CFD EXPERTS

Simulate the Future

WWW.CFDEXPERTS.NET

Ansys 2021 R2

Engineering What's Ahead.

Ansys CFD-Post Tutorials

Ansys

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2021 R2
July 2021

ANSYS, Inc. and Ansys Europe, Ltd. are UL registered ISO 9001: 2015 companies.

Published in the U.S.A.

Table of Contents

List of Figures

Chapter 1: Introduction to the Tutorials

The tutorials are designed to introduce the capabilities of CFD-Post. The following tutorials are available:

- Postprocessing Fluid Flow and Heat Transfer in a Mixing Elbow (p. 9) illustrates how to use CFD-Post to visualize a three-dimensional turbulent fluid flow and heat transfer problem in a mixing elbow.
- Turbo Postprocessing (p. 47) demonstrates the turbomachinery postprocessing capabilities of CFD-Post to visualize flow in a centrifugal compressor.
- Quantitative Postprocessing (p. 65) demonstrates the quantitative postprocessing capabilities of CFD-Post using a 3D model of a circuit board with a heat-generating electronic chip mounted on it.

Using Help

To open the Ansys Help, from the menu bar select **Help** > **Contents**.

You may also use context-sensitive help, which is provided for many of the details views and other parts of the interface. To invoke the context-sensitive help for a particular details view or other feature, ensure that the feature is active, place the mouse pointer over it, then press **F1**. Not every area of the interface supports context-sensitive help.

Chapter 2: Postprocessing Fluid Flow and Heat Transfer in a Mixing Elbow

This tutorial illustrates how to use 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:

2.1. Problem Description

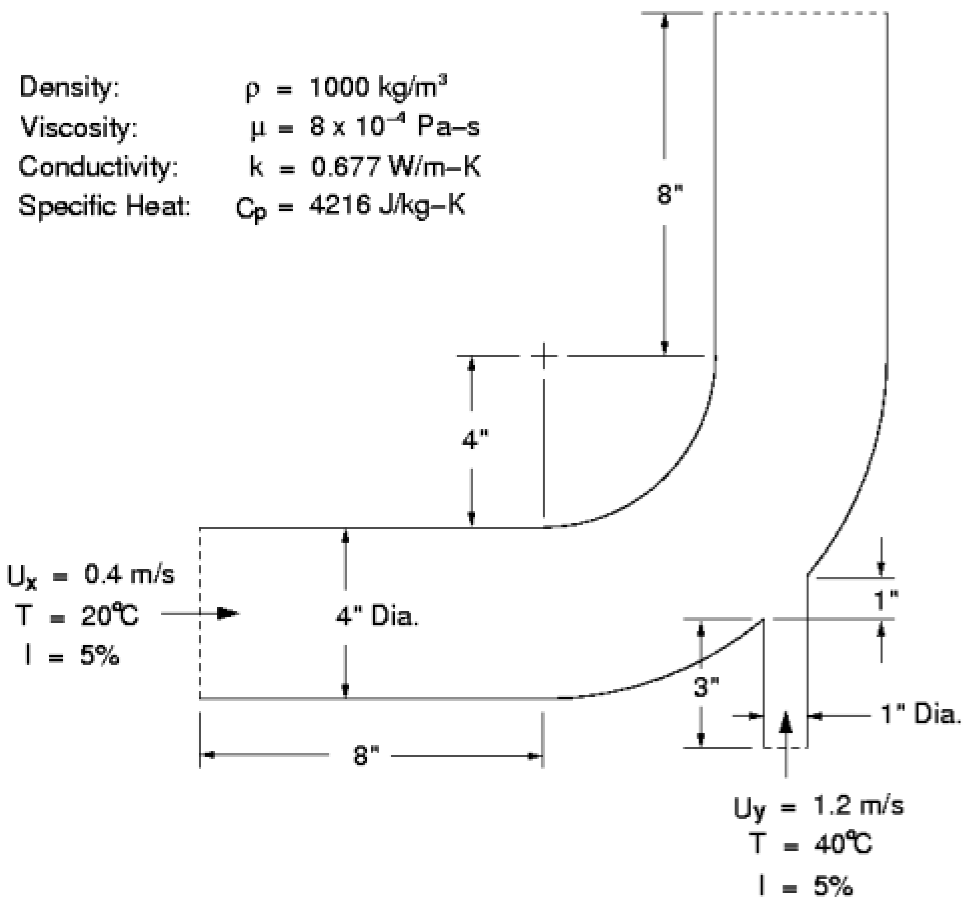
The problem to be considered is shown schematically in Figure 2.1: Problem Specification (p. 10). A cold fluid at 20° C flows into the pipe through a large inlet and mixes with a warmer fluid at 40° C that enters through a smaller inlet located at the elbow. The pipe dimensions are in inches, but the 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 Fluent case. The combination of SI and Imperial units is not typical, but follows a Fluent example.

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

Figure 2.1: Problem Specification



If this is the first tutorial you are working with, it is important to review Introduction to the Tutorials (p. 7) before beginning.

2.2. Preparing the Working Directory

1. 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.

2. Download the `mixing_elbow.zip` file here.

3. Unzip `mixing_elbow.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `elbow_tracks.xml`
   - `elbow1.cas.gz`
   - `elbow1.cdat.gz`
   - `elbow3.cas.gz`

- `elbow3.cdat.gz`

2.3. Setting the Working Directory and Starting 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 launch CFD-Post stand-alone, from the Ansys CFX Launcher, or from Ansys Workbench:

- To run CFD-Post stand-alone
  - On Windows:
    1. From the **Start** menu, right-click **All Programs > Ansys 2021 R2 > Fluid Dynamics > CFD-Post 2021 R2** and select **Properties**.
    2. Type the path to your working directory in the **Start in** field and click **OK**.
    3. Click **All Programs > Ansys 2021 R2 > Fluid Dynamics > CFD-Post 2021 R2** to launch CFD-Post.
  - On Linux, enter `cfdpost` in a terminal window that has its path set up to run CFD-Post. The path will be something similar to `/usr/ansys_inc/v212/CFD-Post/bin`.
- To run Ansys CFX Launcher
  1. Start the launcher.

     You can run the launcher in any of the following ways:
     - On Windows:
       - → From the **Start** menu, select **All Programs > Ansys 2021 R2 > Fluid Dynamics > CFX 2021 R2**.
       - → In a Command Prompt that has its path set up correctly to run Ansys CFX, enter `cfx5launch`. If the path is not set up correctly, you will need to type the full pathname of the `cfx5launch` command, which will be something similar to `C:\Program Files\ANSYS Inc\v212\CFX\bin`.
     - On Linux, 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/v212/CFX/bin`.
  2. Set the working directory.
  3. Click the **CFD-Post 2021 R2** 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 the A2 **Results** cell and select **Edit**. **CFD-Post** opens.

2.4. Display the Solution in CFD-Post

In the steps that follow, you will explore the solution using CFD-Post.

2.4.1. Prepare the Case and Set the Viewer Options

1. 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**.

   Ignore any message boxes that appear regarding global variable ranges or solution history by clicking **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**.

2. Optionally, set the viewer background to white:

   a. Right-click the viewer and select **Viewer Options**.

   b. In the **Options** dialog box, select **CFD-Post** > **Viewer**.

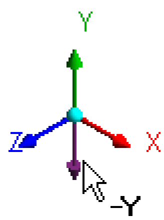
   c. Set:

      - **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.

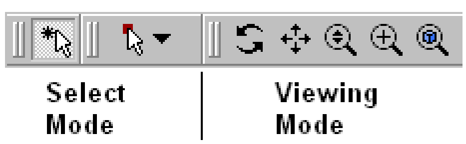
e. Experiment with rotating the object by clicking on the arrows of the triad in the **3D Viewer**. This is the triad:



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 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.

3. 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 (p. 9), the solution units are SI, except for the length, which is measured in inches.

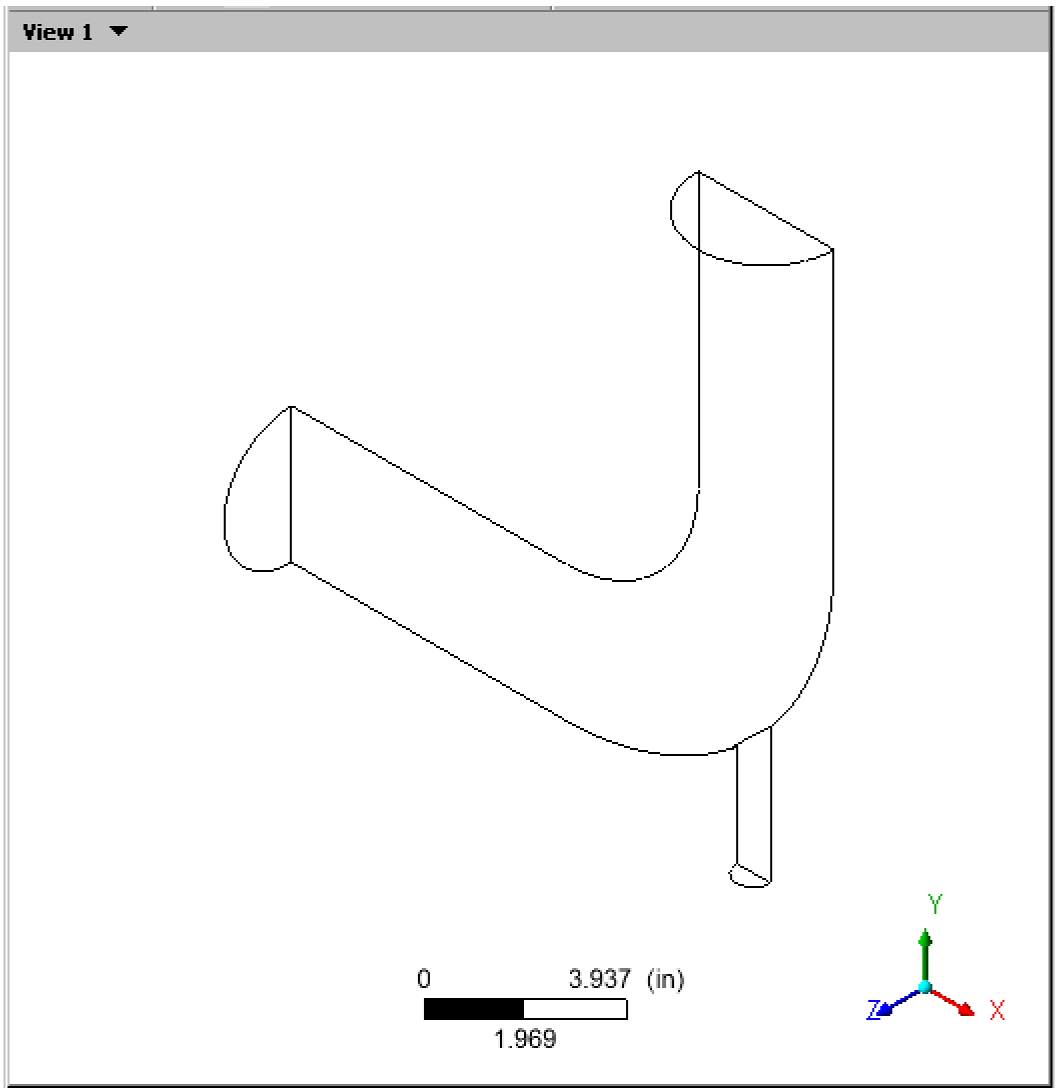
   a. Right-click the viewer and select **Viewer Options**.

   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.



2.4.2. 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:

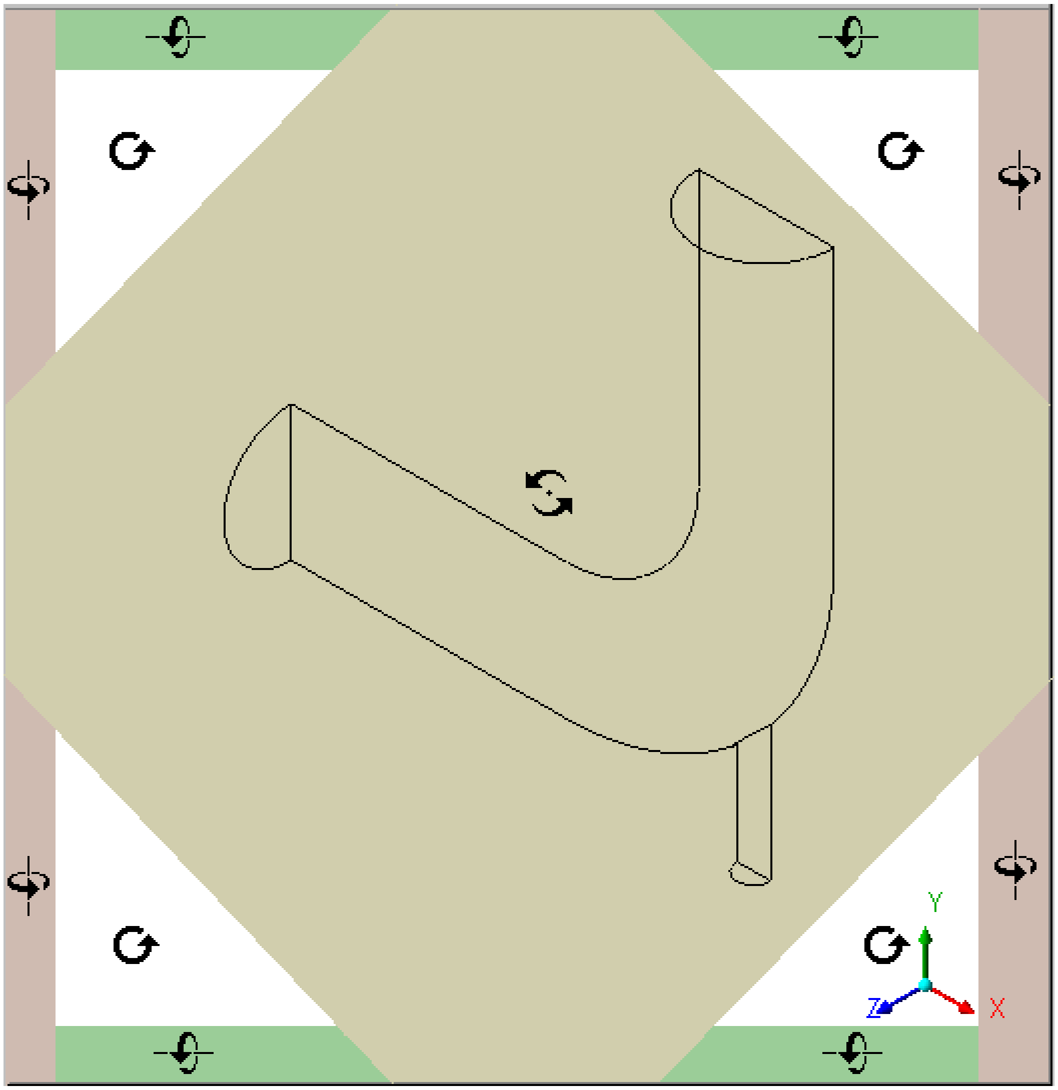
1. Click *Zoom Box* 
2. Click and drag a rectangular box over the geometry.
3. Release the mouse button to zoom in on the selection.

   The geometry zoom changes to display the selection at a greater resolution.
4. Click *Fit View*  to re-center and re-scale the geometry.

Now, the rotation functions:

1. Click *Rotate*  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 2.2: 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 From -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*.

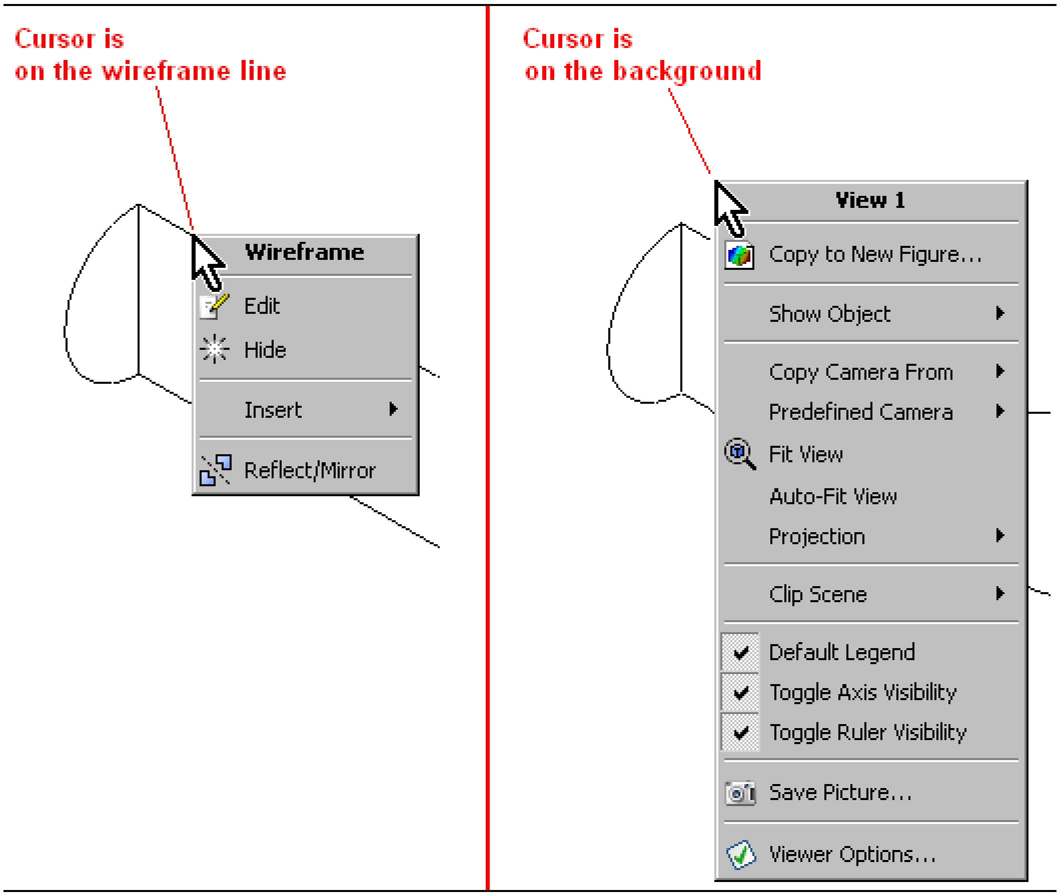
1. Click  to enter select mode.

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 2.3: 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 the wall, a new menu appears.

6. Click 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.

2.4.3. 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 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 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 shortcut menu, as opposed to the shortcut menu for the symmetry plane.) See Figure 2.3: Right-click Menus Vary by Cursor Position (p. 17).

3. From the shortcut 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.

Tip:

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

2.4.4. 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**	symmetry [a]
	Variable	Velocity [b]

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

b. 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.

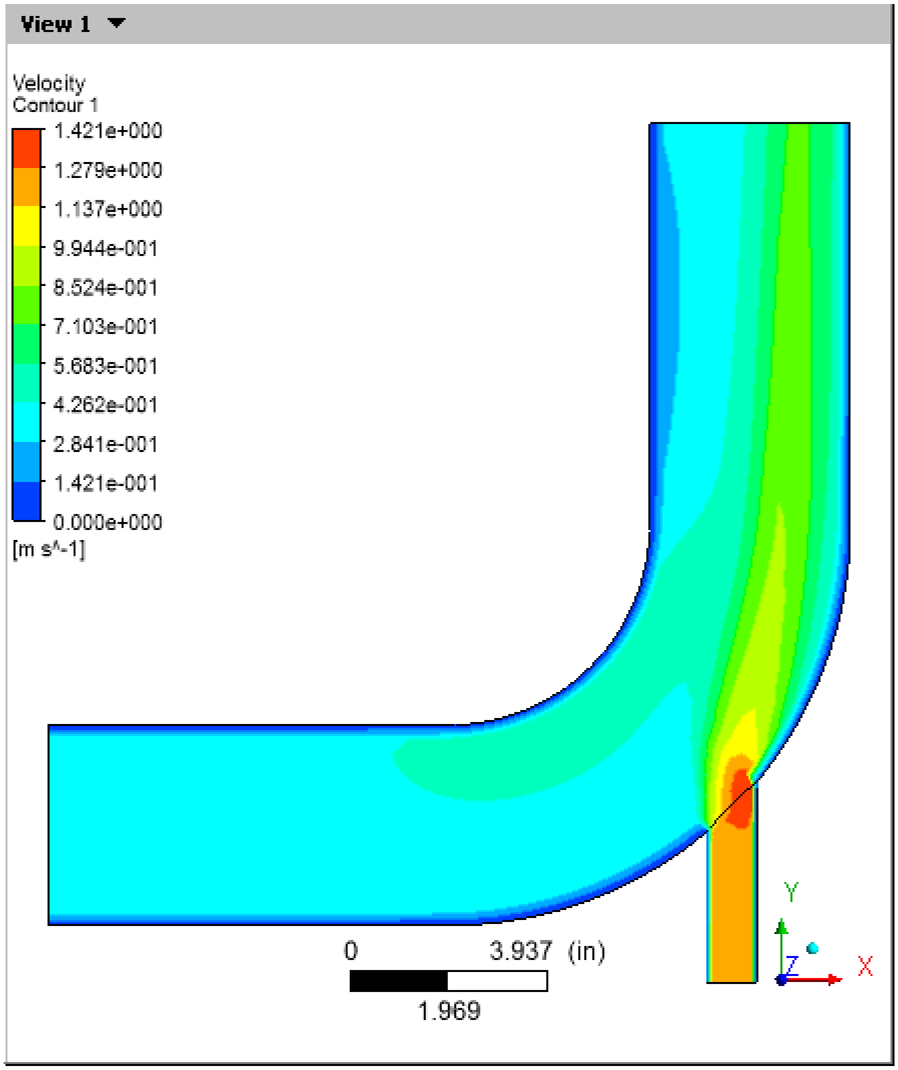
4. The coloring of the contour plot may not correspond to the colors on the legend because the viewer has a light source enabled by default. There are several ways to correct this:

   - 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 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 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 2.4: Velocity on the Symmetry Plane



6. Improve the contrast between the contour regions:

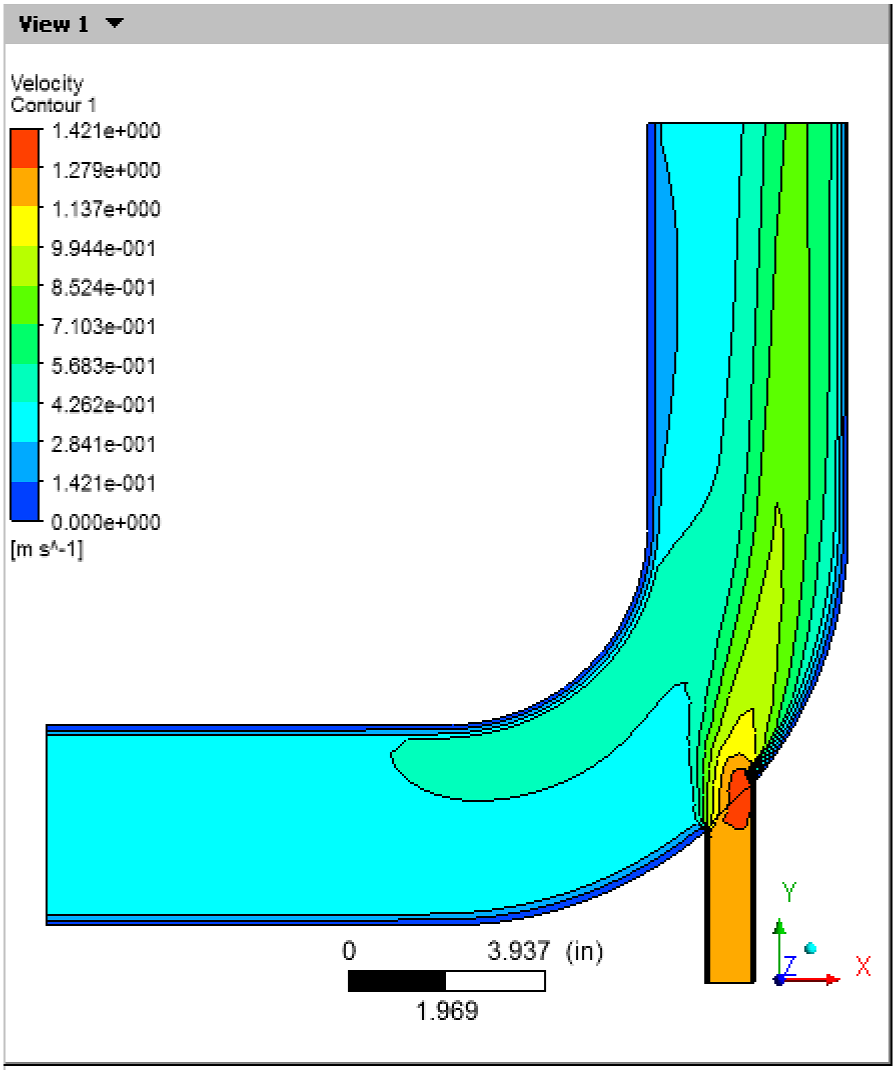
   a. On the **Render** tab, select **Show Contour Lines** and click the plus sign to view more options.

   b. Select **Constant Coloring**.

   c. Set **Color Mode** to `User Specified` and **Line Color** to black (if necessary, click the bar beside **Line Color** until black appears).

   d. Click **Apply**.

Figure 2.5: 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.

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

2.4.5. Show the 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.

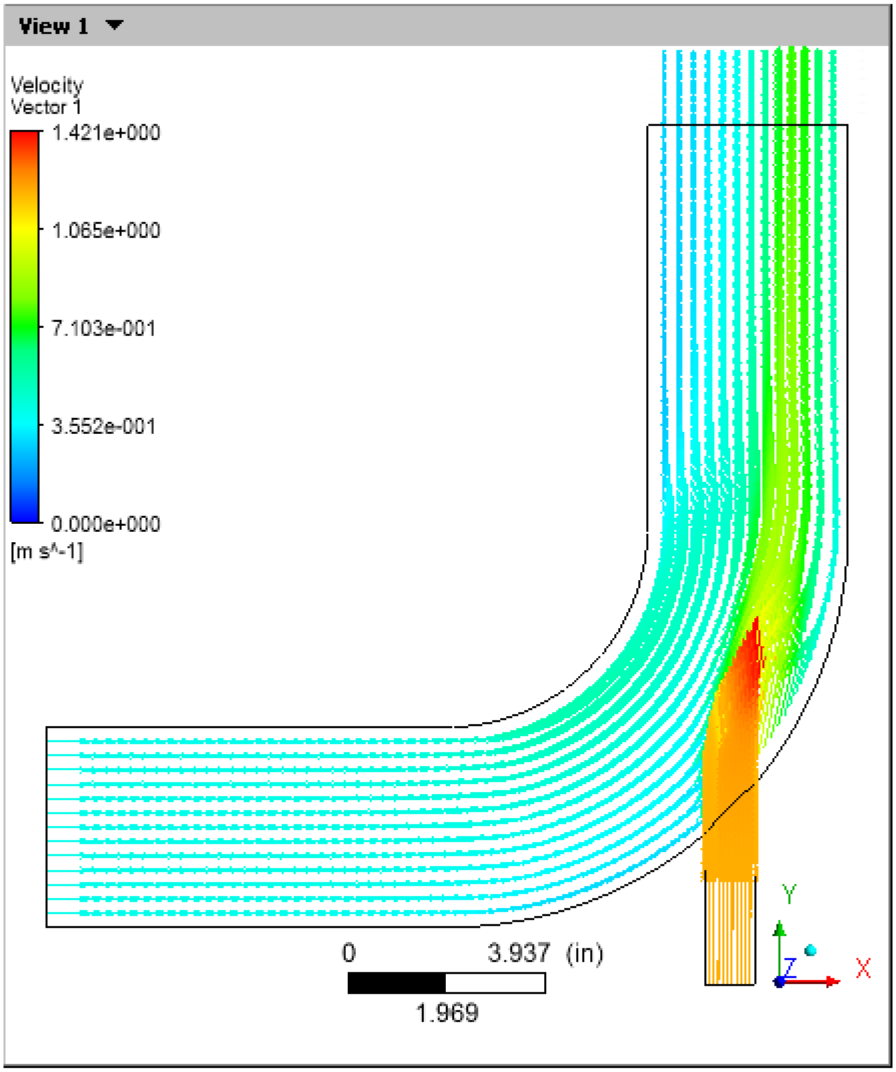
3. On the **Geometry** tab, set **Domains** to `fluid` and **Locations** to `symmetry`.

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 2.6: 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 the **Color** tab.

   b. Set **Mode** to `Variable`.

      The **Variable** field becomes enabled.

   c. Click 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**.

In this example you will create streamlines to show the flow distribution by velocity and color those streamlines to show turbulent kinetic energy. CFD-Post uses the **Variable** setting on the **Geometry** tab to determine how to calculate the streamlines (that is, location). In contrast, the **Variable** setting on the **Color** tab determines the color used when plotting those streamlines.

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 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 *Location editor*. 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 select 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 the **Color** tab and make the following changes:

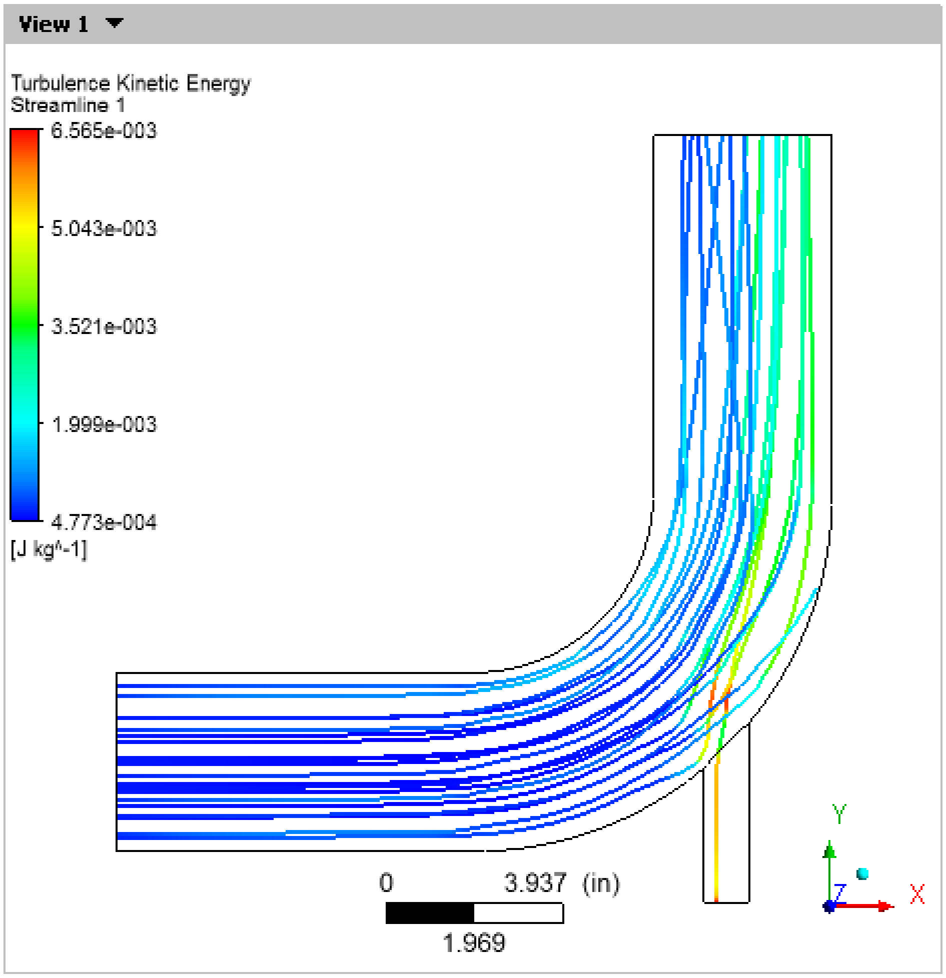
    a. Set **Mode** to `Variable`. The **Variable** field becomes enabled.

    b. Set **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 2.7: 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, select `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.

2.4.6. 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 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 `wall` to edit its properties.

d. On the **Render** tab, set **Transparency** to `0.75`.

e. 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`.

4. 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.

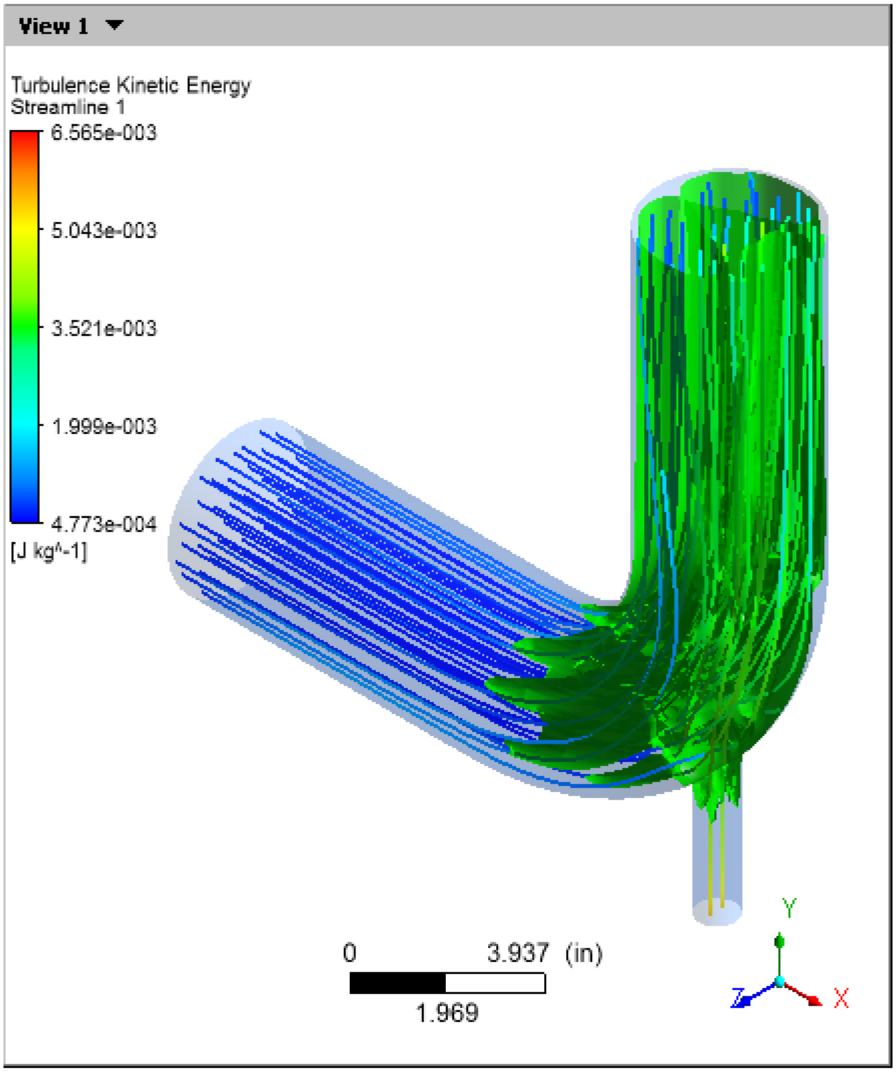
5. On the **Color** tab, click the colored bar in the **Color** field until the bar is green. Click **Apply**.

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

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

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

Figure 2.8: Absolute Helicity Vortex



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

2.4.7. Show Volume Rendering

CFD-Post displays volume rendering to enable you to better understand the processes in your simulation.

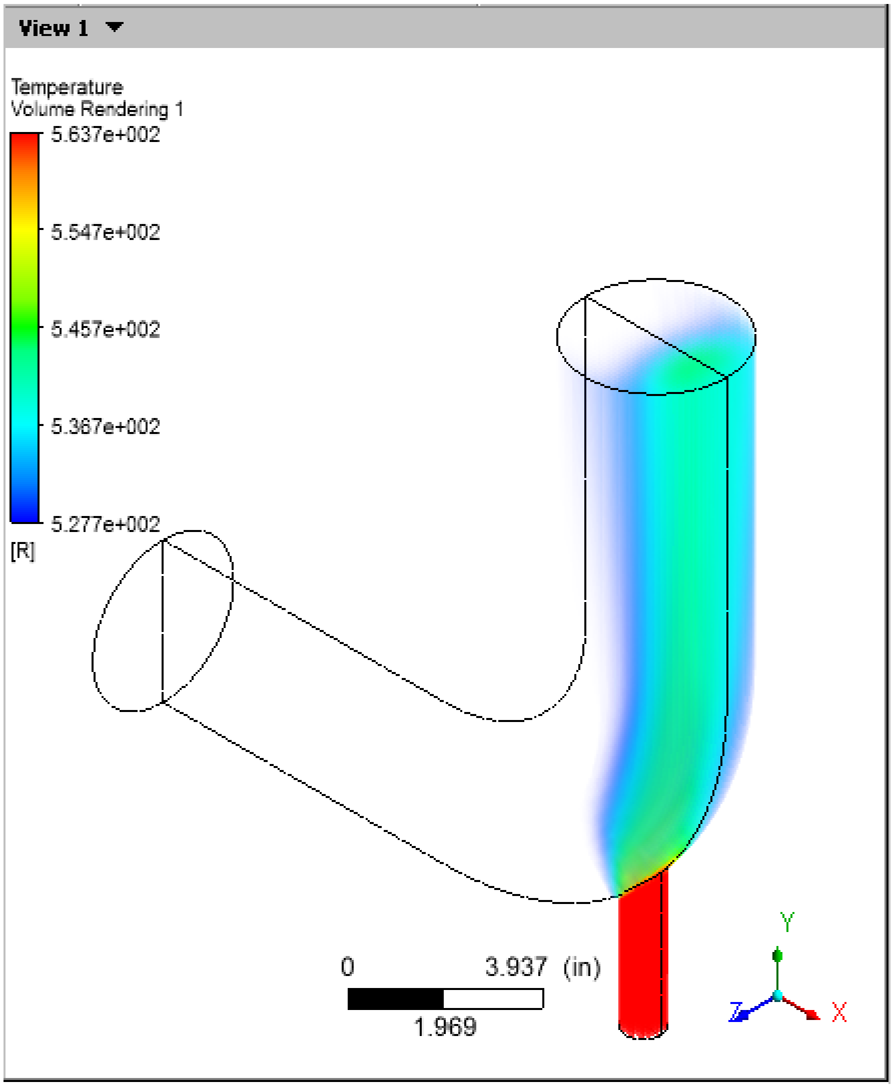
1. From the menu bar, select **Insert** > **Volume Rendering** and click **OK** to accept the default name.

2. In the details view for **Volume Rendering 1** on the **Geometry** tab, set **Variable** to `Temperature`.

3. On the **Color** tab, set **Mode** to `Variable` and **Variable** to `Temperature`. Click **Apply**.

4. If necessary to orient the simulation as shown below, right-click in the **3D Viewer** and select **Predefined Camera** > **Isometric View (Y up)**.

Figure 2.9: Volume Rendering of Temperature

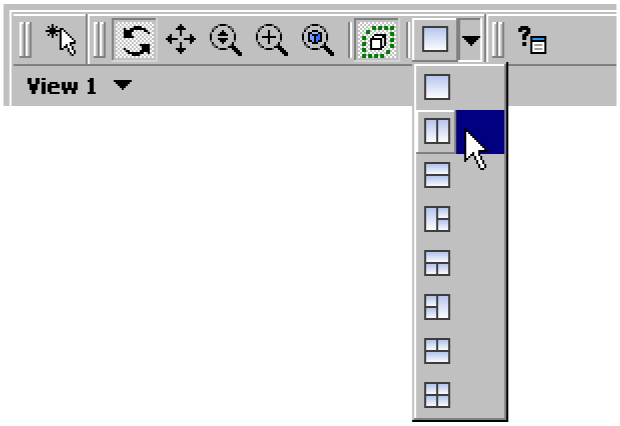


5. Hide the Volume Rendering object by clearing the check box beside **Volume Rendering 1** in the **Outline** tree view.

2.4.8. 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 From -Y**.
3. In the **Outline** tree view, double-click `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 that **Variable** is set to `Pressure`.
   c. Ensure that **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*  is selected.
   e. Click *Synchronize visibility in displayed views*  to disable this feature.

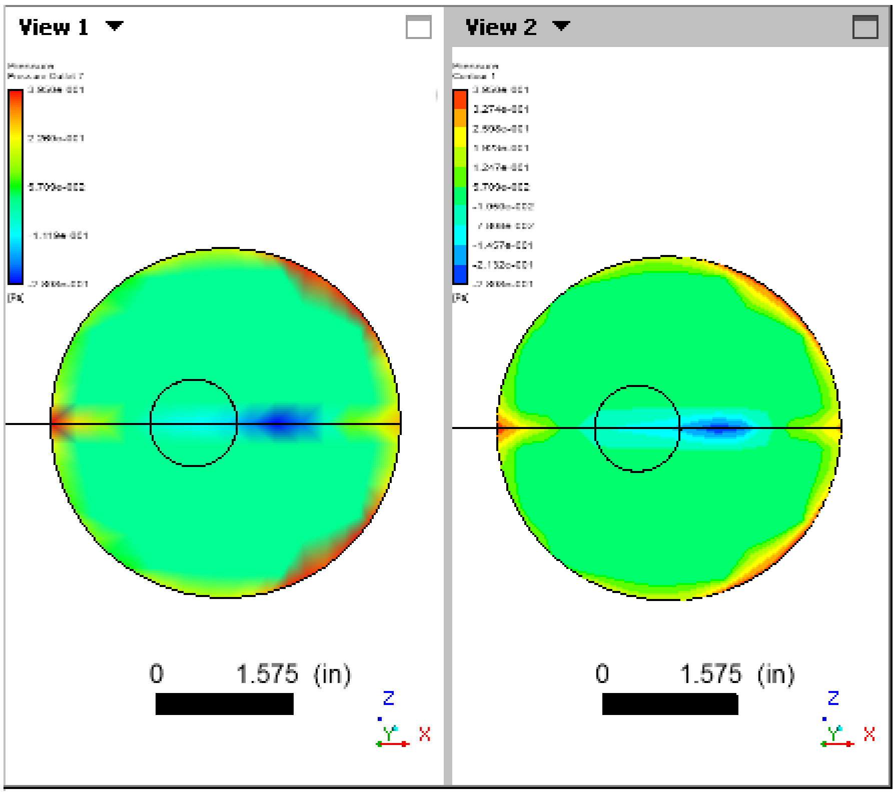
Now, add a contour plot at the same location:

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 **Locations** is set to `pressure outlet 7` and **Variable** is set to `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 2.10: Boundary Pressure vs. a Contour Plot of Pressure



9. Enhance the contrast on the contour bands:

   a. In the **Outline** tree view, right-click **User Locations and Plots** > **Contour 2** and select **Edit**.

   b. In the details view for the contour, on the **Render** tab, select **Show Contour Lines** > **Constant Coloring**, set **Color Mode** to `User Specified`, and click **Apply**.

   c. Click 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 *Synchronize camera in displayed views* . Both viewports take the camera orientation of the active viewport.

    c. Clear the *Synchronize camera in displayed views*  icon and click 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 *Synchronize visibility in displayed views* .

e. In the **Outline** tree view, right-click **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.

2.4.9. Review and Modify a Report

As you work, 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 the **Report Viewer** tab at the bottom of the viewer to view the current report.
2. In the **Outline** tree view, double-click 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 the **3D Viewer**, then click the cyan (ISO) sphere in the triad.
8. Click the **Report Viewer** tab.
9. In the top frame of the **Report Viewer**, click *Refresh* Refresh. 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** tree view and select **Hide All**.

2.4.10. 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*

   Tip:

   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 the **3D Viewer** tab.

   b. Right-click 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** with the mouse cursor in select mode, click the plane and drag it to various places in the object to see how the location changes the DynamicHeadVar values displayed.

f. Right-click 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. In the **Animation** dialog box, click *Stop* ■, then click **Close**.

Tip:

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 in the **Animation** dialog box and click *Play* ▶.

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** tree view, right-click **User Locations and Plots** > **Contour 1** and select **Hide All**.

2.4.11. Load and Compare 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 **View 1** you have **elbow1**, and in **View 2** 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: **Cases** > **Case Comparison**.

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

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 *Synchronize visibility in displayed views* .
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.

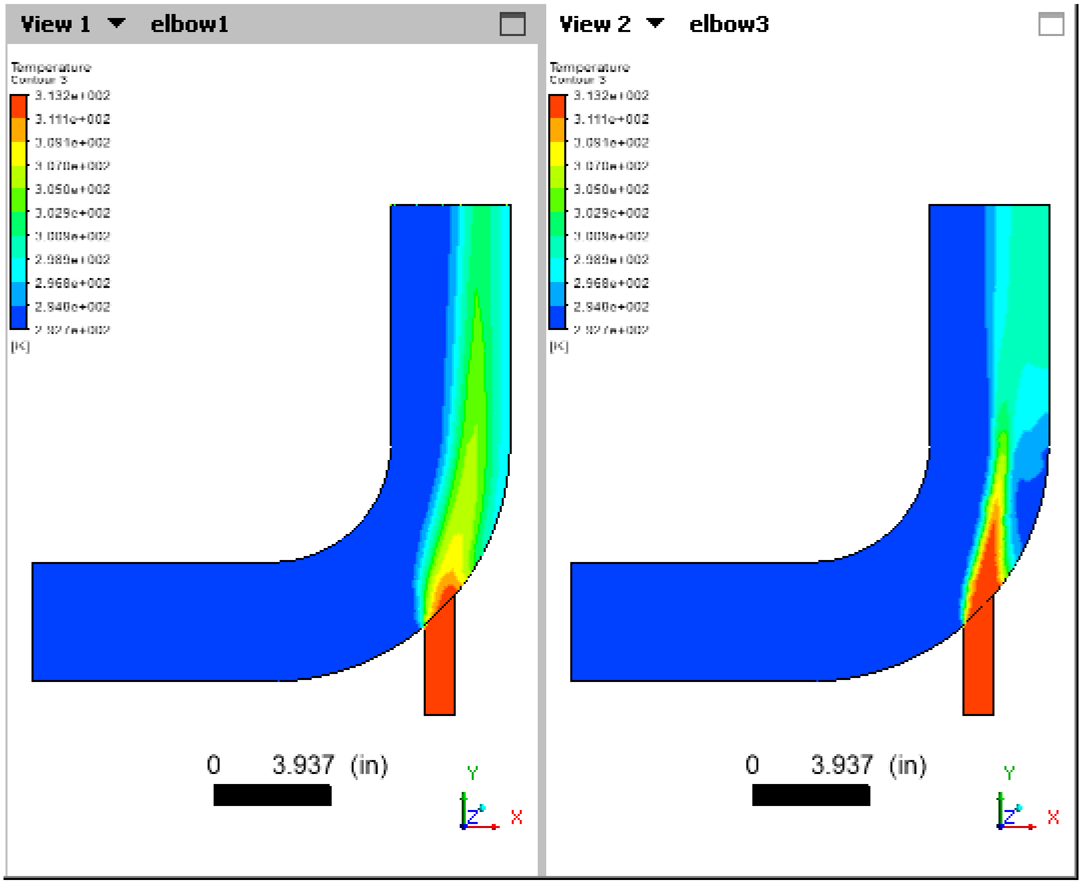
   Note that the contour appears only in View 1. When visibility is not synchronized, changes you make to **User Location and Plots** settings apply only to the currently active view.
3. In either view (while in viewing mode), click the Z axis on the triad. Both views show their cases from the perspective of the Z axis.
4. In the viewer toolbar, select *Synchronize visibility in displayed views* .
5. With the focus on the view that contains `elbow3`, select **Insert** > **Contour**. Accept the default name and click **OK**.
6. Define the contour such that it displays temperature on the symmetry plane:

Tab	Setting	Value
Geometry	**Locations**	symmetry
	Variable	Temperature
	Range	Local
Render	Show Contour Bands > **Lighting**	 (Cleared)

   Click **Apply**.

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

Figure 2.11: Comparing Contour Plots of Temperature on Two Mesh Densities

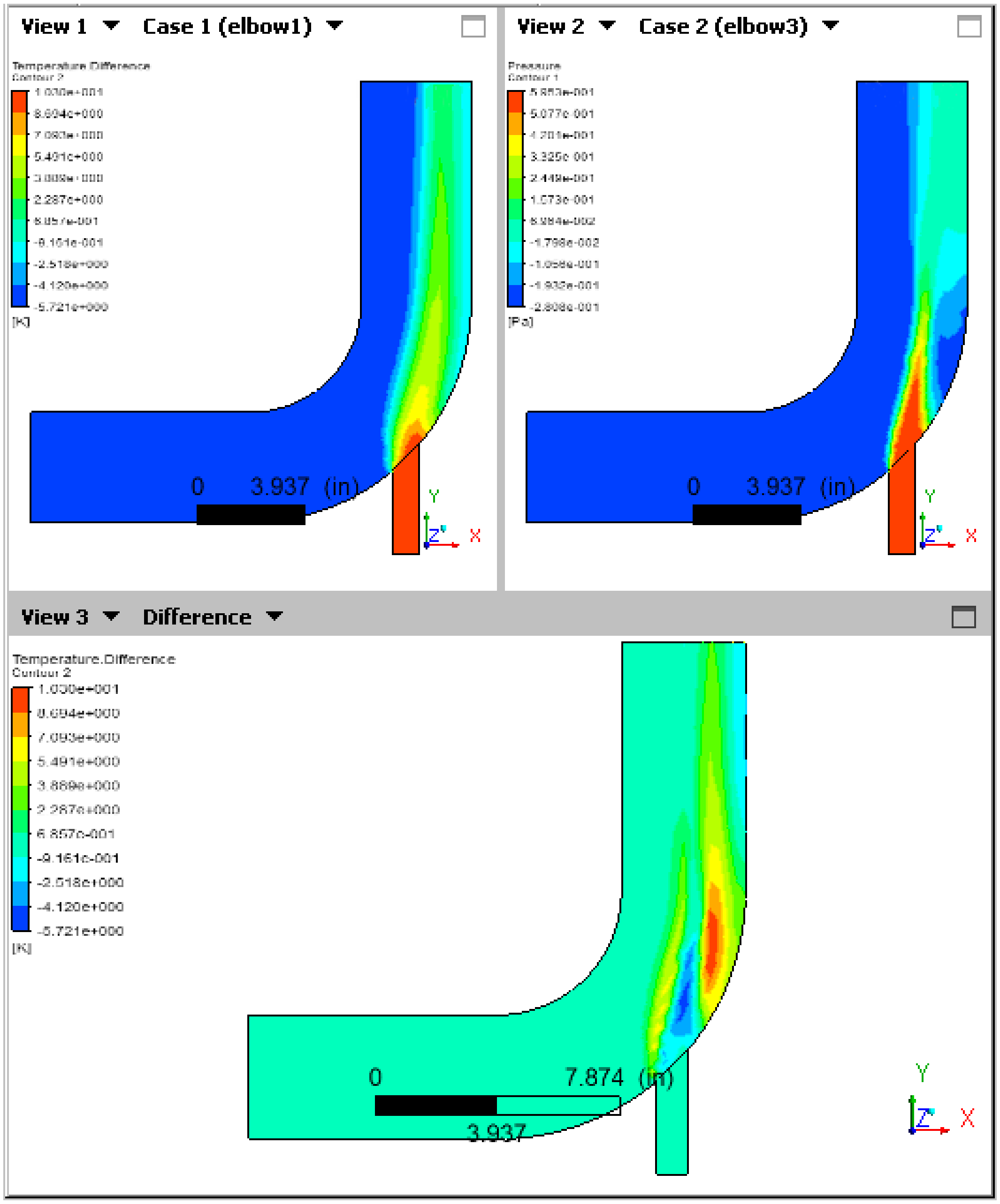


You can now compare the differences between the coarse and refined meshes:

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 2.12: 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**.
3. In the **Outline** tree view, right-click **User Locations and Plots** > **Contour 1** and select **Hide All**.

2.4.12. Display Particle Tracks

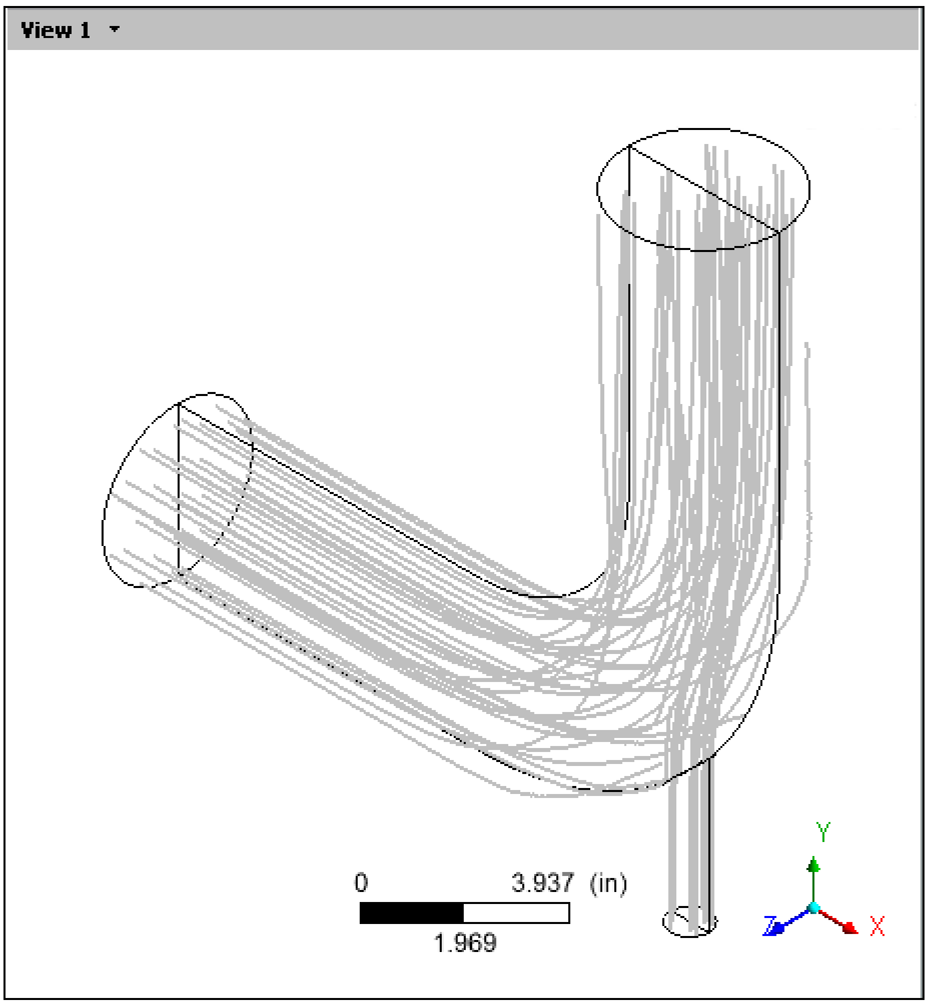
You can export an XML file of Particle Tracks from Fluent and display the tracks in CFD-Post.

To display particle tracks:

1. With only `elbow1.cdat` loaded, load the particle track file `elbow_tracks.xml`:
   - Select **File** > **Import** > **Import Fluent Particle Track File**.

2. In the **Import Fluent Particle Track File** dialog box, select: `elbow_tracks.xml`
3. Click **Open**.
4. Click **OK**.

   Particle tracks appear in the **3D Viewer**. The tracks stretch from the two inlets to the outlet.

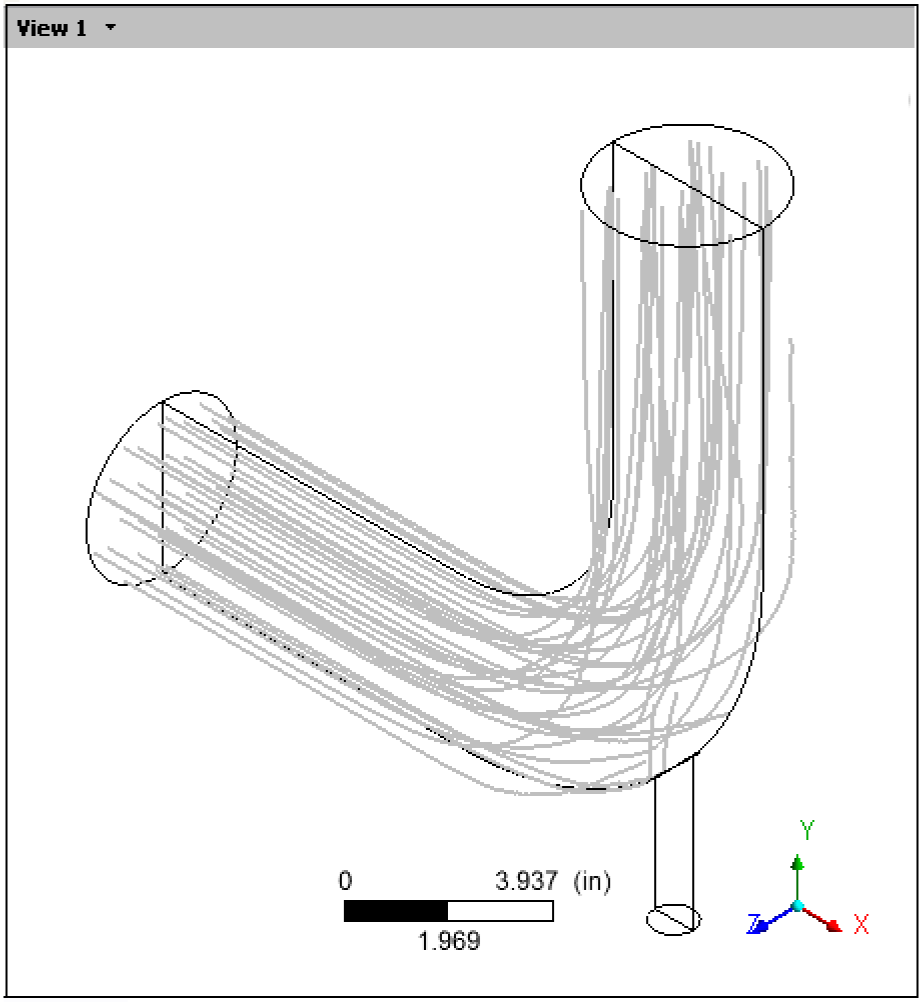


Make only the particle tracks from the large inlet visible:

1. In the **Outline** tree view, double-click **User Locations and Plots**> **Fluent PT for Anthracite** to see the details view for the particle tracks.
2. In the details view, click the drop-down arrow beside the **Injections** field so that you can see the names of the two sets of particle tracks.

3. Select **injection-0**.

4. Click **Apply**.



Display both sets of particle tracks, each set in a different color:

1. First, display both sets of particle tracks again:

   a. Click the drop-down arrow beside the Injections field.

   b. Select **injection-0,injection-1**.

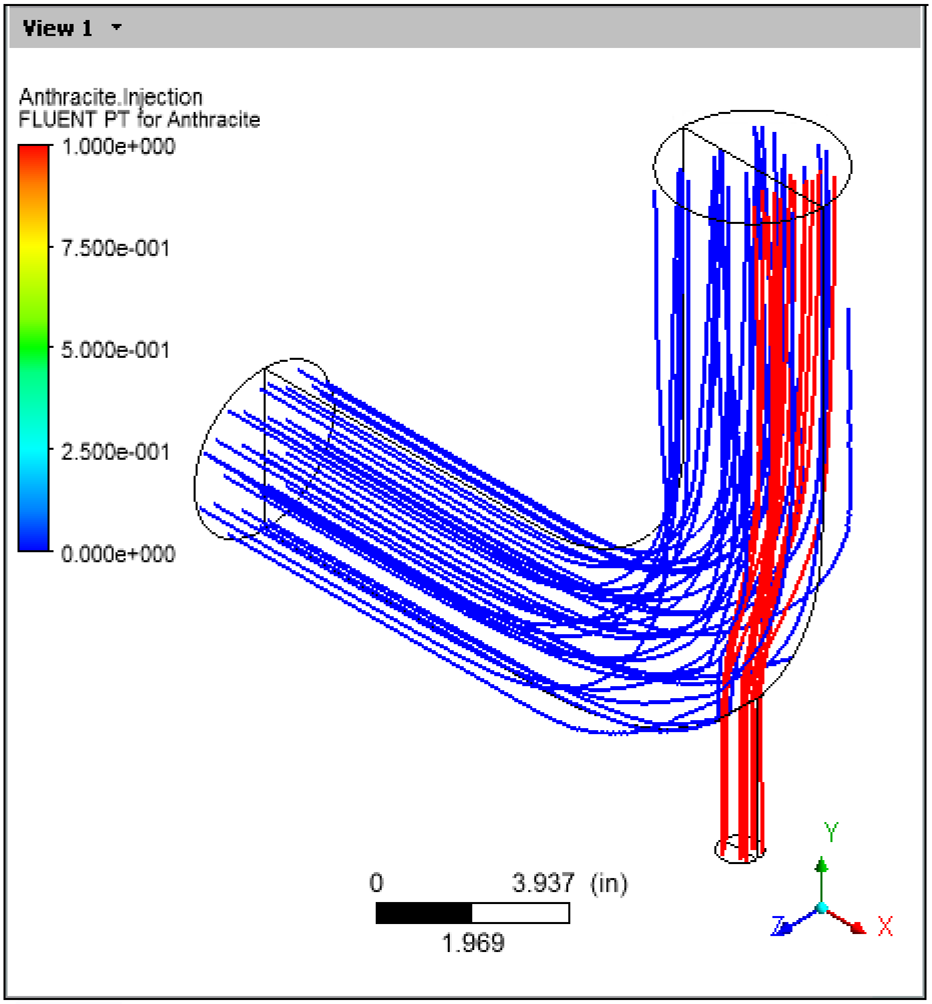
   c. Click **Apply**.

2. Click the **Color** tab.

3. Set **Mode** to `Variable`.

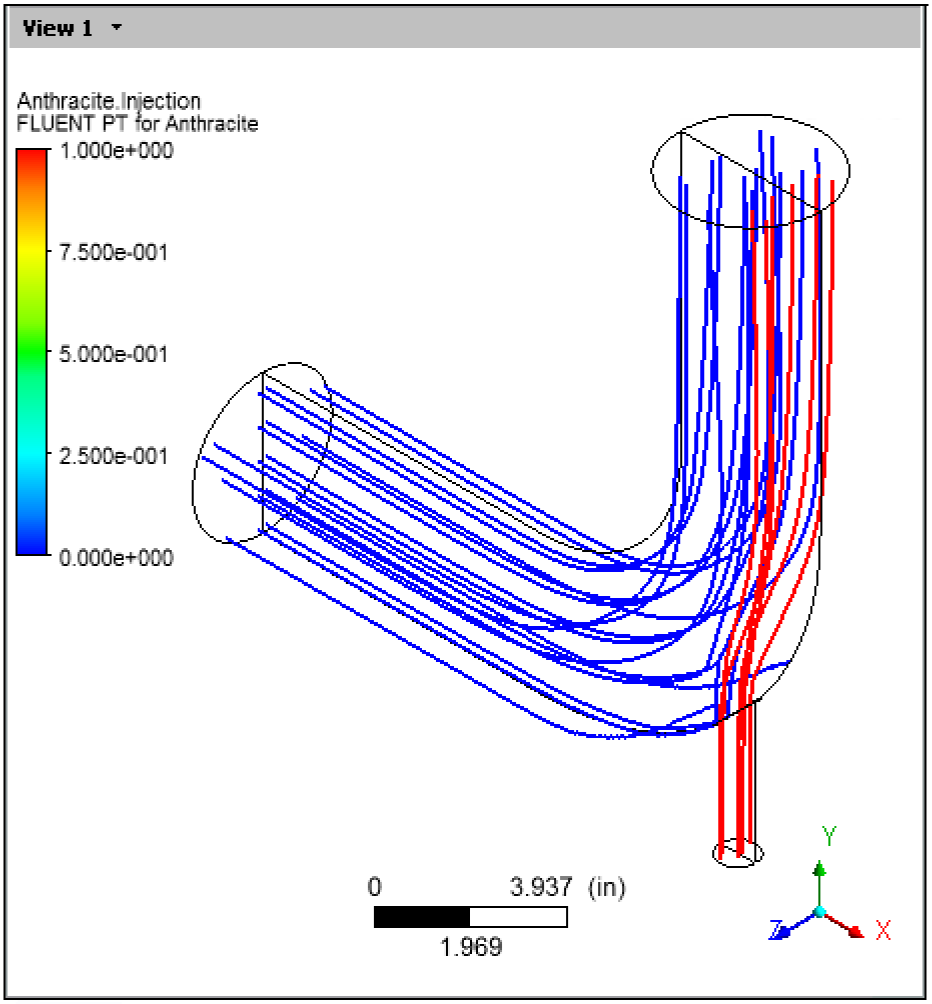
4. Set **Variable** to `Anthracite.Injection`.

5. Click **Apply**.



Show fewer particle tracks:

1. Click the **Geometry** tab.
2. Click the drop-down arrow beside the **Reduction Type** field and select `Reduction Factor`.
3. To display only half of the tracks, set **Reduction** to 2.
4. Click **Apply**.

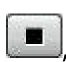


5. Display all tracks again by setting **Reduction** to `1` and clicking **Apply**.

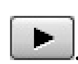
Animate symbols running along the particle track lines:

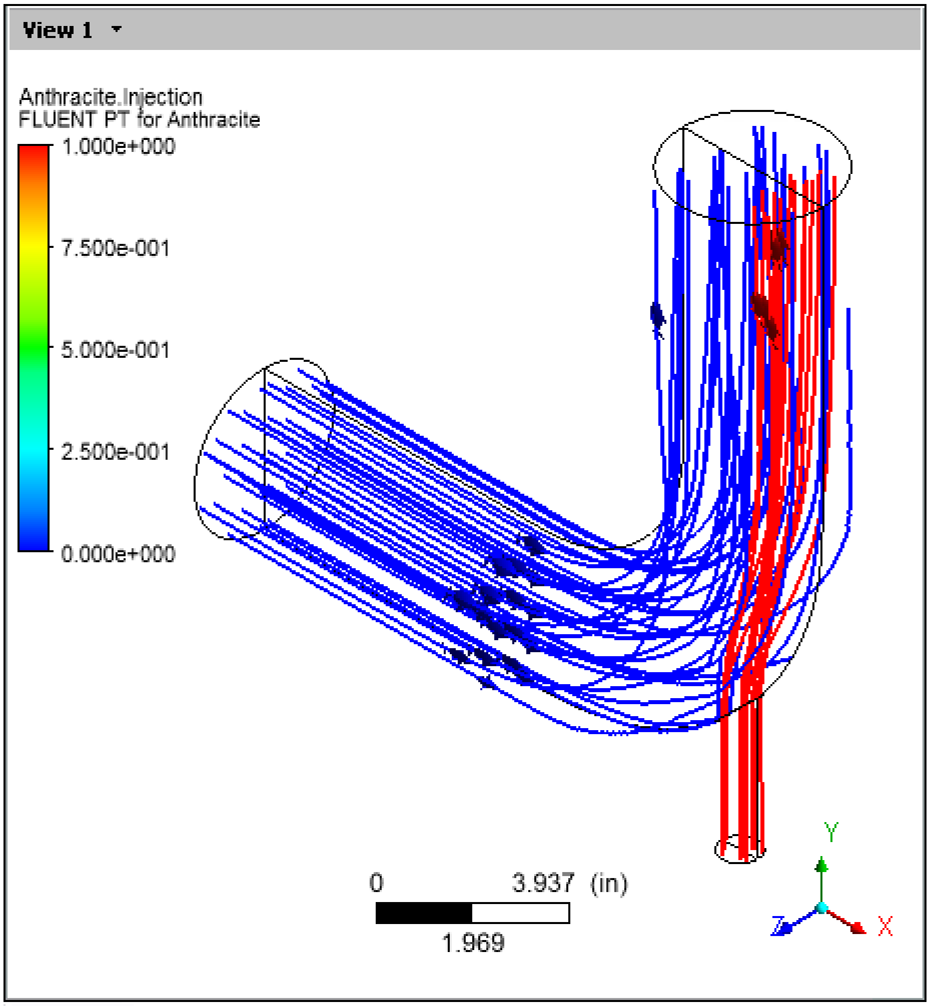
1. Right-click a particle track and select **Animate**.

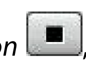
   The animation begins automatically.

2. Click *Stop the animation* , then click **Options**.

3. On the **Options** dialog box, set **Symbol Size** to 2 and **Symbol** to `Fish3D`. Click **OK**.

4. In the **Animations** dialog box, click *Play the animation* .



5. When you have finished viewing the animation, click *Stop the animation* , then close the **Animation** dialog box.

Create a vector plot:

1. Select **Insert** > **Vector**.

2. In the **Insert Vector** dialog box, click **OK** to accept the default name for the vector.

3. On the **Geometry** tab, set:

   - **Locations** to `Fluent PT for Anthracite`

   - **Reduction** to `Reduction Factor`

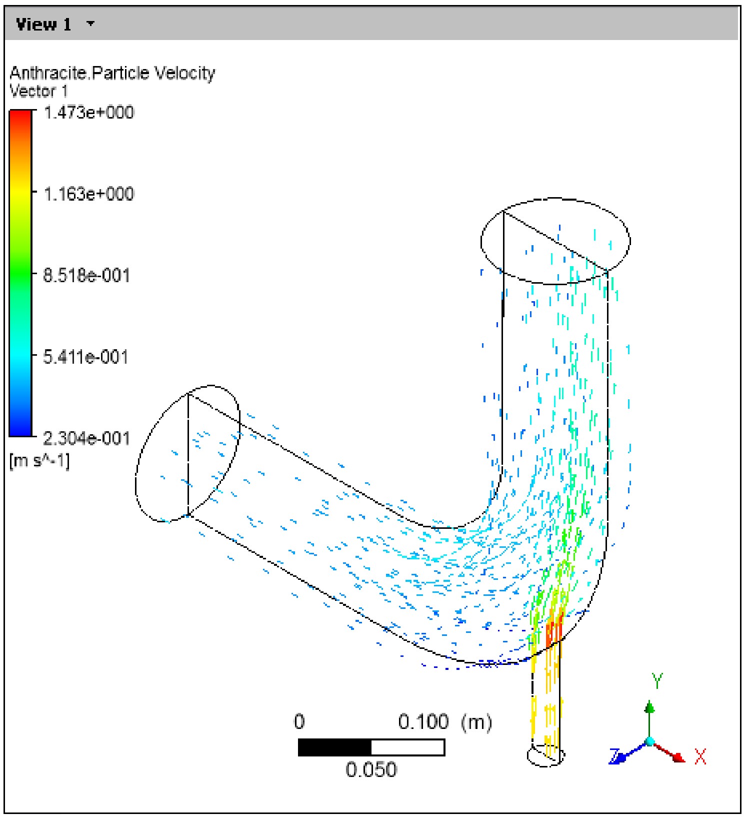
   - **Factor** to `20`

- **Variable** to `Anthracite.Particle Velocity`

Tip:

You need to click the *More Variables* icon to see Anthracite.Particle Velocity.

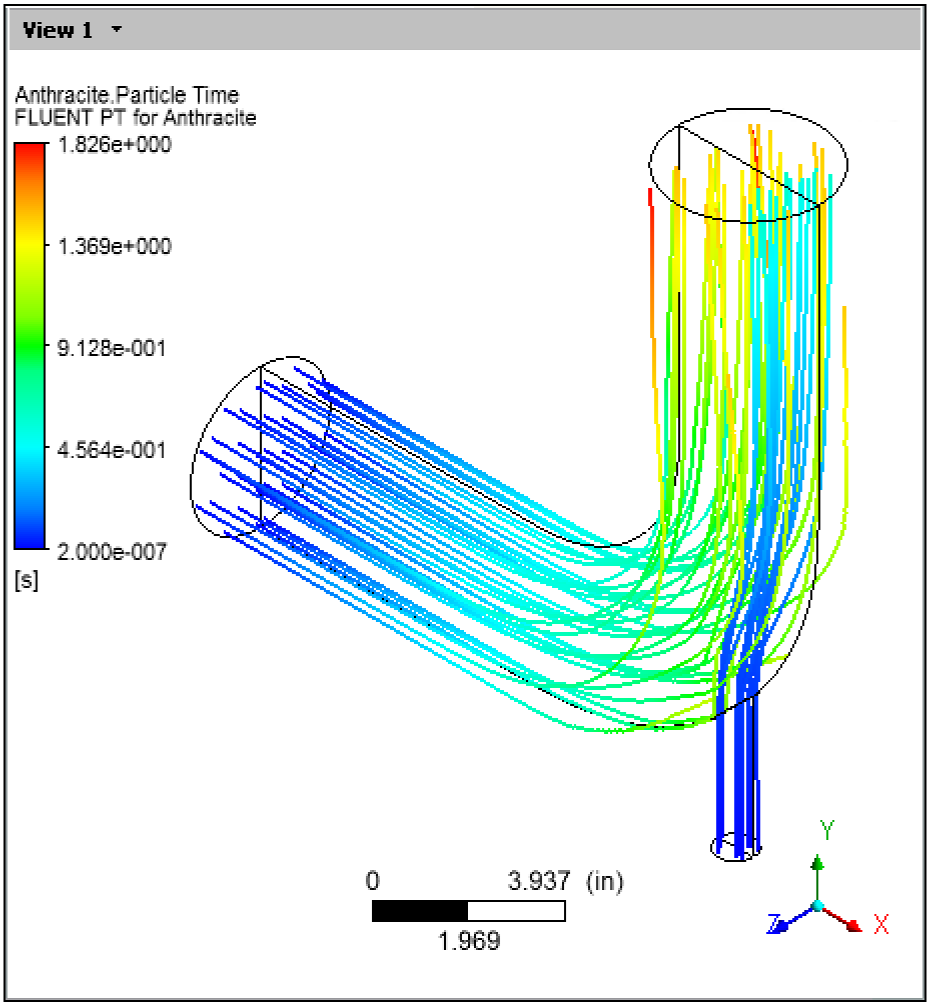
- Click the **Symbol** tab, and set **Symbol Size** to 2.
- Click **Apply**.
- In the Outline tree, clear **User locations and Plots** > **Fluent PT for Anthracite**.



4. After viewing the vector plot, clear **User locations and Plots** > **Vector 2** and select **User locations and Plots** > **Fluent PT for Anthracite** in order to view the particle tracks only.

Color the particle tracks by particle time:

1. Click the **Color** tab.
2. Ensure that **Mode** is set to `Variable`.
3. Set **Variable** to `Anthracite.Particle Time`.
4. Click **Apply**.



Create a chart of particle time vs. particle velocity Y for a single track:

1. On the **Geometry** tab, click the drop-down arrow beside the **Injections** field and select `injection-0`.

2. Click **Apply**.

3. On the **Symbol** tab, select **Show Track Numbers** and click **Apply**.

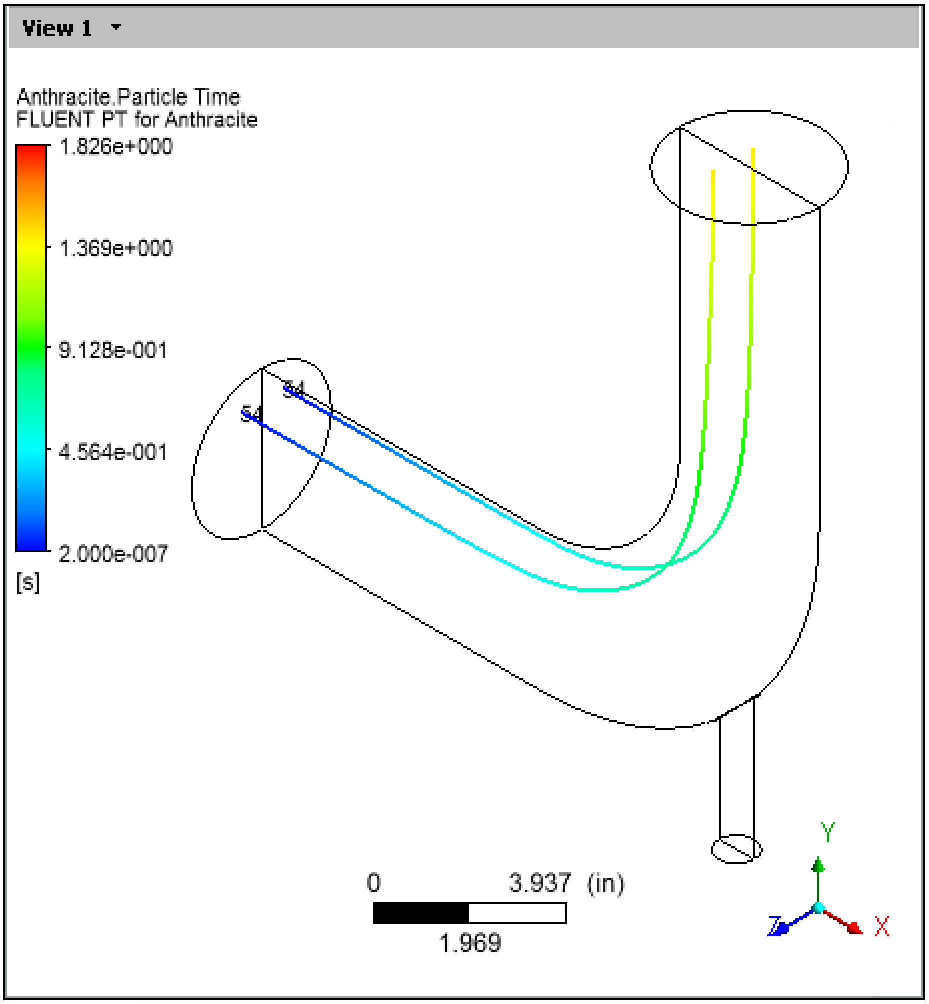
4. On the **Geometry** tab:

   a. Enable the **Filter** option.

   b. Select **Track**.

   c. In the **Track** field, type 54.

   d. Click **Apply**.



5. Create a chart of a particle's velocity over time:

   a. From the menu bar, select **Insert** > **Chart**.

   b. In the **Insert Chart** dialog box, type: `Particle 54` and click **OK**.

      The details view for the chart appears, and the **Chart Viewer** appears.

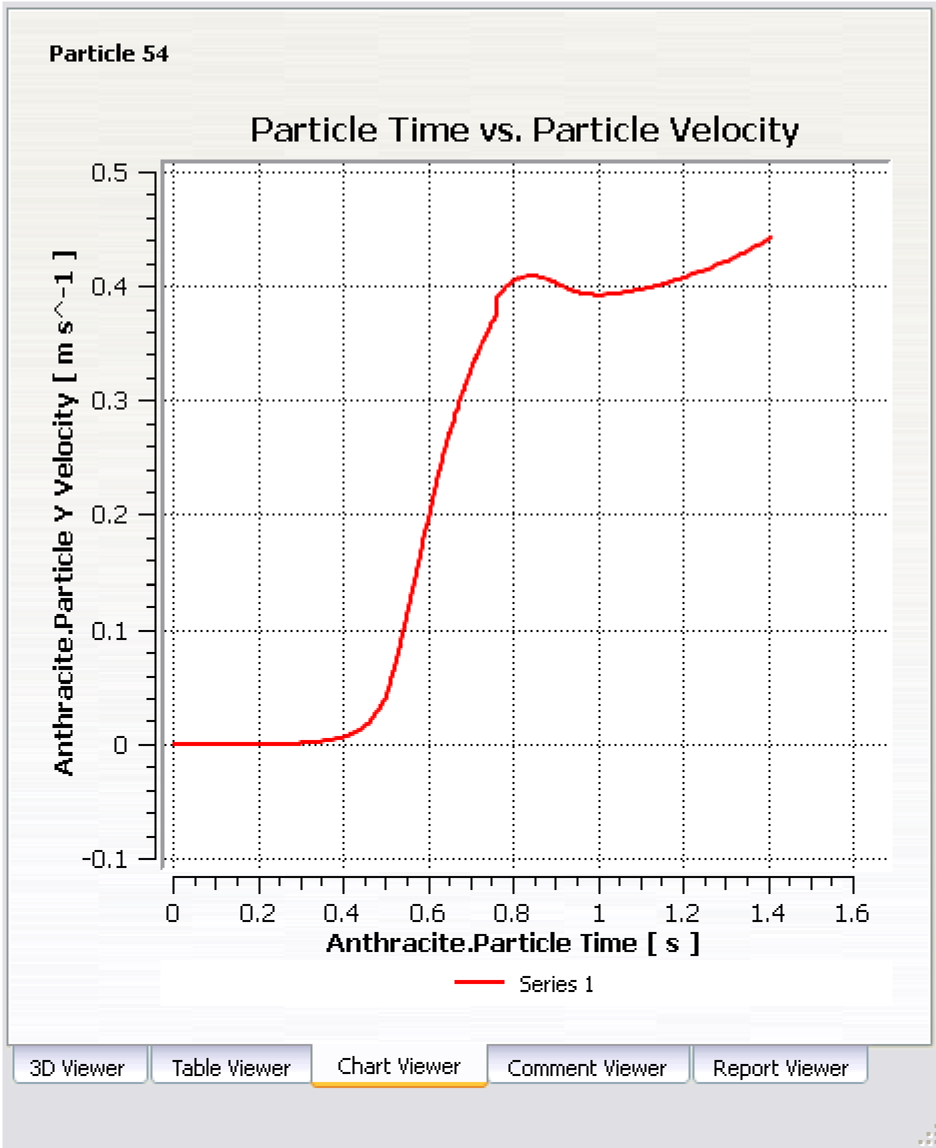
   c. In the **Title** field, type: `Particle Time vs. Particle Velocity`

   d. On the **Data Series** tab, select `Series 1` and set **Location** to `Fluent PT for Anthracite`.

   e. On the **X Axis** tab, set **Variable** to `Anthracite.Particle Time`.

   f. On the **Y Axis** tab, set **Variable** to `Anthracite.Particle Y Velocity`.

   g. Click **Apply**.



Interpolate a field variable onto the track:

1. On the **Y Axis** tab, set **Variable** to `Pressure`.
2. On the **General** tab, change the **Title** to `Particle Time vs. Pressure`.
3. Click **Apply**.

Use the Function Calculator to calculate `lengthAve` of Pressure on the track:

1. From the menu bar, select **Tools** > **Function Calculator**.
2. In the **Function Calculator**, set **Function** to `lengthAve`.
3. Ensure that **Location** is set to `Fluent PT for Anthracite`.
4. Ensure that **Variable** is set to `Pressure`.

5. Enable **Show equivalent expression**.

6. Click **Calculate**.

   The value of `lengthAve(Pressure)@Fluent PT for Anthracite` appears.

2.5. 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 stand-alone or from within Ansys Workbench:

   - From CFD-Post stand-alone:

     1. 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.

     2. A **Warning** dialog box asks if you want to save the animation state. Click **Yes**.

     3. 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:

     1. From the CFD-Post menu bar, select **File** > **Quit**. Ansys Workbench saves the state file automatically.

     2. In the Ansys Workbench **Project Schematic**, double-click 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** tree view, show **Contour 1**.

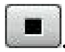
   With the focus in the **3D Viewer**, click *Save Picture*  from the toolbar. In the **Save Picture** dialog box, click **Save**. A PNG file of the current state of the viewer is saved to `<casename>.png` (`elbow1.png`) in your working directory.

3. You can recreate the animation you made previously and save it to a file:

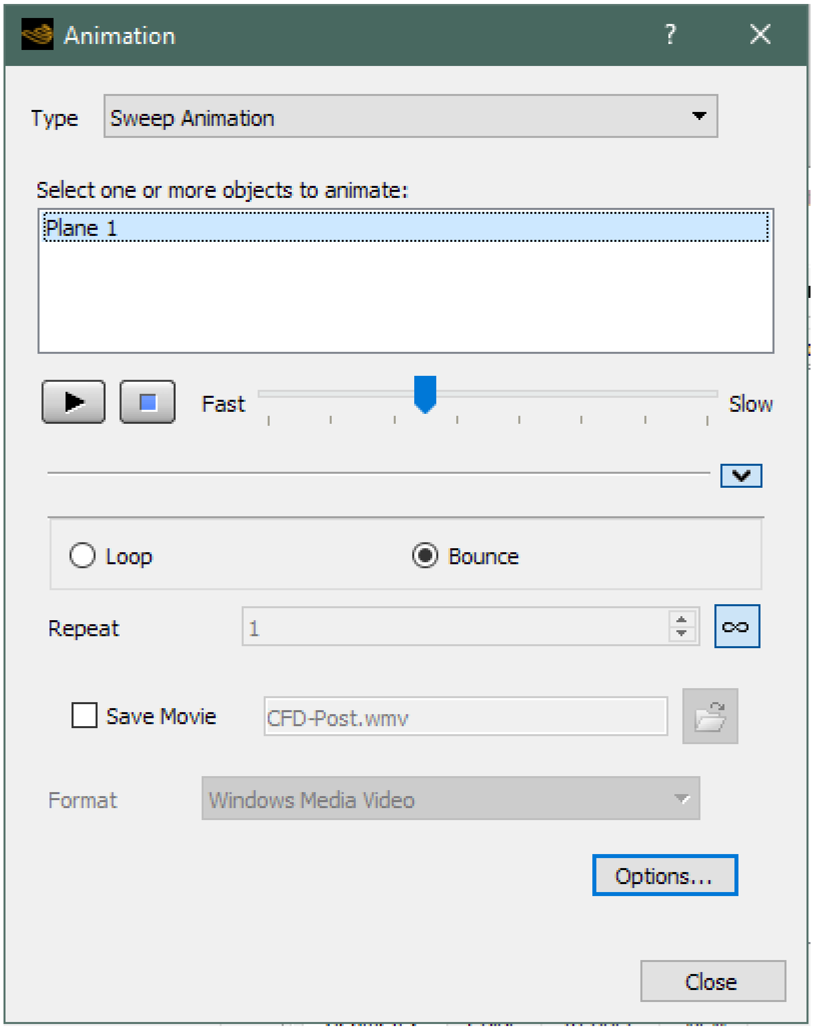
   a. Click the cyan (ISO) sphere in the triad to orient the elbow to display Plane 1.

   b. In the **Outline** tree view, clear **Contour 1** and **Fluent PT for Anthracite**; show **Plane 1**.

   c. Right-click Plane 1 in the **3D Viewer** and select **Animate**. The **Animation** dialog box appears and the plane moves through the entire domain.

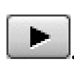
d. Click the stop icon .

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 selected and by default is set to one.

g. Select **Save Movie** to save the animation to the indicated file.

h. Click *Play the animation* .

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 box.

5. Close CFD-Post: from the toolbar select **File** > **Quit**. If prompted, you may save your changes.

2.6. 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.

Chapter 3: Turbo Postprocessing

This tutorial demonstrates the turbomachinery postprocessing capabilities of CFD-Post.

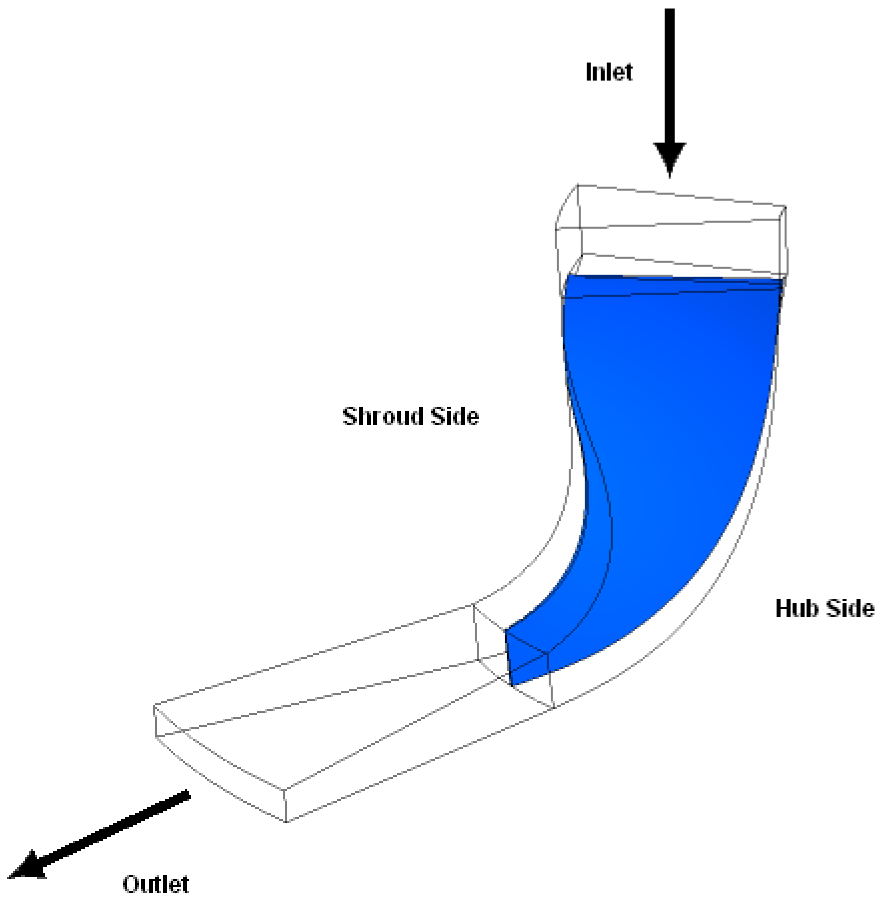
In this example, you will read Fluent case and data files (without doing any calculations) and perform a number of turbomachinery-specific postprocessing operations.

This tutorial demonstrates:

3.1. Problem Description

This tutorial considers the problem of a centrifugal compressor shown schematically in Figure 3.1: Problem Specification (p. 48). The model is composed of a single 3D sector of the compressor to take advantage of the circumferential periodicity in the problem. The flow of air through the compressor is simulated and the postprocessing capabilities of CFD-Post are used to display realistic, full 360° images of the solution obtained.

Figure 3.1: Problem Specification



If this is the first tutorial you are working with, it is important to review Introduction to the Tutorials (p. 7) before beginning.

3.2. Preparing the Working Directory

1. 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.

2. Download the `turbo.zip` file here.

3. Unzip `turbo.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `turbo.cas.gz`
   - `turbo.cdat.gz`

3.3. Setting the Working Directory and Starting 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 run CFD-Post stand-alone, from the Ansys CFX Launcher, or from Ansys Workbench:

- To run CFD-Post stand-alone
  - On Windows:
    1. From the **Start** menu, right-click **All Programs > Ansys 2021 R2 > Fluid Dynamics > CFD-Post 2021 R2** and select **Properties**.
    2. Type the path to your working directory in the **Start in** field and click **OK**.
    3. Click **All Programs > Ansys 2021 R2 > Fluid Dynamics > CFD-Post 2021 R2** to launch CFD-Post.
  - On Linux, enter `cfdpost` in a terminal window that has its path set up to run CFD-Post. The path will be something similar to `/usr/ansys_inc/v212/CFD-Post/bin`.
- To run Ansys CFX Launcher
  1. Start the launcher.

     You can run the launcher in any of the following ways:
     - On Windows:
       - → From the **Start** menu, select **All Programs > Ansys 2021 R2 > Fluid Dynamics > CFX 2021 R2**.
       - → In a Command Prompt that has its path set up correctly to run CFX, enter `cfx5launch`. If the path is not set up correctly, you will need to type the full pathname of the `cfx` command, which will be something similar to `C:\Program Files\ANSYS Inc\v212\CFX\bin`.
     - On Linux, enter `cfx5launch` in a terminal window that has its path set up to run CFX. The path will be something similar to `/usr/ansys_inc/v212/CFX/bin`.
  2. Set the working directory.
  3. Click the **CFD-Post 2021 R2** 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 the A2 **Results** cell and select **Edit**. **CFD-Post** opens.

3.4. Display the Solution in CFD-Post

In the steps that follow, you will explore the solution using CFD-Post.

3.4.1. Prepare the Case and Set the Viewer Options

1. Load the CDAT file (`turbo.cdat.gz`) from the menu bar by selecting **File** > **Load Results**. In the **Load Results File** dialog box, select `turbo.cdat.gz` and click **Open**.

2. If you see a message that discusses Global Variables Ranges, it can be ignored. Click **OK**.

   The turbo blade appears in the viewer in an isometric orientation. The **Wireframe** appears in the **3D Viewer** and there is a check mark beside **Wireframe** in the **Outline** workspace; the check mark indicates that the wireframe is visible in the **3D Viewer**.

3. Set CFD-Post to display 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.

   a. Right-click the viewer and select **Viewer Options**.

   b. In the **Options** dialog box, select **Common** > **Units**.

   c. Set **System** to SI 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.

4. Double-click **Wireframe** in the **Outline** workspace to see the details view. To display the mesh, set **Edge Angle** to 0 degrees and click **Apply**. The edge angle determines how much of the surface mesh is visible. If the angle between two adjacent faces is greater than the edge angle,

then that edge is drawn. If the edge angle is set to 0°, the entire surface mesh is drawn. If the edge angle is large, then only the most significant corner edges of the geometry are drawn.

Tip:

With the mouse focus on CFD-Post and the mouse over the **Details of Wireframe** editor, press **F1** to see help about the Wireframe object.

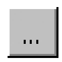
On the **Wireframe** details view, click **Defaults** and **Apply** to restore the original settings.

5. Optionally, set the viewer background to white:

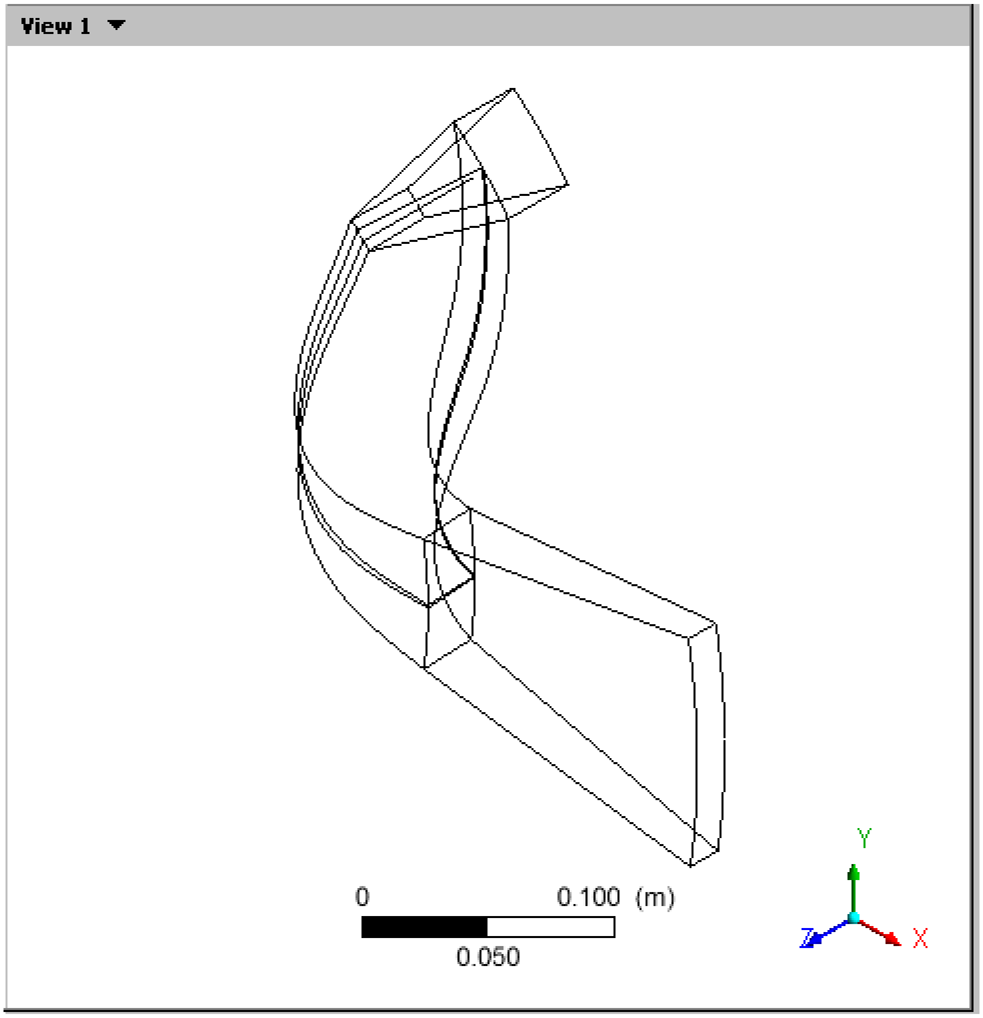
   a. Right-click the viewer and select **Viewer Options**.

   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  icon 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.



3.4.2. Initialize the Turbomachinery Components

Before you can start using the Turbo workspace features, you need to initialize the components of the loaded case (such as hub, blade, periodics, and so on). Among other things, initialization generates span, a (axial), r (radial), and Theta coordinates for each component.

You need to initialize Fluent case and data files manually (automatic initialization is available only for CFX files produced by the Turbo wizard in CFX-Pre). To initialize the components:

1. Click the **Turbo** tab in the upper-left pane of the CFD-Post window. The **Turbo** workspace appears as does a **Turbo initialization** dialog box that offers to auto-initialize all turbo components. Click **No**.

2. In the **Turbo** workspace under **Initialization**, double-click `fluid (fluid)`. The details view of Fluid appears.

3. On the **Definition** tab, the regions of the geometry are listed under **Turbo Regions**. However, not all regions are listed; correct this as follows:

a. Click *Location editor* [...] to the right of the **Hub** region.

b. Hold down the **Ctrl** key and in the **Location Selector** select `wall diffuser hub`, `wall hub`, and `wall inlet hub`.

c. Click **OK**.

   The **Hub** field now lists all three hub locations.

d. Repeat the previous steps for the **Shroud** region, selecting `wall diffuser shroud`, `wall inlet shroud`, and `wall shroud`.

e. Repeat the previous steps for the **Blade** region, selecting *only* `wall blade`.

f. Repeat the previous steps for the **Inlet** region, selecting *only* `inlet`.

g. Repeat the previous steps for the **Outlet** region, selecting *only* `outlet`.

h. Repeat the steps for the **Periodic 1** region, selecting `periodic.33`, `periodic.34`, and `periodic.35`.

   You do not need to initialize the `periodic.*shadow` regions; the `periodic.*` nodes provide the information that the turbo reports require.

4. Click the **Instancing** tab.

   a. Ensure that **Number of Graphical Instances** is set to `1`.

   b. Ensure that **Axis Definition** is set to `Custom`, that **Method** is set to `Principal Axis`, and that **Axis** is set to `Z`.

   c. Set **Instance Definition** to `Custom`.

   d. Select **Full Circle**.

5. Click **Initialize**. This generates variables that you will use later to create reports.

Tip:

If the turbo topology is not correctly defined, an error message is generated and the initialization does not occur. To resolve such an error:

a. Ensure that the rotational axis is correct.

b. Ensure that the turbo regions are correctly set, *and that they enclose the passage without any gaps*.

---

6. Double-click **Initialization** at the top of the **Turbo** tree view. The **Initialization** editor appears.

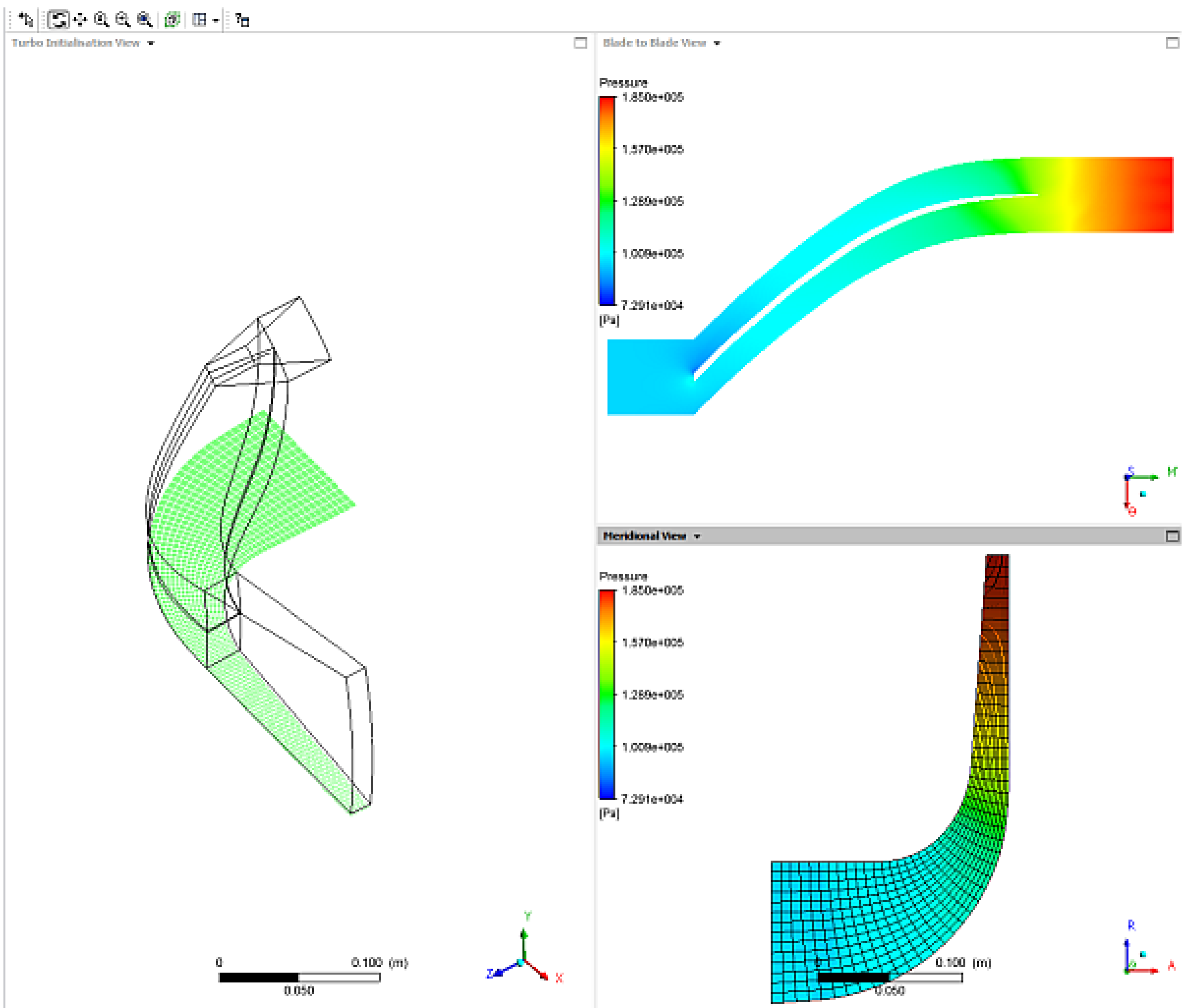
7. Click the **Calculate Velocity Components** button. This generates velocity variables that you will also use in your reports.

The initialization process has created a variety of plots automatically; you will access these from the **Turbo** tab in the sections that follow.

3.4.3. Compare the Blade-to-Blade, Meridional, and 3D Views

To compare the Blade-to-Blade, Meridional, and 3D Views:

- In the **Turbo** workspace, select the **Three Views** option at the bottom of the **Initialization** editor. In the **3D Viewer** you can see the **Turbo Initialization View**, the **Blade to Blade View**, and the **Meridional View**.



The CFD-Post **Blade to Blade View** is equivalent to the Fluent "2D contour on a spanwise surface". By default, the variable shown is `Pressure`. To change this to velocity and to make the image more like the default Fluent equivalent:

1. In the **Blade to Blade View**, right-click the colored area shown in the viewport and select **Edit**.

2. In the details view for the **Blade-to-Blade Plot**, change the **Plot Type** from `Color` to `Contour` (this changes the continuous gradation found in `Color` to the discrete color bands found in `Contour`).

3. Change **Variable** to `Velocity`.

4. Change the **# of Contours** to `21`.

5. Click **Apply**.

The CFD-Post **Meridional View** is equivalent to the Fluent "contour averaged in the circumferential direction". To make the image more like the default Fluent equivalent:

1. In the **Meridional View**, right-click the colored area shown in the viewport and select **Edit**.

2. In the details view for the **Meridional Plot**, change the **Plot Type** from `Color` to `Contour`.

3. Change the **# of Contours** to `21`.

4. Click **Apply**.

3.4.4. Display Pressure on Meridional Isosurfaces

In this example you will define six meridional isosurfaces colored by pressure.

1. Return to the original orientation of the case:

   a. In the **Tree** view, double-click **Plots** and select **Single View**.

   b. Double-click **3D View**.

2. From the menu bar select **Insert** > **Location** > **Isosurface** and accept the default name.

3. Set the following values on the details view for the isosurface:

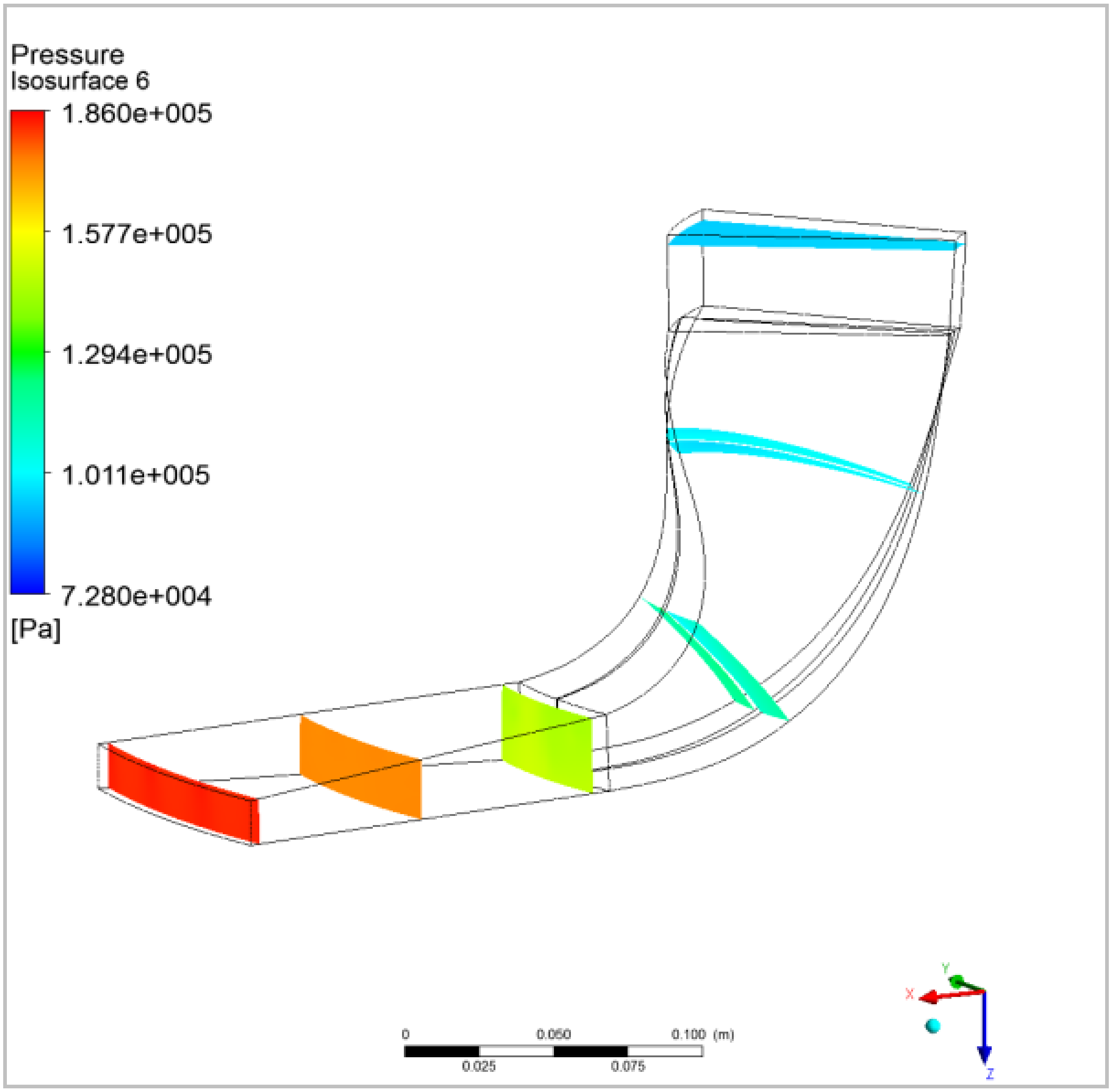
Tab	Field	Value
Geometry	Domains	fluid
	Variable	Linear BA Streamwise Location [a]
	Value	.01
Color	Mode	Variable
	Variable	Pressure
	Range	User Specified
	Min	72800 [Pa]
	Max	186000 [Pa]
Render	Lighting	(Cleared)

a. Click the *Variable Editor* ... to access this variable.

4. Click **Apply** to define the isosurface.

5. Right-click **Isosurface 1** in the **Tree** view and select **Duplicate**, then change **Geometry** > **Value** to `.2` and click **Apply**.

6. Create other duplicates for geometry values `.4`, `.6`, `.8`, and `.99`.



Note:

You can set locator variables other than Linear BA (Blade Aligned) Streamwise Location. For example, edit Isosurface 5 and change `Linear BA Streamwise Location` to `M Length Normalized` to see how the isosurface changes.

3.4.5. Display a 360-Degree View

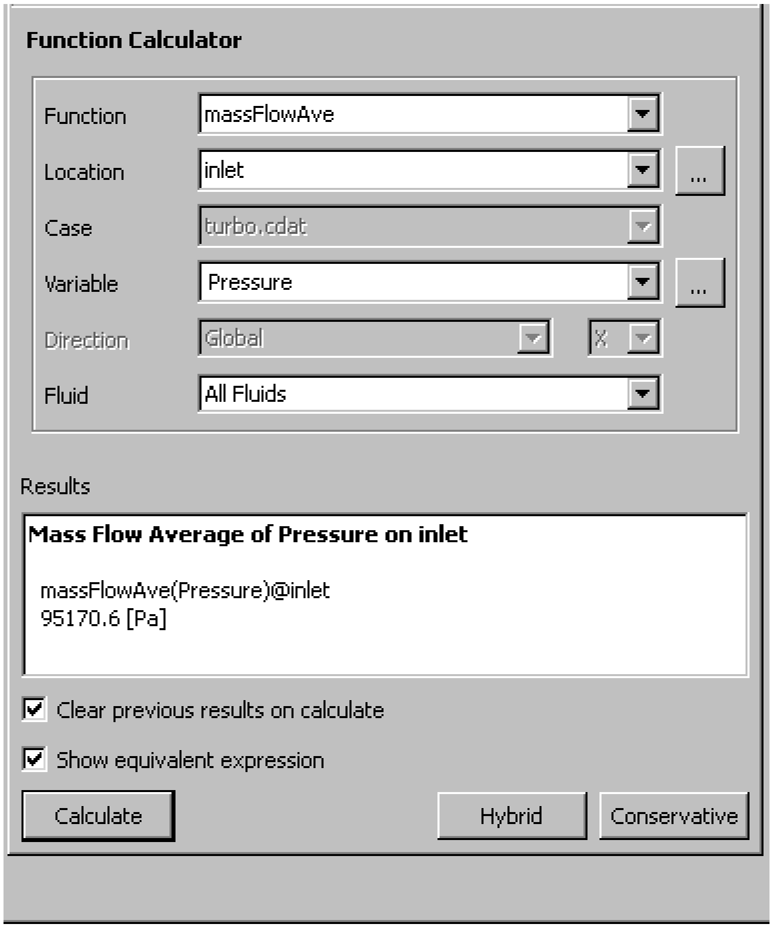
To display a 360° view of the turbomachinery:

1. In the **Outline** tree view, right-click any object under `User Locations and Plots` that has a visibility check box, then select **Hide All**.
2. Under **User Locations and Plots**, ensure that the check box beside `Wireframe` is selected.
3. Under **Cases** > **turbo**, double-click **fluid** to edit that domain.
4. On the **Instancing** tab:
    a. Set **Number of Graphical Instances** to `20`.
    b. Ensure that **Instance Definition** is set to `Custom` and that **Full Circle** is selected.
    c. Ensure that **Axis Definition** is set to `Custom`, that **Method** is set to `Principal Axis`, and that **Axis** is set to `Z`.
5. Click **Apply**.
6. If necessary, click the *Fit View*  icon so that you can see the whole case.

3.4.6. Calculate and Display Values of Variables

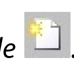
You can calculate the values of variables at locations in the case and display these results in a table. First, use the **Function Calculator** to see how to create a function.

1. From the menu bar, select **Tools** > **Function Calculator**. The **Calculators** tab appears with the **Function Calculator** displayed.
2. Use the **Function Calculator** to calculate the mass flow average of pressure at the inlet as follow:
    a. Use the **Function** drop-down arrow to select `massFlowAve`.
    b. Use the **Location** drop-down arrow to select `inlet`.
    c. Use the **Variable** drop-down arrow to select `Pressure`.
    d. At the bottom of the **Function Calculator** select **Show equivalent expression**.
    e. Click **Calculate** and the expression and results appear:



The **Function Calculator** not only makes it easy to create and calculate a function, it also enables you to see the syntax for functions, which you will use in the subsequent steps.

3. To display functions like this in a table, click the **Table Viewer** tab (at the bottom of the viewer area). The **Table Viewer** appears.

4. In the toolbar at the top of the **Table Viewer**, click *New Table* . The **New Table** dialog box appears. Type in `Inlet and Outlet Values` and click **OK**.

5. Type the following text to make cell headings:

   - In cell B1: Inlet
   - In cell C1: Outlet
   - In cell A2: Mass Flow
   - In cell A3: Average Pressure

6. Now, create functions:

   a. Click in cell B2, then in the **Table Viewer** toolbar, select **Function** > **CFD-Post** > **massFlow**. The definition `=massFlow()@` appears.

b. With the text cursor after the @ symbol, click **Location** > **inlet**.

c. Press **Enter**; the value of the mass flow at the inlet appears.

d. Repeat the above steps for cell C2, but use **Location** > **outlet**.

e. For cell B3, select **Function** > **CFD-Post** > **massFlowAve**. With the text cursor between the parentheses, select **Variable** > **Pressure**. With the text cursor after the @ symbol, click **Location** > **inlet**. Press **Enter**; the value of the mass flow average of pressure at the inlet appears.

f. Repeat the previous step for cell C3, but use **Location** > **outlet**.

Your table should be similar to this:

Inlet and Outlet Values

D6 =

	A	B	C
1		Inlet	Outlet
2	Mass Flow	2.658e-01 [kg s^-1]	-2.606e-01 [kg s^-1]
3	Average Pressure	9.517e+04 [Pa]	1.830e+05 [Pa]

7. Format the cells to make the table easier to read.

a. Click in cell A1 and, while holding down **Shift**, click in cell C1. Now the operations you perform will apply to A1 through C1.

b. Click  to make the heading font bold, then click  to center the heading text. Click  to apply a background color to those cells.

c. Click in cell A2 and, while holding down **Shift**, click in cell A3. Click  to make the row description bold, then click  to right-align the text.

d. Manually resize the cells.

Your table should be similar to this:

Inlet and Outlet Values

D6 =

	A	B	C
1		**Inlet**	**Outlet**
2	**Mass Flow**	2.658e-01 [kg s^-1]	-2.606e-01 [kg s^-1]
3	**Average Pressure**	9.517e+04 [Pa]	1.830e+05 [Pa]

8. Click the **Report Viewer** tab and then click **Refresh** in the **Report Viewer** toolbar. The table data appears at the bottom of the report.

3.4.7. Display the Inlet to Outlet Chart

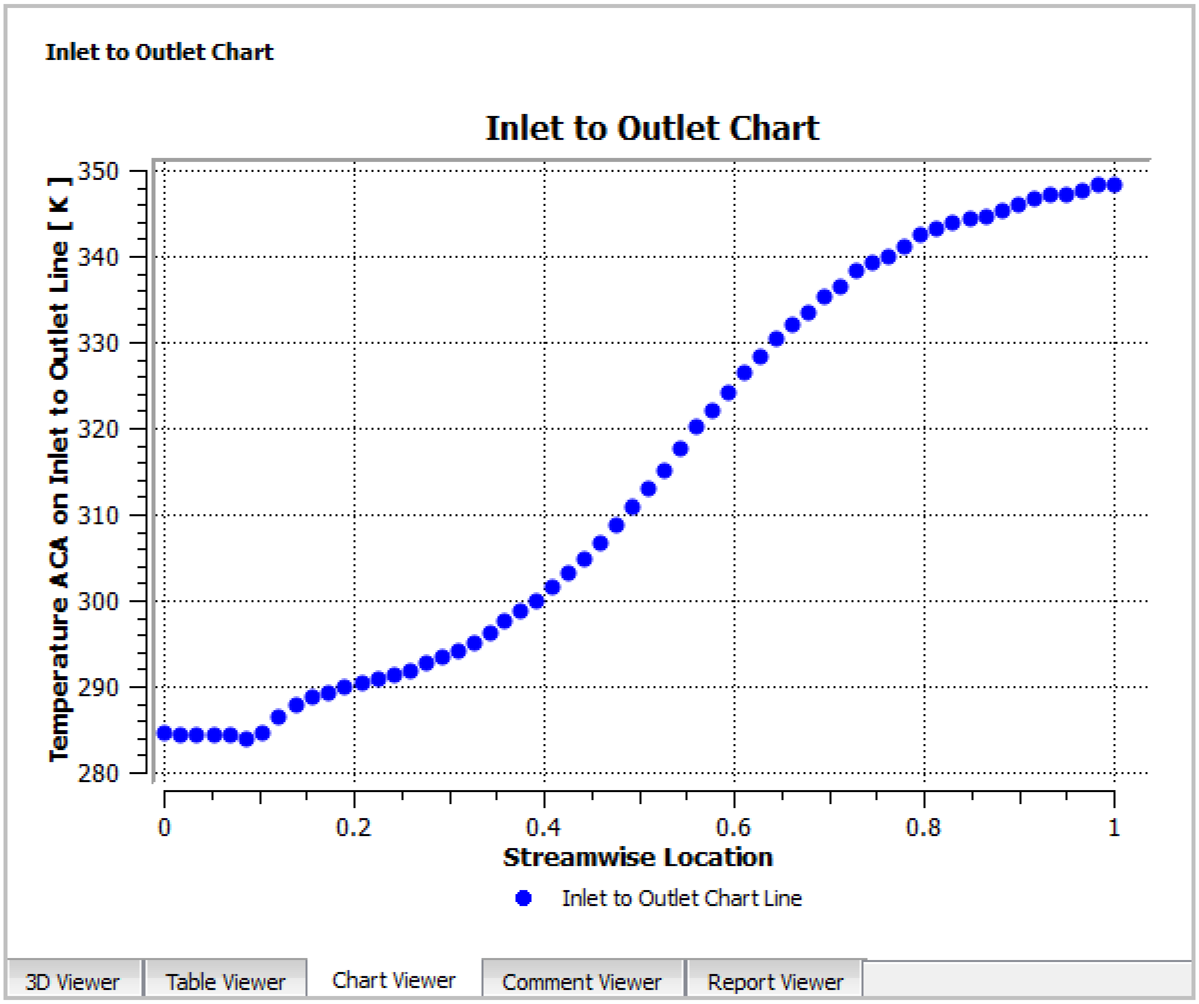
In CFD-Post, displaying the Inlet to Outlet chart is equivalent to displaying averaged XY plots in Fluent. To display the **Inlet to Outlet** chart:

1. In the **Turbo** workspace's **Turbo Charts** area, double-click **Inlet to Outlet**.
2. Now, change the chart to compare temperature to streamwise location (the latter being called "meridional location" in Fluent) and make the chart look more like the Fluent default:

   a. Set the following:

Setting	Value
Domains	fluid
Samples/Comp	60
Y Axis > **Variable**	 Temperature

   b. Click **Apply**. The chart appears:



3. Click the **Report Viewer** tab at the bottom of the viewer area.

4. In the **Report Viewer** toolbar, click the **Refresh** button. The **Inlet to Outlet Chart** appears in the **User Data** section of the report.

Tip:

You can also explore the other **Turbo Charts**:

- **Blade Loading**
- **Circumferential**
- **Hub to Shroud**

3.4.8. Generate and View a Turbo Report

Turbo reports give performance results, component data summary tables, meanline 1D charts, stage plots, and spanwise loading charts for the blade.

Note:

The Turbo report is generated from the values set when you initialized the case, so if there were any changes required to those values, you would make them now and run the initialization procedure again. For this tutorial, that will not be necessary.

To generate a Turbo report:

1. Create a new variable that the report expects (which would be available with CFX results files for rotating machinery applications, but which is not available from Fluent case and data files).

    a. From the toolbar, click *Variable*. The **Insert Variable** dialog box appears.

    b. In the **Name** field, type `Rotation Velocity` and click **OK**.

        The details view for **Rotation Velocity** appears.

    c. In the **Expression** field, type `Radius * abs(omega) / 1 [rad]` and click **Apply**. This expression calculates the angular speed (in units of length per unit time) as a product of the local radius and the rotational speed.

2. In the **3D Viewer** toolbar, click *Fit View*. This ensures that the graphics will not be truncated in the report you are about to generate.

3. In the **Outline** tree view, right-click **Report** and select **Report Templates**. The **Report Templates** dialog box appears.

4. Select an appropriate report template; in this case, select `Centrifugal Compressor Report`.

5. Click **Load**. The **Report Templates** dialog box disappears and you can watch the report's progress in the status bar in the bottom-right corner of CFD-Post.

    Note:

    A dialog box appears that warns that hybrid values do not exist and that conservative values will be used. This is expected behavior when using data loaded from Fluent. An error about "Mach Number in Stn Frame" is also mentioned; this prevents a line in the report from appearing. Click **OK**.

    When the report has been generated, there are new entries in the **Outline** tree view under **Report**.

6. Under **User Locations and Plots**, double-click `fluid Instance Transform`. This is an instance transform generated by the report to facilitate showing two blades in the figures that show blade-to-blade views.

7. Ensure that **Number of Passages** is set to `20` and click **Apply**.

8. Click the **Expressions** tab. Double-click the expression **fluid Components in 360** to edit it. Change the definition to `20` and click **Apply**.

To view the Turbo report:

1. In the **Report Viewer**, click **Refresh**. The turbo report appears.

2. Optionally, you can remove pieces from the report by clearing the appropriate check boxes in the **Report** section of the **Outline** tree. When you have made your selections, return to the **Report Viewer** and click **Refresh** (in the **Report Viewer** toolbar). The edited version of the turbo report appears.

3. To produce an HTML version of the report that you can share with others, click **Publish** (at the top of the viewer area). The report is saved in a filename of your choosing in your working directory (by default).

Chapter 4: Quantitative Postprocessing

This tutorial demonstrates the quantitative postprocessing capabilities of CFD-Post using a 3D model of a circuit board with a heat-generating electronic chip mounted on it. The flow over the chip is laminar and involves conjugate heat transfer.

The heat transfer involves conduction in the chip and conduction and convection in the surrounding fluid. The physics of conjugate heat transfer such as this is common in many engineering applications, not just the design and cooling of electronic components.

In this tutorial, you will read the case and data files and perform a number of postprocessing exercises.

This tutorial demonstrates how to do the following:

Note:

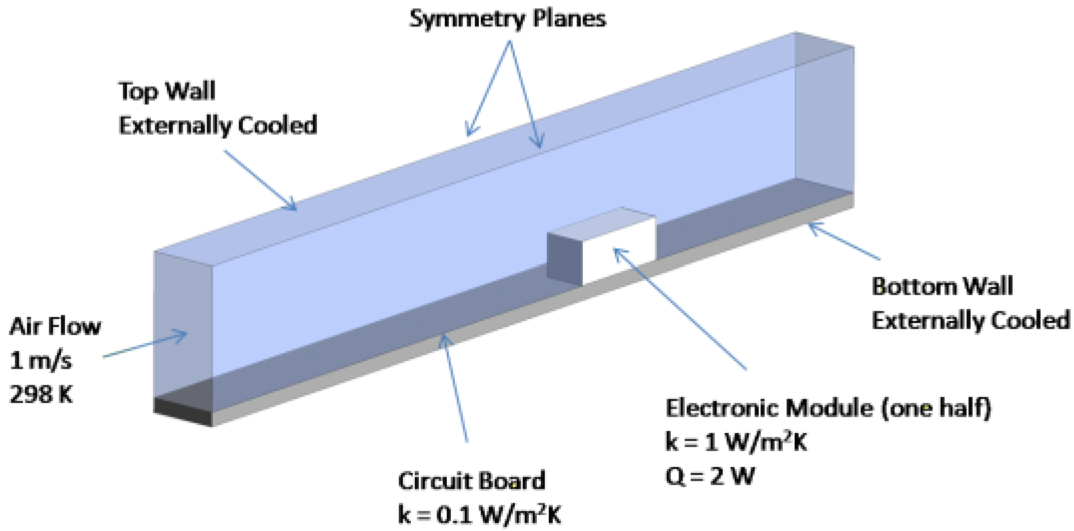
These tutorials are prepared on a Windows system. The screen shots in the tutorials may be slightly different than the appearance on your system, depending on the operating system or graphics card.

4.1. Problem Description

The problem is shown schematically in Figure 4.1: Problem Specification (p. 66). The configuration consists of a series of side-by-side electronic chips, or modules, mounted on a circuit board. Air flow, confined between the circuit board and an upper wall, cools the modules. To take advantage of the symmetry present in the problem, the model will extend from the middle of one module to the plane of symmetry between it and the next module.

As shown in the figure, each half-module is assumed to generate 2.0 Watts and to have a bulk conductivity of 1.0 W/m^2K. The circuit board conductivity is assumed to be one order of magnitude lower: 0.1 W/m^2K. The air flow enters the system at 298 K with a velocity of 1 m/s. The Reynolds number of the flow, based on the module height, is about 600. The flow is therefore treated as laminar.

Figure 4.1: Problem Specification



If this is the first tutorial you are working with, it is important to review Introduction to the Tutorials (p. 7) before beginning.

4.2. Preparing the Working Directory

1. 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.

2. Download the `quantitative.zip` file here.

3. Unzip `quantitative.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `chip.cas.gz`
   - `chip.cdat.gz`

4.3. Setting the Working Directory and Starting 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 run CFD-Post stand-alone, from the Ansys CFX Launcher, or from Ansys Workbench:

- To run CFD-Post stand-alone
  - On Windows:
    1. From the **Start** menu, right-click **All Programs > Ansys 2021 R2 > Fluid Dynamics > CFD-Post 2021 R2** and select **Properties**.

2. Type the path to your working directory in the **Start in** field and click **OK**.

3. Click **All Programs > Ansys 2021 R2 > Fluid Dynamics > CFD-Post 2021 R2** to launch CFD-Post.

– On Linux, enter `cfdpost` in a terminal window that has its path set up to run CFD-Post. The path will be something similar to `/usr/ansys_inc/v212/CFD-Post/bin`.

- To run Ansys CFX Launcher

1. Start the launcher.

   You can run the launcher in any of the following ways:

   – On Windows:

     → From the **Start** menu, select **All Programs > Ansys 2021 R2 > Fluid Dynamics > CFX 2021 R2**.

     → In a Command Prompt that has its path set up correctly to run CFX, enter `cfx5launch`. If the path is not set up correctly, you will need to type the full pathname of the `cfx` command, which will be something similar to `C:\Program Files\ANSYS Inc\v212\CFX\bin`.

   – On Linux, enter `cfx5launch` in a terminal window that has its path set up to run CFX .The path will be something similar to `/usr/ansys_inc/v212/CFX/bin`.

2. Set the working directory.

3. Click the **CFD-Post 2021 R2** 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 the A2 **Results** cell and select **Edit**. **CFD-Post** opens.

4.4. Display the Solution in CFD-Post

In the steps that follow, you will explore the solution using CFD-Post.

4.4.1. Prepare the Case and CFD-Post

Before you perform various quantitative analyses of the case, prepare the case and CFD-Post:

1. Start CFD-Post now and load the CDAT file (`chip.cdat.gz`) from the menu bar by selecting **File** > **Load Results**. In the **Load Results File** dialog box, select `chip.cdat.gz` and click **Open**.

2. Set CFD-Post to display 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.

   a. Right-click the viewer and select **Viewer Options**.

   b. In the **Options** dialog box, select **Common** > **Units**.

   c. Set **System** to `SI` and click **OK**.

   Note:

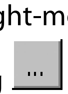
   The display units you set are saved between sessions and projects. This means that you can load results files from diverse sources and always see familiar units displayed.

3. Optionally, set the viewer background to white:

   a. Right-click 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  icon 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.

4.4.2. 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:

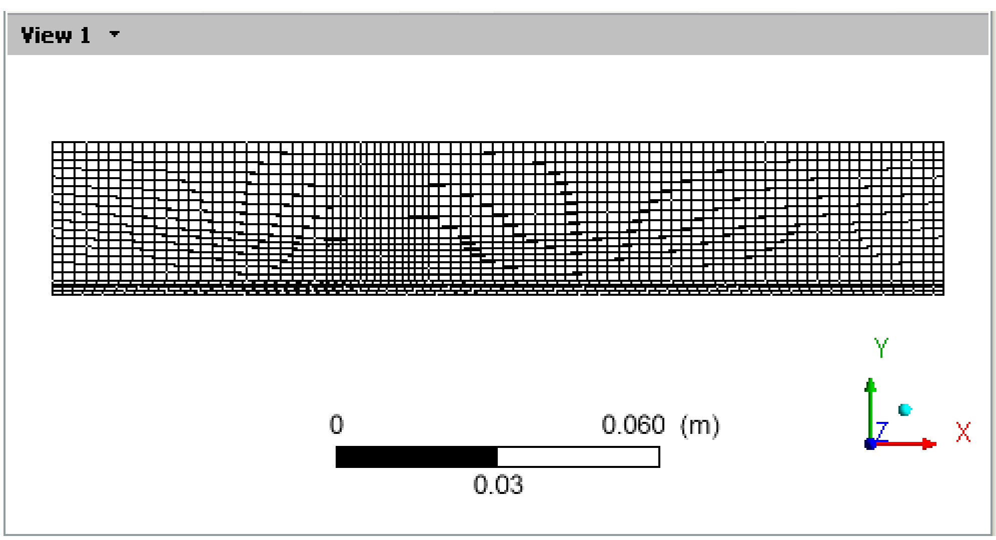
1. Right-click a line of the wireframe in the **3D Viewer** and select **Show surface mesh** to display the mesh.

2. Click the "Z" axis of triad in the viewer to get a side view of the object.

Note:

The **3D Viewer** toolbar has to be in viewing mode for you to be able to select the triad elements.

Figure 4.2: The Hexahedral Grid for the Simulation



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

Tip:

Click 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 chip (wall 4 shadow)):

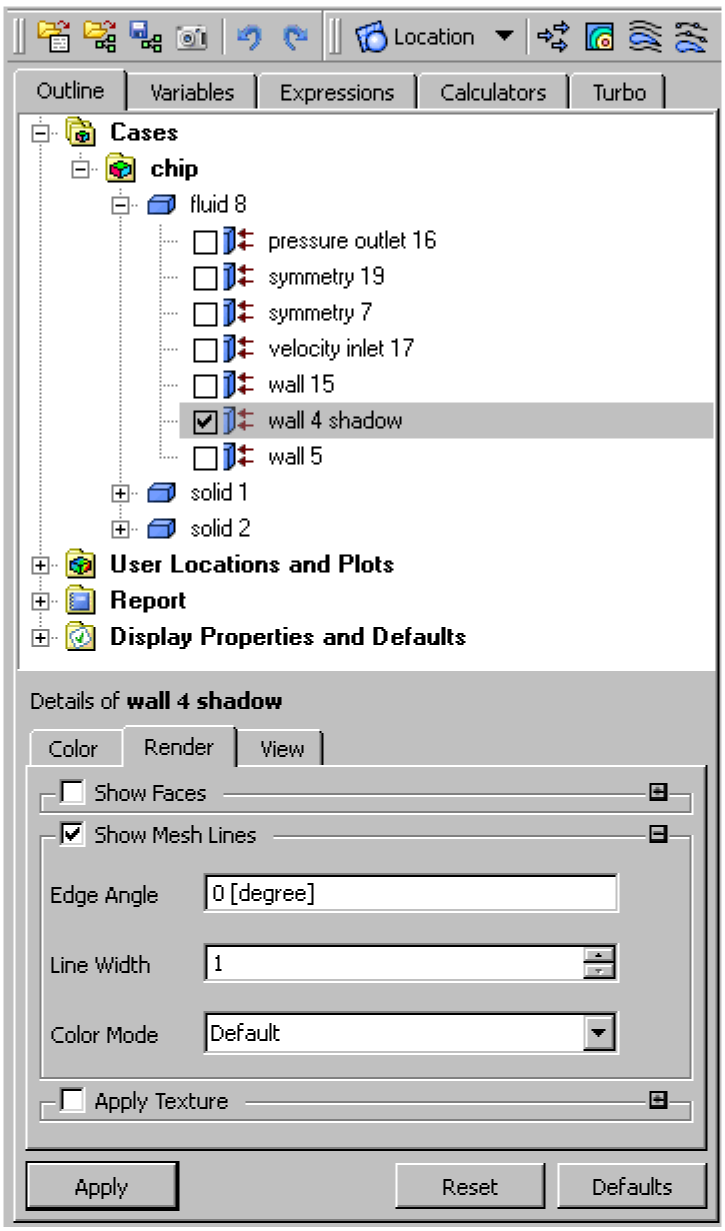
1. In the **Outline** tree view, select the check box beside **Cases** > **chip** > **fluid 8** > **wall 4 shadow**, then double-click **wall 4 shadow** to edit its properties in its details view.

2. In the details view:

   a. On the **Render** tab, clear **Show Faces**.

   b. Select **Show Mesh Lines**.

   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 chip.

e. Now, clear the display of the chip wireframe. In the details view:

   i. Clear **Render** > **Show Mesh Lines**.

   ii. Select **Show Faces**.

   iii. Click **Apply**.

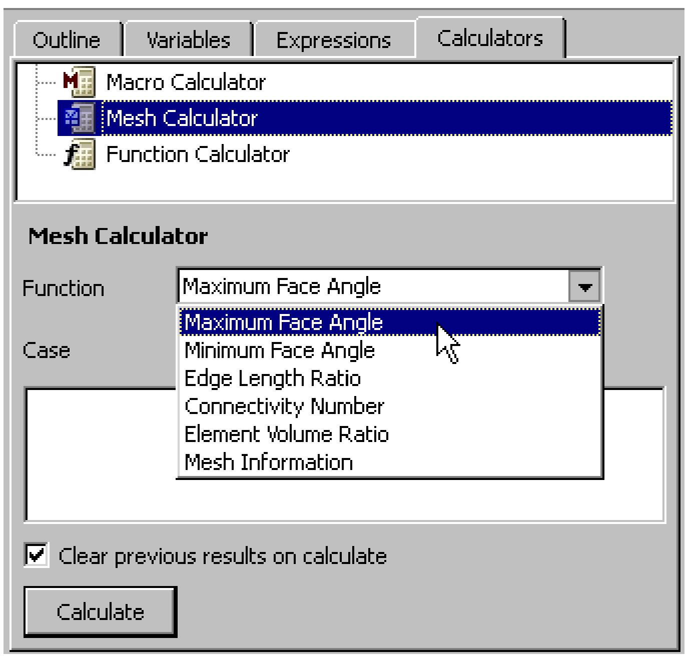
   The chip reappears.

3. In the **Outline** tree view, clear the check box beside **Cases** > **chip** > **fluid 8** > **wall 4 shadow**.

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`.

4.4.3. 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 `minVal` as the function to evaluate.

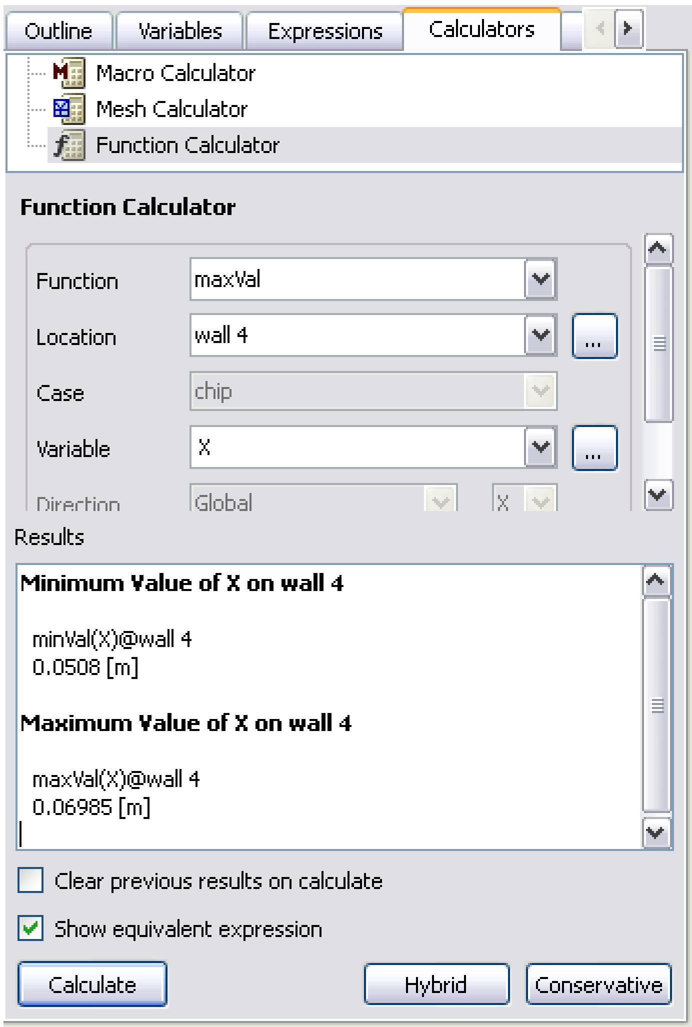
3. In the **Location** field, select `wall 4`.

4. Beside the **Variable** field, select X in the **Variable Selector** dialog box.

5. Clear the **Clear previous results on calculate** setting and select **Show equivalent expression**.

6. Click **Calculate** to see the result of the calculation of the minimum X value of the chip.

7. Repeat the operation, but in the **Function** field, select `maxVal` as the function to evaluate. Click **Calculate** to see the result of the calculation of the maximum X value of the chip.



You will use these values in subsequent steps.

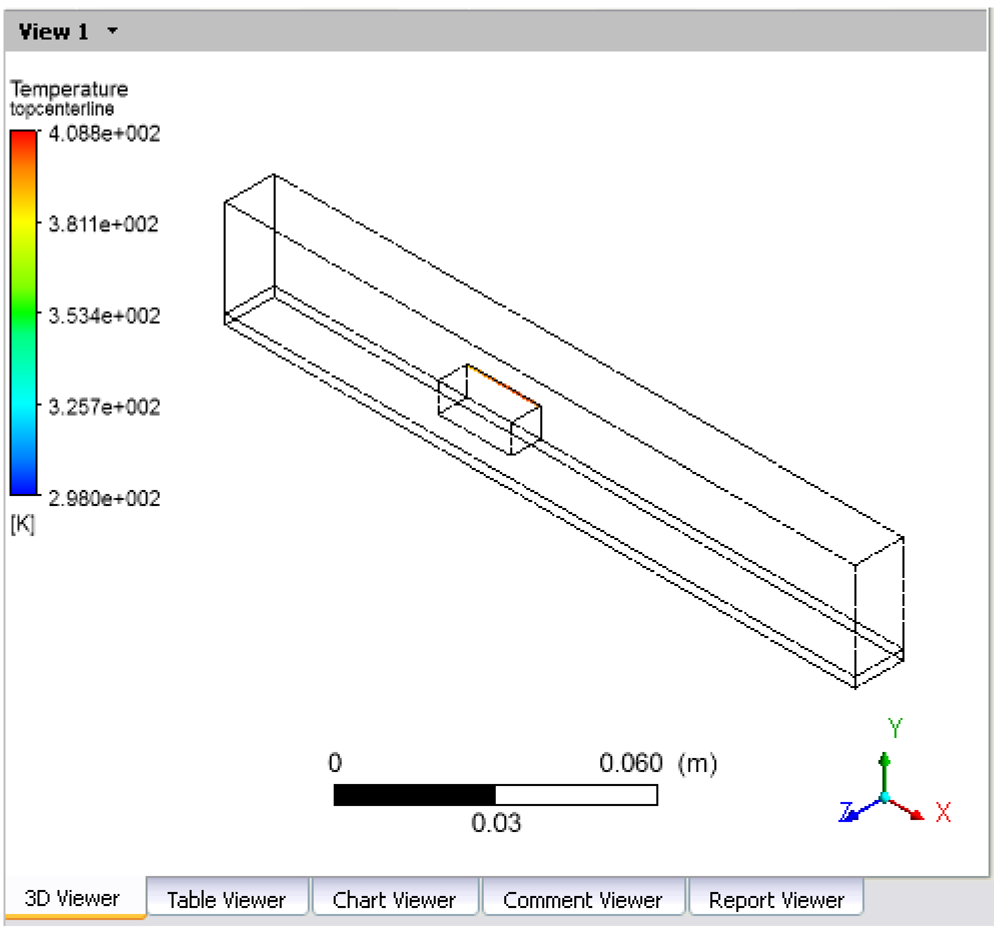
4.4.4. Create a Line

Lines can be used to display quantitative results of your CFD simulations. Here, you will create a line along which to plot the temperature distribution along the top center of the chip.

1. Select **Insert** > **Location** > **Line**.
2. For the name, type `topcenterline` and click **OK**.
3. On the **Details of topcenterline** > **Geometry** tab:
    a. Set **Method** to `Two Points`.
    b. Set **Point 1** to `0.0508`, `0.01`, `0`.
    c. Set **Point 2** to `0.06985`, `0.01`, `0`.
    d. Ensure that **Line Type** is set to `Sample`.

    Those coordinates define a line along the top center of the chip.
4. On the **Color** tab:
    a. Set **Mode** to `Variable`.
    b. Set **Variable** to `Temperature`.

    These steps will color the line by temperature and cause the legend to be displayed.
5. Click **Apply**.

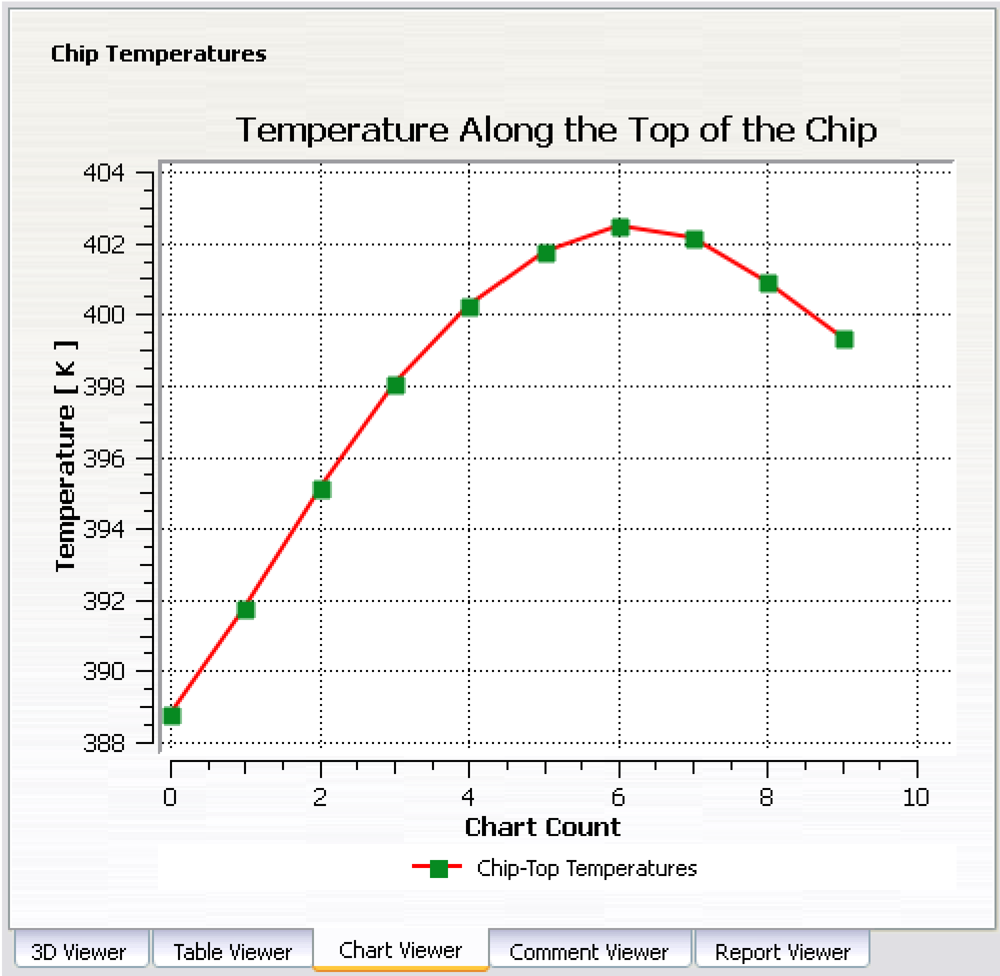


4.4.5. Create a Chart

Here you will plot the temperature distribution along a line along the top center of the chip.

1. Select **Insert** > **Chart**.
2. For the name, type `Chip Temperatures` and click **OK**.

   The **Details of Chip Temperatures** view appears.
3. On the **General** tab:
   a. Set **Title** to `Temperature Along the Top of the Chip`.
   b. Set **Caption** to `Graph of the Temperature Along the Top of the Chip`.
4. On the **Data Series** tab
   a. Set **Location** to `topcenterline`.
   b. Change the name `Series 1` to `Chip-Top Temperatures`.
5. On the **X Axis** tab, set **Variable** to `Chart Count`.
6. On the **Y Axis** tab, set **Variable** to `Temperature`.
7. On the **Line Display** tab, select **Chip-Top Temperatures** and set **Symbols** to `Rectangle`.
8. Make the **Symbol Color** a darker shade of green: beside the **Symbol Color** field, click *Color Selector*, select a new shade of green, and click **OK**.
9. Click **Apply**.



4.4.6. Add a Second Line

Here you will create a second line near the bottom of the chip so that you can compare that to the temperature distribution along the top center of the chip.

1. Select **Insert** > **Location** > **Line**.

2. For the name, type `bottomsideline` and click **OK**.

3. On the **Details of bottomsideline** > **Geometry** tab:

   a. Set **Method** to `Two Points`.

   b. Set **Point 1** to `0.0508`, `0.0027`, `0`.

   c. Set **Point 2** to `0.06985`, `0.0027`, `0`.

   d. Set **Line Type** to `Sample`.

   Those coordinates define a line near board level beside the chip.

4. Click **Apply**.

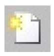
4.4.7. Create a Chart

Here you will plot the temperature distribution along the second line.

1. In the **Report** area of the Tree view, double-click **Chip Temperatures**.

2. On the **General** tab, change **Title** to `Temperature Differences on the Chip` and **Caption** to `Graph of the Temperature Along the Top and Bottom of the Chip`.

3. On the **Data Series** tab:

   a. Click *New*  to create `Series 2`.

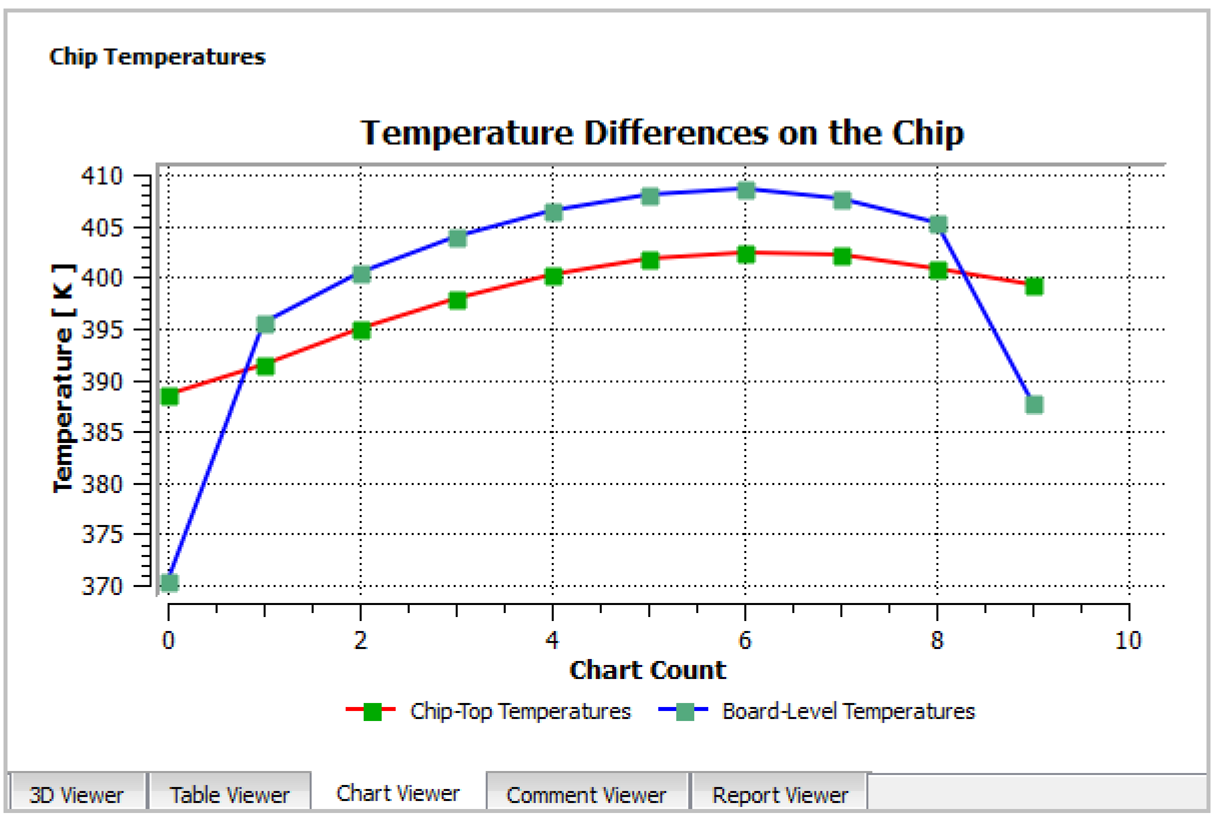
   b. Set **Location** to `bottomsideline`.

   c. Change the name `Series 2` to `Board-Level Temperatures`.

4. On the **Line Display** tab, select **Board-Level Temperatures** and set **Symbols** to `Rectangle`.

5. Beside the **Symbol Color** field, click *Color Selector* , select a new shade of green, and click **OK**.

6. Click **Apply**.



4.4.8. Create a Table to Show Heat Transfer

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 the cyan-colored ball on the triad to make it easier for you to see the temperature planes that you will create.

3. From the toolbar, select **Location** > **Plane**. In the **Insert Plane** dialog box, type `Table Plane 1` and click **OK**.

4. In the details view for Table Plane 1, set the following values:

Tab	Field	Value
Geometry	Domains	fluid 8
	Definition > **Method**	YZ Plane
	Definition > **X**	0.051 [m]
Color	Mode	Variable
	Variable	Temperature
	Range	Local
Render	Lighting	(Cleared)

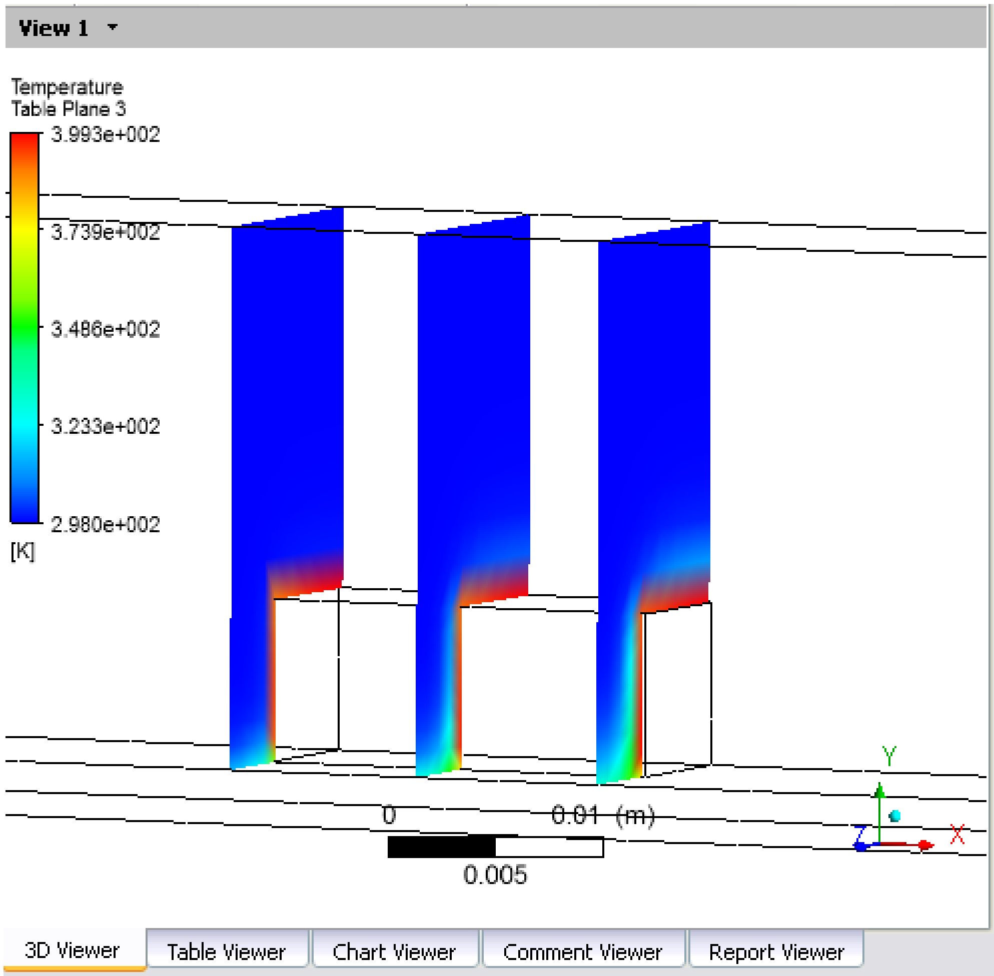
5. Click **Apply**.

6. Right-click **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** tree view, double-click **Table Plane 2** and on the **Geometry** tab change **Definition > X** to 0.0605. Click **Apply**.

7. Repeat the previous step, duplicating `Table Plane 2` to make `Table Plane 3` and changing **Definition > X** to 0.0697. Click **Apply**.



Now, create a table:

1. From the menu bar, select **Insert** > **Table**. Accept the default table name and click **OK**.

   The **Table Viewer** opens.

2. Type in the following headings:

	A	B	C	D
1	Distance Along Chip	Min. Temperature	Max. Temperature	Difference

3. For the "Distance Along Chip" column, create an equation that gives the distance from the beginning of the chip (which is available from "wall 4" in "solid 2"). Click 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 X. Move the cursor after the @ sign and either type `Table Plane 1` or select **Insert** > **Location** > **Table Plane 1**.

```
=minVal(X)@Table Plane 1 - minVal(X)@wall 4
```

When you click away from cell A2, the equation is solved.

Note:

The expressions in the equation are what you created in the Function Calculator. You can copy expressions from the Function Calculator and paste them into table cells, adding other characters in the cell definition field as required.

---

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

Cell A2

=minVal(X)@Table Plane 1 - minVal(X)@wall 4

Cell A3

=minVal(X)@Table Plane 2 - minVal(X)@wall 4

Cell A4

=minVal(X)@Table Plane 3 - minVal(X)@wall 4

Cell B2

=minVal(T)@Table Plane 1

Cell B3

=minVal(T)@Table Plane 2

Cell B4

=minVal(T)@Table Plane 3

Cell C2

=maxVal(T)@Table Plane 1

Cell C3

=maxVal(T)@Table Plane 2

Cell C4

=maxVal(T)@Table Plane 3

Cell D2

=maxVal(T)@Table Plane 1 - minVal(T)@Table Plane 1

Cell D3

=maxVal(T)@Table Plane 2 - minVal(T)@Table Plane 2

Cell D4

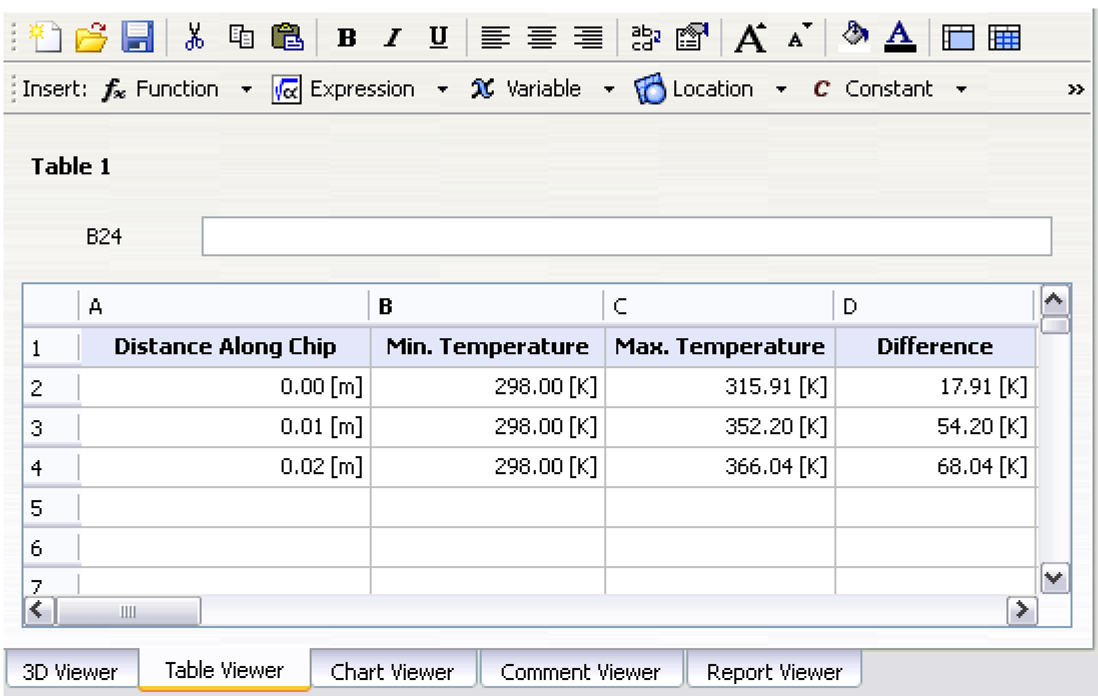
=maxVal(T)@Table Plane 3 - minVal(T)@Table Plane 3

As you complete the table, notice that the minimum temperature values stay constant, but the maximum values increase as the chip heats the passing air.

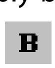
5. The default format for cell data is appropriate for some variables, but it is not appropriate here. Click cell A2, then while depressing the **Shift** key, click in the lower-right cell (D4). Click the *Number Formatting*  icon in the **Table Viewer** toolbar. In the **Cell Formatting** dialog box, set **Precision** to 2, change **Scientific** to `Fixed`, and click **OK**.

6. Optionally, apply some formatting to the table.

   You can view the table in three places: in the **Table Viewer** (where you can apply formatting), in the **Report Viewer** (where some of the formatting you applied in the **Table Viewer** will be visible), and in the published report (which has default formatting for tables that you cannot see in either the **Table Viewer** or the **Report Viewer**, but which are overridden by any formatting changes you make in the **Table Viewer**). It is useful to view the published report (see Publish a Report (p. 81)) before applying formatting in the **Table Viewer**.



To format the table as shown above:

a. For cells A1-D1: Apply bold font, background color, and text centering. Manually resize cell widths individually.   

b. For cells A2-D4: Right-justified text. 

4.4.9. Publish a Report

To save a report as an HTML file:

1. Click the **Report Viewer** tab.
2. Click **Publish** at the top of the **Report Viewer**.
3. In the **Publish Report** dialog box, specify a meaningful name for the report, such as `IC_Cooling_Simulation.htm`.

Tip:

Click the Browse icon in the **Publish Report** dialog box to control where the report is stored.

---

4. Click **OK**.