
Tutorial 1. Introduction to Using CFD-Post: Fluid Flow and Heat Transfer in a Mixing Elbow

Introduction

This tutorial illustrates how to use ANSYS CFD-Post to visualize a three-dimensional turbulent fluid flow and heat transfer problem in a mixing 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.

This tutorial demonstrates how to do the following:

- Create a Working Directory
- Launch ANSYS CFD-Post
- Display the Solution in ANSYS CFD-Post
- Save Your Work
- Generate a Report

Problem Description

The problem to be considered is shown schematically in Figure 1.1. A cold fluid at 20°C flows into the pipe through a large inlet and mixes with a warmer fluid at 40°C that enters through a smaller inlet located at the elbow. The pipe dimensions are in inches, but the fluid properties and boundary conditions are given in SI units. The Reynolds number for the flow at the larger inlet is 50,800, so the flow has been modeled as being turbulent.

Note: *This tutorial is derived from an existing ANSYS FLUENT case. The combination of SI and Imperial units is not typical, but follows an ANSYS FLUENT example.*

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

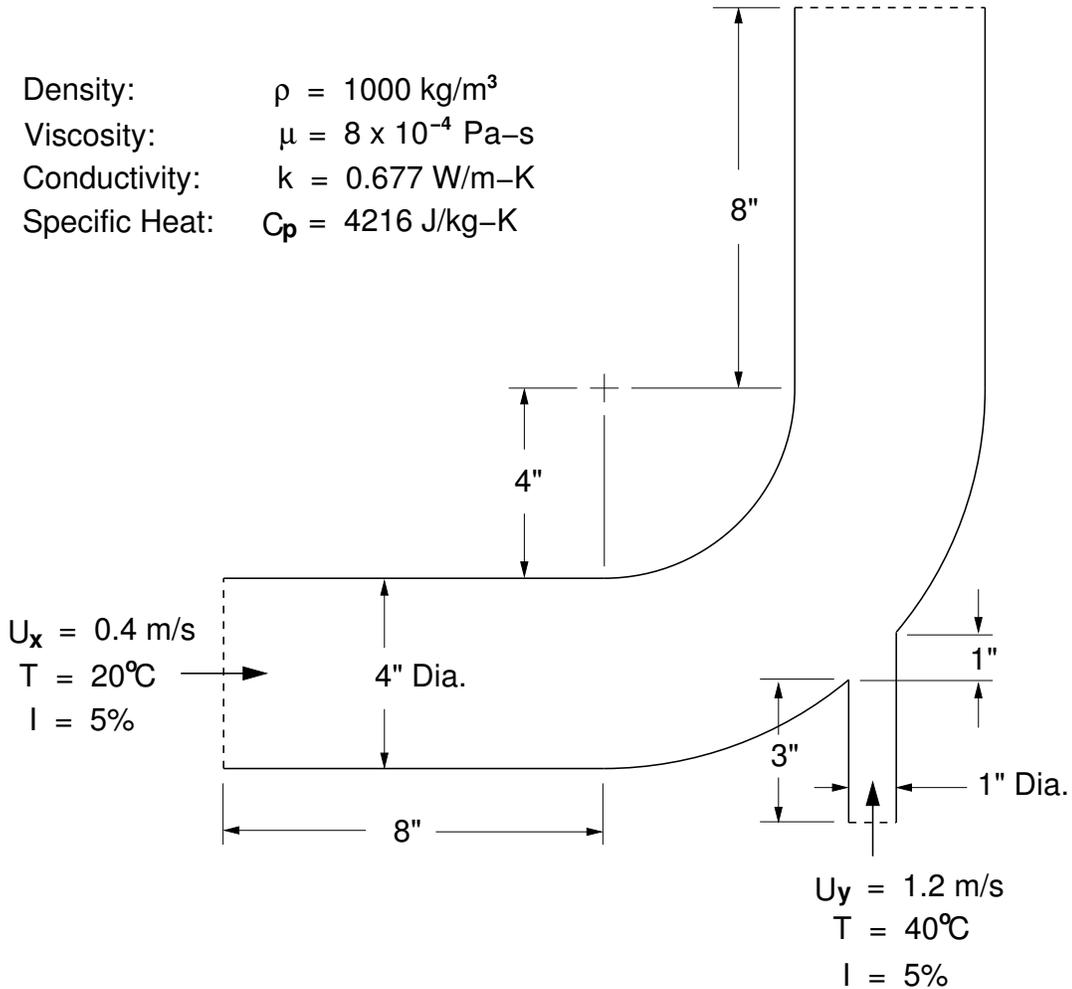


Figure 1.1: Problem Specification

Setup

Create a Working Directory

CFD-Post uses a *working directory* as the default location for loading and saving files for a particular session or project. Before you run a tutorial, use your operating system's commands to create a working directory where you can store your sample files and results files.

By working in that new directory, you prevent accidental changes to any of the sample files.

Copying the CAS and CDAT Files

Sample files are provided so that you can begin using CFD-Post immediately. You may find sample files in a variety of places, depending on which products you have:

- *If you have CFD-Post or ANSYS CFX*, sample files are in <CFXROOT>\examples, where <CFXROOT> is the installation directory for ANSYS CFX. Copy the CASE and CDAT files (`elbow1.cas.gz`, `elbow1.cdat.gz`, `elbow3.cas.gz`, and `elbow3.cdat.gz`) to your working directory.
- *If you have ANSYS FLUENT 12:*
 1. Download `cfd-post-elbow.zip` from the User Services Center (www.fluentusers.com) to your working directory. This file can be found by using the Documentation link on the FLUENT product page.
 2. Extract the CAS files and CDAT files (`elbow1.cas.gz`, `elbow1.dat.gz`, `elbow3.cas.gz`, and `elbow3.dat.gz`) from `cfd-post-elbow.zip` to your working directory.

Note: – *These tutorials are prepared 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.*

– *The case name is derived from the name of the results file that you load with the final extension removed. Thus, if you load `elbow1.cdat.gz` the case name will be `elbow1.cdat`; if you load `elbow1.cdat`, the case name will be `elbow1`. The case names used in this tutorial are `elbow1` and `elbow3`.*

Step 1: Launch ANSYS CFD-Post

Before you start CFD-Post, set the working directory. The procedure for setting the working directory and starting CFD-Post depends on whether you will launch CFD-Post standalone, from the ANSYS CFX Launcher, from ANSYS Workbench, or from ANSYS FLUENT:

- To run CFD-Post standalone, from the Start menu, right-click All Programs / ANSYS 12.0 / Fluid Dynamics / CFD-Post and select Properties. Type the path to your working directory in the Start in field and click OK, then click All Programs / ANSYS 12.0 / Fluid Dynamics / CFD-Post to launch CFD-Post.
- ANSYS CFX Launcher
 1. Start the ANSYS CFX Launcher.

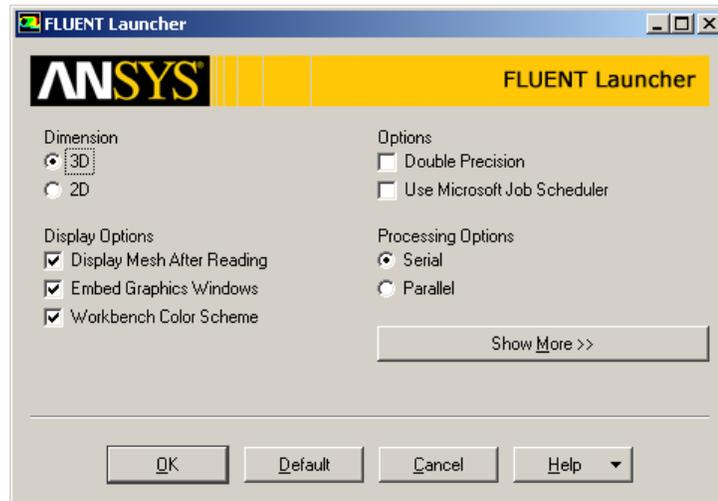
You can run the ANSYS CFX Launcher in any of the following ways:

 - On Windows:
 - * From the Start menu, go to All Programs / ANSYS 12.0 / Fluid Dynamics / CFX.
 - * In a DOS window that has its path set up correctly to run ANSYS CFX, enter `cfx5launch` (otherwise, you will need to type the full pathname of the `cfx` command, which will be something similar to `C:\Program Files\ANSYS Inc\v120\CFX\bin`).
 - On UNIX, enter `cfx5launch` in a terminal window that has its path set up to run ANSYS CFX (the path will be something similar to `/usr/ansys.inc/v120/CFX/bin`).
 2. Select the Working Directory (where you copied the sample files).
 3. Click the CFD-Post 12.0 button.
- ANSYS Workbench
 1. Start ANSYS Workbench.
 2. From the menu bar, select File/Save As and save the project file to the directory that you want to be the working directory.
 3. Open the Component Systems toolbox and double-click Results. A Results system opens in the Project Schematic.
 4. Right click on the A2 Results cell and select Edit. CFD-Post opens.

- ANSYS FLUENT

1. Click the ANSYS FLUENT icon () in the ANSYS program group to open FLUENT Launcher.

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



2. Ensure that the proper options are enabled.

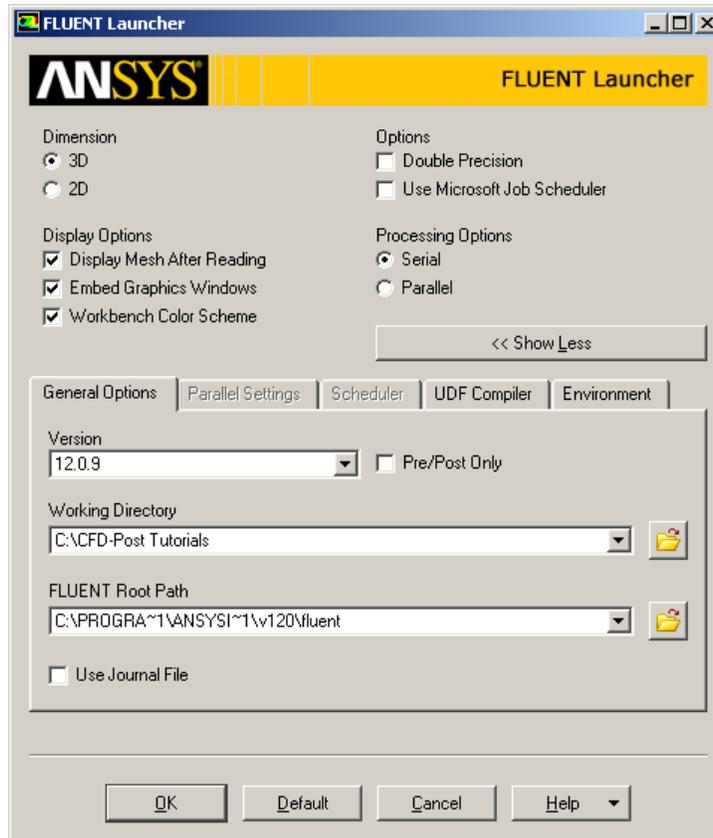
FLUENT Launcher retains settings from the previous session.

- (a) Select 3D from the Dimension list.
- (b) Select Serial from the Processing Options list.
- (c) Make sure that the Display Mesh After Reading and Embed Graphics Windows options are enabled.
- (d) Make sure that the Double-Precision option is disabled.

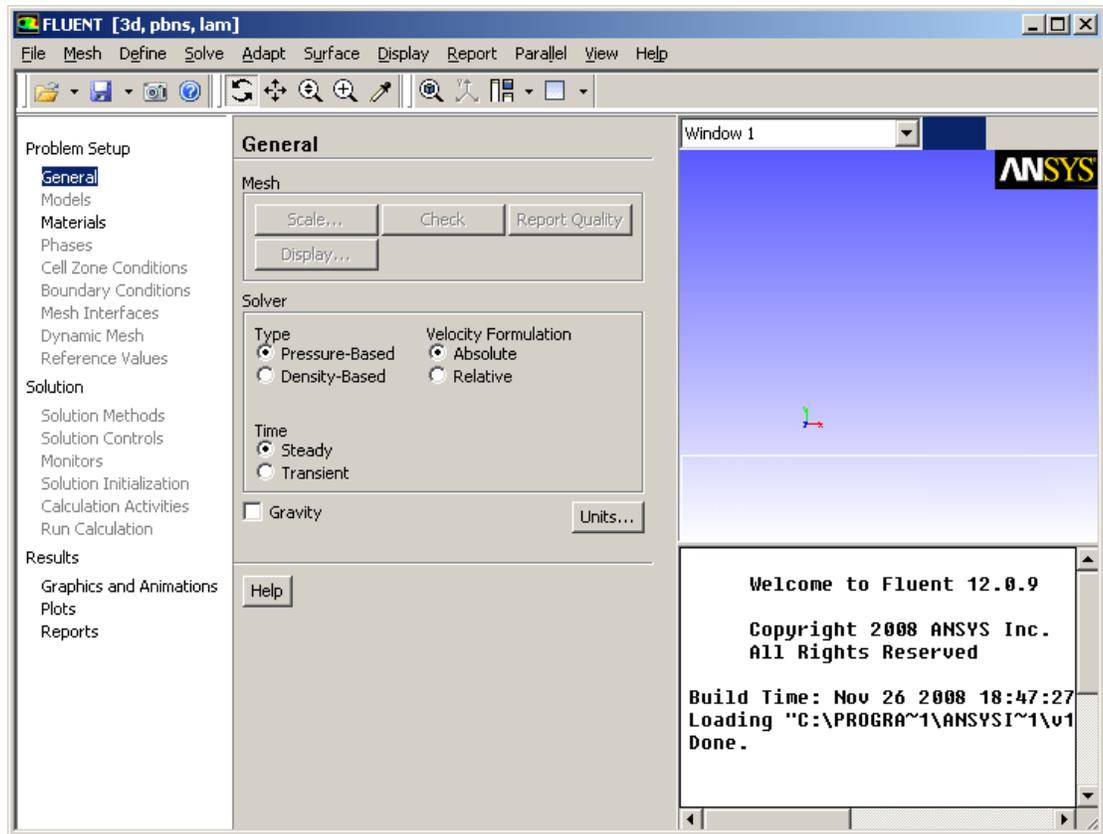
You can also restore the default settings by clicking the Default button.

3. Set the working path to the directory created when you unzipped `cfd-post-elbow.zip`.
 - (a) Click Show More.
 - (b) Enter the path to your working directory for Working Directory by double-clicking the text box and typing.

Alternatively, you can click the browse button () next to the Working Directory text box and browse to the directory, using the Browse For Folder dialog box.



4. Click OK to launch FLUENT.



5. Select File/Read/Case & Data and choose the elbow1.cas.gz file.
6. Select File/Export to CFD-Post.
7. In the Select Quantities list that appears, highlight the following variables:
 - Static Pressure
 - Density
 - X Velocity
 - Y Velocity
 - Z Velocity
 - Static Temperature
 - Turbulent Kinetic Energy (k)
8. Click Write.
CFD-Post starts with the tutorial file loaded.
9. In the ANSYS FLUENT application, select File / Read / Case & Data and choose the elbow3.cas.gz file.
10. On the Export to CFD-Post dialog, clear the Open CFD-Post option and click Write. Accept the default name and click OK to save the files.
11. Close ANSYS FLUENT.

Step 2: Display the Solution in ANSYS CFD-Post

In the steps that follow, you will visualize various aspects of the flow for the solution using CFD-Post. You will:

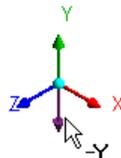
- Prepare the case and set the viewer options
- View the mesh and check it by using the Mesh Calculator
- View simulation values using the Function Calculator
- Become familiar with the 3D Viewer controls
- Create an instance reflection
- Show fluid velocity on the symmetry plane
- Create a vector plot to show the flow distribution in the elbow
- Create streamlines to show the flow distribution in the elbow
- Show the vortex structure
- Use multiple viewports to compare a contour plot to the display of a variable on a boundary
- Chart the changes to temperature at two places along the pipe
- Create a table to show mixing
- Review and modify a report
- Create a custom variable and cause the plane to move through the domain to show how the values of a custom variable change at different locations in the geometry
- Compare the results to those in a refined mesh
- Save your work
- Create an animation of a plane moving through the domain.

Step 3: Prepare the Case and Set the Viewer Options

1. If you have launched CFD-Post from ANSYS FLUENT, proceed to the next step. For all other situations, load the simulation from the data file (elbow1.cdat.gz) from the menu bar by selecting File/Load Results. In the Load Results File dialog box, select elbow1.cdat.gz and click Open.
2. If you see a pop-up that discusses Global Variables Ranges, it can be ignored. Click OK.

The mixing elbow appears in the 3D Viewer in an isometric orientation. The wireframe appears in the view and there is a check mark beside User Location and Plots / Wireframe in the Outline tree view; the check mark indicates that the wireframe is visible in the 3D Viewer.

3. Optionally, set the viewer background to white:
 - (a) Right-click on the viewer and select Viewer Options.
 - (b) In the Options dialog box, select CFD-Post / Viewer.
 - (c) Set:
 - Background / Color Type to Solid.
 - Background / Color to white. To do this, click the bar beside the Color label to cycle through 10 basic colors. (Click the right-mouse button to cycle backwards.) Alternatively, you can choose any color by clicking  to the right of the Color option.
 - Text Color to black (as above).
 - Edge Color to black (as above).
 - (d) Click OK to have the settings take effect.
 - (e) Experiment with rotating the object by clicking on the arrows of the triad in the 3D Viewer. This is the triad:



In the picture of the triad above, the cursor is hovering in the area opposite the positive Y axis, which reveals the negative Y axis.

Note: *The viewer must be in “viewing mode” for you to be able to click on the triad. You set viewing mode or select mode by clicking the icons in the viewer toolbar:*

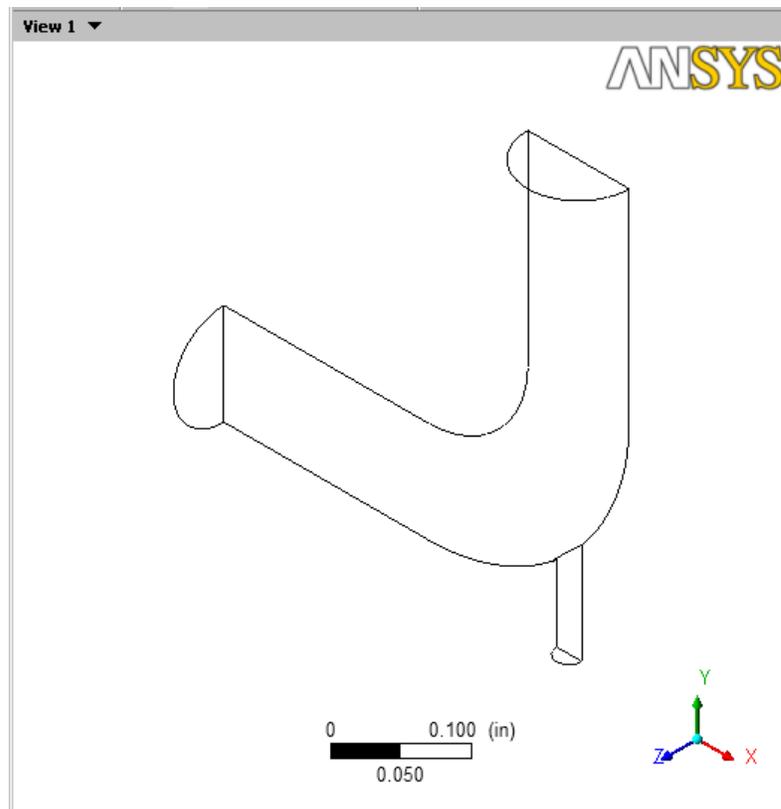


When you have finished experimenting, click the cyan (ISO) sphere in the triad to return to the isometric view of the object.

4. Set CFD-Post to display objects in the units you want to see. These display units are not necessarily the same types as the units in the results files you load; however, for this tutorial you will set the display units to be the same as the solution units for consistency. As mentioned in the Problem Description, the solution units are SI, except for the length, which is measured in inches.
 - (a) Right-click on the viewer and select **Viewer Options**.
The Options dialog box is where you set your preferences.
 - (b) In the Options dialog box, select **Common / Units**.
 - (c) Notice that **System** is set to **SI**. In order to be able to change an individual setting (length, in this case) from SI to imperial, set **System** to **Custom**. Now set **Length** to **in** (inches) and click **OK**.

Note:

- *The display units you set are saved between sessions and projects. This means that you can load results files from diverse sources and always see familiar units displayed.*
- *You have set only length to inches; volume will still be reported in meters. To change volume as well, in the Options dialog box, select **Common / Units**, then click **More Units** to find the full list of settings.*



Step 4: View and Check the Mesh

There are two ways to view the mesh: you can use the wireframe for the entire simulation or you can view the mesh for a particular portion of the simulation.

To view the mesh for the entire simulation:

1. Right-click on the wireframe in the 3D Viewer and select **Show surface mesh** to display the mesh.
2. Click the “Z” axis of triad in the viewer to get a side view of the object. (Remember that the 3D Viewer toolbar has to be in viewing mode for you to be able to select the triad elements.)

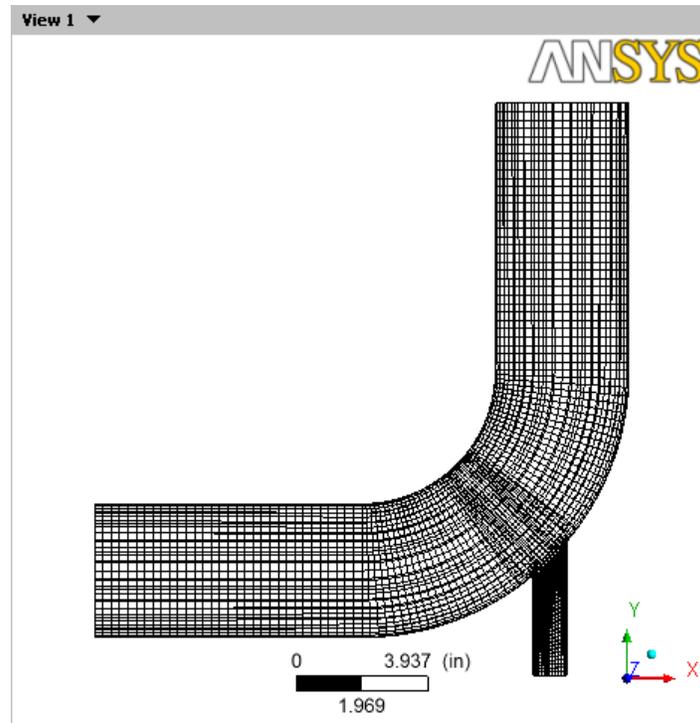


Figure 1.2: The Hexahedral Grid for the Mixing Elbow

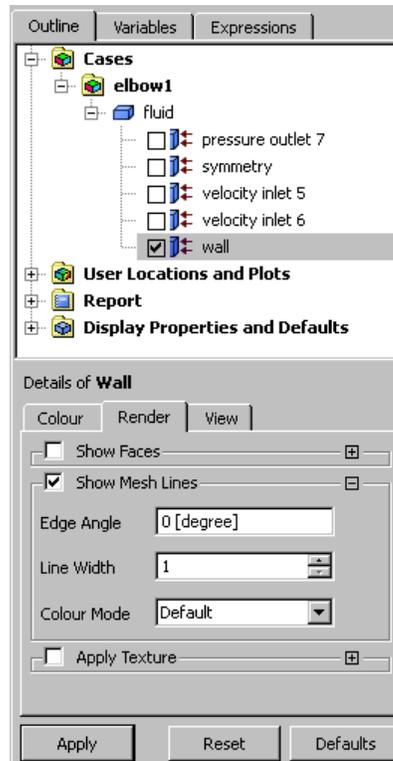
3. In the Outline tree view, double-click on User Locations and Plots / Wireframe to display the wireframe's editor.

Click on the Details of Wireframe editor and press F1 to see help about the Wireframe object.

4. On the Wireframe Details view, click Defaults and Apply to restore the original settings.

To view the mesh for a particular portion of the simulation (in this case, the wall):

1. In the Outline tree view, select the check box beside Cases / elbow1 / fluid / wall, then double-click wall to edit its properties in its Details view.
2. In the Details view:
 - (a) On the Render tab, clear the Show Faces check box.
 - (b) Select the Show Mesh Lines check box.
 - (c) Ensure that Edge Angle is set to 0 [degree].



(d) Click **Apply**.

The mesh appears and is similar to the mesh shown by the previous procedure, except that the mesh is shown only on the wall.

(e) Now, clear the display of the wall wireframe. In the **Details** view:

- i. Clear the **Render / Show Mesh Lines** check box.
- ii. Select the **Show Faces** check box.
- iii. Click **Apply**.

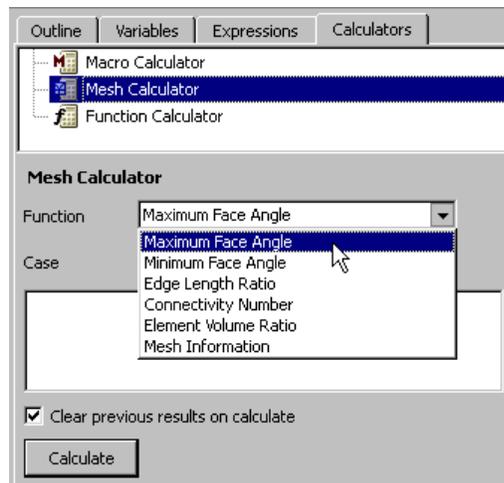
The wall reappears.

3. In the **Outline** tree view, clear the check box beside **Cases / elbow1 / fluid / wall**.

Note: *The rest of the tutorial assumes that the wall is not visible or, if it is visible, that it is showing faces, not lines.*

To check the mesh:

1. Select the **Calculators** tab at the top of the workspace area, then double-click **Mesh Calculator**. The **Mesh Calculator** appears.
2. Using the drop-down arrow beside the **Function** field, select a function such as **Maximum Face Angle**.

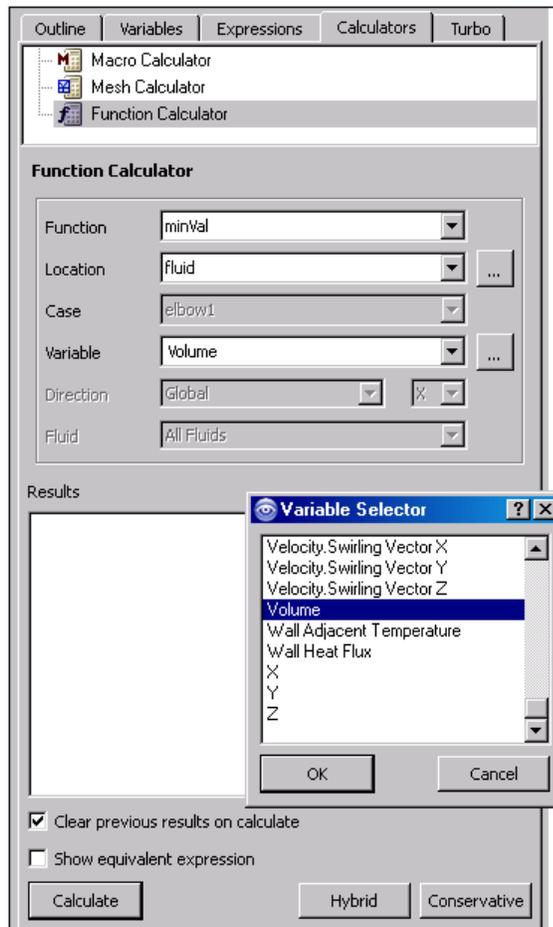


3. Click **Calculate**. The results of the calculation appear.
4. Repeat the previous steps for other functions, such as **Mesh Information**.

Step 5: View Simulation Values using the Function Calculator

You can view values in the simulation by using the Function Calculator:

1. In the **Calculators** view, double-click **Function Calculator**. The **Function Calculator** appears.
2. In the **Function** field, select a function to evaluate. This example uses `minVal`.
3. In the **Location** field, select `fluid`.
4. Beside the **Variable** field, click the *More Variables* icon  and select **Volume** in the **Variable Selector** dialog box.



Click OK.

5. Click **Calculate** to see the result of the calculation of the minimum value of element volumes found in the fluid region. Note that even though the length of the elbow is measured in inches, the volume is returned in cubic meters.

Step 6: Become Familiar with the Viewer Controls

Optionally, take a few moments to become familiar with the viewer controls. These icons switch the mouse from selecting items in the viewer to controlling the orientation and display of the view. First, the sizing controls:

1. Click the *Zoom Box* icon .
2. Click and drag a rectangular box over the geometry.
3. Release the mouse button to zoom in on the selection.

The geometry zoom changes to display the selection at a greater resolution.

4. Click the *Fit View* icon  to re-center and re-scale the geometry.

Now, the rotation functions:

1. Click the *Rotate* icon  on the viewer toolbar.
2. Click and drag repeatedly within the viewer to test the rotation of the geometry. Notice how the mouse cursor changes depending on where you are in the viewer, particularly near the edges:

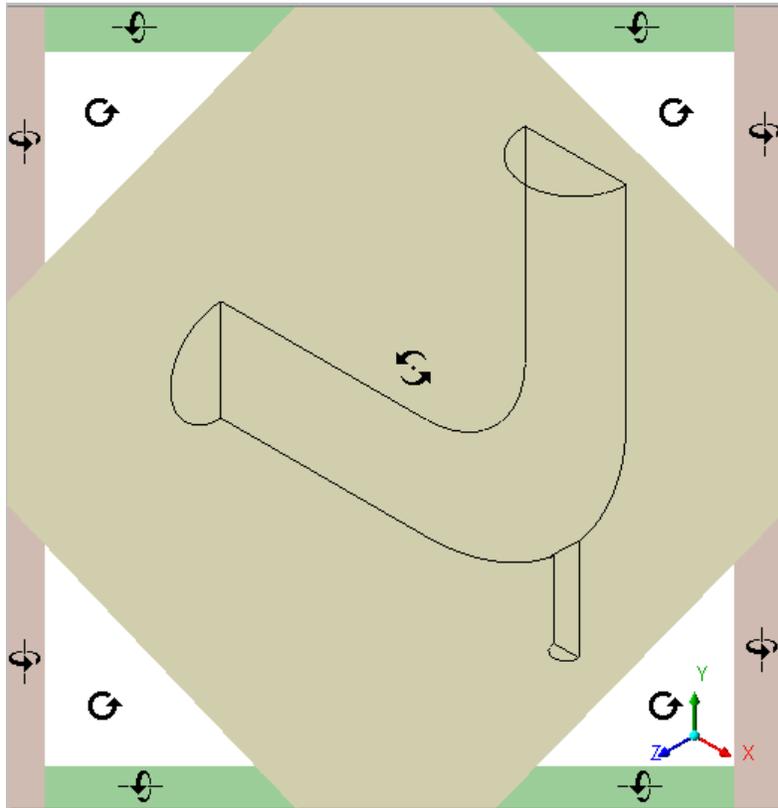


Figure 1.3: Orientation Control Cursor Types

The geometry rotates based on the direction of movement. If the mouse cursor has an axis (which happens around the edges), the object rotates around the axis shown in the cursor. The axis of rotation is through the pivot point, which defaults to be in the center of the object.

Now explore orientation options:

1. Right-click a blank area in the viewer and select Predefined Camera / View Towards -X.
2. Right-click a blank area in the viewer and select Predefined Camera / Isometric View (Z Up).
3. Click the “Z” axis of triad in the viewer to get a side view of the object.
4. Click the three axes in the triad in turn to see the vector objects in all three planes; when you are done, click the cyan (ISO) sphere.

Now explore the differences between the orienting controls you just used and *select mode*.

1. Click the  icon to enter select mode.
2. Hover over one of the wireframe lines and notice that the cursor turns into a box.
3. Click a wireframe line and notice that the Details view for the wireframe appears.
4. Right-click away from a wireframe line and then again on a wireframe line. Notice how the menu changes:

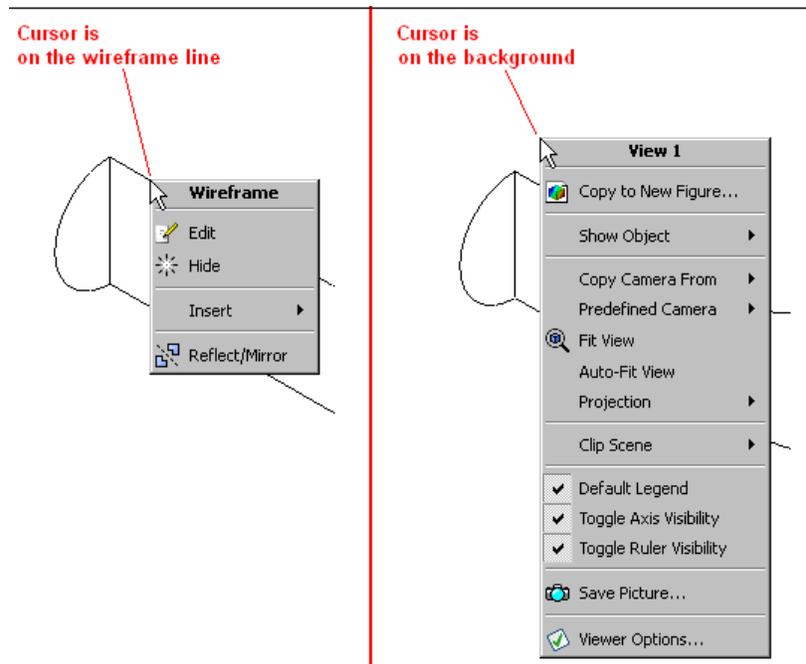


Figure 1.4: Right-click Menus Vary by Cursor Position

5. In the Outline tree view, select the `elbow1 / fluid / wall` check box; the outer wall of the elbow becomes solid. Notice that as you hover over the colored area, the cursor again becomes a box, indicating that you can perform operations on that region. When you right-click on the wall, a new menu appears.
6. Click on the triad and notice that you cannot change the orientation of the viewer object. (The triad is available only in viewing mode, not select mode.)
7. In the Outline tree view, clear the `elbow1 / fluid / wall` check box; the outer wall of the elbow disappears.

Step 7: Create an Instance Reflection

Create an instance reflection on the symmetry plane so that you can see the complete case:

1. With the 3D Viewer toolbar in viewing mode, click on the cyan (ISO) sphere in the triad. This will make it easy to see the instance reflection you are about to create.
2. Right-click on one of the wireframe lines on the symmetry plane. (If you were in select mode, the mouse cursor would have a “box” image added when you are on a valid line. As you are in viewing mode there is no change to the cursor to show that you are on a wireframe line, so you may see the general right-click menu, as opposed to the right-click menu for the symmetry plane.) See Figure 1.4.
3. From the right-click menu, select `Reflect/Mirror`. If you see a dialog box prompting you for the direction of the normal, choose the Z axis. The mirrored copy of the wireframe appears.

If the reflection you create is on an incorrect axis, click the Undo  toolbar icon twice.

Step 8: Show Velocity on the Symmetry Plane

Create a contour plot of velocity on the symmetry plane:

1. From the menu bar, select **Insert/Contour**. In the **Insert Contour** dialog box, accept the default name, and click **OK**.
2. In the **Details** view for **Contour 1**, set the following:

Tab	Setting	Value
Geometry	Locations Variable	symmetry* Velocity**

* Notice how the available locations are highlighted in the viewer as you move the mouse over the objects in the **Locations** drop-down list. You could also create a slice plane at a location of your choice and define the contour to be at that location.

** Velocity is just an example of a variable you can use.

3. Click **Apply**. The contour plot for velocity appears and a legend is automatically generated.
4. The coloring of the contour plot may not correspond to the colors on the legend because the viewer has a light source enabled by default. There are several ways to correct this:
 - You can change the orientation of the objects in the viewer.
 - You can experiment with changing the position of the light source by holding down the **Ctrl** key and dragging the cursor with the right mouse button.
 - You can disable lighting for the contour plot. To disable lighting, click on the **Render** tab and clear the check box beside **Lighting**, then click **Apply**.

Disabling the lighting is the method that provides you with the most flexibility, so change that setting now.

5. Click on the Z on the triad to better orient the geometry (the 3D Viewer must be in viewing mode, not select mode, to do this).

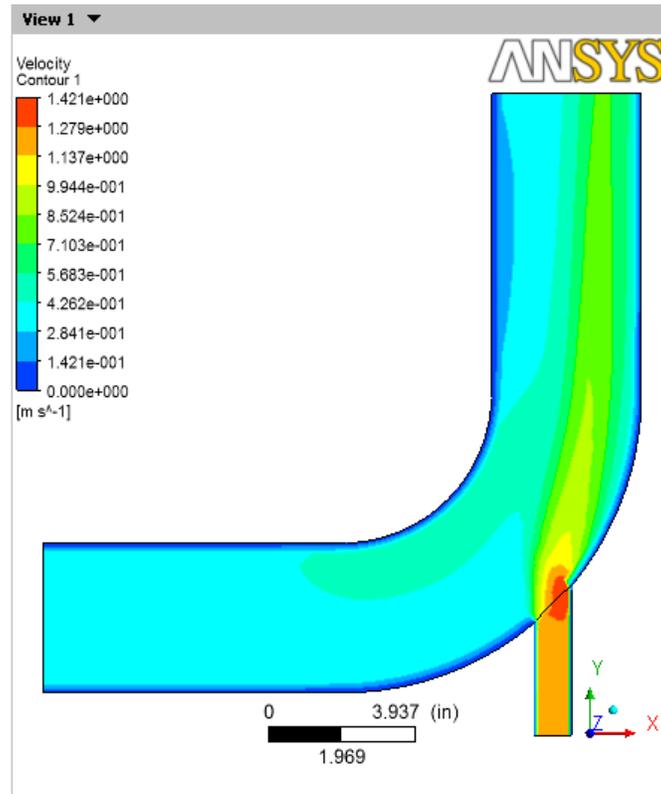


Figure 1.5: Velocity on the Symmetry Plane

6. Improve the contrast between the contour regions:
 - (a) On the Render tab, select Show Contour Lines and click the plus sign to view more options.
 - (b) Select Constant Coloring.
 - (c) Set Color Mode to User Specified and set Line Color to black (if necessary, click on the bar beside Line Color until black appears).
 - (d) Click Apply.

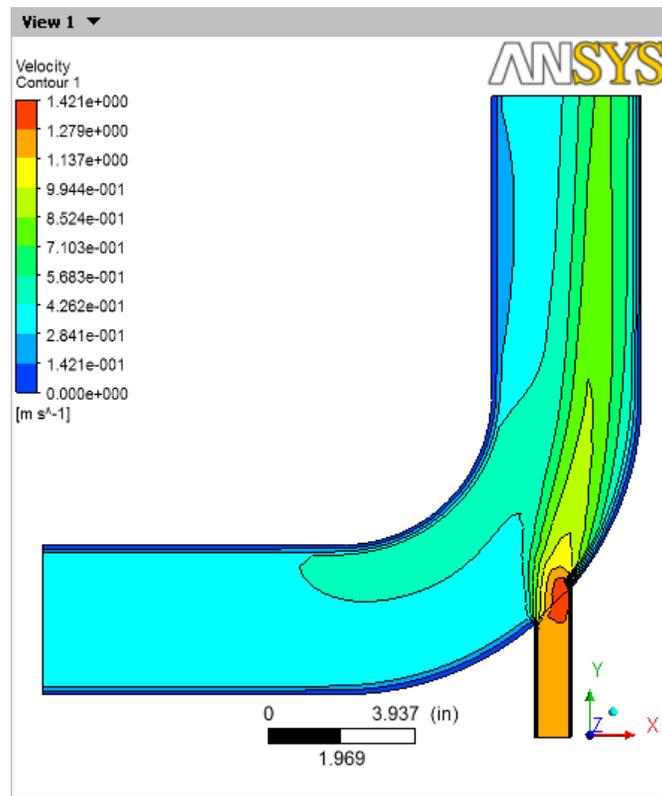


Figure 1.6: Velocity on the Symmetry Plane (Enhanced Contrast)

7. Hide the contour plot by clearing the check box beside User Locations and Plots / Contour 1 in the Outline tree view.

You can also hide an object by right-clicking on its name in the Outline tree view and selecting Hide.

Step 9: Show Flow Distribution in the Elbow

Create a vector plot to show the flow distribution in the elbow:

1. From the menu bar, select **Insert/Vector**.
2. Click **OK** to accept the default name. The **Details** view for the vector appears.
3. On the **Geometry** tab, set **Domains** to **fluid** and **Locations** to **symmetry**.
4. Click **Apply**.
5. On the **Symbol** tab, set **Symbol Size** to 4.

6. Click Apply and notice the changes to the vector plot.

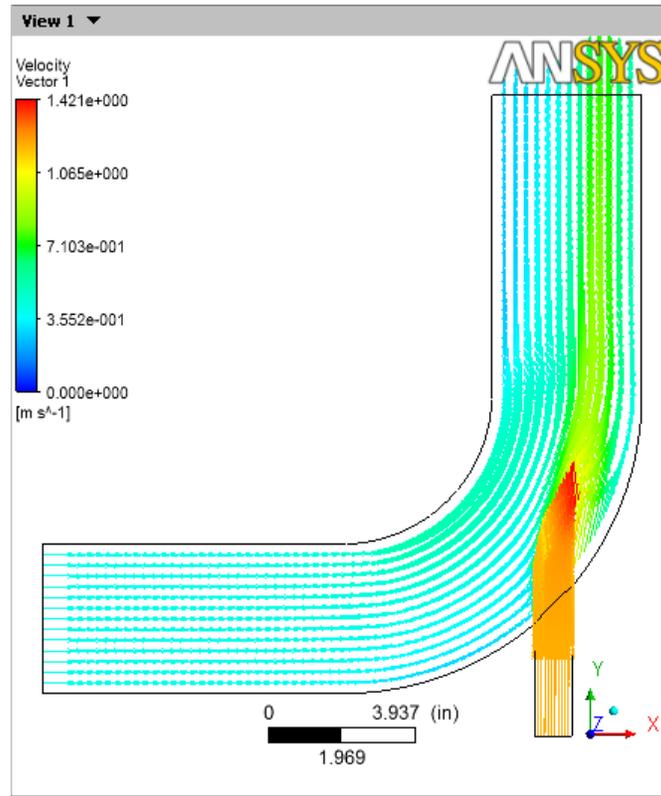


Figure 1.7: Vector Plot of Velocity

7. Change the vector plot so that the vectors are colored by temperature:
 - (a) In the Details view for Vector 1, click on the Color tab.
 - (b) Set the Mode to Variable. The Variable field becomes enabled.
 - (c) Click on the down arrow  beside the Variable field to set it to Temperature.
 - (d) Click Apply and notice the changes to the vector plot.
8. Optionally, change the vector symbol. In the Details view for the vector, go to the Symbol tab and set Symbol to Arrow3D. Click Apply.
9. Hide the vector plot by right-clicking on a vector symbol in the plot and selecting Hide.

CFD-Post uses the **Variable** setting on the **Geometry** tab to determine where to place objects to best illustrate changes in that variable. Once the object has been put in place, you can have CFD-Post measure other variables along those streamlines by using the **Variable** setting on the **Color** tab. In this example you will create streamlines to show the flow distribution by velocity, then recolor those streamlines to show turbulent kinetic energy:

1. From the menu bar select **Insert/Streamline**. Accept the default name and click **OK**.
2. In the **Details** view for Streamline 1, choose the points from which to start the streamlines. Click on the down arrow  beside the **Start From** drop-down widget to see the potential starting points. Hover over each point and notice that the area is highlighted in the **3D Viewer**. It would be best to show how streamlines from both inlets interact, so, to make a multi-selection, click the *Location editor* icon . The **Location Selector** dialog box appears.
3. In the **Location Selector** dialog box, hold down the **Ctrl** key and click **velocity inlet 5** and **velocity inlet 6** to highlight both locations, then click **OK**.
4. Click **Preview Seed Points** to see the starting points for the streamlines.
5. On the **Geometry** tab, ensure that **Variable** is set to **Velocity**.
6. Click on the **Color** tab and make the following changes:
 - (a) Set the **Mode** to **Variable**. The **Variable** field becomes enabled.
 - (b) Set the **Variable** to **Turbulence Kinetic Energy**.
 - (c) Set **Range** to **Local**.
7. Click **Apply**. The streamlines show the flow of massless particles through the entire domain.

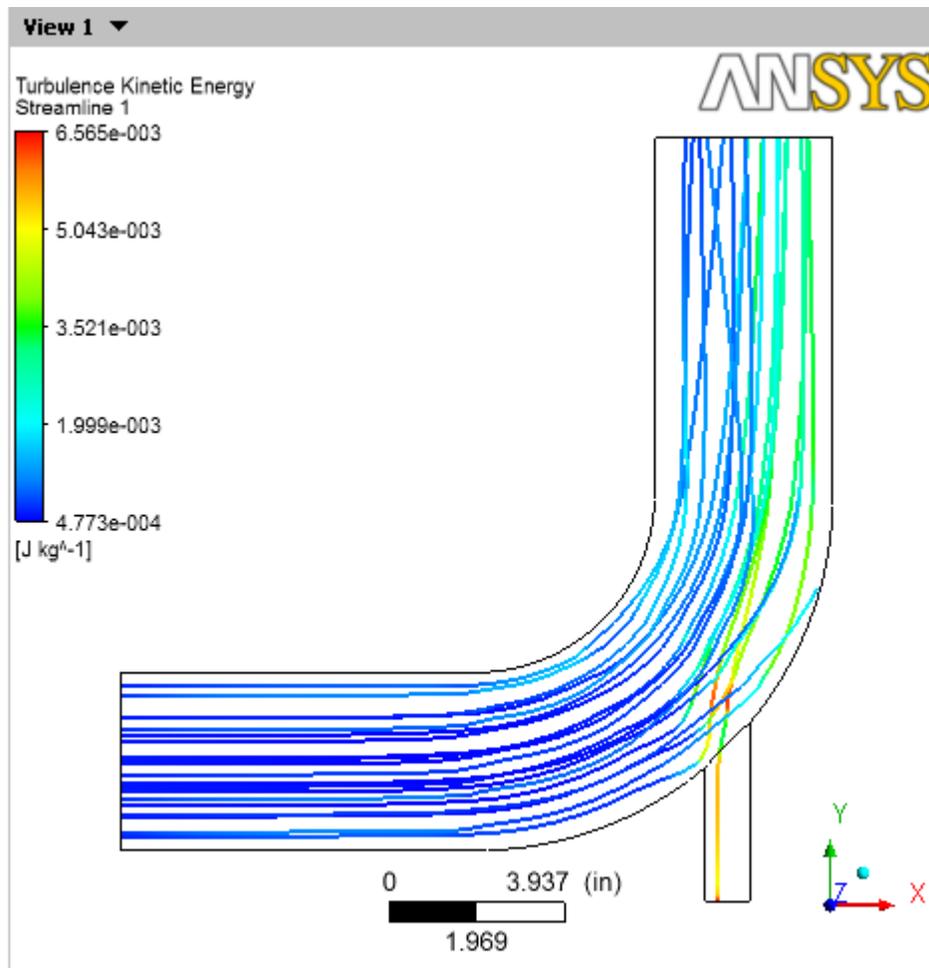


Figure 1.8: Streamlines of Turbulence Kinetic Energy

8. Select the check box beside Vector 1. The vectors appear, but are largely hidden by the streamlines. To correct this, highlight **Streamline 1** in the Outline tree view and press **Delete**. The vectors are now clearly visible, but the work you did to create the streamlines is gone. Click the *Undo* icon  to restore **Streamline 1**.
9. Hide the vector plot and the streamlines by clearing the check boxes beside **Vector 1** and **Streamline 1** in the Outline tree view.

Step 10: Show the Vortex Structure

CFD-Post displays vortex core regions to enable you to better understand the processes in your simulation. In this example you will look at the helicity method for vortex cores, but in your own work you would use the vortex core method that you find most instructive.

1. In the Outline tree view:

- (a) Under **User Locations and Plots**, clear the check box for **Wireframe**.
- (b) Under **Cases / elbow1 / fluid**, select the check box for **wall**.
- (c) Double-click on **wall** to edit its properties. On the **Render** tab, set **Transparency** to **0.75** and click **Apply**.

This makes the pipe easy to see while also making it possible to see objects inside the pipe.

2. From the menu bar, select **Insert/Location/Vortex Core Region** and click **OK** to accept the default name.

3. In the **Details** view for **Vortex Core Region 1** on the **Geometry** tab, set **Method** to **Absolute Helicity** and **Level** to **.01**. On the **Render** tab, set **Transparency** to **0.2**. Click **Apply**.

The absolute helicity vortex that is displayed is created by a mixture of effects from the walls, the curve in the main pipe, and the interaction of the fluids. If you had chosen the vorticity method instead, wall effects would dominate.

4. On the **Color** tab, click on the colored bar in the **Color** field until the bar is green. Click **Apply**.

This improves the contrast between the vortex region and the blue walls.

5. Right-click in the **3D Viewer** and select **Predefined Camera / Isometric View (Y up)**.

6. In the Outline tree view, select the check box beside Streamline 1. This shows how the streamlines are affected by the vortex regions.

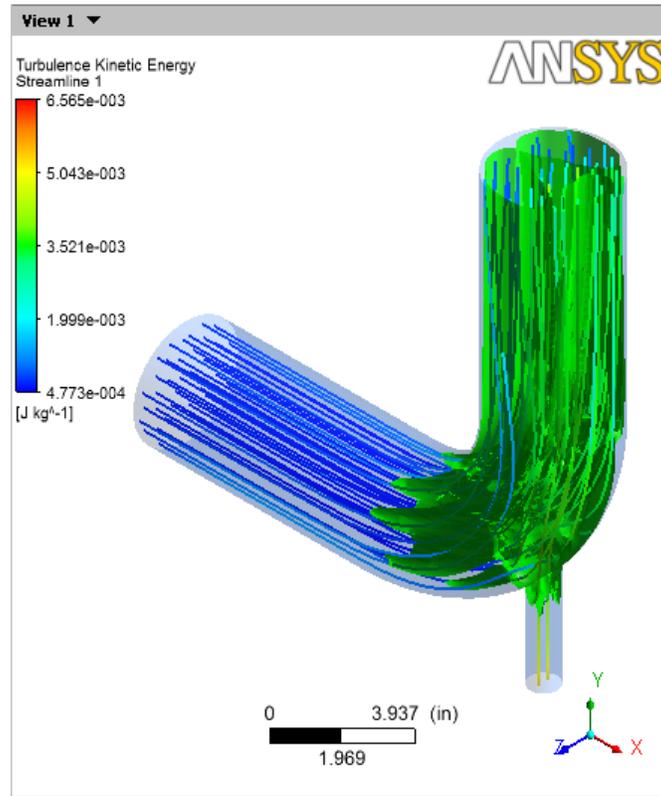


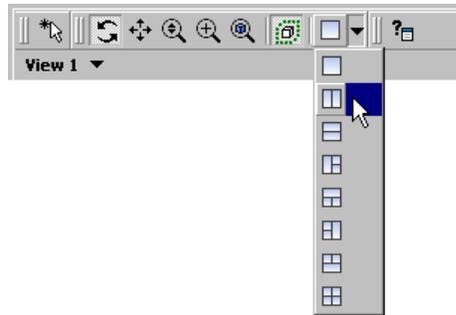
Figure 1.9: Absolute Helicity Vortex

7. Clear the check boxes beside wall, Streamline 1, and Vortex Core Region 1. Select the check box beside Wireframe.

Step 11: Compare a Contour Plot to the Display of a Variable on a Boundary

A contour plot with color bands has discrete colored regions while the display of a variable on a locator (such as a boundary) shows a finer range of color detail by default. The instructions that follow will illustrate a variable at the outlet and create a contour plot that displays the same variable at that same location.

1. To do the comparison, split the 3D Viewer into two viewports by using the Viewport Layout toolbar in the 3D Viewer toolbar:



2. Right-click in both viewports and select **Predefined Camera / View Towards -Y**.
3. In the **Outline** tree view, double-click on **pressure outlet 7** (which is under **elbow1 / fluid**). The **Details** view of **pressure outlet 7** appears.
4. Click in the **View 1** viewport.
5. In the **Details** view for **pressure outlet 7** on the **Color** tab:
 - (a) Change **Mode** to **Variable**.
 - (b) Ensure **Variable** is set to **Pressure**.
 - (c) Ensure **Range** is set to **Local**.
 - (d) Click **Apply**. The plot of pressure appears and the legend shows a smooth spectrum that goes from blue to red. Notice that this happens in both viewports; this is because *Synchronize visibility in displayed views* icon  is enabled.
 - (e) Click the *Synchronize visibility in displayed views* icon  to disable this feature.

Now, add a contour plot at the same location:

1. Click in View 2 to make it active, the title bar for that viewport becomes highlighted.
2. In the Outline tree view, clear the check box beside fluid / pressure outlet 7.
3. From the menu bar, select **Insert/Contour**.
4. Accept the default contour name and click OK.
5. In the Details view for the contour, ensure that the **Locations** setting is **pressure outlet 7** and the **Variable** setting is **Pressure**.
6. Set **Range** to **Local**.
7. Click **Apply**. The contour plot for pressure appears and the legend shows a spectrum that steps through 10 levels from blue to red.
8. Compare the two representations of pressure at the outlet. Pressure at the Outlet is on the left and a Contour Plot of pressure at the Outlet is on the right:

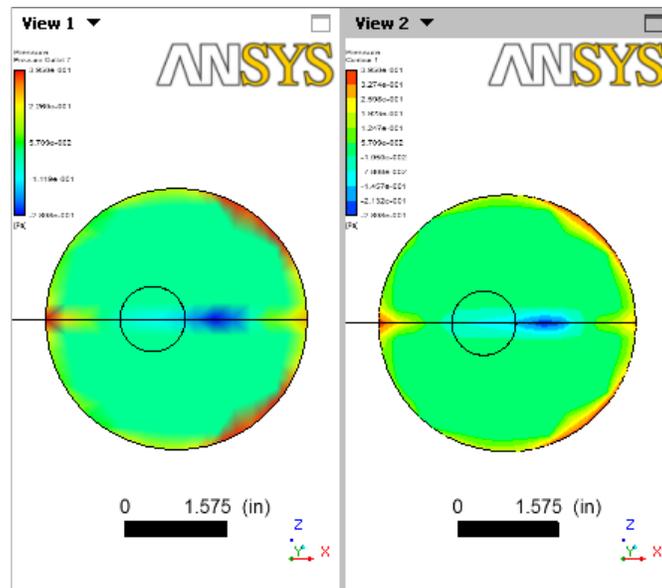


Figure 1.10: Boundary Pressure vs. a Contour Plot of Pressure

9. Enhance the contrast on the contour bands:
 - (a) In the **Outline** tree view, right-click on **User Locations and Plots / Contour 2** and select **Edit**.
 - (b) In the **Details** view for the contour, click on the **Render** tab, expand the **Show Contour Lines** area, and select the **Constant Coloring** check box. Then set the **Color Mode** to **User Specified**. Click **Apply**.
 - (c) Click on the **Labels** tab and select **Show Numbers**. Click **Apply**.
10. Explore the viewer synchronization options:
 - (a) In **View 1**, click the cyan (ISO) sphere in the triad so that the two viewports show the elbow in different orientations.
 - (b) In the **3D Viewer** toolbar, click the *Synchronize camera in displayed views* icon . Both viewports take the camera orientation of the active viewport.
 - (c) Clear the *Synchronize camera in displayed views* icon  and click on the Z arrow head of the triad in **View 1**. The object again moves independently in the two viewports.
 - (d) In the **3D Viewer** toolbar, click the *Synchronize visibility in displayed views* icon .
 - (e) In the **Outline** tree view, right-click on **fluid / wall** and select **Show**. The wall becomes visible in both viewports. (Synchronization applies only to events that take place after you enable the synchronize visibility function.)
11. When you are done, use the viewport controller to return to a single viewport. The synchronization icons disappear.

Step 12: Chart Temperature vs. Distance

We want to show temperature vs. distance at the outlet of the pipe. We will create a chart to do that, but the chart requires a line where we can sample values.

1. In the Outline tree view in the User Location and Plots area, right-click and select Hide All, then click in the check boxes to enable the Default Legend View 1 and the Wireframe.
2. Click the cyan (ISO) sphere in the triad so that you can again see the full elbow.
3. Create a line at the centerline of the outlet:
 - (a) On the menu bar, select Insert/Location/Line. In the Insert Line dialog box, accept the default name and click OK.
 - (b) Accept the default values and click Apply. A yellow line appears near the inside of the elbow.
 - (c) In the Details view for Line 1, set the following:

Tab	Setting	Value
Geometry	Point 1*	4, 7.999, 0.001
	Point 2	8, 7.999, 0.001**
	Line Type	Cut***
Color	Mode	Variable
	Variable	Temperature
	Range	Local
Render	Line Width	2

* The units are taken from the 3D Viewer options setting, which you can access from the menu bar by selecting Edit/Options.

** The Z value ensures that the line unambiguously passes through the domain.

*** This setting constrains the line to be within the object.

- (d) Click Apply.

4. Create a chart that samples the points on Line 1:
 - (a) On the menu bar, select **Insert/Chart**. In the **Insert Chart** dialog box, accept the default name and click **OK**.
 - (b) In the **Details** view for **Chart 1**, on the **General** tab set **Title** to **Output Temperatures**.
 - (c) On the **Data Series** tab, right-click in the empty list box and select **New**, then set **Name** to **Temperature at Line 1** and **Location** to **Line 1**.
 - (d) On the **X Axis** tab, set **Variable** to **X**.
 - (e) On the **Y Axis** tab, set **Variable** to **Temperature**.
 - (f) Click **Apply**. A chart similar to the following appears in the **Chart Viewer**:

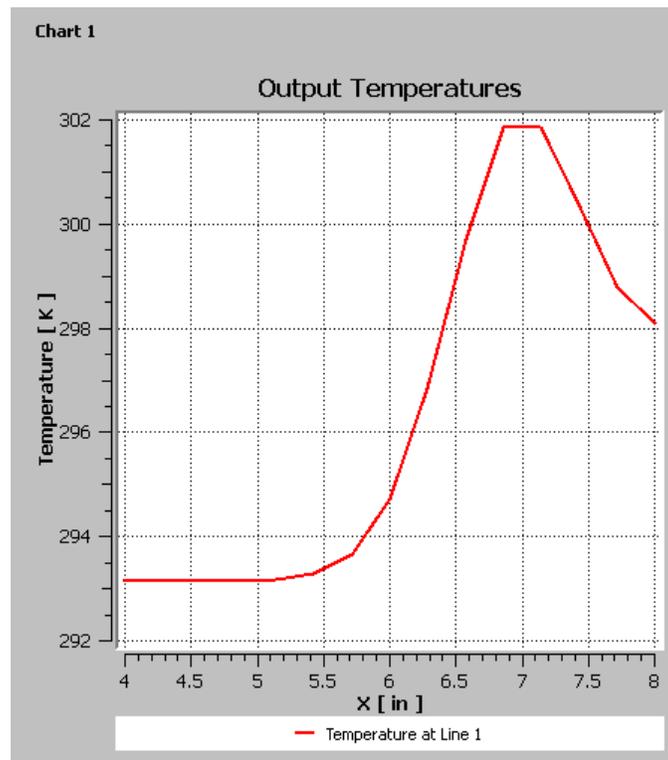


Figure 1.11: Chart of Output Temperatures at the Outlet

The Chart Viewer tab is at the bottom of the viewers area.

Now add a second line to the simulation and to the chart so that you can compare temperature distributions at two points:

1. Click on the **3D Viewer** tab so that you can see the line you are about to create.
2. Create a new line midway towards the outlet:
 - (a) On the menu bar, select **Insert/Location/Line**. In the **Insert Line** dialog box, accept the default name and click **OK**.
 - (b) In the **Details** view for Line 2, set the following:

Tab	Setting	Value
Geometry	Point 1*	4, 1.25, 0.001 **
	Point 2	8, 1.25, 0.001
	Line Type	Cut***
Color	Mode	Variable
	Variable	Temperature
	Range	Local
Render	Line Width	2

* The units are taken from the **3D Viewer options** setting, which you can access from the menu bar by selecting **Edit/Options**.

** The *Z* value ensures that the line unambiguously passes through the domain.

*** This setting constrains the line to be within the object.

- (c) Click **Apply**. A second line appears near the middle of the top section of pipe.
3. Add Line 2 to the existing chart:
 - (a) Right-click on **Outline / Report / Chart 1** and select **Edit**.
 - (b) In the **Details** view for Chart 1, click on the **Data Series** tab, then click the **New** icon . A new series appears. Select Series 2 and set the following:

Tab	Setting	Value
Series	Name	Temperature at Line 2
	Location	Line 2

- (c) Click **Apply**. A chart similar to the following appears in the **Chart Viewer**:

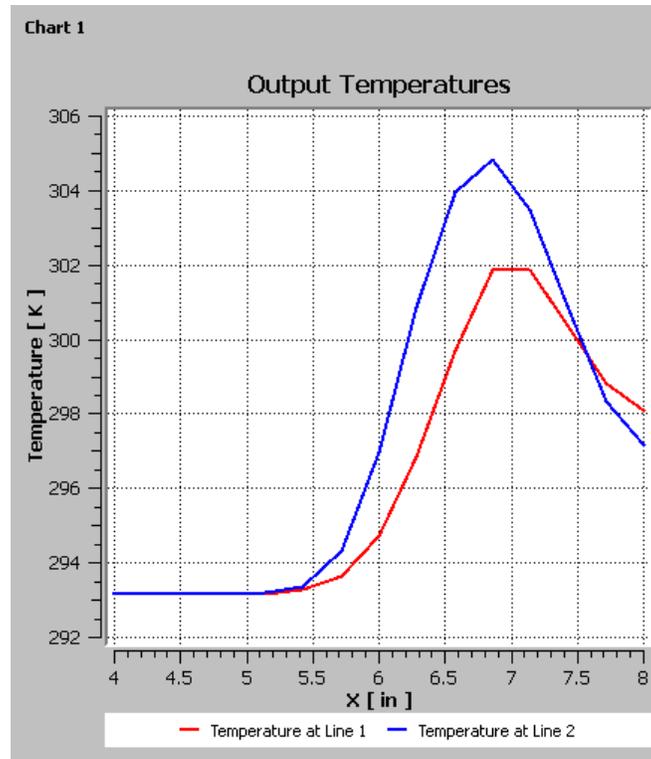


Figure 1.12: Chart of Output Temperatures at Two Locations

The temperatures at Line 2 are higher, which shows that less mixing has occurred than at Line 1.

Step 13: Create a Table to Show Mixing

You can create a table to show how values change at different locations, provided that the locations have been defined. In this section you will create three planes along the mixing region and measure the temperatures on those planes. You will then create a table and define functions that show temperature minimums and maximums, and the differences between those values.

1. In the 3D Viewer, ensure that only the wireframe is visible.
2. Click on the +Z axis on the triad to get a side-view of the elbow, then rotate the top of the elbow slightly towards you. This will make it easier for you to see the temperature planes that you will create.
3. From the tool bar, select Location / Plane. In the Insert Plane dialog box, type Table Plane 1 and click OK.

- In the Details view for Table Plane 1, set the following values:

Tab	Field	Value
Geometry	Domains	fluid
	Definition / Method	ZX Plane
	Definition / Y	-6.04
Color	Mode	Variable
	Variable	Temperature
	Range	Local
Render	Lighting	(clear)

- Click Apply.
- Right-click on Table Plane 1 and select Duplicate. The Duplicate dialog box appears. In the Duplicate dialog box, accept the default name Table Plane 2 and click OK. In the Outline view, double-click on Table Plane 2 and change Definition / Y to -0.55. Click Apply.
- Repeat the previous step, duplicating Table Plane 2 to make Table Plane 3 and changing Definition / Y to 2.18. Click Apply.
- Repeat the previous step, duplicating Table Plane 3 to make Table Plane 4 and changing Definition / Y to 5.04. Click Apply.

Now, create a table:

- From the menu bar, select Insert/Table. Accept the default table name and click OK. The Table Viewer opens.
- Type in the following headings:

	A	B	C	D
1	Distance to Outlet	Min. Temperature	Max. Temperature	Difference

- For the "Distance to Outlet" column, create an equation that gives the distance from the outlet (which is at the 8" mark). Click on cell A2, then in the Table Viewer's Insert bar, select Function / CFD-Post / minVal. In the cell definition field you see =minVal()@, which will be the base of the equation. With the cursor between the parentheses, type Y. Move the cursor after the @ sign and either type Table Plane 1 or select Insert / Location / Table Plane 1. The expression you have created gives the value of Y for Table Plane 1. To determine the distance of Table Plane 1 from the outlet, complete the equation as follows, being careful to add 8[in]- to the beginning of the equation to set the location of the outlet:

=8[in]-minVal(Y)@Table Plane 1

When you click away from cell A2, the equation is solved as 1.404e+01 [in].

4. Complete the rest of the table by entering the following cell definitions:

	A	B	C	D
1	Distance to Outlet	Min. Temperature	Max. Temperature	Difference
2	=8[in]-minVal(Y)@Table Plane 1	=minVal(T)@Table Plane 1	=maxVal(T)@Table Plane 1	=maxVal(T)@Table Plane 1-minVal(T)@Table Plane 1
3	=8[in]-minVal(Y)@Table Plane 2	=minVal(T)@Table Plane 2	=maxVal(T)@Table Plane 2	=maxVal(T)@Table Plane 2-minVal(T)@Table Plane 2
4	=8[in]-minVal(Y)@Table Plane 3	=minVal(T)@Table Plane 3	=maxVal(T)@Table Plane 3	=maxVal(T)@Table Plane 3-minVal(T)@Table Plane 3
5	=8[in]-minVal(Y)@Table Plane 4	=minVal(T)@Table Plane 4	=maxVal(T)@Table Plane 4	=maxVal(T)@Table Plane 4-minVal(T)@Table Plane 4
6	=8[in]-minVal(Y)@pressure outlet 7	=minVal(T)@pressure outlet 7	=maxVal(T)@pressure outlet 7	=maxVal(T)@pressure outlet 7-minVal(T)@pressure outlet 7

As you complete the table, notice that the minimum temperature values stay constant, but the maximum values decrease as mixing occurs.

Note: *The sixth row determines the values at the outlet, rather than on a plane you defined.*

5. The default format for cell data is appropriate for some variables, it is not appropriate here. Click on cell A2, then while depressing the **Shift** key, click in the lower-right cell (D6). Click on the *Number Formatting* icon  icon in the Table Viewer toolbar. In the Cell Formatting dialog box, set Precision to 2, change Scientific to Fixed, and click OK.

Step 14: Review and Modify a Report

As you work, ANSYS CFD-Post automatically updates a report, which you can see in the Report Viewer. At any time you can publish the report to an HTML file. In this section you will add a picture of the elbow and produce an HTML report:

1. Click on the **Report Viewer** tab at the bottom of the viewer to view the current report.
2. In the **Outline** tree view, double-click on the **Report / Title Page**. In the **Title** field on the **Content** tab of the **Details of Report Title Page**, type: **Analysis of Heat Transfer in a Mixing Elbow**.
3. Click **Apply**, then **Refresh Preview** to update the contents of the **Report Viewer**.
4. In the **Outline** tree view, ensure that only **User Location and Plots / Contour 1**, **Default Legend View 1**, and **Wireframe** are visible, then double-click **Contour 1**. On the **Geometry** tab, set **Variable** to **Temperature** and click **Apply**.
5. On the menu bar, select **Insert/Figure**. The **Insert Figure** dialog box appears. Accept the default name and click **OK**.
6. In the **Outline** tree view, double-click **Report / Figure 1**. In the **Caption** field, type **Temperature on the Symmetry Plane** and click **Apply**.
7. Click on the **3D Viewer**, then click on the cyan (ISO) sphere in the triad.
8. Click on the **Report Viewer**.
9. On the top frame of the **Report Viewer**, click the **Refresh** icon . The report is updated with a picture of the mixing elbow at the end of the report.
10. Optionally, click **Publish** to create an HTML version of the report. In the **Publish Report** dialog box, click **OK**. The report is written to **Report.htm** in your working directory.
11. Right-click in the **Outline** view and select **Hide All**, then select **Wireframe**.

Step 15: Create a Custom Variable and Animate the Display

In this section you will generate an expression using the CFX Expression Language (CEL), which you can then use in CFD-Post in place of a numeric value. You will then associate the expression with a variable, which you will also create. Finally, you will create a plane that displays the new variable, then move the plane to see how the values for the variable change.

1. Define a custom expression for the dynamic head formula $(\rho|V|^2)/2$ as follows:
 - (a) On the tab bar at the top of the workspace area, select **Expressions**. Right-click in the **Expressions** area and select **New**.
 - (b) In the **New Expression** dialog box, type: `DynamicHeadExp`
 - (c) Click **OK**.
 - (d) In the **Definition** area, type in this definition:

```
Density * abs(Velocity)^2 / 2
```

where:
 - `Density` is a *variable*
 - `abs` is a *CEL function* (`abs` is unnecessary in this example, it simply illustrates the use of a CEL function)
 - `Velocity` is a *variable*

*You can learn which predefined functions, variables, expression, locations, and constants are available by right-clicking in the **Definition** area.*
 - (e) Click **Apply**.
2. Associate the expression with a variable (as the plane you define in the next step can display only variables):
 - (a) On the tab bar at the top of the workspace area, select **Variables**. Right-click in the **Variables** area and select **New**.
 - (b) In the **New Variable** dialog box, type: `DynamicHeadVar`
 - (c) Click **OK**.
 - (d) In the **Details** view for `DynamicHeadVar`, click the drop-down arrow beside **Expression** and choose `DynamicHeadExp`. Click **Apply**.

3. Create a plane and animate it:
 - (a) Click on the 3D Viewer tab.
 - (b) Right-click on the wireframe and select **Insert / YZ Plane**.
 - (c) If you see a dialog box that asks in which direction you want the normal to point, choose the direction appropriate for your purposes.

A plane that maps the distribution of the default variable (Pressure) appears.
 - (d) On the **Color** tab, set **Variable** to “DynamicHeadVar”. On the **Render** tab, clear **Lighting**. Click **Apply**.

The plane now maps the dynamic head distribution.
 - (e) In the **3D Viewer** in with the mouse cursor in select mode, click on the plane and drag it to various places in the object to see how the location changes the DynamicHeadVar values displayed.
 - (f) Right-click on the plane and select **Animate**. The **Animation** dialog box appears and the plane moves through the entire domain, displaying changes to the DynamicHeadVar values as it moves.
 - (g) On the **Animation** dialog box, click the *Stop* icon , then click **Close**.

*You can define multiple planes and animate them concurrently. First, stop any animations currently running, then create a new plane. To animate both planes, hold down **Ctrl** to select multiple planes on the **Animation** dialog box and press the **Play** icon .*
4. In the upper-left corner of the 3D Viewer, click the down arrow beside **Figure 1** and change it to **View 1**.
5. In the Outline view, right-click beside **User Locations and Plots** and select **Hide All**, then select **Wireframe** and **Default Legend View 1** to make them visible.

Step 16: Loading and Comparing the Results to Those in a Refined Mesh

To this point you have been working with a coarse mesh. In this section you will compare the results from that mesh to those from a refined mesh:

1. Select **File/Load Results**. The **Load Results File** dialog box appears.
2. On the **Load Results File** dialog box, select **Keep current cases loaded** and keep the other settings unchanged.

3. Select `elbow3.cdat.gz` (or `elbow3.cdat`) and click Open.

In the 3D Viewer, there are now two viewports: in the title bar for View1 you have `elbow1`, and in View2 you have `elbow3`. In the Outline tree view under Cases you have `elbow1` and `elbow3`; all boundaries associated with each case are listed separately and can be controlled separately. You also have a new entry: Case / Case Comparison.

4. In the Toolbar, select the *Synchronize camera in displayed views* icon .

If the two cases are not oriented in the same way, clear the *Synchronize camera in displayed views* icon  and then select it again.

Examine the operation of CFD-Post when the two views are not synchronized and when they are synchronized:

1. In the viewer toolbar, clear the *Synchronize visibility in displayed views* icon .

2. With the focus in View 1, select **Insert / Contour** and create a contour of pressure on pressure outlet 7 that displays values in the local range.

Note that the contour appears only in View 1. When visibility is not synchronized, changes you make to **User Location and Plots** settings apply only to the currently active view.

3. In either view (while in viewing mode), click on the Z axis on the triad. Both views show their cases from the perspective of the Z axis.

4. In the viewer toolbar, select the *Synchronize visibility in displayed views* icon .

5. With the focus on the view that contains `elbow3`, select **Insert / Contour**. Accept the default name and click **OK**. Define a contour that displays temperature on the symmetry plane:

Tab	Field	Value
Geometry	Locations Variable Range	symmetry Temperature Local
Render	Lighting	(clear)

Click **Apply**.

Note that the contour appears in both views. You can see the differences between the coarse and refined meshes:

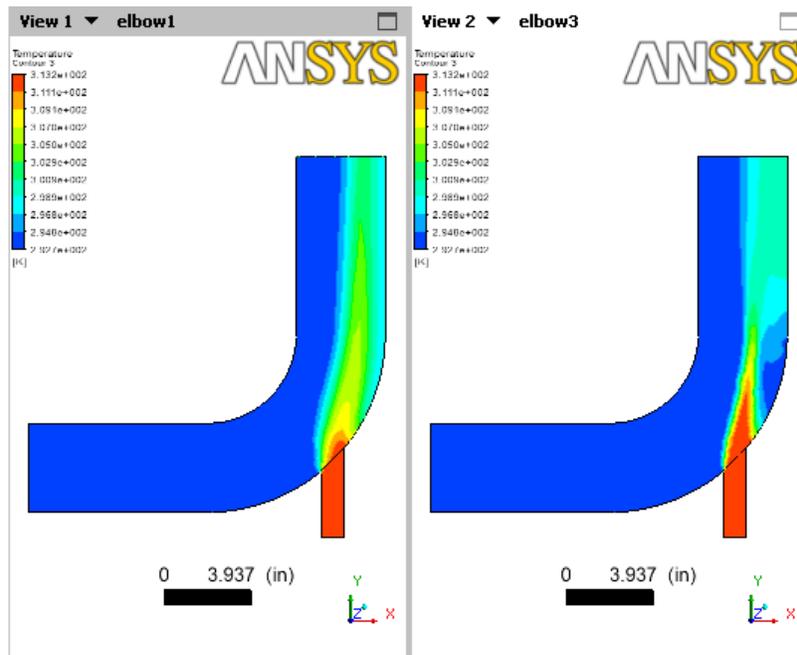


Figure 1.13: Comparing Contour Plots of Temperature on Two Mesh Densities

You can now compare the differences between the coarse and refined meshes:

1. In the Outline tree view, double-click Cases / Case Comparison.
2. In the Case Comparison Details view, select Case Comparison Active and click Apply. The differences between the values in the two cases appear in a third view. Click the Z axis of the triad to restore the orientation of the views.

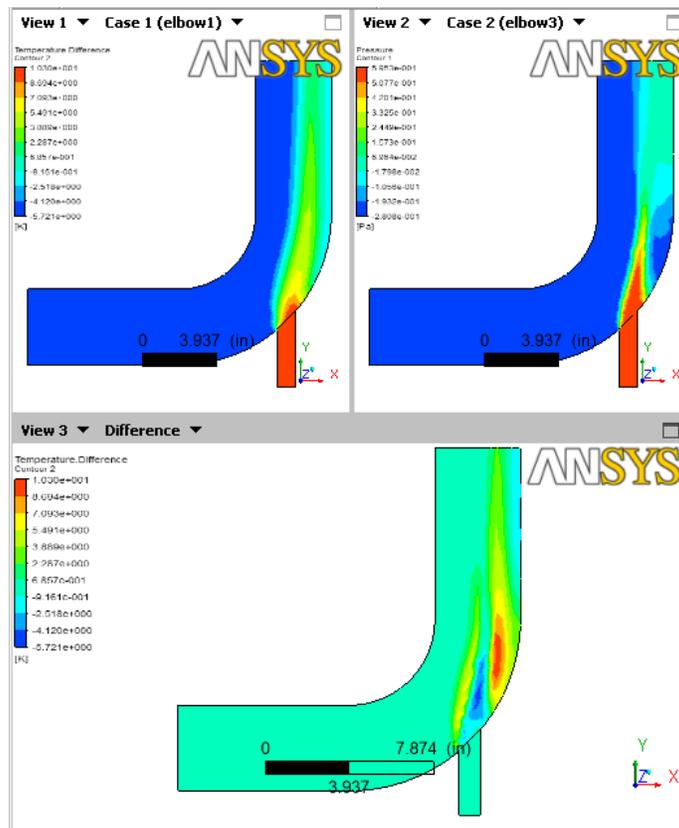


Figure 1.14: Displaying Differences in Contour Plots of Temperature on Two Mesh Densities

Now, revert to a single view that shows the original case:

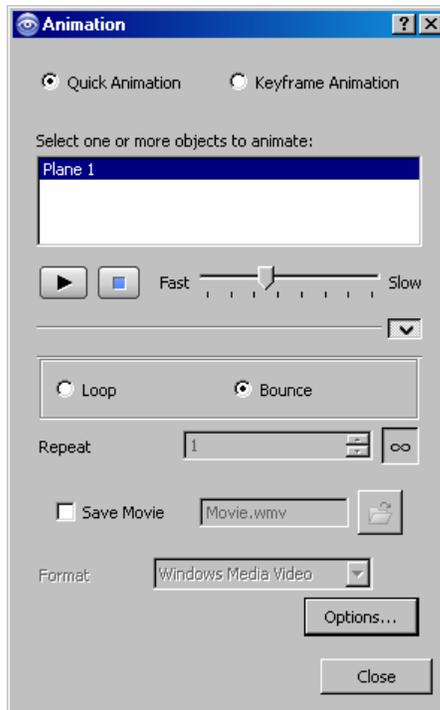
1. To remove the Difference view, clear Case Comparison Active and click Apply.
2. To remove the refined mesh case, in the Outline tree view, right-click elbow3 and select Unload.

Step 17: Save Your Work

When you began this tutorial, you loaded a solver results file. When you save the work you have done in CFD-Post, you save the current state of CFD-Post into a CFD-Post State file (.cst).

1. How you save your work depends on whether you are running CFD-Post standalone or from within ANSYS Workbench:
 - From CFD-Post standalone:
 - (a) From the menu bar, select **File/Save State**.

This operation saves the expression, custom variable, and the settings for the objects in a .cst file and saves the state of the animation in a .can file. The .cas.gz and .cdat.gz files remain unchanged.
 - (b) A **Warning** dialog asks if you want to save the animation state. Click **OK**.
 - (c) Optionally, confirm the state file's contents: close the current file from the menu bar by selecting **File / Close** (or press **Ctrl+W**) then reload the state file (select **File / Load State** and choose the file that you saved in step 1.)
 - From ANSYS Workbench:
 - (a) From the CFD-Post menu bar, select **File/Quit**. ANSYS Workbench saves the state file automatically.
 - (b) In the ANSYS Workbench Project Schematic, double-click on the **Results** cell. CFD-Post re-opens with the state file loaded.
2. Save a picture of the current state of the simulation: in the **Outline** view, show **Contour 1**. With the focus in the **3D Viewer**, click the *Save Picture* icon  from the tool bar. In the **Save Picture** dialog box, click **Save**. A PNG file of the current state of the viewer is saved to <casename>.png (e.g., elbow1.png) in your working directory.
3. You can recreate the animation you made in the previous section and save it to a file:
 - (a) Click on the cyan (ISO) sphere in the triad to orient the elbow to display **Plane 1**.
 - (b) In the **Outline** view, clear **Contour 1** and show **Plane 1**.
 - (c) Right-click on **Plane 1** in the **3D Viewer** and select **Animate**. The **Animation** dialog appears and the plane moves through the entire domain.
 - (d) Click the stop icon .
 - (e) If necessary, display the full animation control set by clicking .



- (f) The **Repeat** is set to infinity; change the value to 1 by clicking the infinity button. The **Repeat** field becomes enabled and by default is set to one.
- (g) Enable **Save Movie** to save the animation to the indicated file.
- (h) Click the *Play the animation* icon .

The plane moves through one cycle.

You can now go to your working directory and play the animation file in an appropriate viewer.

4. Click **Close** to close the **Animation** dialog.
5. Close **CFD-Post**: from the tool bar select **File/Quit**. If prompted, you may save your changes.

Step 18: Generated Files

As you worked through this tutorial you generated the following files in your working directory (default names are given):

- `elbow1.cst`, the state file, and `elbow1.can`, the animation associated with that state file
- `elbow1.wmv`, the animation
- `elbow1.png`, a picture of the contents of the 3D Viewer
- `Report.htm`, the report.