m) = 2.032000e-001
    y-coordinate: min (m) = -2.286000e-001, max (m) = 2.032000e-001
    z-coordinate: min (m) = -2.332875e-018, max (m) = 5.079992e-002
  Volume statistics:
    minimum volume (m3): 1.106292e-009
    maximum volume (m3): 1.406825e-006
      total volume (m3): 2.607593e-003
  Face area statistics:
    minimum face area (m2): 1.042914e-006
    maximum face area (m2): 2.850931e-004
  Checking number of nodes per cell.
  Checking number of faces per cell.
  Checking thread pointers.
  Checking number of cells per face.
  Checking face cells.
  Checking cell connectivity.
  Checking bridge faces.
  Checking right-handed cells.
  Checking face handedness.
  Checking face node order.
  Checking closed cells.
  Checking contact points.
  Checking element type consistency.
  Checking boundary types:
  Checking face pairs.
  Checking wall distance.
  Checking node count.
  Checking nosolve cell count.
  Checking nosolve face count.
  Checking face children.
  Checking cell children.
  Checking storage.
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, since* ANSYS FLUENT *cannot begin a calculation when this is the case.*
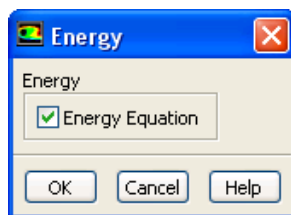
3. Set up your models for the CFD simulation.

  ✛ Models
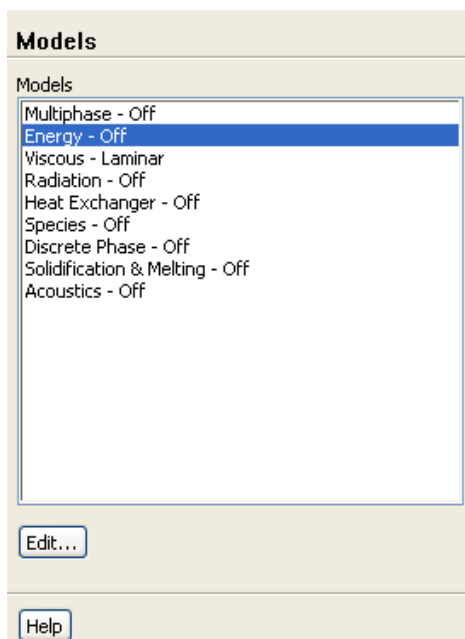
(a) Enable heat transfer by activating the energy equation.

  ✛ Models ⟶ ☰ Energy ⟶ Edit...

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-ϵ turbulence model.

✛ Models ⟶ ≣ Viscous ⟶ Edit...
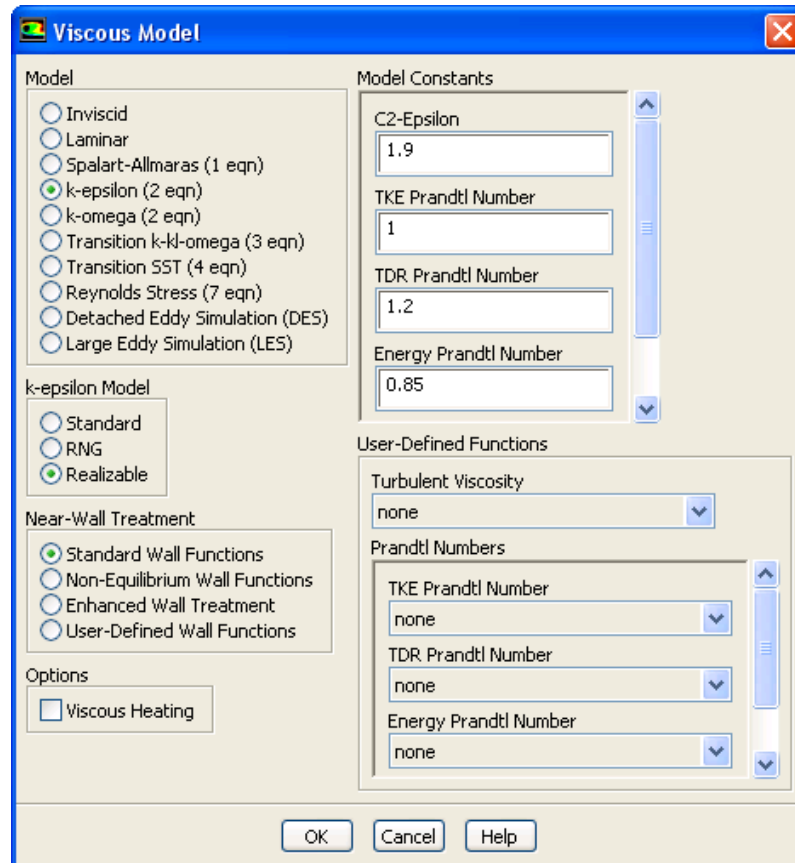
i. Select k-epsilon from the Model list.

The Viscous Model *dialog box will expand.*

ii. Select Realizable from the k-epsilon Model list.

iii. Click OK to accept the model and close the Viscous Model dialog box.

4. Set up your materials for the CFD simulation.

⬩ Materials

Materials

Materials
Fluid
air
Solid
aluminum

Create/Edit...   Delete

Help
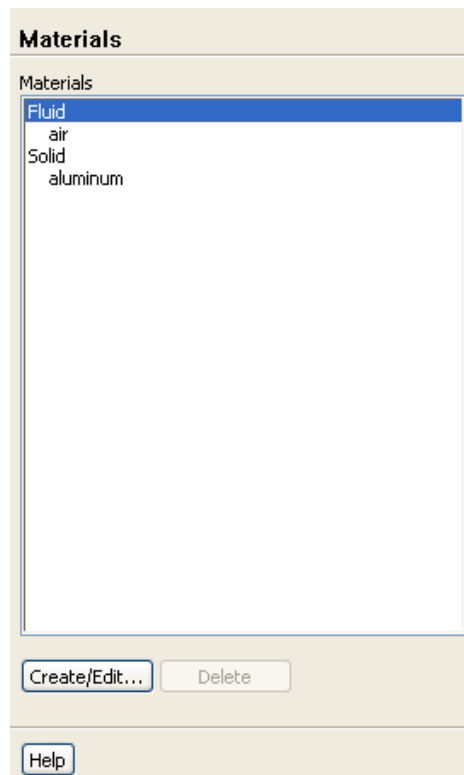
(a) Create a new material called water using the Create/Edit Materialsdialog box (Figure 1.18).

⬩ Materials ⟶ ☰ Fluid ⟶ Create/Edit...

i. Enter water for Name.

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

Property	Value
Density	1000 kg/m^3
c_p	4216 J/kg $-$ K
Thermal Conductivity	0.677 W/m $-$ K
Viscosity	8e-04 kg/m $-$ s
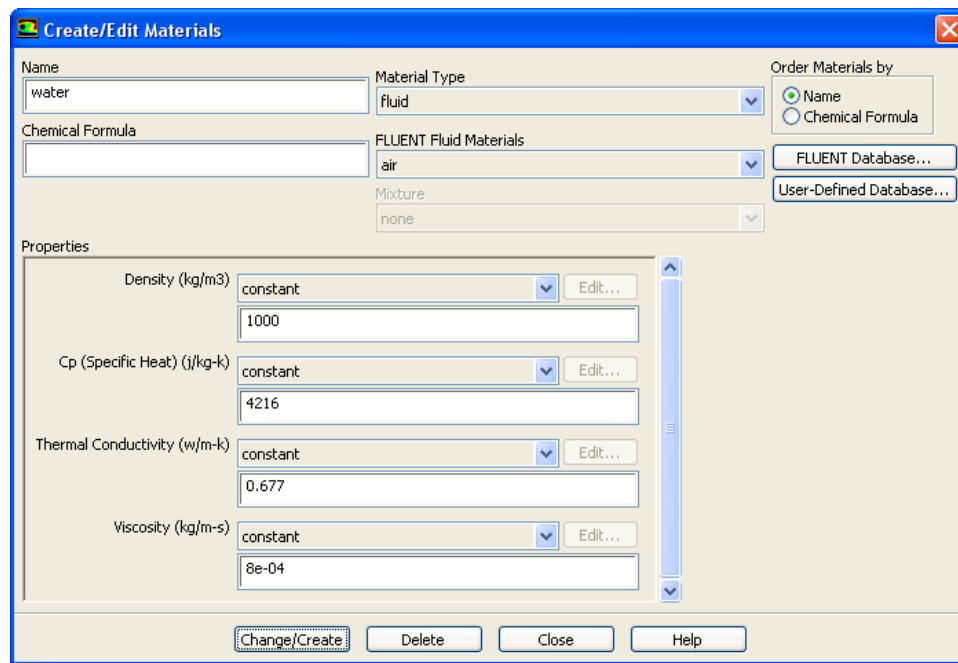
Figure 1.18: The Create/Edit Materials Dialog Box

iii. Click Change/Create.

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 list of materials which originally contained only* air.
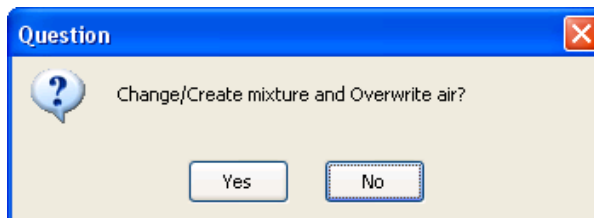
Extra: *You could have copied the material* water-liquid (h2o<l>) *from the materials database (accessed by clicking the* FLUENT Database... *button). If the properties in the database are different from those you wish to use, you can edit the values in the* Properties *group box in the* Create/Edit 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.

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

v. Close the Create/Edit Materials dialog box.

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

5. Set up the cell zone conditions for the CFD simulation.

   ✛ Cell Zone Conditions
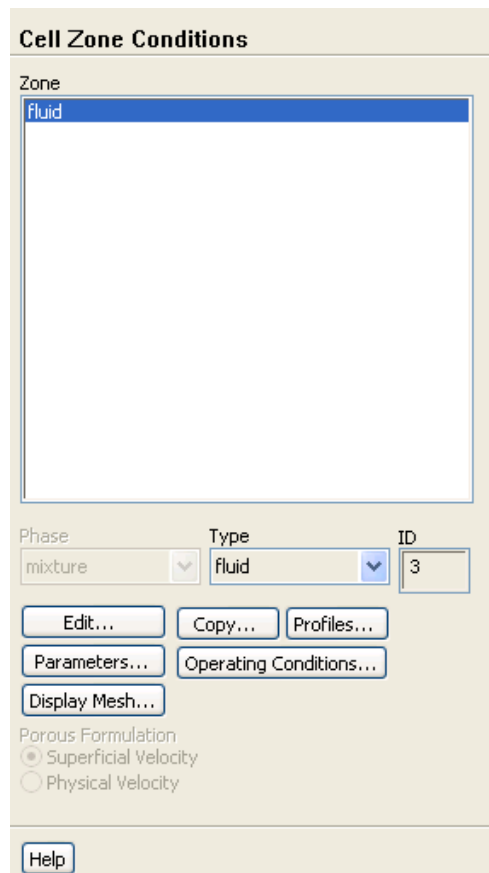
(a) Set the cell zone conditions for the fluid zone.

   i. Select fluid in the Zones list in the Cell Zone Conditions task page, then click the Edit... button to open the Fluid dialog box.
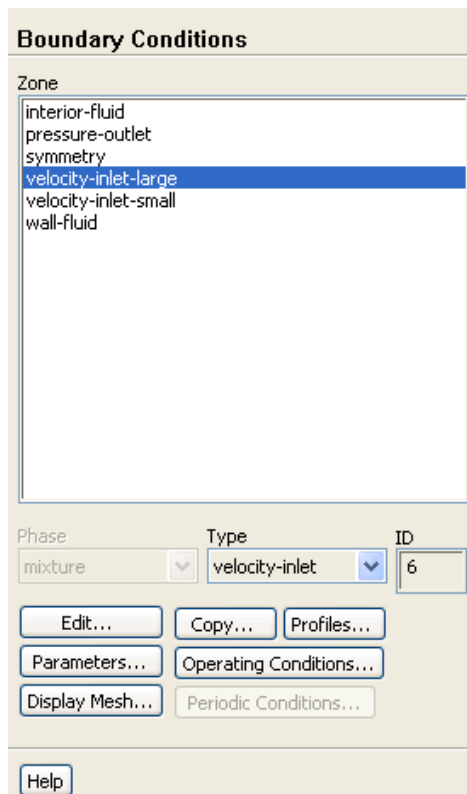
   You can also double-click a list item in order to open the corresponding dialog box.

1-32

Release 12.1 © ANSYS, Inc. September 10, 2009
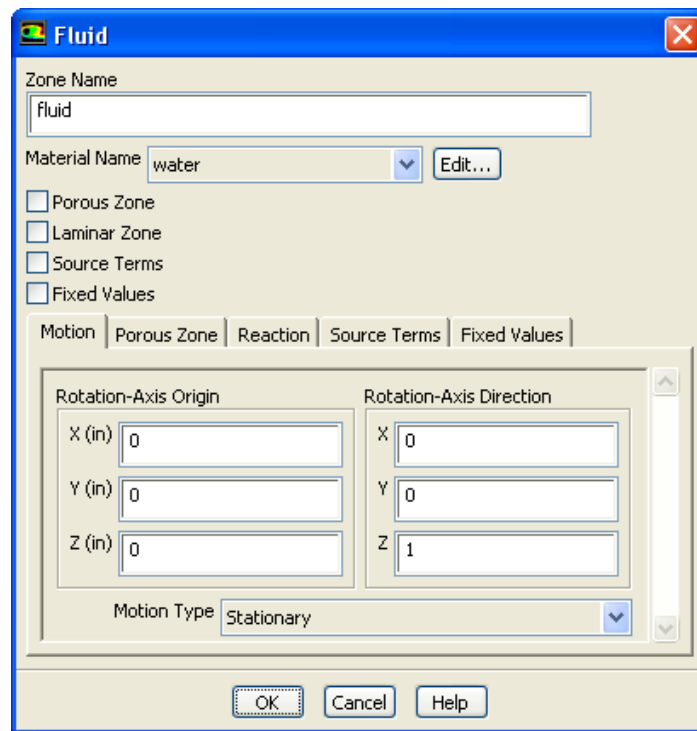
   ii. In the **Fluid** dialog box, select **water** from the **Material Name** drop-down
       list.

  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...

Hint: *If you are unsure of which inlet zone corresponds to the cold inlet, you can probe the mesh display using the right mouse button or the probe toolbar button (   ) as described in a previous step. The information will be displayed in the* ANSYS FLUENT *console, and the zone you probed will be automatically selected from the* Zone *selection list in the* Boundary Conditions *task page.*
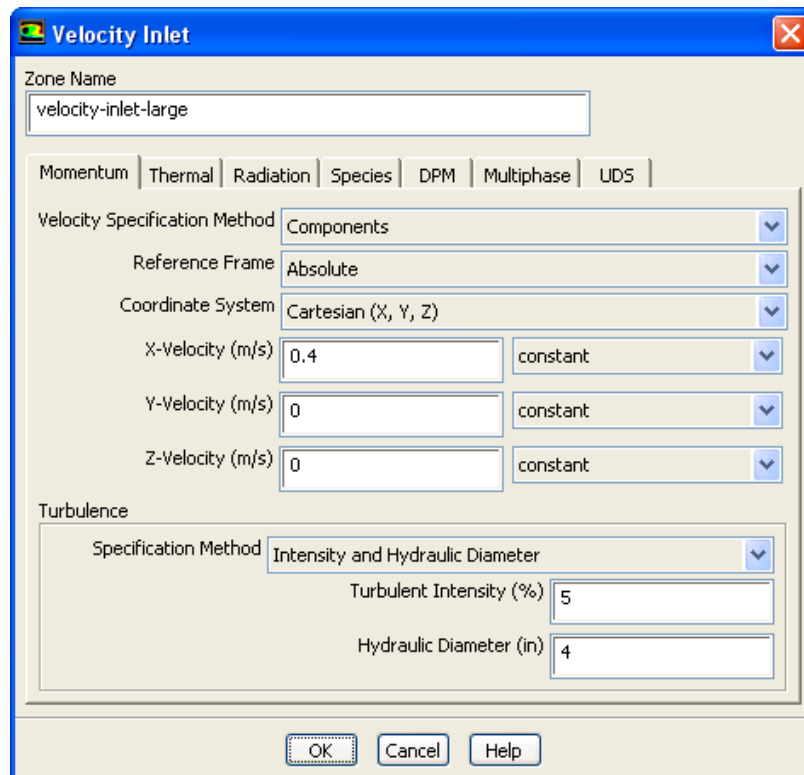
i. Select Components from the Velocity Specification Method drop-down list.

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. Select Intensity and Hydraulic Diameter from the Specification Method drop-down list in the Turbulence group box.
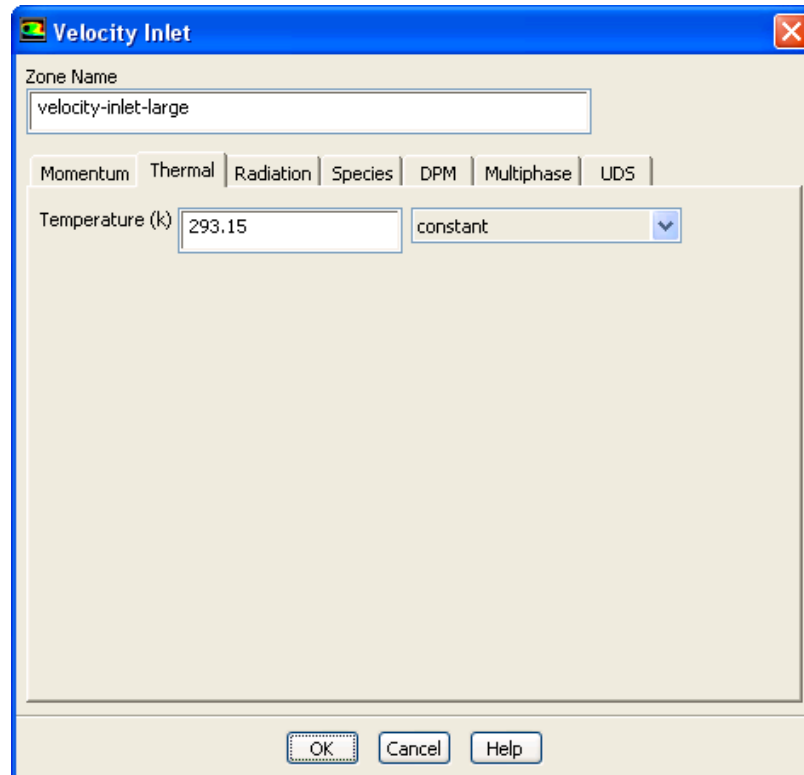
v. Enter 5% for Turbulent Intensity.

vi. Enter 4 in for Hydraulic Diameter.

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
Z-Velocity	0 m/s
Specification Method	Intensity & Hydraulic Diameter
Turbulent Intensity	5%
Hydraulic Diameter	1 in
Temperature	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.

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) Change the convergence criteria for the continuity equation residual.

⬥ Monitors ⟶ ☰ Residuals ⟶ Edit...
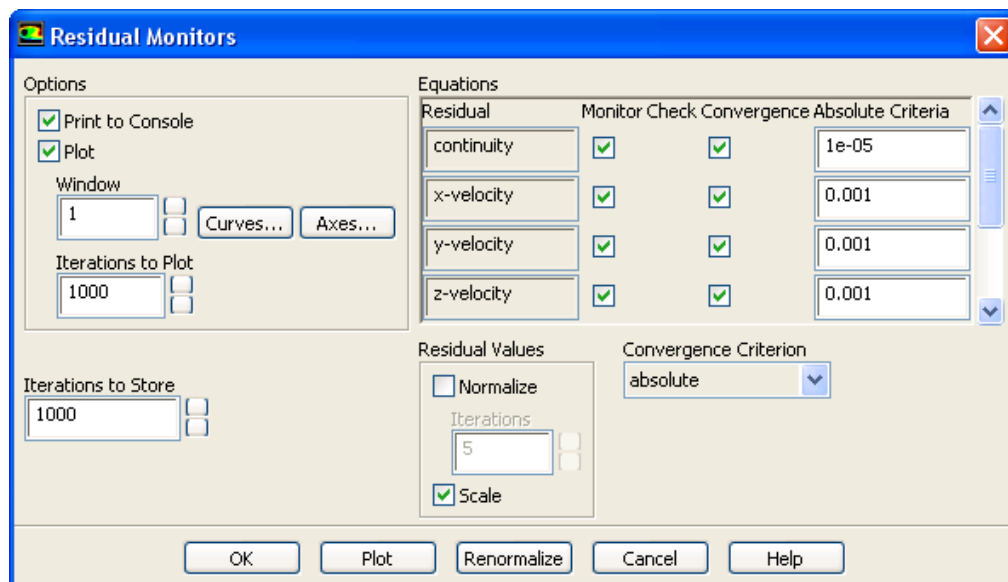
i. Make sure that Plot is enabled in the Options group box.

ii. Enter 1e-05 for the Absolute Criteria of continuity, as shown in the Residual Monitor dialog box.

iii. Click OK to close the Residual Monitors dialog box.

Note: *By default, all variables will be monitored and checked by* ANSYS FLU-ENT *as a means to determine the convergence of the solution.*

(b) Initialize the flow field, using the boundary conditions settings at the cold inlet (**velocity-inlet-large**) as a starting point.

✥ Solution Initialization

i. Select **velocity-inlet-large** from the **Compute From** drop-down list.

ii. Enter 1.2 m/s for **Y Velocity** in the **Initial Values** group box.

   Note: *While an initial* X Velocity *is an appropriate guess for the horizontal section, the addition of a* Y Velocity *component will give rise to a better initial guess throughout the entire elbow.*

iii. Click **Initialize**.

(c) Check to see if the case conforms to best practices.

✧ Run Calculation ⟶ Check Case
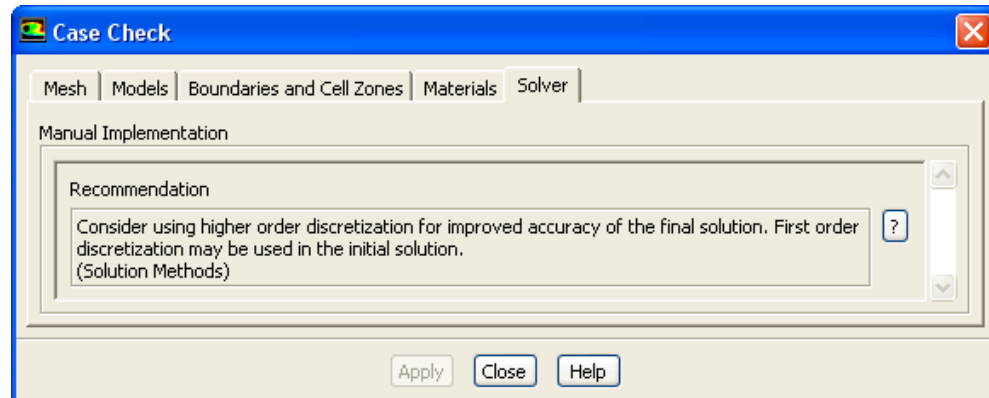
i. Click the Solver tab and examine the Recommendation in the Manual Implementation group box.

The only recommendation for this mesh is to use discretization of a higher order. This recommendation can be ignored for the time being, as it was performed in the first tutorial in the separate Tutorial Guide.

ii. Close the Case Check dialog box.

i Note that, while you are working in the ANSYS FLUENT application, the states of the Setup and Solution cells in the fluid flow 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 Solution Initialization 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).

For more information, see the separate FLUENT in Workbench User's Guide.

8. Calculate a solution.

   (a) Start the calculation by requesting 250 iterations.

   ✣ Run Calculation
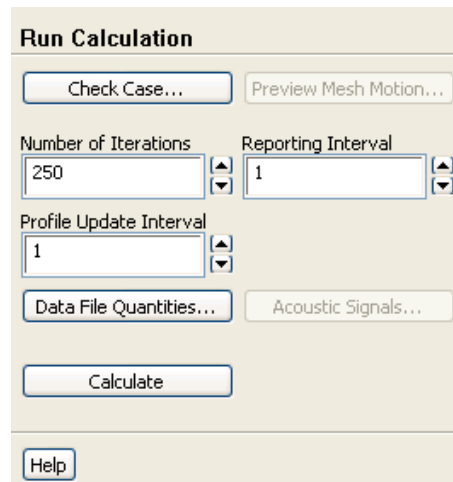
   i. Enter 250 for Number of Iterations.

   ii. Click Calculate.

   ℹ️ Note that ANSYS FLUENT settings file is written before the calculation begins. For more information about settings files, see the separate FLUENT in Workbench User's Guide.

As the calculation progresses, the residuals will be plotted in the graphics window (Figure 1.19).
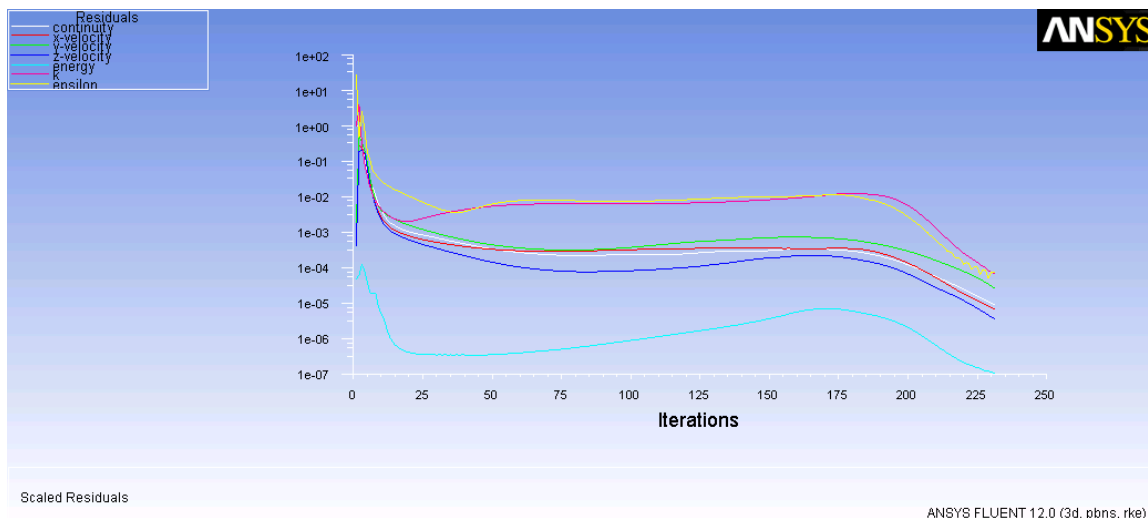

Figure 1.19: Residuals for the Converged Solution

Note: *The solution will be stopped by* ANSYS FLUENT *after approximately 230 iterations, when the residuals reach their specified values. 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.*

Since 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.

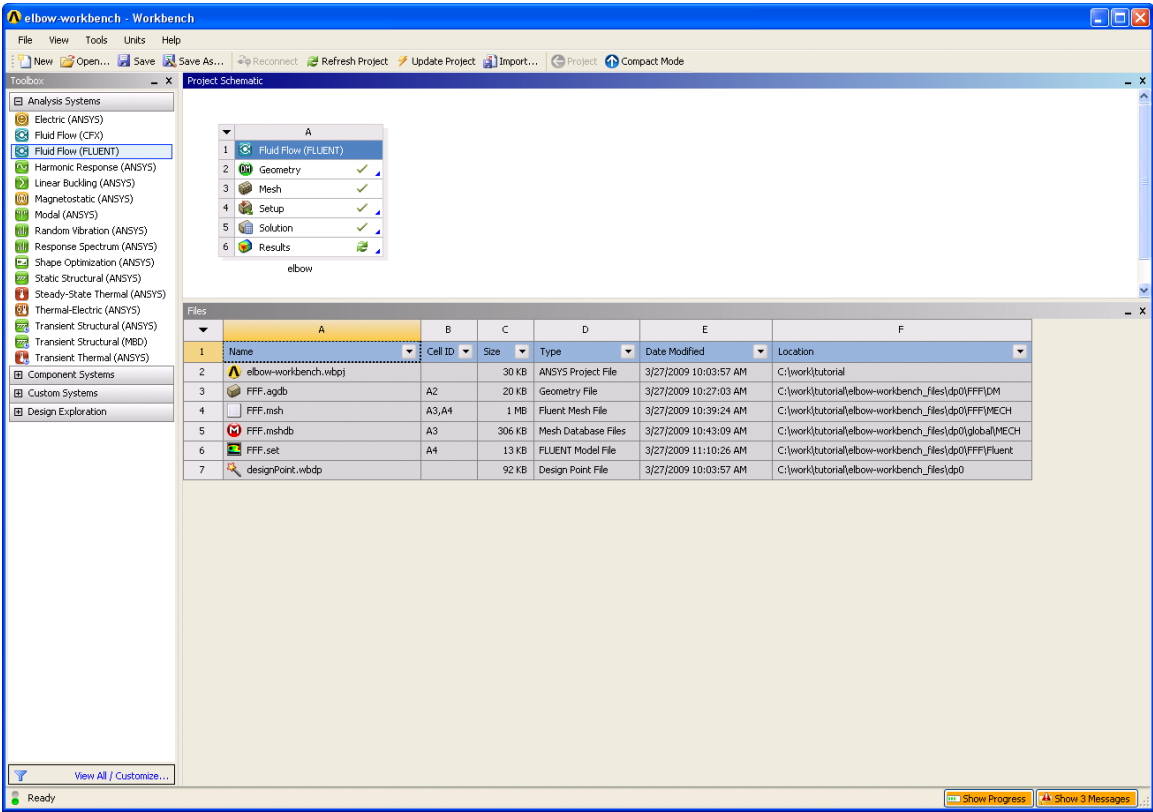9. View the files generated by ANSYS Workbench.

   View ⟶Files

Figure 1.20: ANSYS Workbench Displaying the Files View for the Project After Generating a Solution

Note the addition of the ANSYS FLUENT settings file (`FFF.set`) to the list of files. Also note that the status of the Solution cell is now up-to-date.

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

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 ANSYS CFD-Post (from within ANSYS Workbench) to perform the same evaluation.

   (a) Display filled contours of velocity magnitude on the symmetry plane (Figure 1.21).

   ◈ Graphics and Animations ⟶ ▤ Contours ⟶ Set Up...

   You can also double-click a list item in order to open the corresponding dialog box.
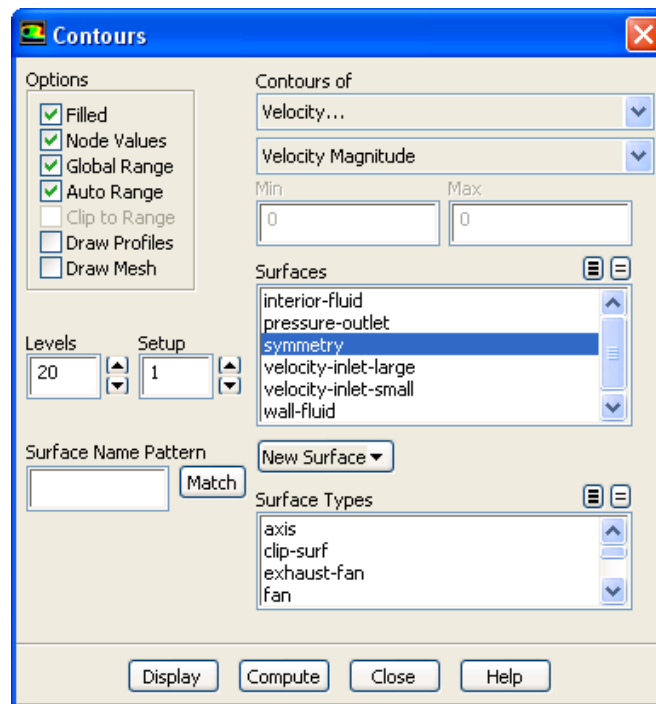
   i. Enable Filled in the Options group box.
   ii. Make sure that Node Values is enabled in the Options group box.

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

iv. Select symmetry from the Surfaces selection list.

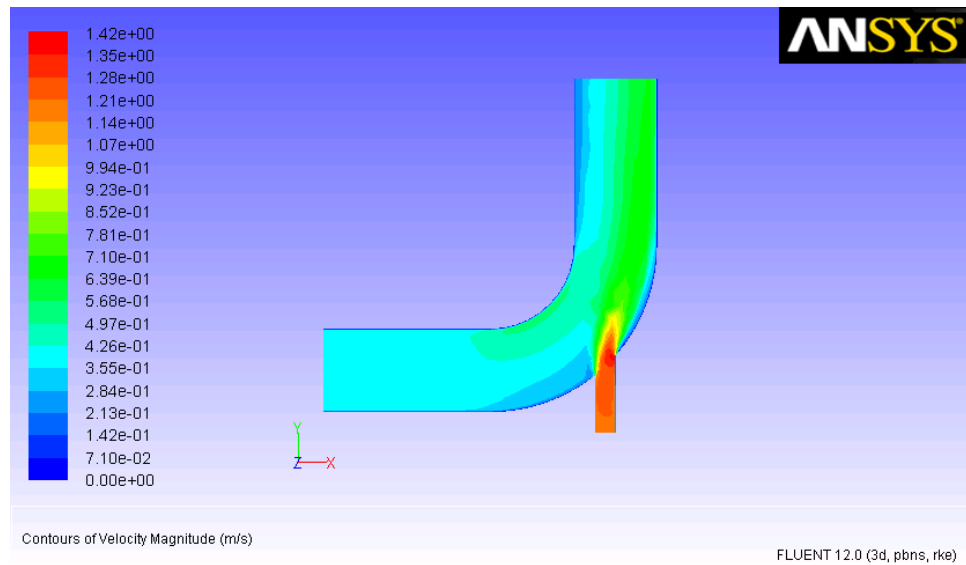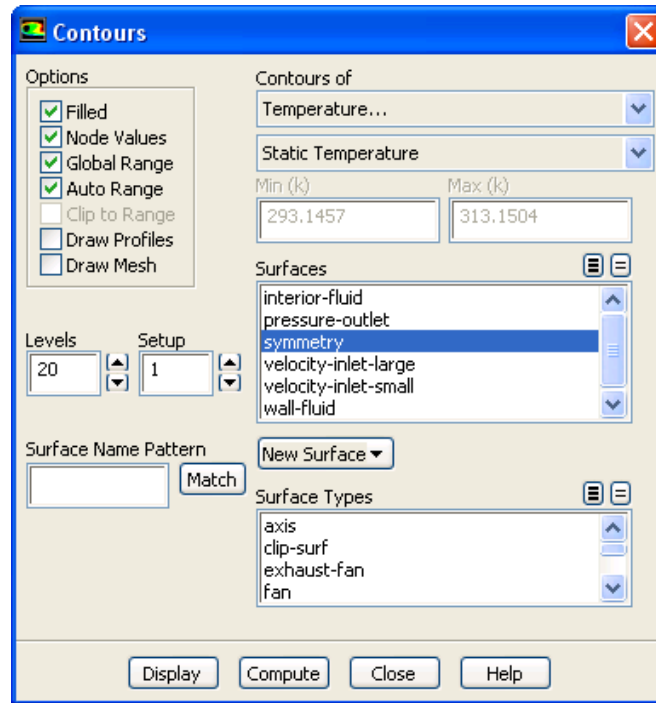v. Click Display to display the contours in the active graphics window.

Figure 1.21: Velocity Distribution Along Symmetry Plane

(b) Display filled contours of temperature on the symmetry plane (Figure 1.22).

✣ Graphics and Animations ⟶ ▤ Contours ⟶ Set Up...

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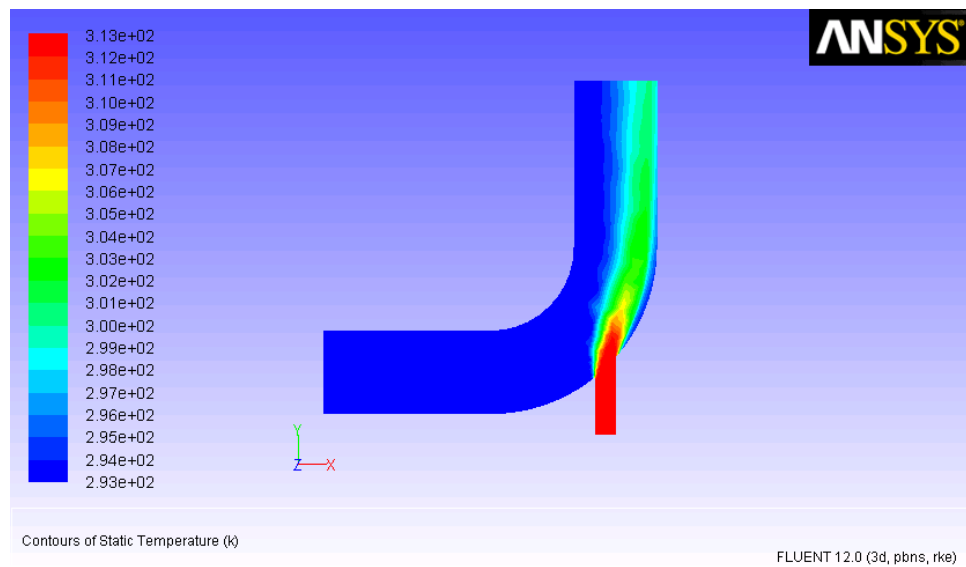
ii. Click Display and close the Contours dialog box.

Figure 1.22: Temperature Distribution Along Symmetry Plane

(c) Close the ANSYS FLUENT application.

File —→Close FLUENT

> **i** 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 files generated by ANSYS Workbench.
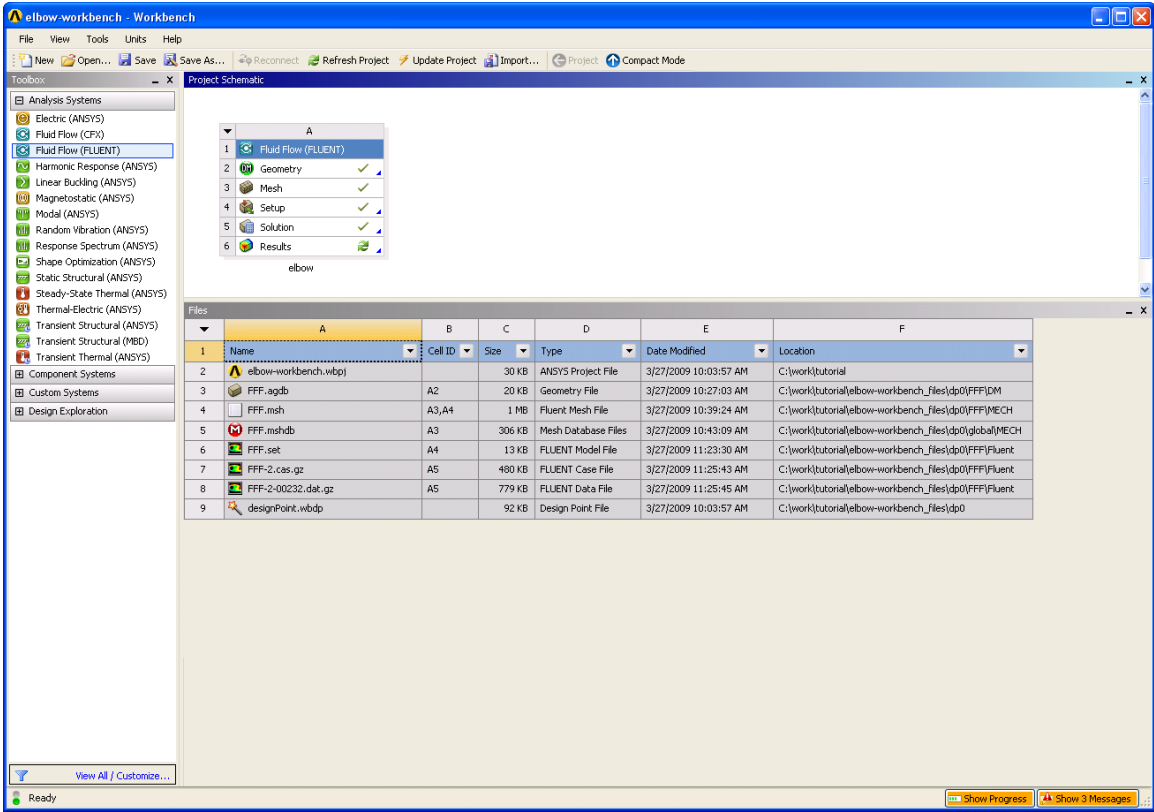
               View ⟶Files

Figure 1.23: ANSYS Workbench Displaying the Files View for the Project
After Exiting ANSYS FLUENT

Note the addition of the compressed ANSYS FLUENT case file (`FFF-1.cas.gz`)
and corresponding data file (`FFF-1-00231.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 ANSYS CFD-Post.

(a) Start ANSYS 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 ANSYS CFD-Post application. You can also right-click on the Results cell to display the context menu where you can select the Edit option.

This displays the ANSYS CFD-Post *application with the elbow geometry already loaded (displayed in outline mode). Note that* ANSYS FLUENT *results (e.g., case and data files) are also automatically loaded into* ANSYS CFD-Post.
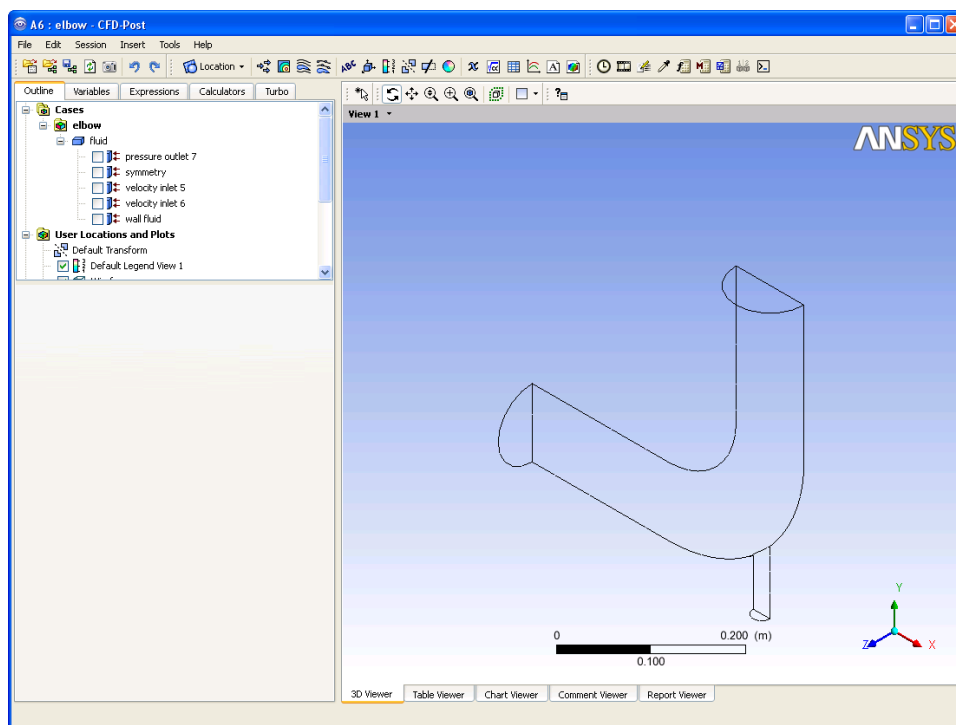

Figure 1.24: The Elbow Geometry Loaded into ANSYS CFD-Post

(b) Re-orient 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) Display filled contours of velocity magnitude on the symmetry plane (Figure 1.25).

i. Insert a contour object.

Insert $\longrightarrow$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* ANSYS 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.

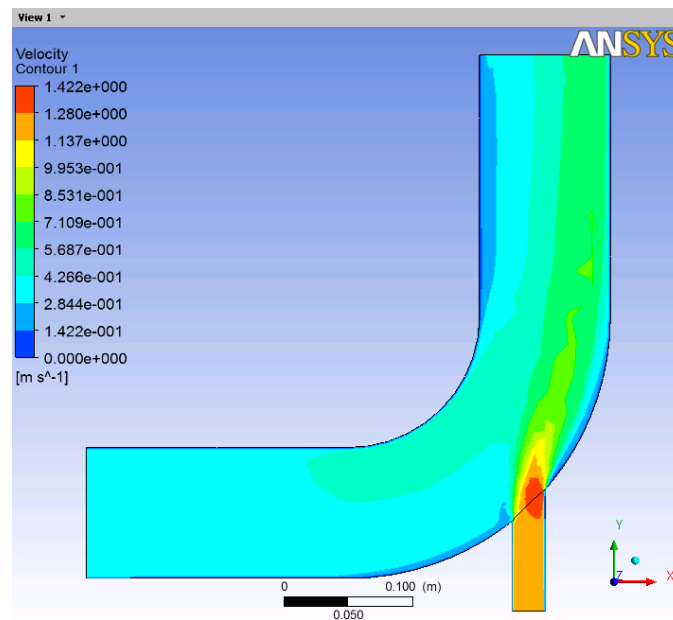v. Select Velocity in the Variable list.

vi. Click Apply.

Figure 1.25: Velocity Distribution Along Symmetry Plane

(d) Display filled contours of temperature on the symmetry plane (Figure 1.26).

   i. Deselect the Contour 1 object under User Location and Plots in ANSYS CFD-Post to 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.

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

   v. Select symmetry in the Locations list.

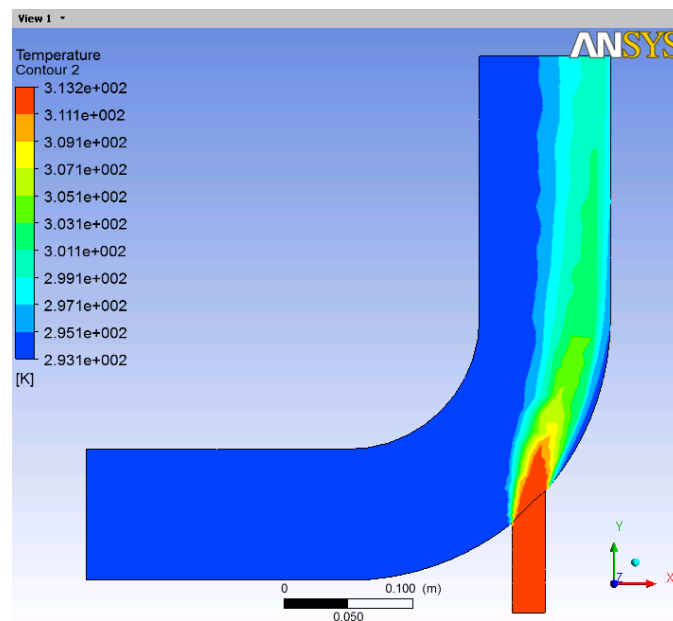   vi. Select Temperature in the Variable list.

   vii. Click Apply.

Figure 1.26: Temperature Distribution Along Symmetry Plane

3. Close the ANSYS CFD-Post application.

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

4. Save the `elbow-workbench` project in ANSYS Workbench.

5. View the files generated by ANSYS Workbench.
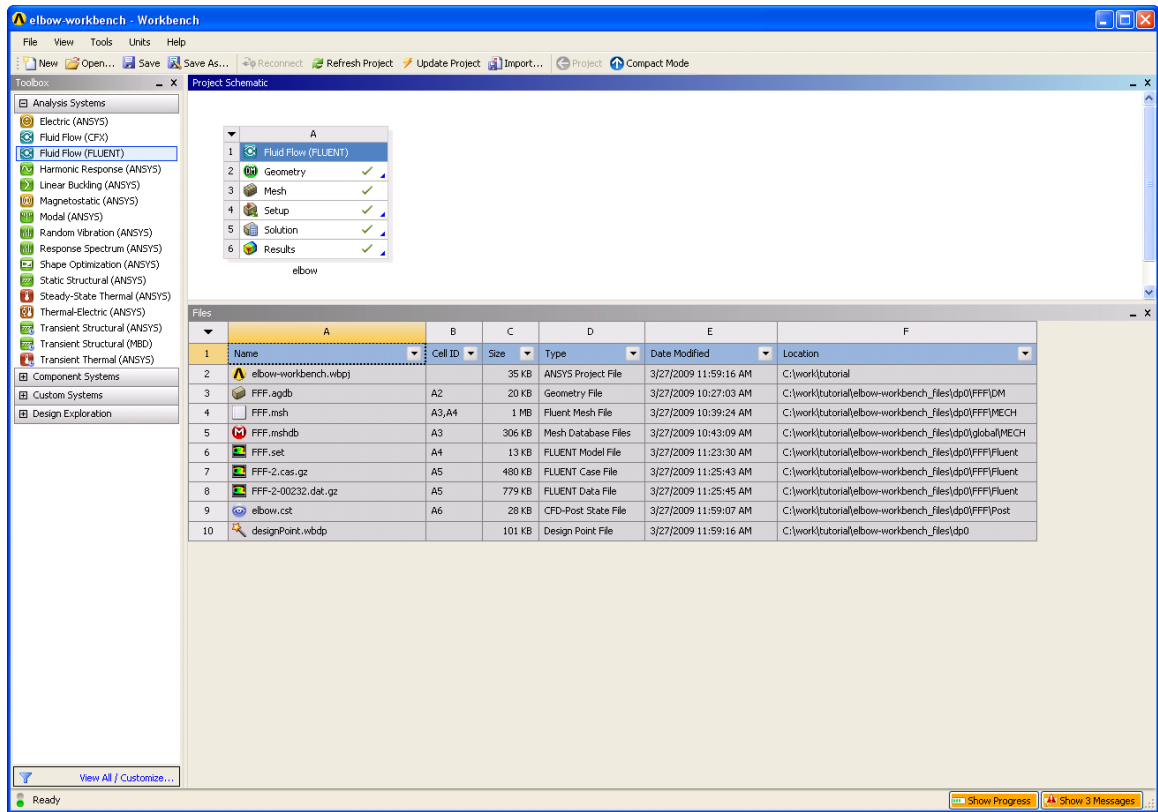
   View —→Files

Figure 1.27: ANSYS Workbench Displaying the Files View for the Project
After Viewing Results in ANSYS CFD-Post

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

Step 6: Duplicating the ANSYS 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 ANSYS FLUENT-based fluid flow system.

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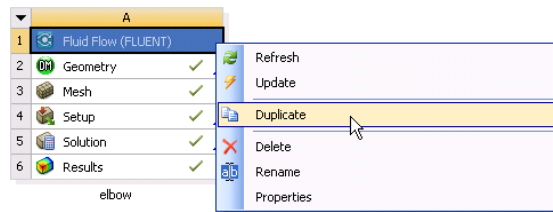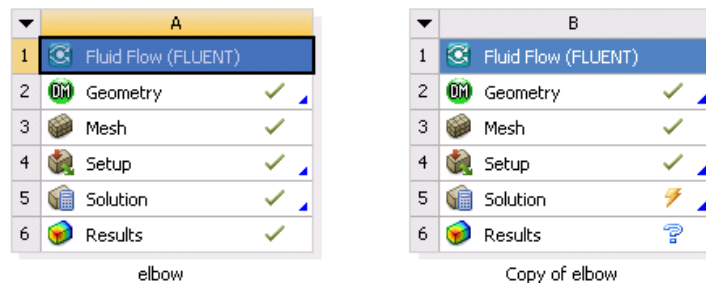

Figure 1.28: Duplicating the Fluid Flow System

Figure 1.29: The Original Fluid Flow System and Its Duplicate

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.

Step 7: Changing the Geometry in ANSYS DesignModeler

Now that you have two separate, but equivalent, ANSYS 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.

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` in to `0.75` in.

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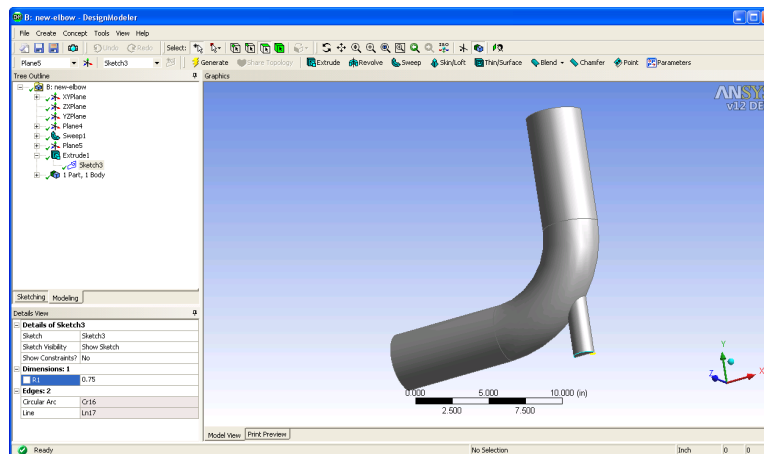

Figure 1.30: Changing the Diameter of the Small Inlet in ANSYS DesignModeler

3. Close ANSYS DesignModeler.

4. View the files generated by ANSYS Workbench.

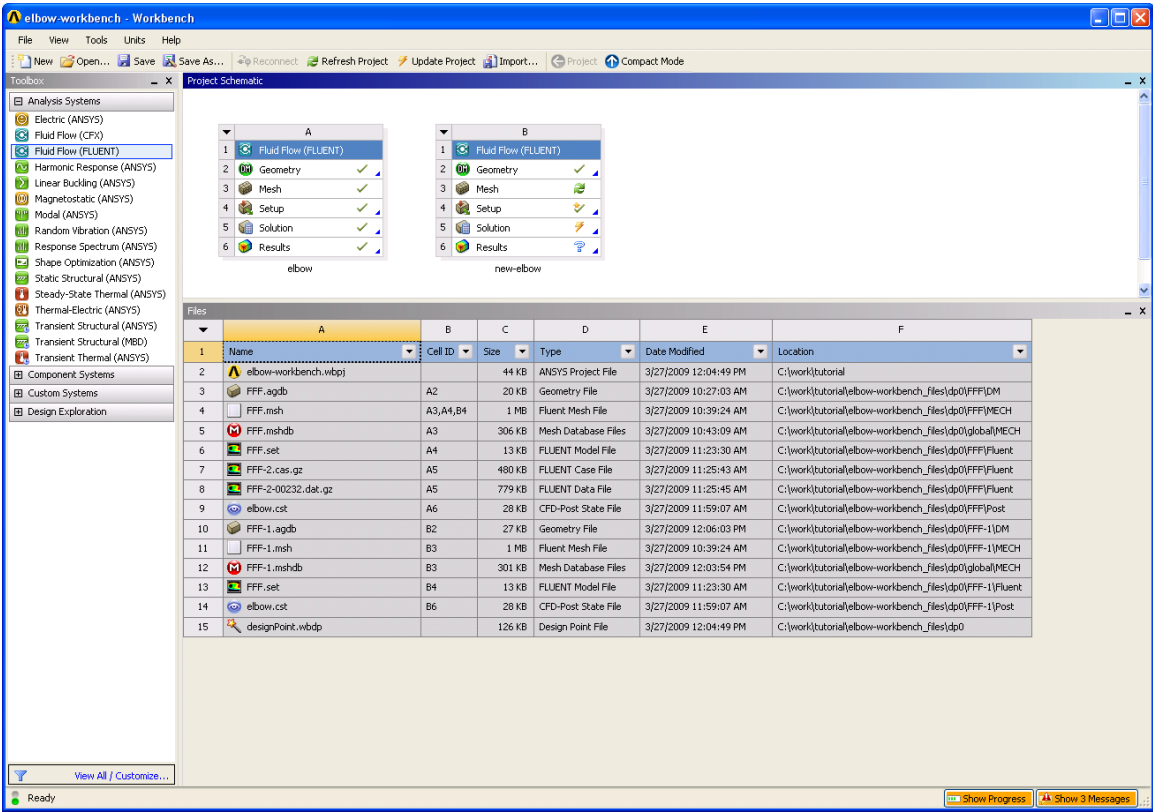$\boxed{\text{View}}$ $\longrightarrow$ Files

Figure 1.31: ANSYS Workbench Displaying the Files View for the Project After Duplicating the System and Changing the Geometry

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

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, so all you need to do is update the mesh based on the mesh settings from the original system.

In the Project Schematic, right-click on 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.
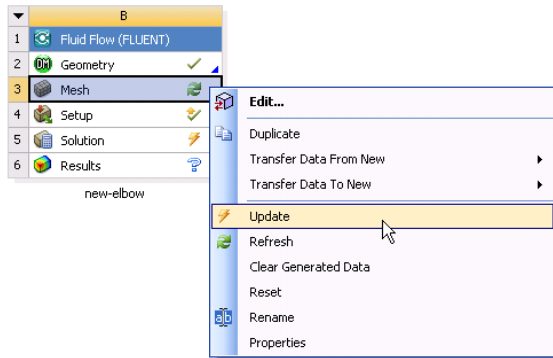

Figure 1.32: Updating the Mesh for the Changed Geometry

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 is displayed below.

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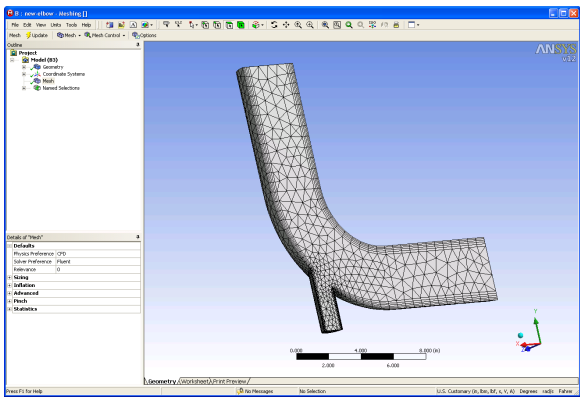
View ⟶ Files

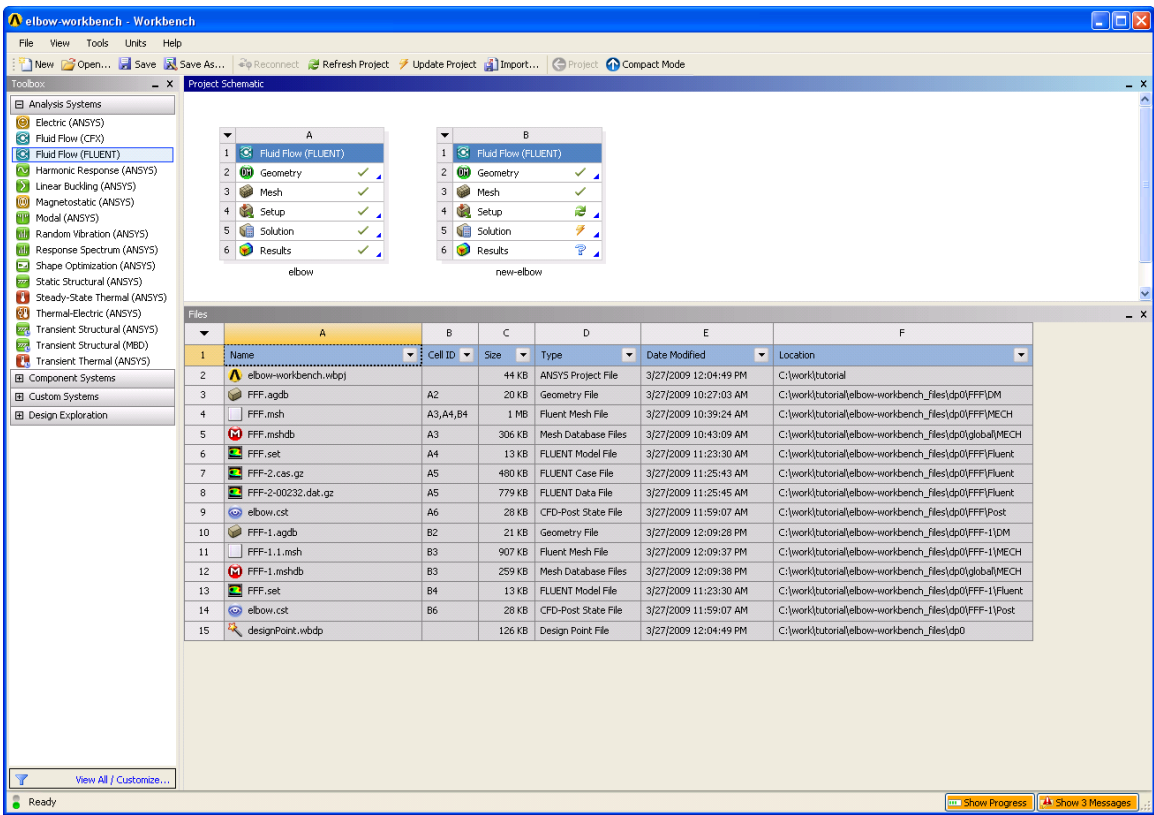Figure 1.33: The Updated Geometry and Mesh in the ANSYS Meshing Application

Figure 1.34: ANSYS Workbench Displaying the Files View for the Project After Updating the Mesh for the Altered Geometry

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, then view the new results.

1. Open ANSYS FLUENT.

   In the Project Schematic, right-click on 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.
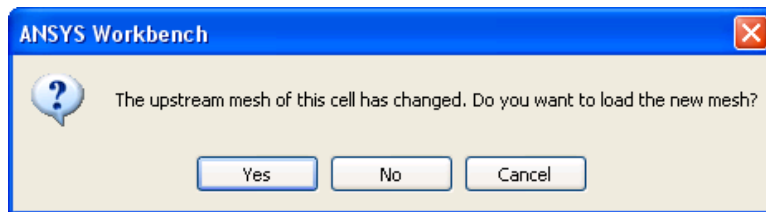

   Figure 1.35: ANSYS Workbench Prompt When the Upstream Mesh Has Changed

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

Again, use the `velocity-inlet-large` boundary condition to initialize the solution, adding the Y Velocity component of `1.2` as you did before.

6. Recalculate the solution.

✥ Run Calculation

Keep the Number of Iterations *set to* 250*. This time, the solution converges after approximately 220 iterations.*

7. Close ANSYS FLUENT.

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

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

9. Close ANSYS CFD-Post.

10. Save the `elbow-workbench` project in ANSYS Workbench.

11. View the files generated by ANSYS Workbench.
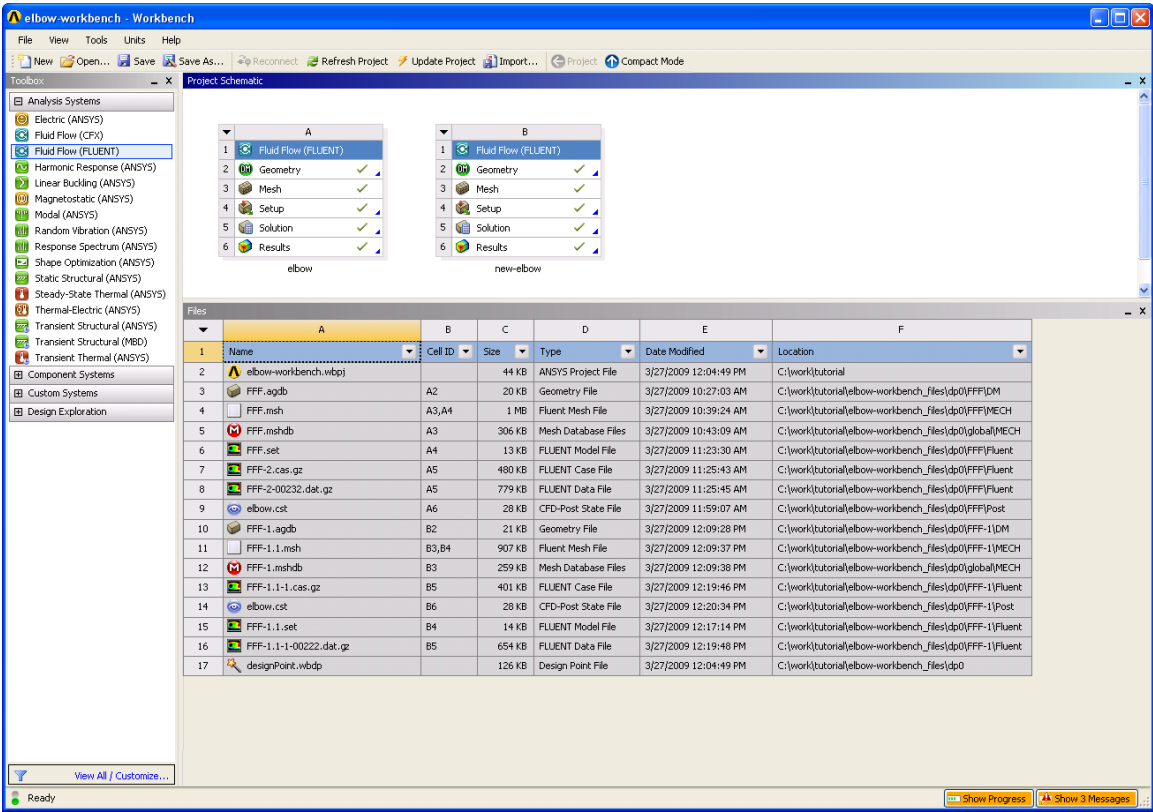
   View ⟶ Files

Figure 1.36: ANSYS Workbench Displaying the Files View for the Project After Viewing the New Results in ANSYS CFD-Post

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

Step 10: **Comparing the Results of Both Systems in** ANSYS CFD-Post

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

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.
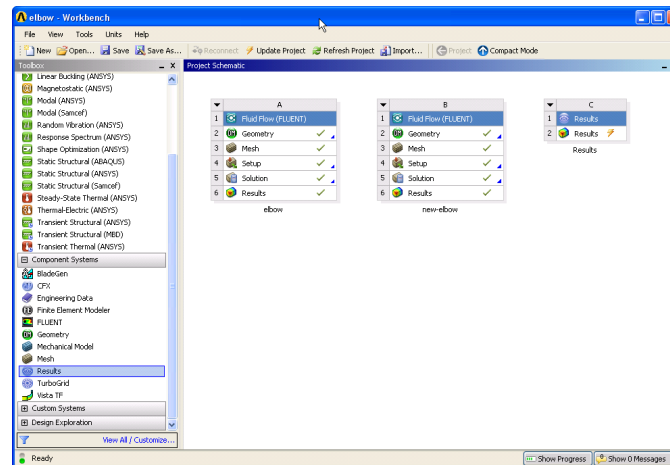

Figure 1.37: The New Results System in the Project Schematic

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.
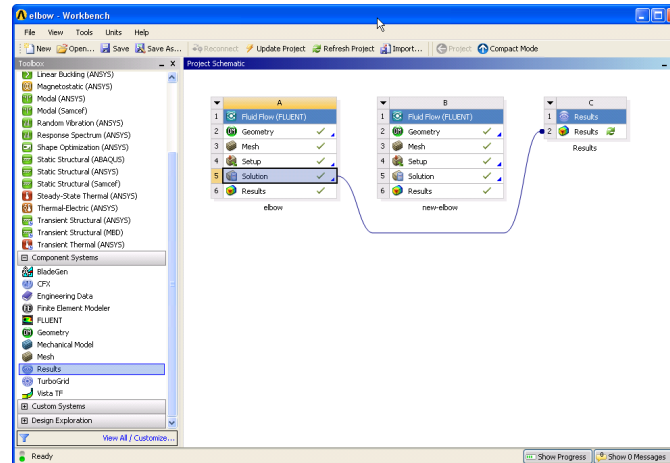

Figure 1.38: Connecting the First Fluid Flow System to the New Results System

(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.
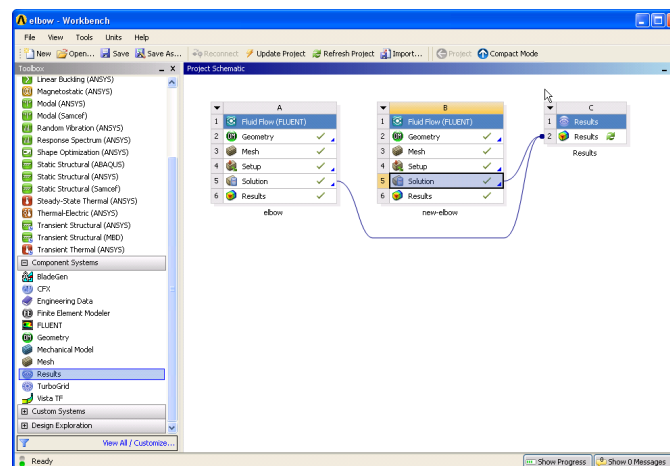

Figure 1.39: Connecting the Second Fluid Flow System to the New Results System

3. Open ANSYS 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 ANSYS CFD-Post. Within ANSYS CFD-Post, both geometries are displayed side by side.
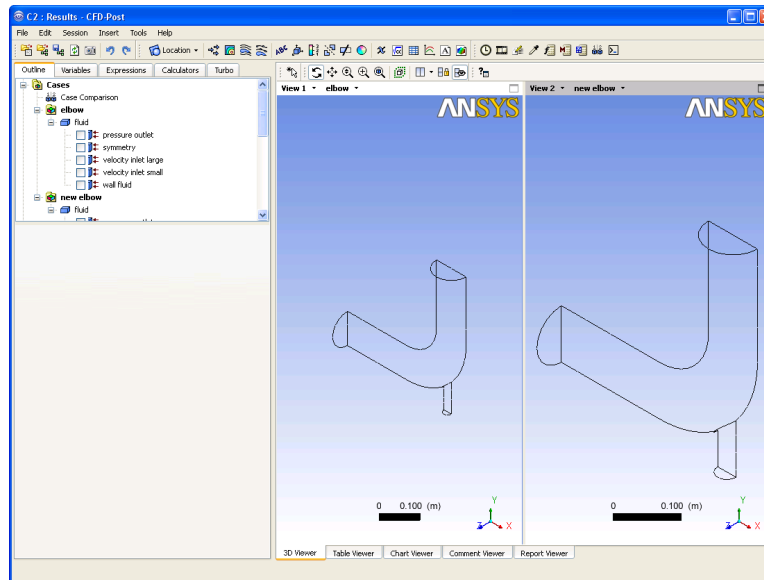

Figure 1.40: ANSYS CFD-Post with Both Fluid Flow Systems Displayed

(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.

    i 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* ANSYS 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.

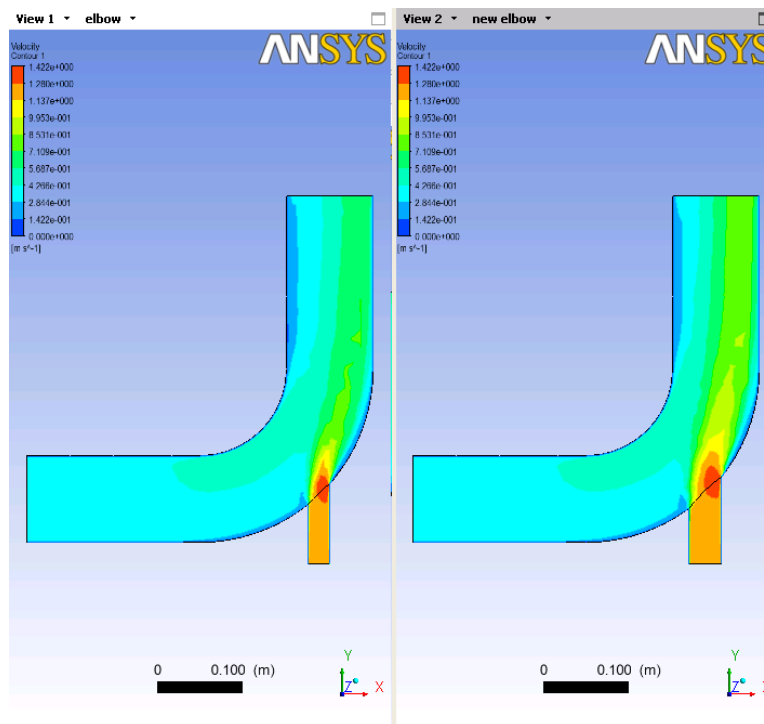   *In each view, the velocity contours are displayed.*

Figure 1.41: ANSYS 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 Location and Plots in ANSYS 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* ANSYS CFD-Post. *This view contains all of the settings for a contour object.*

 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.
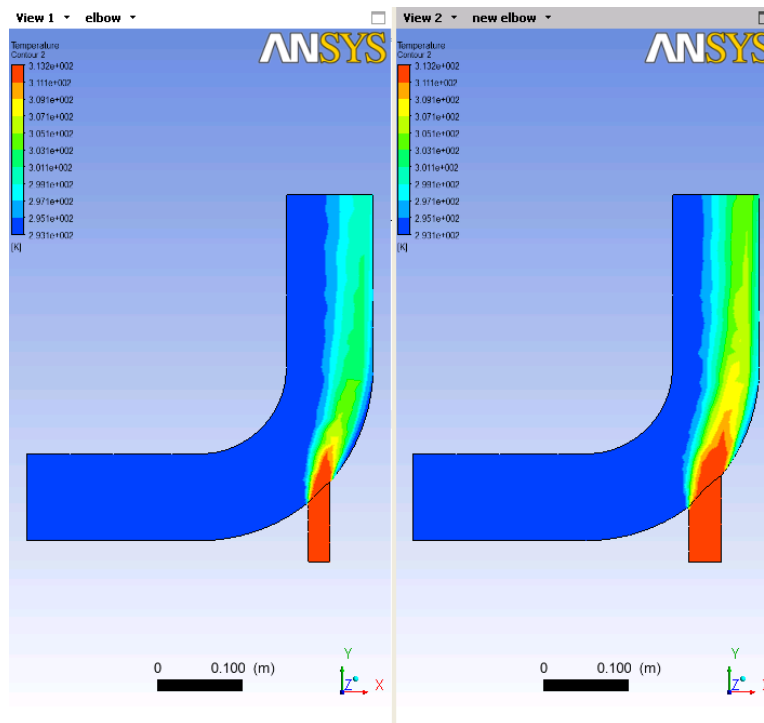
 In each view, the temperature contours are displayed.

Figure 1.42: ANSYS CFD-Post Displaying Temperature Contours for Both Geometries

4. Close the **ANSYS CFD-Post** application.

5. Save the `elbow-workbench` project in **ANSYS Workbench**.

6. View the files generated by **ANSYS Workbench**.

   View the files associated with your project using the **Files** view.
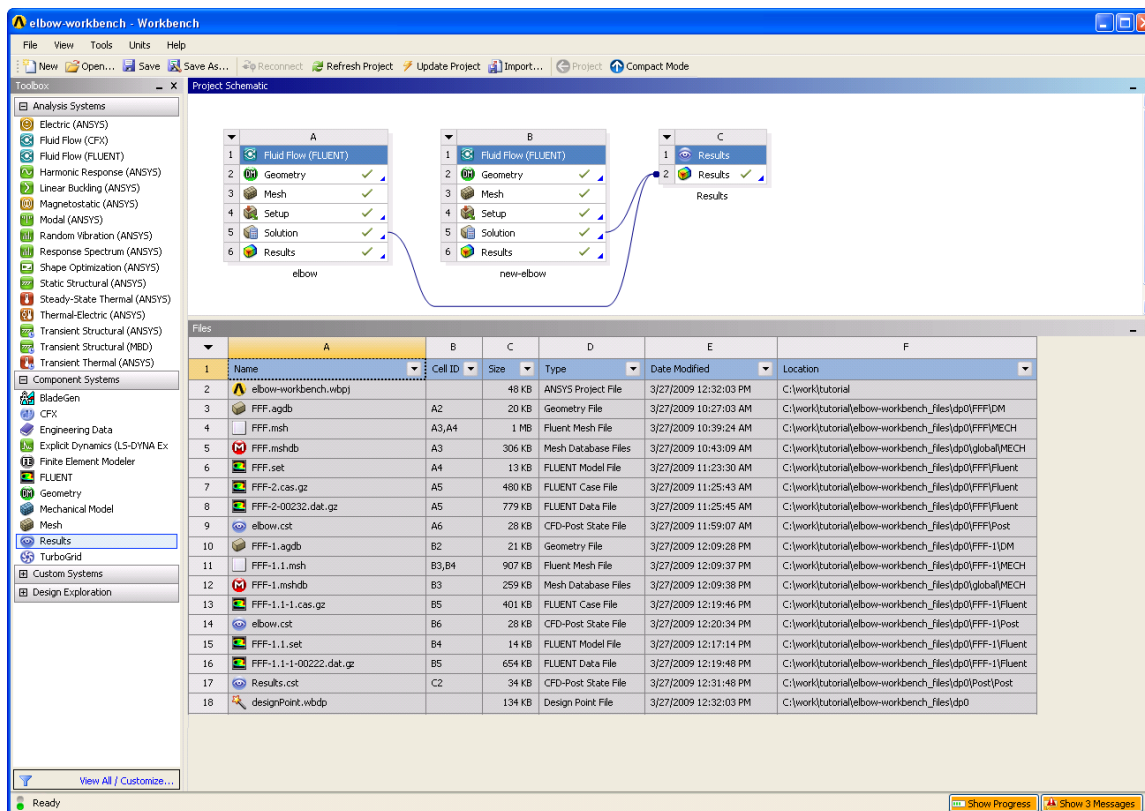
   View ⟶ Files

Figure 1.43: ANSYS Workbench Displaying the Files View for the Project After Viewing the Combined Results in ANSYS CFD-Post

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

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 ANSYS 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 used to allow ANSYS CFD-Post to directly compare the results of both calculations at the same time.