Chapter 1: Fluid Flow and Heat Transfer in a Mixing Elbow

This tutorial is divided into the following sections:

1.1. Introduction

This tutorial illustrates the setup and solution of 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:

- Launch ANSYS Fluent.
- Read an existing mesh file into ANSYS Fluent.
- Use mixed units to define the geometry and fluid properties.
- Set material properties and boundary conditions for a turbulent forced-convection problem.
- Create a surface report definition and use it as a convergence criterion.
- Calculate a solution using the pressure-based solver.
- Visually examine the flow and temperature fields using the postprocessing tools available in ANSYS Fluent.
- Adapt the mesh based on the temperature gradient to further improve the prediction of the temperature field.

1.2. Prerequisites

This tutorial assumes that you have little or no experience with ANSYS Fluent, and so each step will be explicitly described.

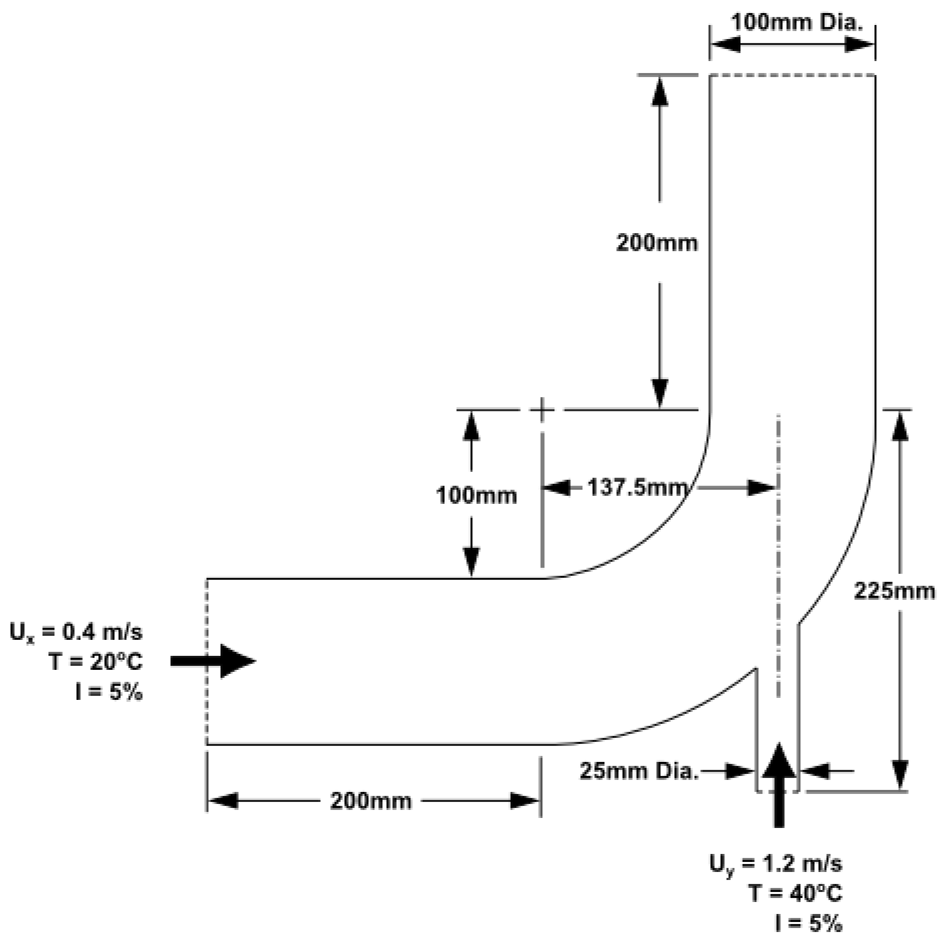
1.3. Problem Description

The problem to be considered is shown schematically in Figure 1.1: Problem Specification (p. 2). 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 and 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 a turbulent flow model will be required.

Note:

Since the geometry of the mixing elbow is symmetric, only half of the elbow must be modeled in ANSYS Fluent.

Figure 1.1: Problem Specification



1.4. Setup and Solution

To help you quickly identify graphical user interface items at a glance and guide you through the steps of setting up and running your simulation, the ANSYS Fluent Tutorial Guide uses several type styles and mini flow charts. See Typographical Conventions Used In This Manual (p. xxiii) for detailed information.

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

1.4.1. Preparation

1. Download the `introduction.zip` file here.

2. Unzip `introduction.zip` to your working directory.

3. The `elbow.msh` can be found in the folder.

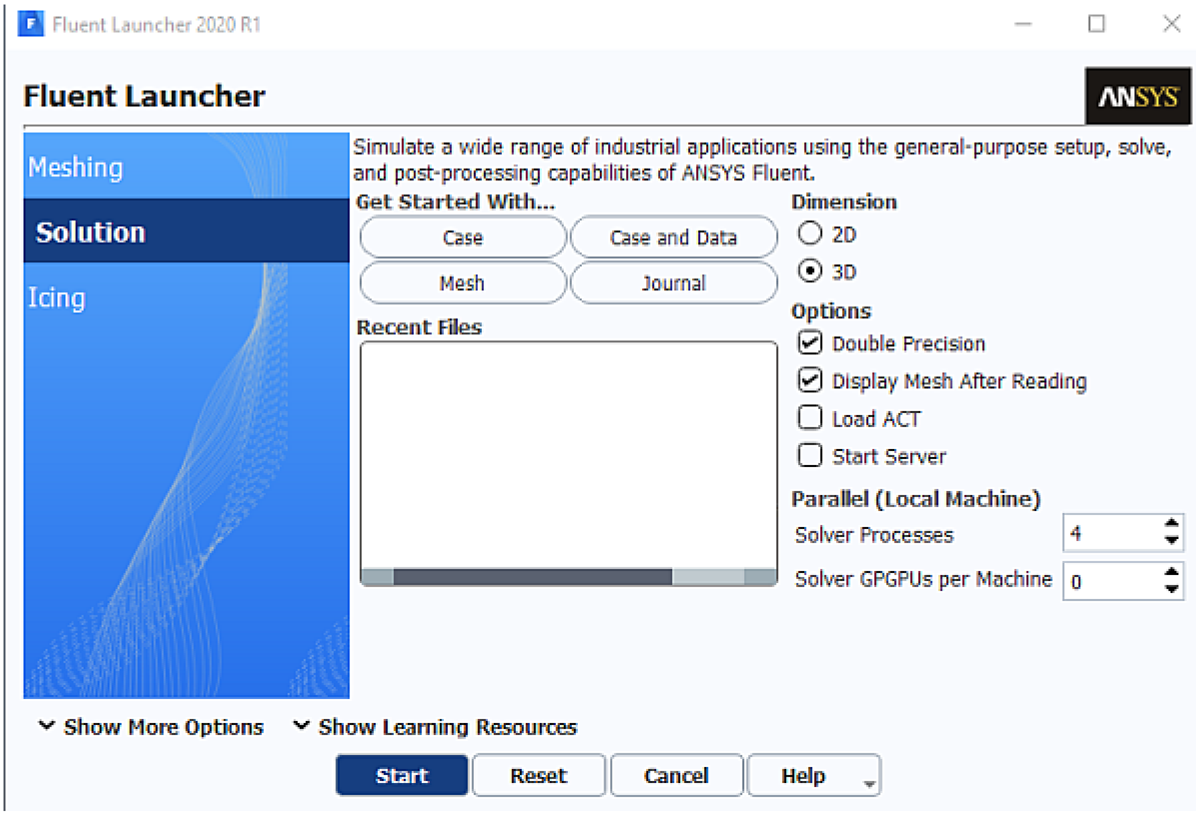
Note:

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

1.4.2. Launching ANSYS Fluent

1. From the Windows **Start** menu, select **Start > ANSYS 2020 R1 > Fluent 2020 R1** to start 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.

   a. Select **3D** from the **Dimension** list by clicking the radio button or the text.

   b. Ensure that the **Double Precision** option is selected.

   c. Ensure that the **Display Mesh After Reading** option is enabled.

   d. Set **Processes** to 4 under the **Parallel (local Machine)**.

   Note:

   Fluent will retain your preferences for future sessions.

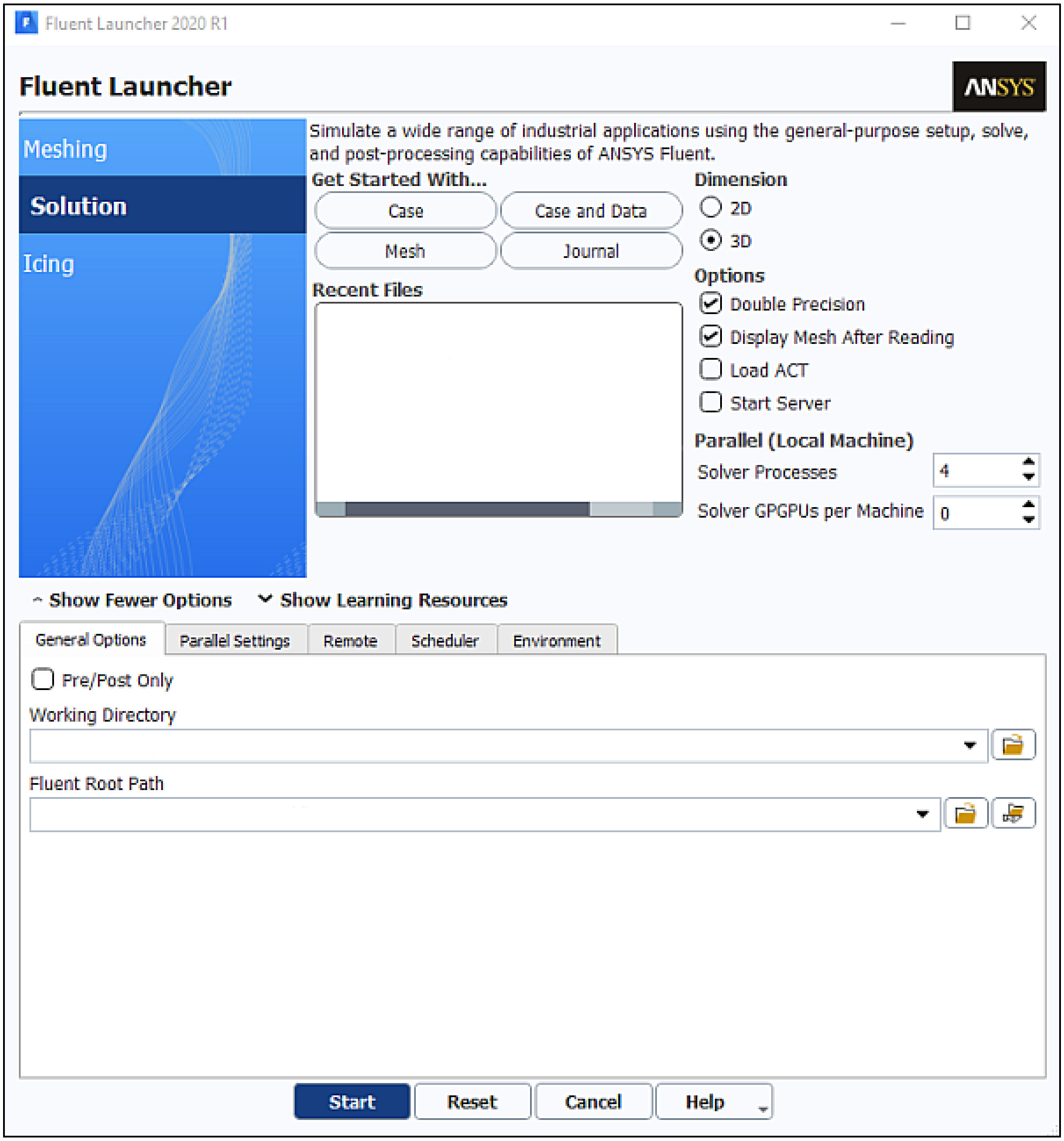
   ---

3. Set the working folder to the one created when you unzipped `introduction.zip`.

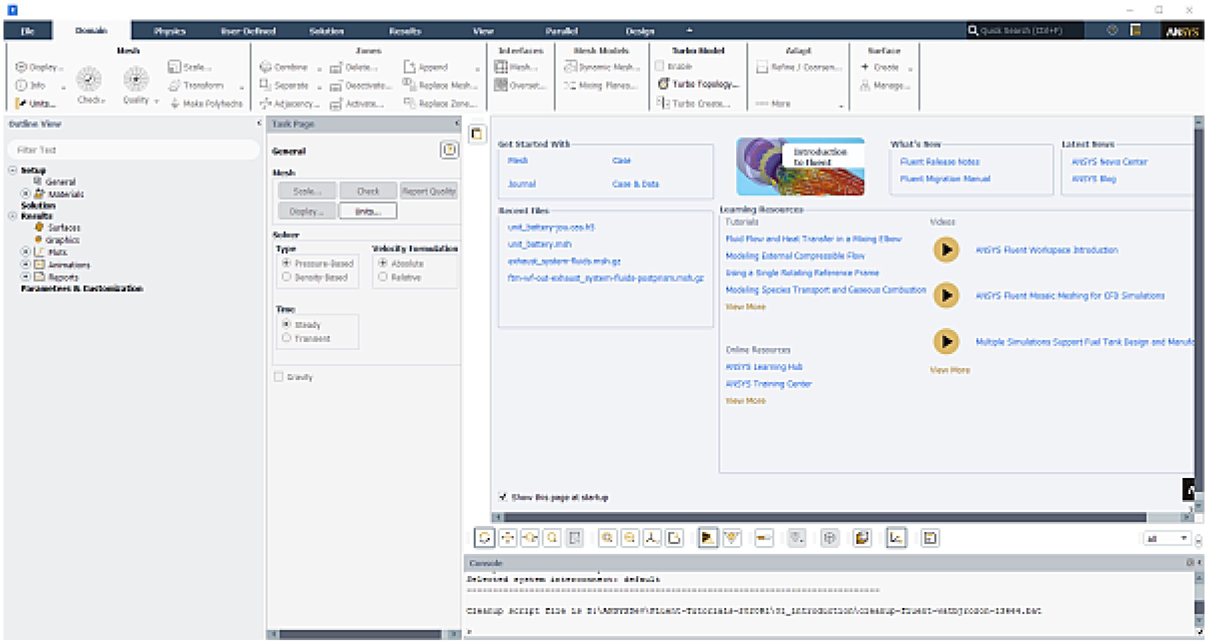
   a. Click the **Show More Options** button to reveal additional options.

   b. Enter the path to your working folder 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 ANSYS Fluent.

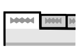


For more information about the Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

1.4.3. Reading the Mesh

1. Read the mesh file `elbow.msh`.

   Click the **File** ribbon tab, then click **Read** and **Mesh...** in the menus that open in order to open the **Select File** dialog box.

    **File** → **Read** → **Mesh...**

a. Select the mesh file by clicking **elbow.msh** in the **introduction** folder created when you unzipped the original file.

b. Enable the **Display Mesh After Reading** in the **Select File** dialog box.

c. Click **OK** to read the file and close the **Select File** dialog box.

As the mesh file is read by ANSYS Fluent, messages will appear in the console reporting the progress of the conversion. ANSYS Fluent will report that 13,852 hexahedral fluid cells have been read, along with a number of boundary faces with different zone identifiers.

After having completed reading mesh, ANSYS Fluent displays the mesh in the graphics window.

Extra:

You can use the mouse to probe for mesh information in the graphics window. If you click the right mouse button with the pointer on any node in the mesh, information about the associated zone will be displayed in the console, including the name of the zone.

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

For this 3D problem, you can make it easier to probe particular nodes by changing the view. The following table describes how to manipulate objects in the graphics window:

Table 1.1: View Manipulation Instructions

Action	Using Graphics Toolbar Buttons and the Mouse
Rotate view (vertical, horizontal)	After clicking the **Rotate View** icon, , press and hold the left mouse button and drag the mouse. Dragging side to side rotates the view about the vertical axis, and dragging up and down rotates the view about the horizontal axis.
Translate or pan view	After clicking the **Pan** icon, , press and hold the left mouse button and drag the object with the mouse until the view is satisfactory.
Zoom in and out of view	After clicking the **Zoom In/Out** icon, , press and hold the left mouse button and drag the mouse up and down to zoom in and out of the view.
Zoom to selected area	After clicking the **Zoom to Area** icon, , press and hold the left mouse button and drag the mouse diagonally to the right. This action will cause a rectangle to appear in the display. When you release the mouse button, a new view will be displayed that consists entirely of the contents of the rectangle. Note that to zoom in, you must drag the mouse to the right, and to zoom out, you must drag the mouse to the left.

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

After you have clicked a button in the graphics toolbar, you can return to the default mouse button settings by clicking .

To judge the scale of your 3D geometry, you can click the **Orthographic Projection** icon, . This will display the length scale ruler near the bottom of the graphics window.

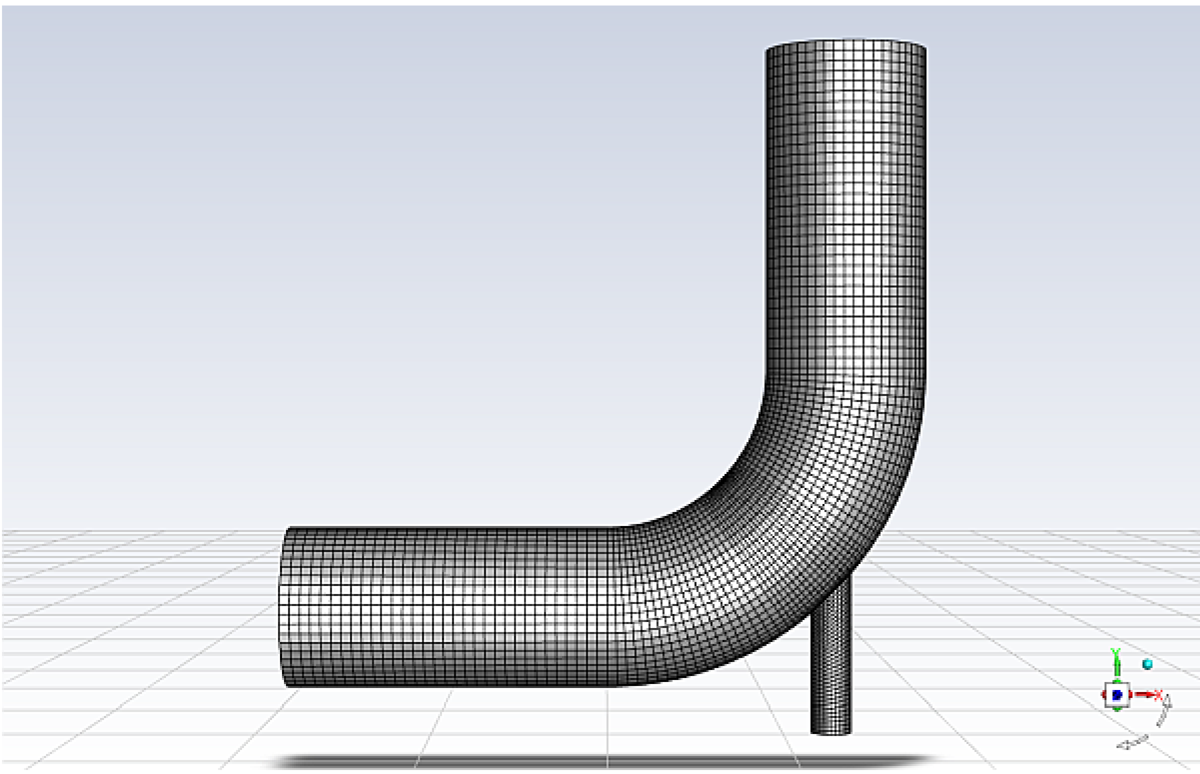
Note that you can change the default mouse button actions in the **View** tab (in the **Mouse** group box). For more information, see the Fluent User's Guide.

2. Manipulate the mesh display using the axis triad to obtain a front view as shown in Figure 1.2: The Hexahedral Mesh for the Mixing Elbow (p. 9).

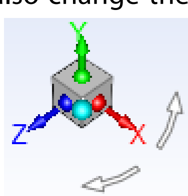
   a. Click the z-axis.

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

   c. **Figure 1.2: The Hexahedral Mesh for the Mixing Elbow**



Extra:

You can also change the orientation of the objects in the graphics window using the axis triad 

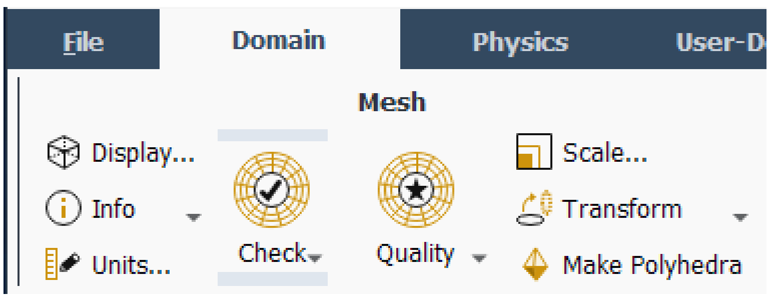
as follows:

- To orient the model in the positive/negative direction, click an axis/semi-sphere.
- To orient the model in the negative/positive direction, right-click an axis/semi-sphere.
- To set the isometric view, click the cyan iso-ball.

- To perform in-plane clockwise or counterclockwise 90° rotations, click the white rotational arrows .
- To perform free rotations in any direction, click and hold—in the vicinity of the triad—and use the mouse. Release the left mouse button to stop rotating.

1.4.4. Setting Up Domain

In this step, you will perform the mesh-related activities using the **Domain** ribbon tab (**Mesh** group box).



1. Check the mesh.

   Domain → **Mesh** → **Check** → **Perform Mesh Check**

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

```
 Domain Extents:
   x-coordinate: min (m) = -8.000000e+00, max (m) = 8.000000e+00
   y-coordinate: min (m) = -9.134634e+00, max (m) = 8.000000e+00
   z-coordinate: min (m) = 0.000000e+00, max (m) = 2.000000e+00
 Volume statistics:
   minimum volume (m3): 5.098304e-04
   maximum volume (m3): 2.330736e-02
     total volume (m3): 1.607154e+02
 Face area statistics:
   minimum face area (m2): 4.865882e-03
   maximum face area (m2): 1.017924e-01
 Checking mesh...................................
Done.
```

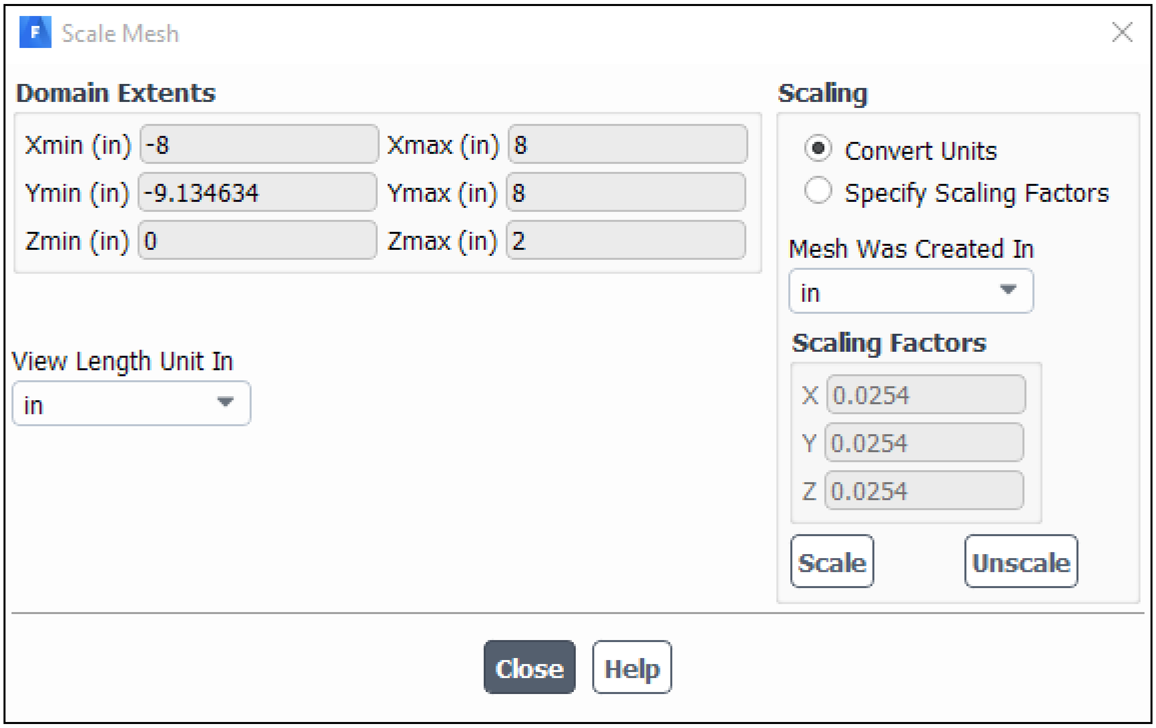
   The mesh check will list the minimum and maximum x, y, and z values from the mesh in the default SI unit of meters. It will also report a number of other mesh features that are checked. Any errors in the mesh will be reported at this time. Ensure that the minimum volume is not negative, since ANSYS Fluent cannot begin a calculation when this is the case.

   Note:

   The minimum and maximum values may vary slightly when running on different platforms.

2. Scale the mesh.

Domain → Mesh → Scale...



a. Ensure that **Convert Units** is selected in the **Scaling** group box.

b. From the **Mesh Was Created In** drop-down list, select **in** by first clicking the down-arrow button and then clicking the **in** item from the list that appears.

c. Click **Scale** to scale the mesh.

> **Warning:**
>
> Be sure to click the **Scale** button only once.

__Domain Extents__ will continue to be reported in the default SI unit of meters.

d. Select **in** from the **View Length Unit In** drop-down list to set inches as the working unit for length.

e. Confirm that the domain extents are as shown in the previous dialog box.

f. Close the **Scale Mesh** dialog box.

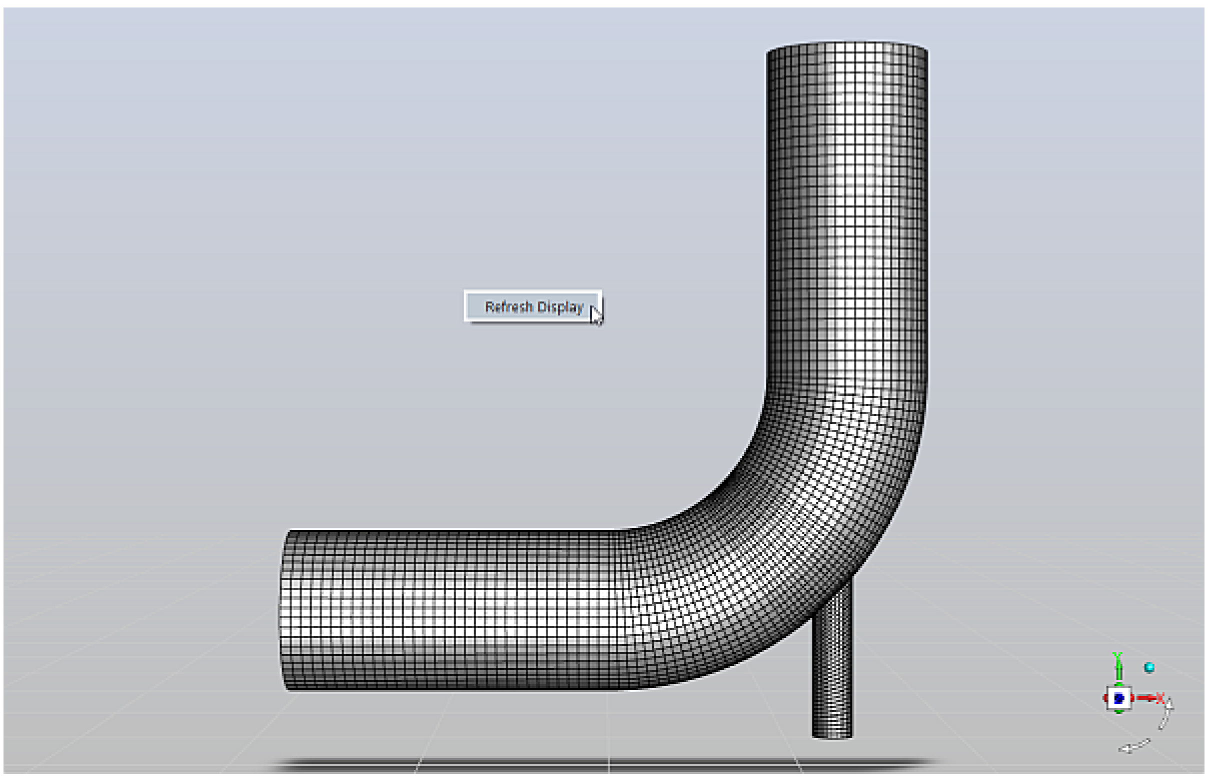
The mesh is now sized correctly and the working unit for length has been set to inches.

Note:

Because the default SI units will be used for everything except length, there is no need to change any other units in this problem. The choice of inches for the unit of length has been made by the actions you have just taken. If you want a different working unit for length, other than inches (for example, millimeters), click **Units...** in the **Domain** ribbon tab (**Mesh** group box) and make the appropriate change in the **Set Units** dialog box.

---

3. Right click in the graphics window and select **Refresh Display**



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

5. Check the mesh.

Domain → **Mesh** → **Check** → **Perform Mesh Check**

Note:

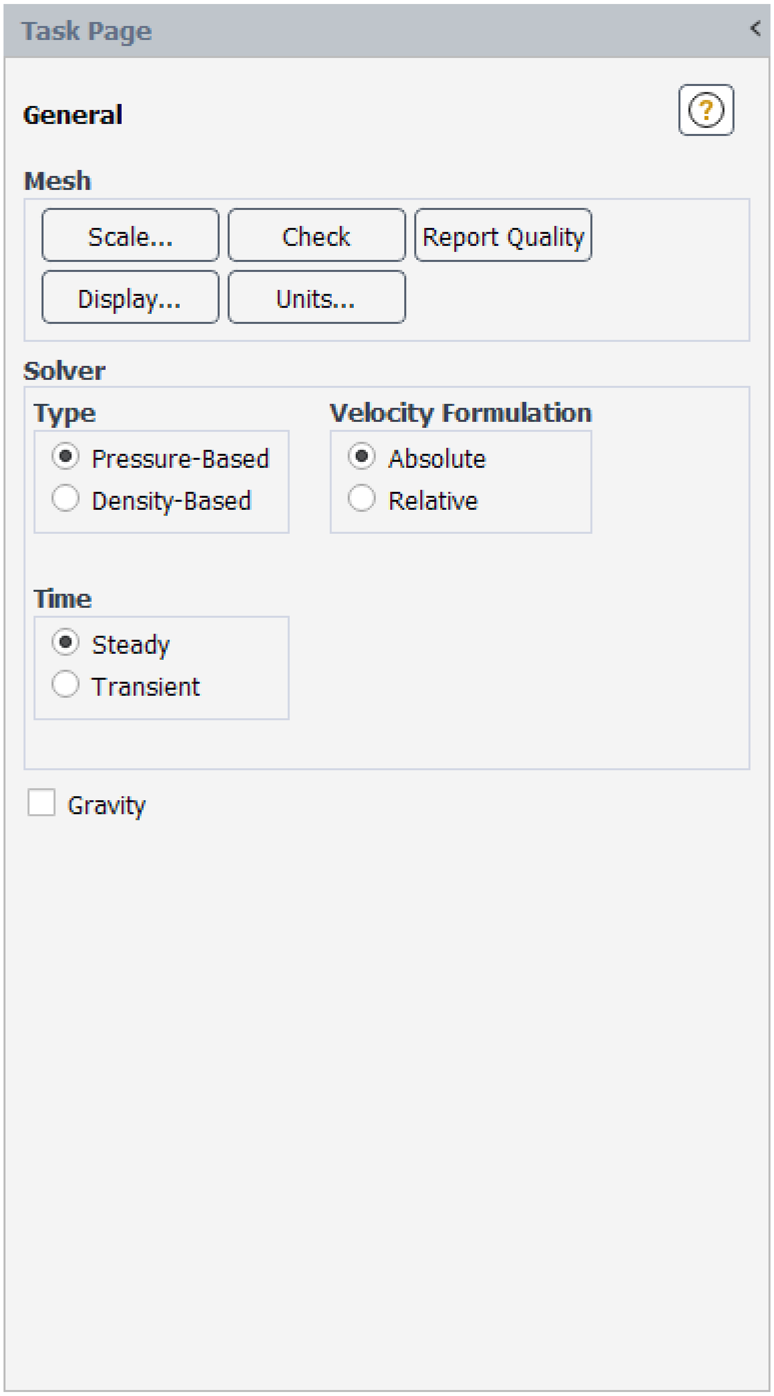
It is a good idea to check the mesh after you manipulate it (that is, scale, convert to polyhedra, merge, separate, fuse, add zones, or smooth and swap). This will ensure that the quality of the mesh has not been compromised.

1.4.5. Setting Up Physics

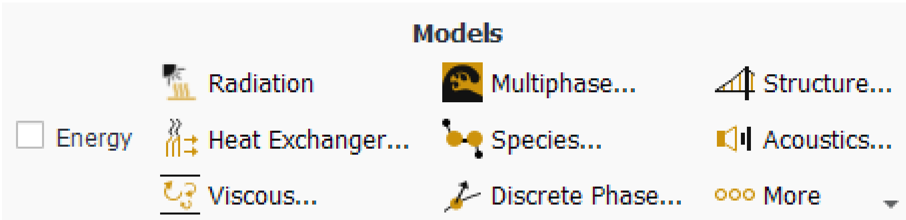
In the steps that follow, you will select a solver and specify physical models, material properties, and zone conditions for your simulation using the **Physics** ribbon tab.

1. In the **Solver** group box of the **Physics** ribbon tab, retain the default selection of the steady pressure-based solver.

   Physics → **Solver** → **General**



2. Set up your models for the CFD simulation using the **Models** group box of the **Physics** ribbon tab.



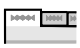
Note:

You can also use the **Models** task page, which can be accessed from the tree by expanding **Setup** and double-clicking the **Models** tree item.

---

a. Enable heat transfer by activating the energy equation.

   In the **Physics** ribbon tab, enable **Energy** (**Models** group box).

    **Physics** → **Models** → **Energy**

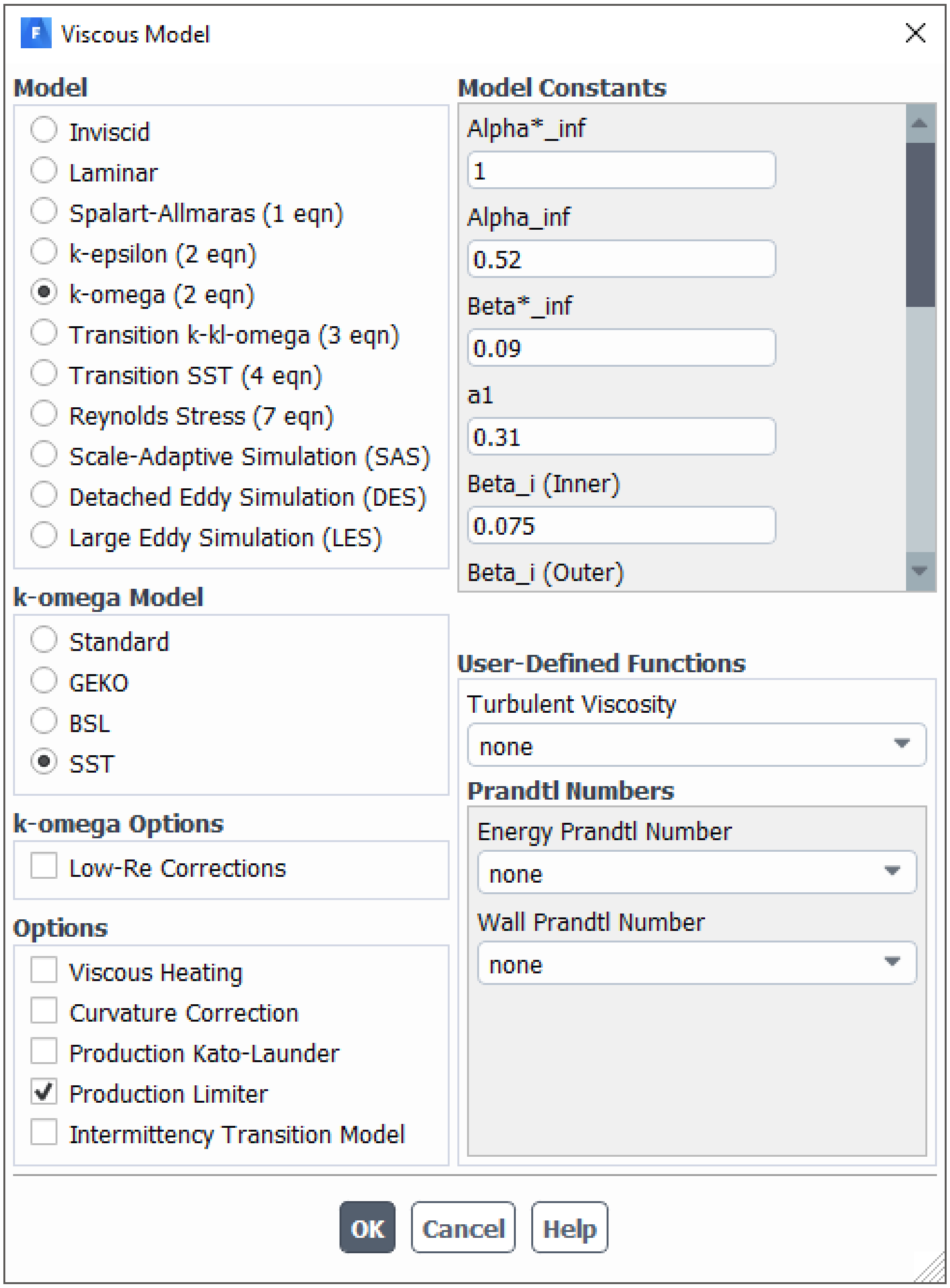
   Note:

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

   ---

b. Enable the k-ω turbulence model.

    **Physics** → **Models** → **Viscous...**



i. Retain the default selection of **k-omega** from the **Model** list.

ii. Retain the default selection of **SST** in the **k-omega Model** group box.

iii. Click **OK** to accept all the other default settings and close the **Viscous Model** dialog box.

Note that the **Viscous...** label in the ribbon is displayed in blue to indicate that the Viscous model is enabled. Also **Energy** and **Viscous** appear as enabled under the **Setup/Models** tree branch.

Note:

While the ribbon is the primary tool for setting up and solving your problem, the tree is a dynamic representation of your case. The models, materials, conditions, and other settings that you have specified in your problem will appear in the tree. Many of the frequently used ribbon items are also available via the right-click functionality of the tree.

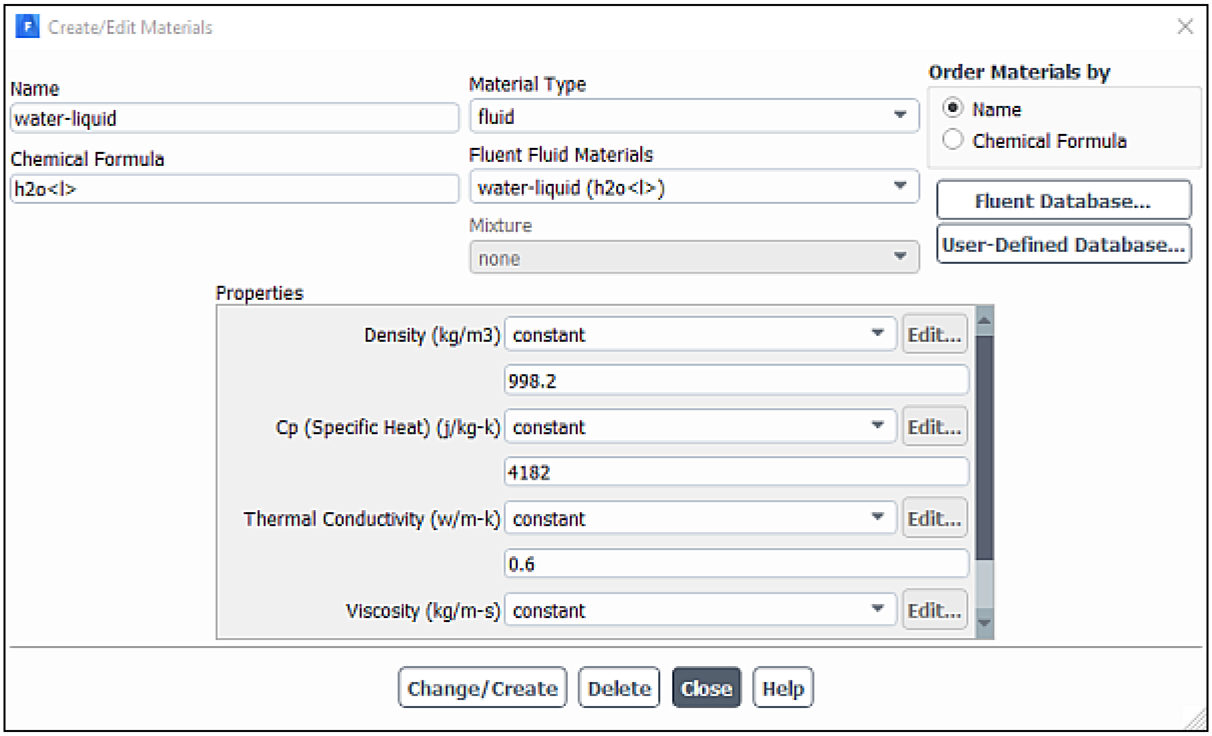
---

3. Set up the materials for the CFD simulation using the **Materials** group box of the **Physics** ribbon tab.

Create a new material called **water** using the **Create/Edit Materials** dialog box.

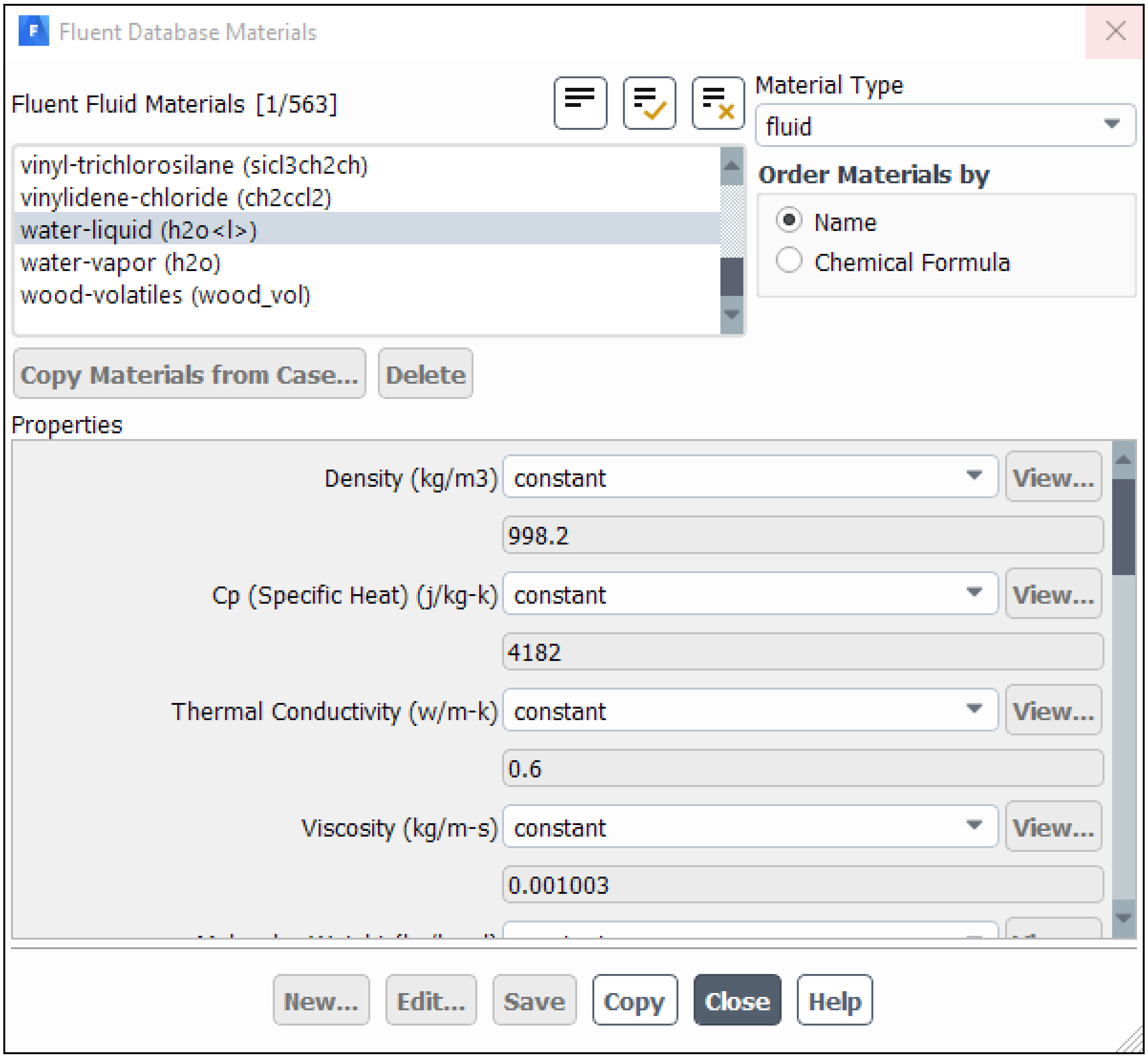
a. In the **Physics** ribbon tab, click **Create/Edit...** (**Materials** group box).

Physics → **Materials** → **Create/Edit...**



b. Click the **Fluent Database...** button to access pre-defined materials.

c. Select **water-liquid (h2o < l >)** from the materials list and click **Copy**, then close the **Fluent Database...** dialog box.

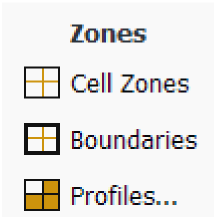


d. Ensure that there are now two materials (water-liquid and air) defined locally by examining the **Fluent Fluid Materials** drop-down list.

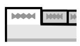
   *Both the materials will also be listed under **Fluid** in the **Materials** task page and under the **Materials** tree branch.*

e. Close the **Create/Edit Materials** dialog box.

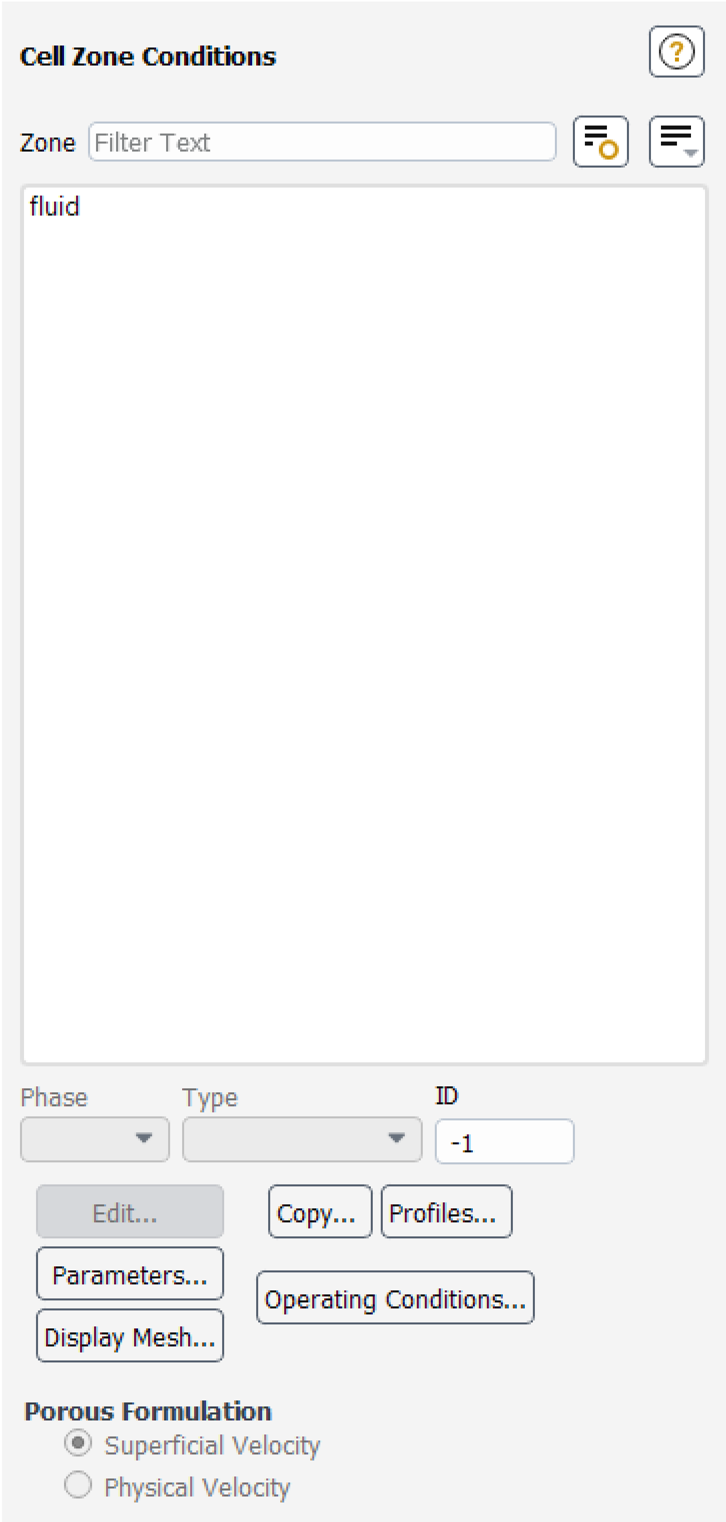
4. Set up the cell zone conditions for the fluid zone (**fluid**) using the **Zones** group box of the **Physics** ribbon tab.



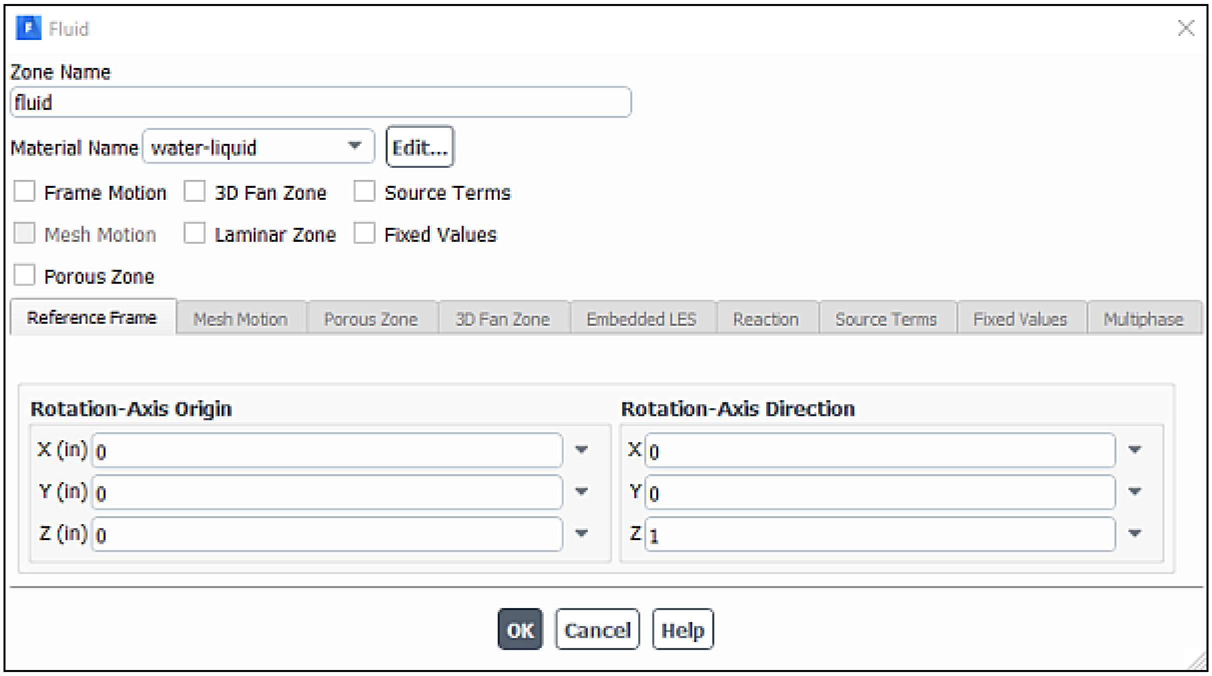
a. In the **Physics** tab, click **Cell Zones** (**Zones** group box).

 **Physics** → **Zones** → **Cell Zones**

This opens the **Cell Zone Conditions** task page.



b. Double-click **fluid** in the **Zone** list to open the **Fluid** dialog box.



Note:

You can also double-click the **Setup/Cell Zone Conditions/fluid** tree item in order to open the corresponding dialog box.

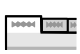
---

c. Select **water-liquid** from the **Material Name** drop-down list.

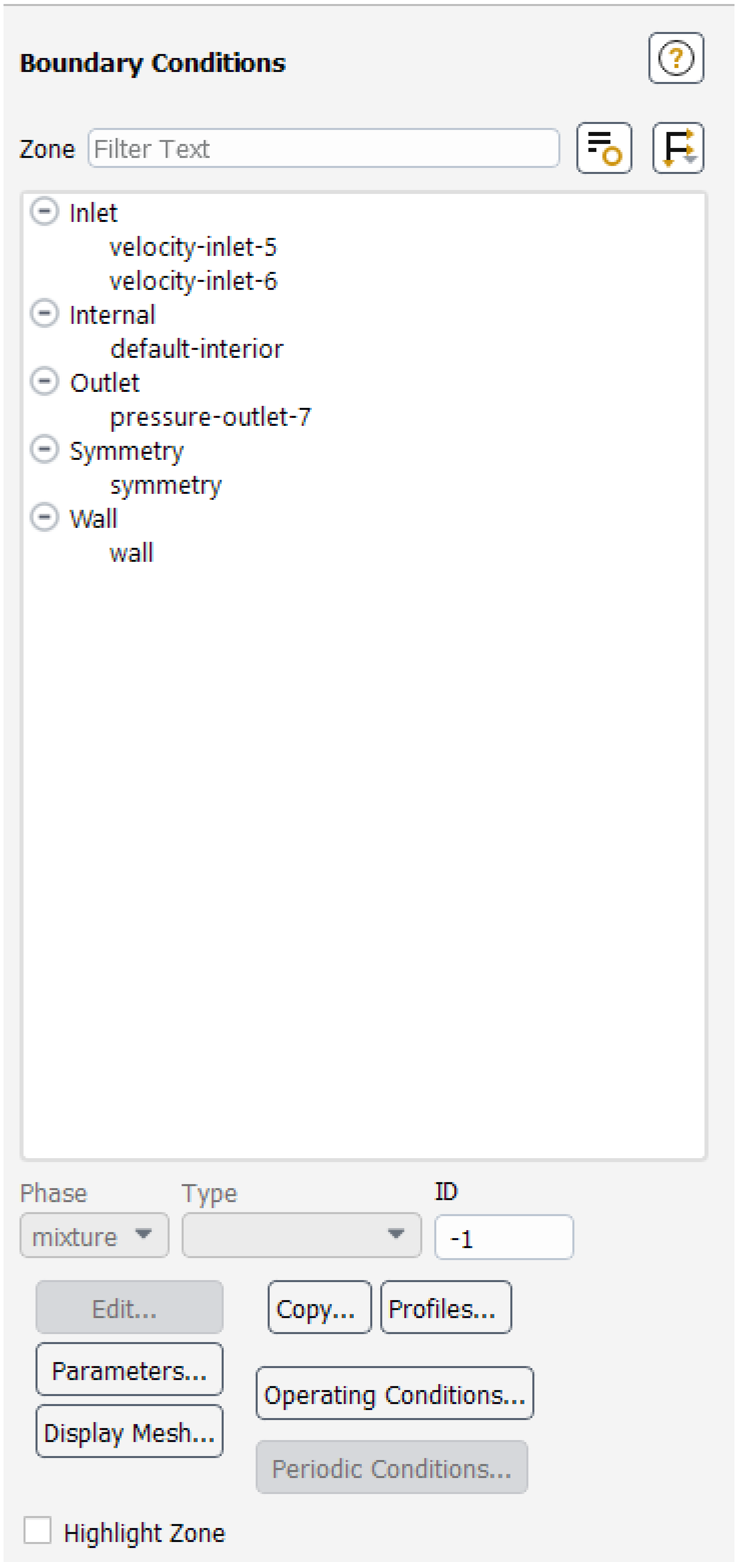
d. Click **OK** to close the **Fluid** dialog box.

5. Set up the boundary conditions for the inlets, outlet, and walls for your CFD analysis using the **Zones** group box of the **Physics** ribbon tab.

a. In the **Physics** tab, click **Boundaries** (**Zones** group box).

 **Physics → Zones → Boundaries**

This opens the **Boundary Conditions** task page where the boundaries defined in your simulation are displayed in the **Zone** selection list.


Boundary Conditions
Zone
Filter Text
Inlet
velocity-inlet-5
velocity-inlet-6
Internal
default-interior
Outlet
pressure-outlet-7
Symmetry
symmetry
Wall
wall
Phase
Type
ID
mixture
-1
Edit...
Copy...
Profiles...
Parameters...
Operating Conditions...
Display Mesh...
Periodic Conditions...
Highlight Zone

Note:

To display boundary zones grouped by zone type (as shown previously), click the **Toggle Tree View** button () in the upper right corner of the **Boundary Conditions** task page and select **Zone Type** under **Group By**.

Here the zones have names with numerical identifying tags. It is good practice to give boundaries meaningful names in a meshing application to help when you set up the model. You can also change boundary names in Fluent by simply editing the boundary and making revisions in the **Zone Name** text box.

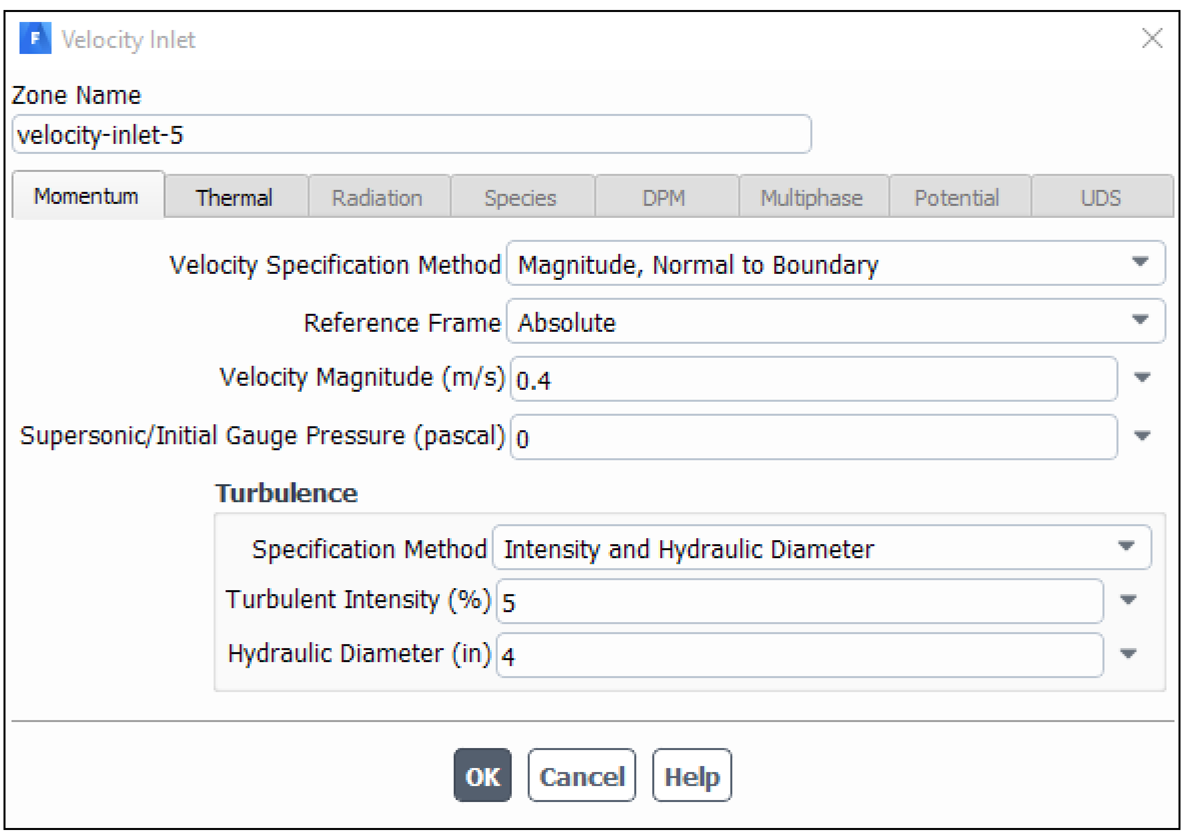
b. Set the boundary conditions at the cold inlet (**velocity-inlet-5**).

Tip:

If you are unsure of which inlet zone corresponds to the cold inlet, you can probe the mesh display using the right mouse button or the probe toolbar button () as described previously in this tutorial. The information will be displayed in the ANSYS Fluent console, and the zone you probed will be automatically selected from the **Zone** selection list in the **Boundary Conditions** task page.

---

i. Double-click **velocity-inlet-5** to open the **Velocity Inlet** dialog box.



ii. Retain the default selection of **Magnitude, Normal to Boundary** from the **Velocity Specification Method** drop-down list.

iii. Enter `0.4` [m/s] for **Velocity Magnitude**.

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

v. Retain the default value of 5 [%] for **Turbulent Intensity**.

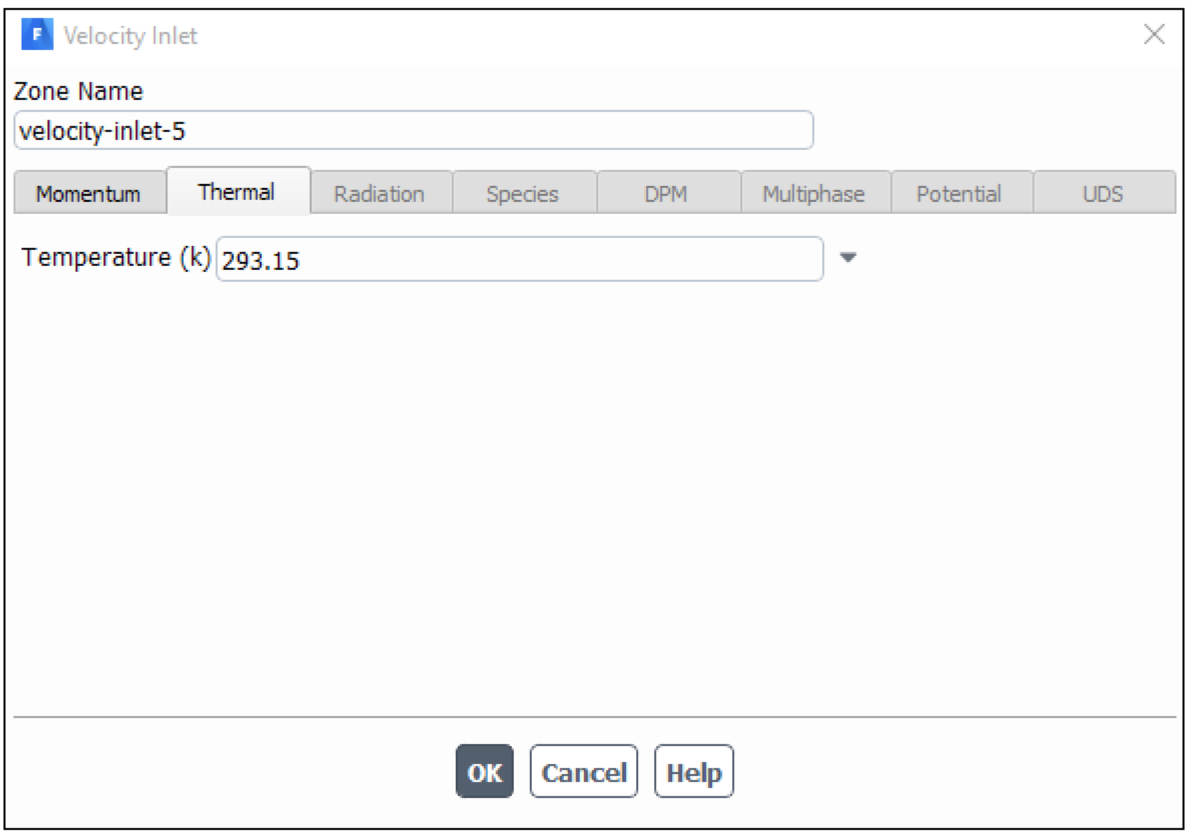
vi. Enter 4 [inches] for **Hydraulic Diameter**.

The hydraulic diameter D_h is defined as:

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

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

vii. Click the **Thermal** tab.



viii. Enter `293.15` [K] for **Temperature**.

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

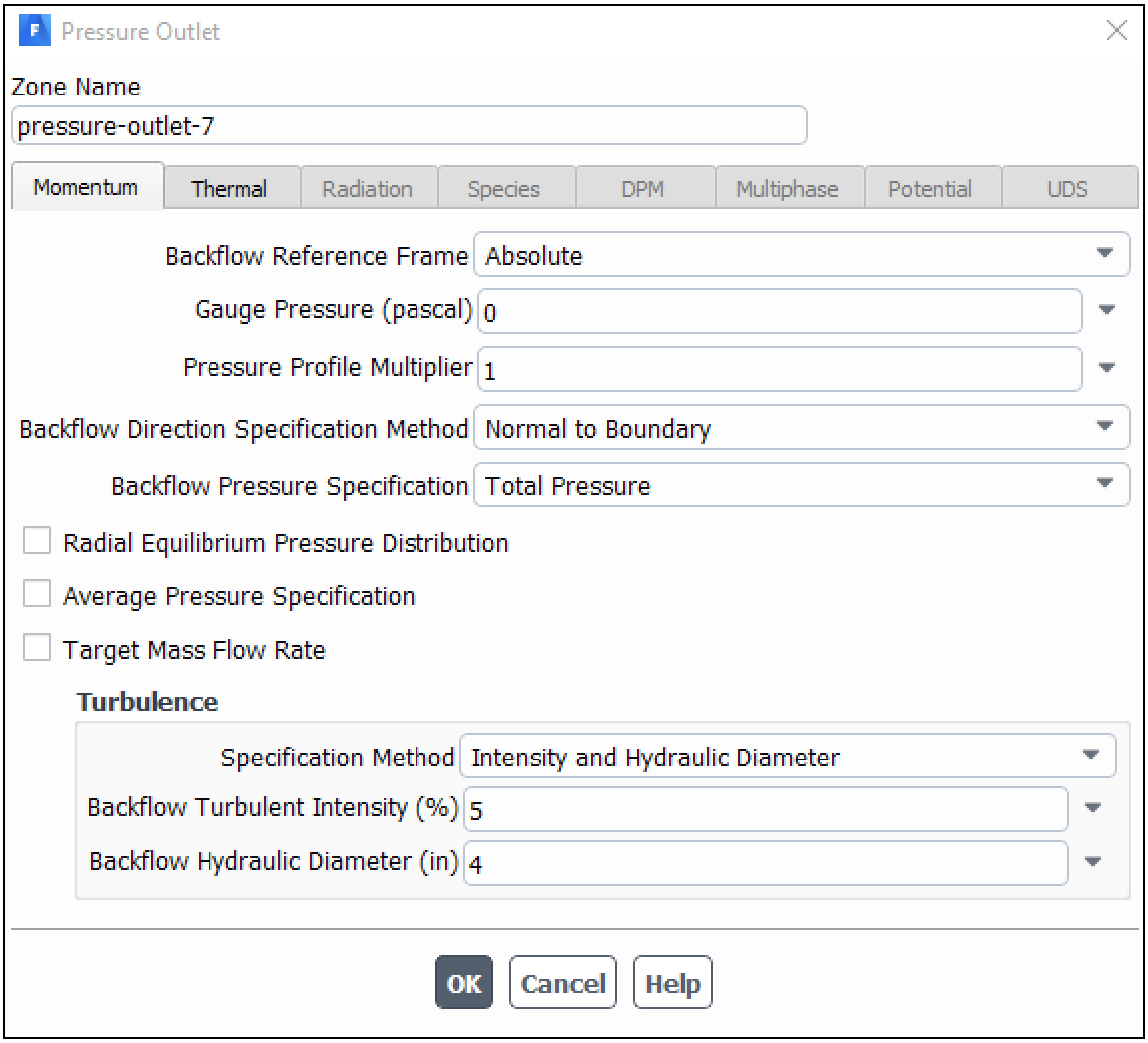
Note:

You can also access the **Velocity Inlet** dialog box by double-clicking the **Setup/Boundary Conditions/velocity-inlet-5** tree item.

c. In a similar manner, set the boundary conditions at the hot inlet (**velocity-inlet-6**), using the values in the following table:

Setting	Value
Velocity Specification Method	Magnitude, Normal to Boundary
Velocity Magnitude	`1.2` [m/s]
Specification Method	Intensity and Hydraulic Diameter
Turbulent Intensity	5 [%]
Hydraulic Diameter	1 [inch]
Temperature	`313.15` [K]

d. Double-click **pressure-outlet-7** in the **Zone** selection list and set the boundary conditions at the outlet, as shown in the following figure.

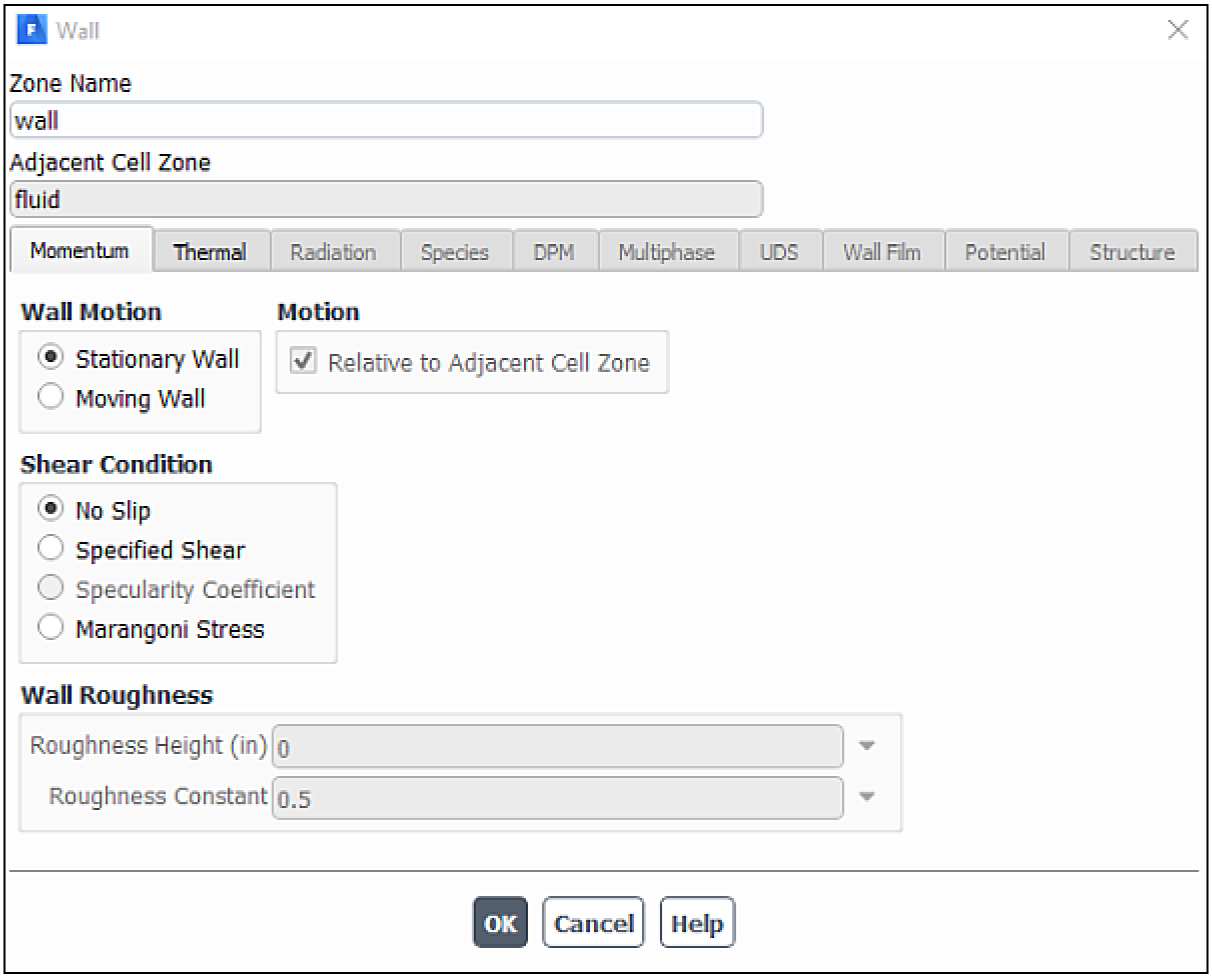


Note:

- You do not need to set a backflow temperature in this case (in the **Thermal** tab) because the material properties are not functions of temperature. If they were, a flow-weighted average of the inlet conditions would be a good starting value.

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

e. For the wall of the pipe (**wall**), retain the default value of `0` W/m^2 for **Heat Flux** in the **Thermal** tab.



1.4.6. Solving

In the steps that follow, you will set up and run the calculation using the **Solution** ribbon tab.

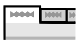
Note:

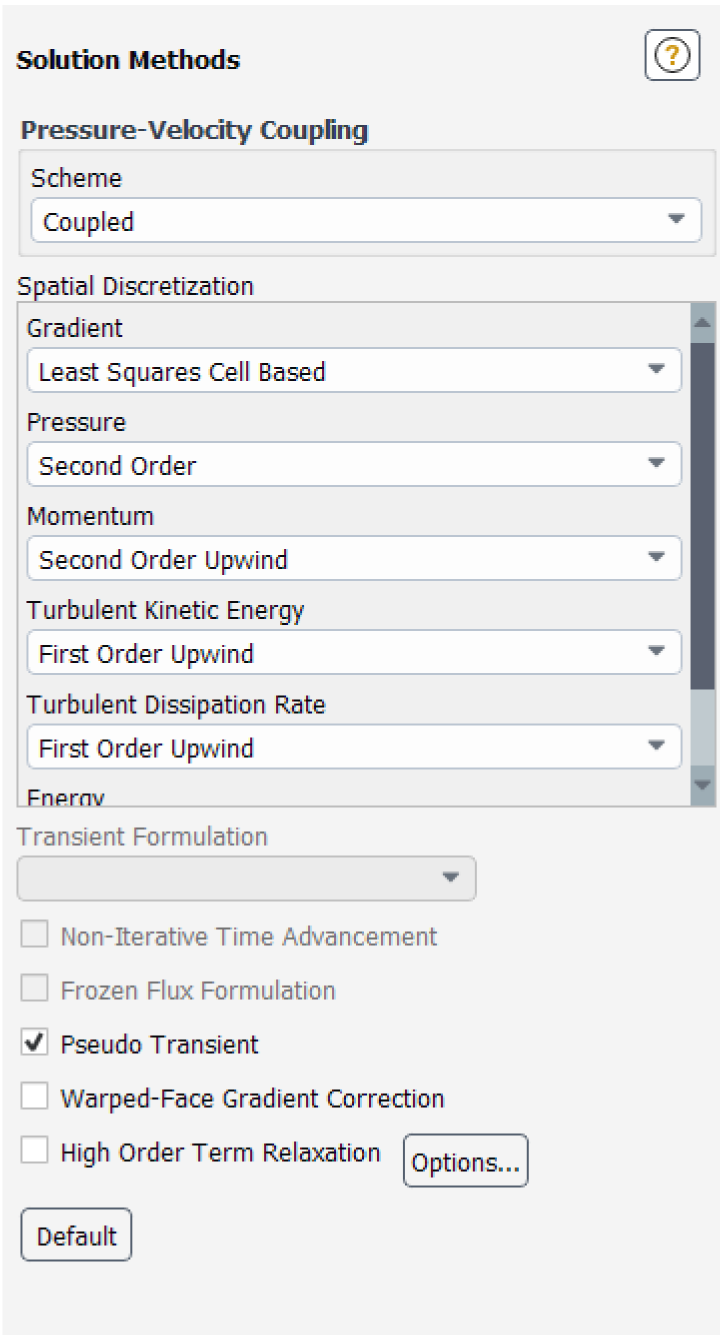
You can also use the task pages listed under the **Solution** tree branch to perform solution-related activities.

---

1. Select a solver scheme.

   a. In the **Solution** ribbon tab, click **Methods...** (**Solution** group box).

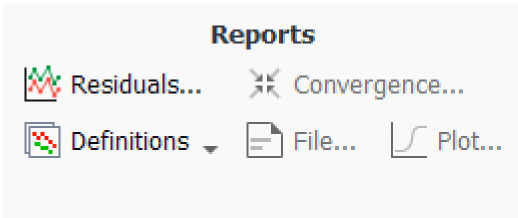
 **Solution → Solution → Methods...**



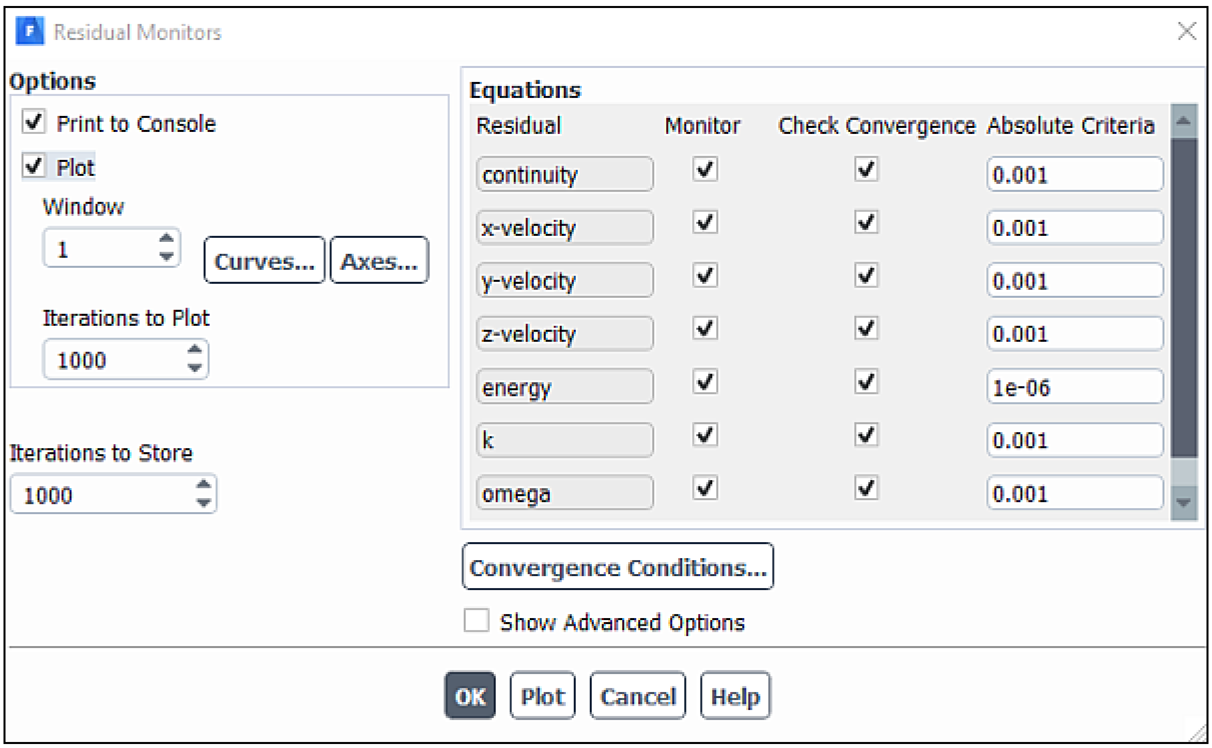
b. Retain the default selections.

2. Enable the plotting of residuals during the calculation.

   a. In the **Solution** ribbon tab, click **Residuals...** (**Reports** group box).



Solution → **Reports** → **Residuals...**



Note:

You can also access the **Residual Monitors** dialog box by double-clicking the **Solution/Monitors/Residual** tree item.

---

b. Ensure that **Plot** is enabled in the **Options** group box.

c. Retain the default value of `0.001` for the **Absolute Criteria** of **continuity**.

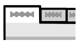
d. Click **OK** to close the **Residual Monitors** dialog box.

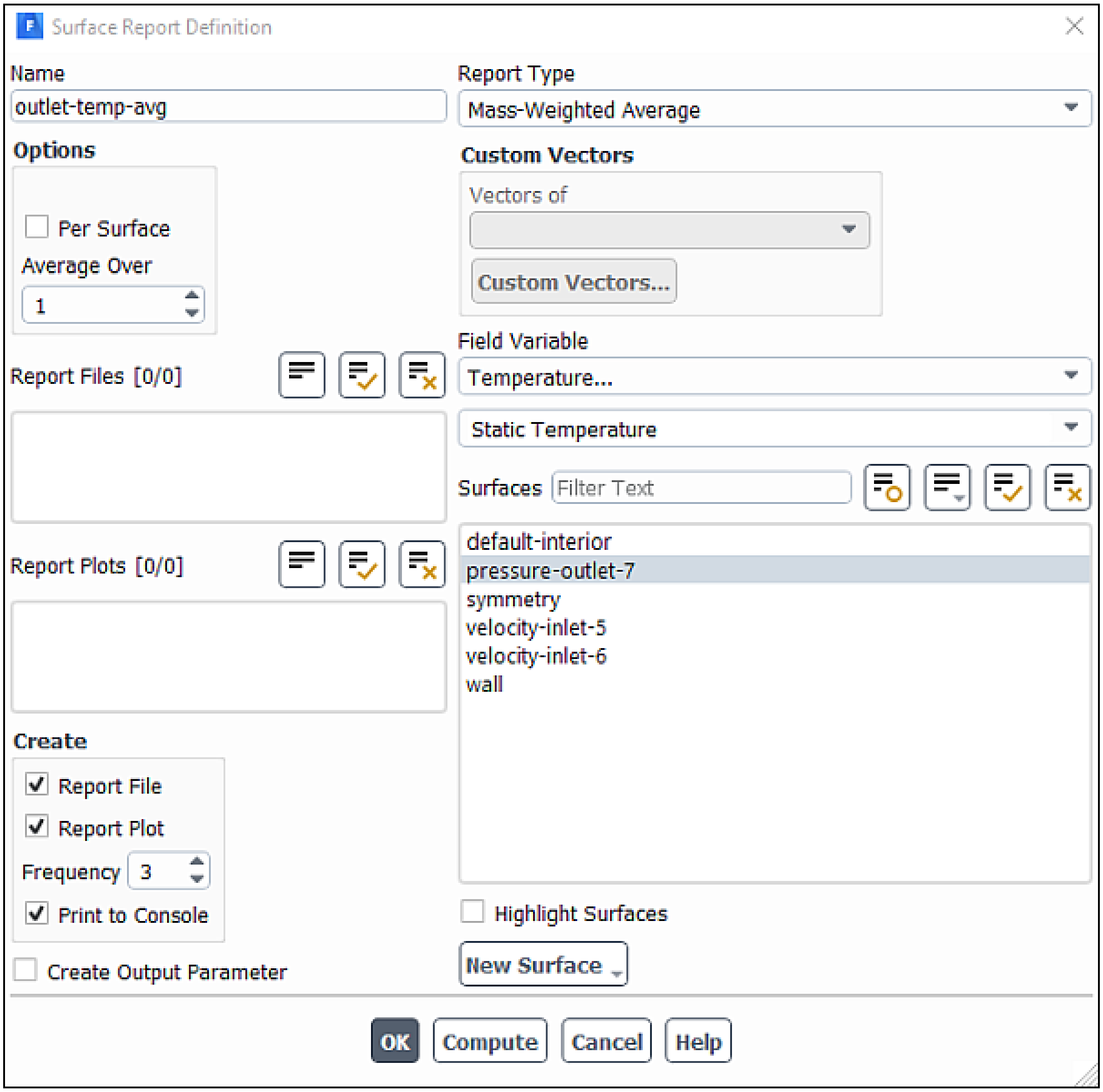
Note:

By default, the residuals of all of the equations solved for the physical models enabled for your case will be monitored and checked by ANSYS Fluent as a means to determine the convergence of the solution. It is a good practice to also create and plot a surface report definition that can help evaluate whether the solution is truly converged. You will do this in the next step.

---

3. Create a surface report definition of average temperature at the outlet (**pressure-outlet-7**).

 **Solution** → **Reports** → **Definitions** → **New** → **Surface Report** → **Mass-Weighted Average...**



Note:

You can also access the **Surface Report Definition** dialog box by right-clicking **Report Definitions** in the tree (under **Solution**) and selecting **New/Surface Report/Mass-Weighted Average...** from the menu that opens.

a. Enter `outlet-temp-avg` for the **Name** of the report definition.

b. Enable **Report File**, **Report Plot**, and **Print to Console** in the **Create** group box.

During a solution run, ANSYS Fluent will write solution convergence data in a report file, plot the solution convergence history in a graphics window, and print the value of the report definition to the console.

c. Set **Frequency** to 3 by clicking the up-arrow button.

This setting instructs ANSYS Fluent to update the plot of the surface report, write data to a file, and print data in the console after every 3 iterations during the solution.

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

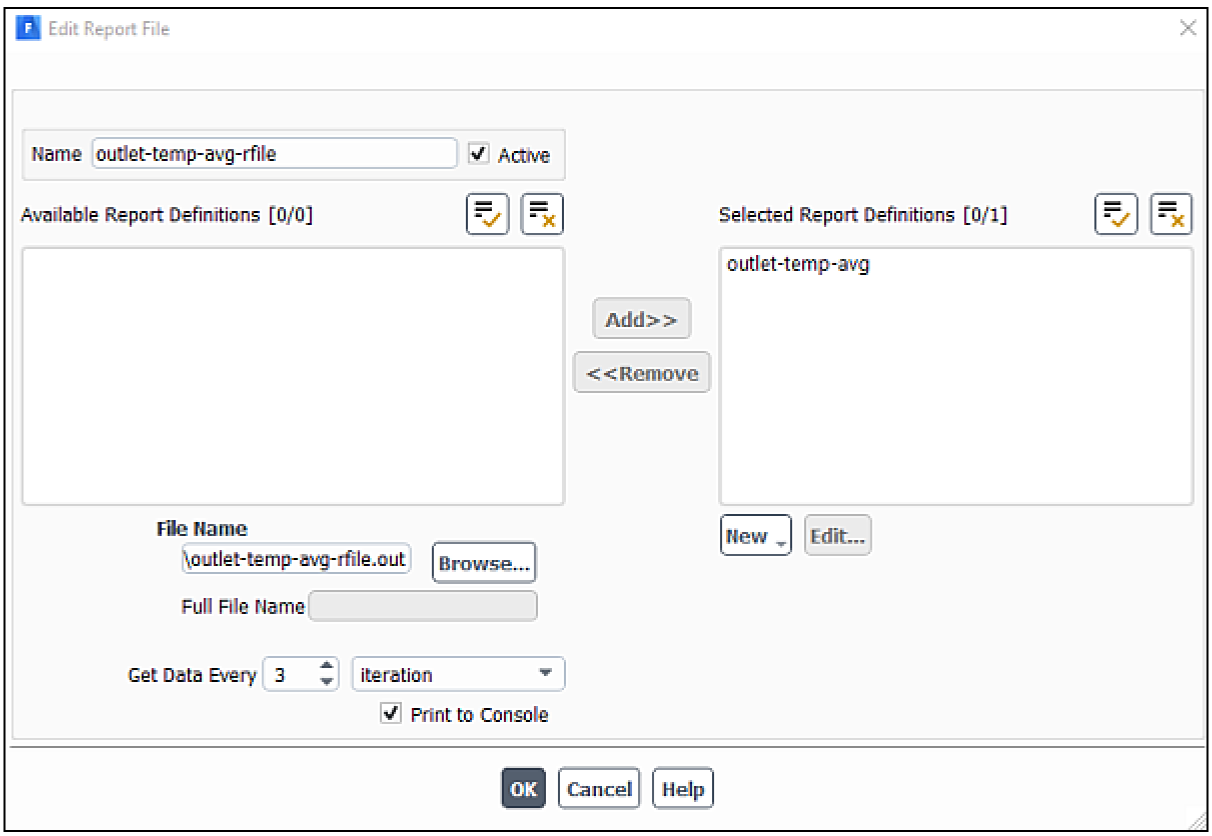
e. Select **pressure-outlet-7** from the **Surfaces** selection list.

f. Click **OK** to save the surface report definition and close the **Surface Report Definition** dialog box.

The new surface report definition **outlet-temp-avg** will appear under the **Solution/Report Definitions** tree item. ANSYS Fluent also automatically creates the following items:

- **outlet-temp-avg-rfile** (under the **Solution/Monitors/Report Files** tree branch)
- **outlet-temp-avg-rplot** (under the **Solution/Monitors/Report Plots** tree branch)

4. In the tree, double-click **outlet-temp-avg-rfile** (under **Solution/Monitors/Report Files**) and examine the report file settings in the **Edit Report File** dialog box.



The dialog box is automatically populated with data from the **outlet-temp-avg** report definition.

a. Verify that **outlet-temp-avg** is in the **Selected Report Definitions** list.

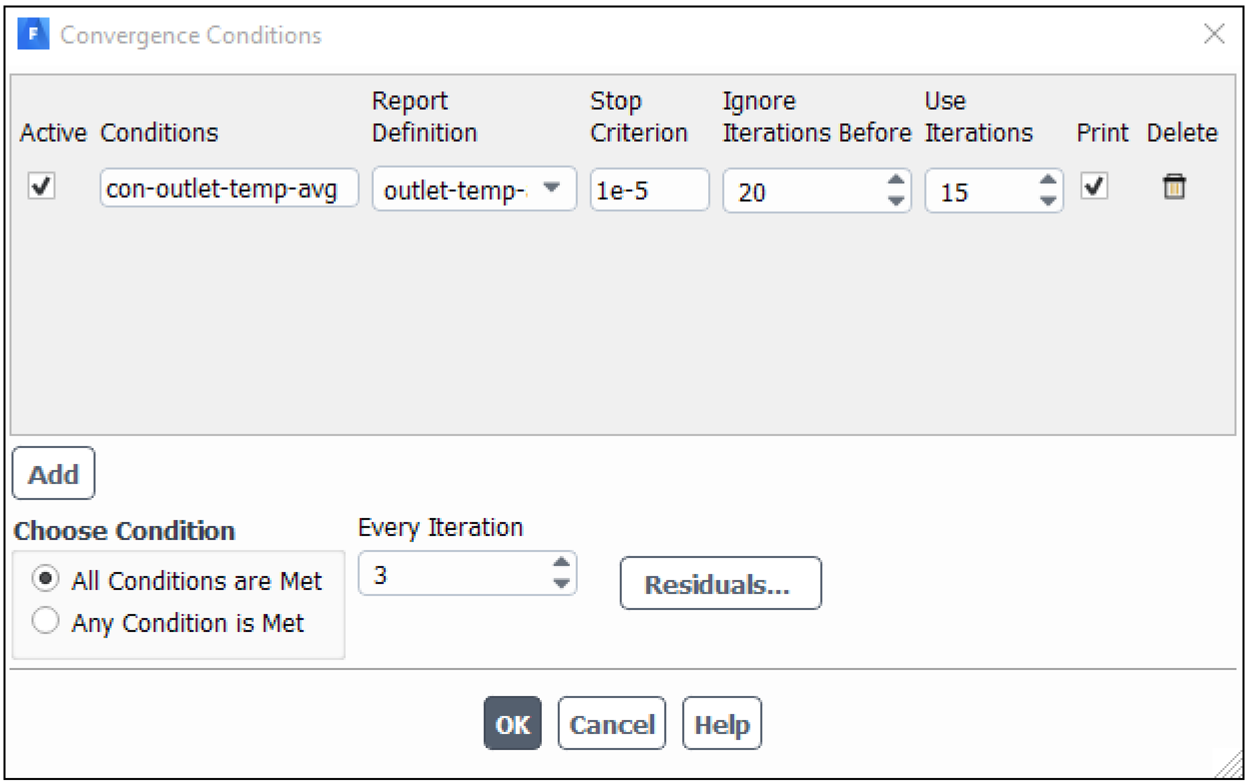
If you had created multiple report definitions, the additional ones would be listed under **Available Report Definitions**, *and you could use the* **Add>>** *and* **<<Remove** *buttons to manage which were written in this particular report definition file.*

b. (optional) Edit the name and location of the resulting file as necessary using the **File Name** field or **Browse...** button.

c. Click **OK** to close the **Edit Report File** dialog box.

5. Create a convergence condition for **outlet-temp-avg**.

Solution → **Reports** → **Convergence...**



a. Click the **Add** button.

b. Enter `con-outlet-temp-avg` for **Conditions**.

c. Select **outlet-temp-avg** from the **Report Definition** drop-down list.

d. Enter `1e-5` for **Stop Criterion**.

e. Enter `20` for **Ignore Iterations Before**.

f. Enter `15` for **Use Iterations**.

g. Enable **Print**.

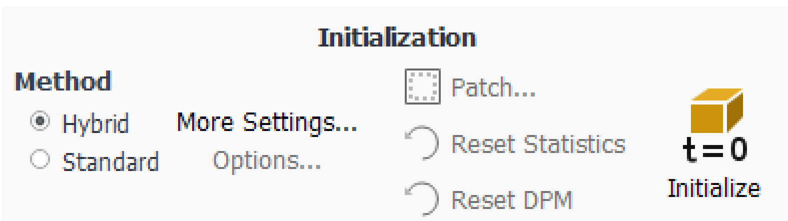
h. Set **Every Iteration** to 3.

i. Click **OK** to save the convergence condition settings and close the **Convergence Conditions** dialog box.

These settings will cause Fluent to consider the solution converged when the surface report definition value for each of the previous 15 iterations is within 0.001% of the current value. Convergence of the values will be checked every 3 iterations. The first 20 iterations will be ignored, allowing for any initial solution dynamics to settle out. Note that the value printed to the console is the deviation between the current and previous iteration values only.

6. Initialize the flow field using the **Initialization** group box of the **Solution** ribbon tab.

Solution → **Initialization**

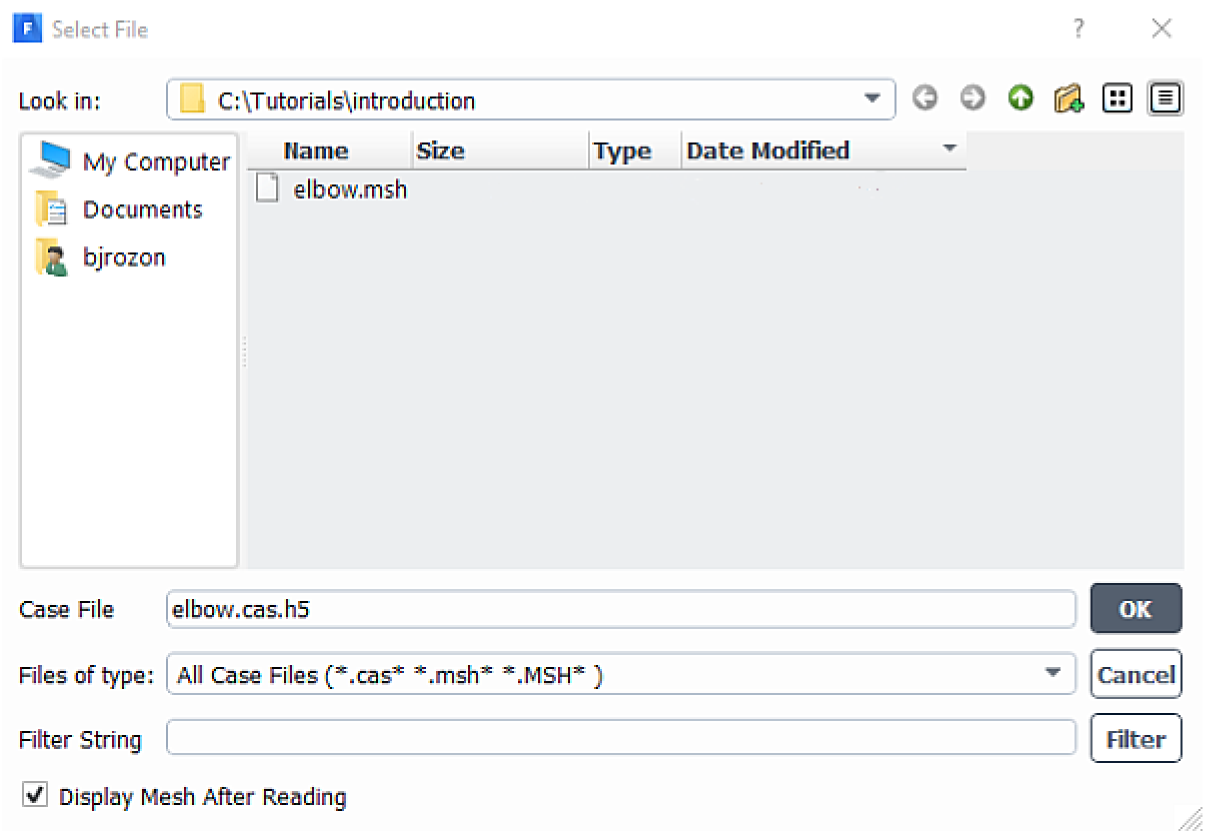


a. Retain the default selection of **Hybrid** from the **Method** list.

b. Click **Initialize**.

7. Save the case file (`elbow1.cas.h5`).

File → **Write** → **Case...**



a. (optional) Indicate the folder in which you would like the file to be saved.

By default, the file will be saved in the folder from which you read in `elbow.msh` *(that is, the* `introduction` *folder). You can indicate a different folder by browsing to it or by creating a new folder.*

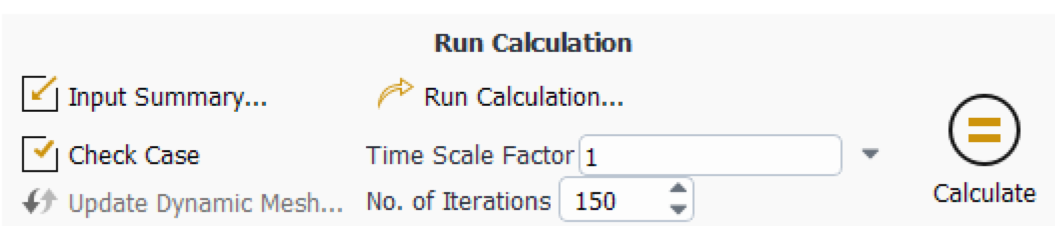
b. Enter `elbow1.cas.h5` for **Case File**.

c. Ensure that the default **Write Binary Files** option is enabled, so that a binary file will be written.

d. Click **OK** to save the case file and close the **Select File** dialog box.

8. Start the calculation by requesting 150 iterations in the **Solution** ribbon tab (**Run Calculation** group box).

Solution → **Run Calculation**



a. Enter `150` for **No. of Iterations**.

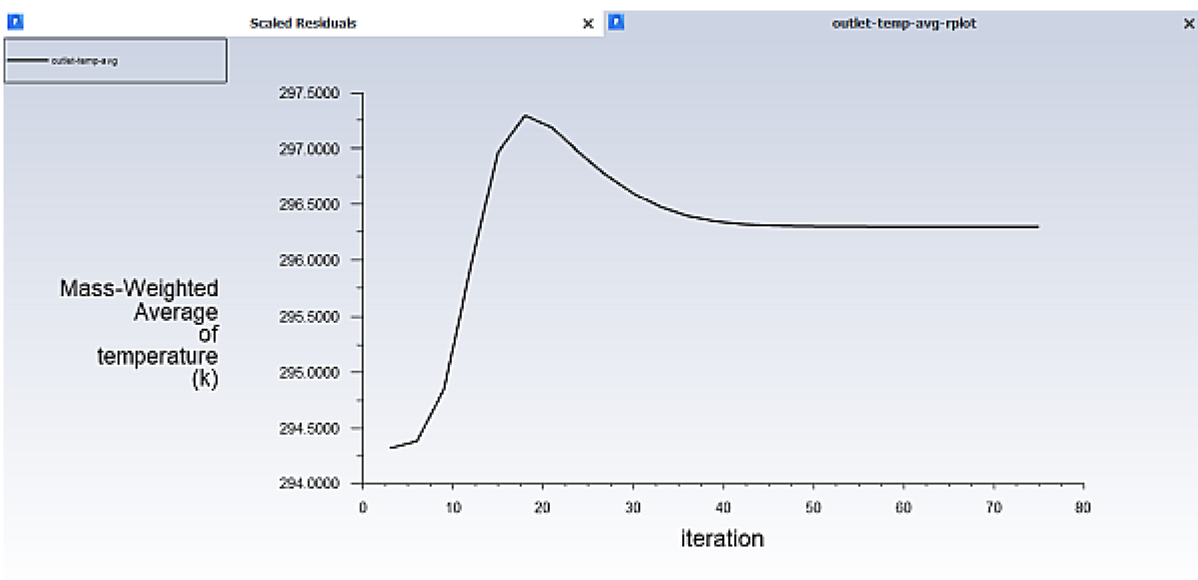
b. Click **Calculate**.

Note:

By starting the calculation, you are also starting to save the surface report data at the rate specified in the **Surface Report Definition** dialog box. If a file already exists in your working directory with the name you specified in the **Edit Report File** dialog box, then a **Question** dialog box will open, asking if you would like to append the new data to the existing file. Click **No** in the **Question** dialog box, and then click **OK** in the **Warning** dialog box that follows to overwrite the existing file.

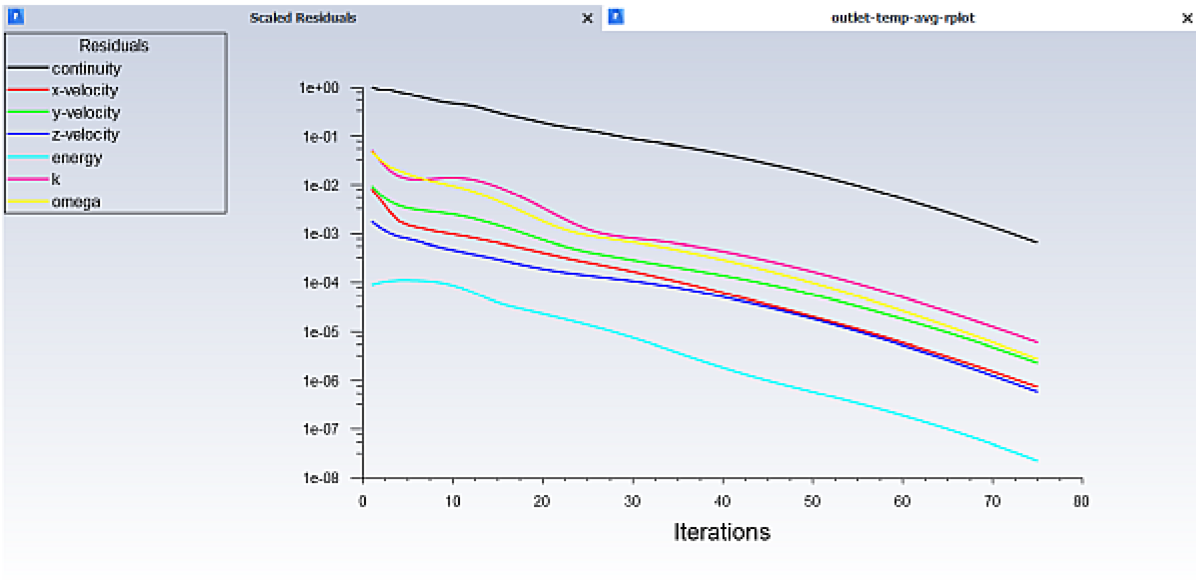
As the calculation progresses, the surface report history will be plotted in the **outlet-temp-avg-rplot** tab in the graphics window (Figure 1.3: Convergence History of the Mass-Weighted Average Temperature (p. 37)).

Figure 1.3: Convergence History of the Mass-Weighted Average Temperature



Similarly, the residuals history will be plotted in the **Scaled Residuals** tab in the graphics window (Figure 1.4: Residuals (p. 38)).

Figure 1.4: Residuals



Note:

You can monitor the two convergence plots simultaneously by right-clicking a tab in the graphics window and selecting **SubWindow View** from the menu that opens. To return to a tabbed graphics window view, right-click a graphics window title area and select **Tabbed View**.

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

The solution will be stopped by ANSYS Fluent when any of the following occur:

- the surface report definition converges to within the tolerance specified in the **Convergence Conditions** dialog box
- the residual monitors converge to within the tolerances specified in the **Residual Monitors** dialog box
- the number of iterations you requested in the **Run Calculation** task page has been reached

In this case, the solution is stopped when the convergence criterion on outlet temperature is satisfied. The exact number of iterations for convergence will vary, depending on the platform being used. An **Information** dialog box will open to alert you that the calculation is complete. Click **OK** in the **Information** dialog box to proceed.

9. Examine the plots for convergence (Figure 1.3: Convergence History of the Mass-Weighted Average Temperature (p. 37) and Figure 1.4: Residuals (p. 38)).

Note:

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

There are three indicators that convergence has been reached:

- The residuals have decreased to a sufficient degree.

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

- The solution no longer changes with more iterations.

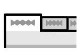
  Sometimes the residuals may not fall below the convergence criterion set in the case setup. However, monitoring the representative flow variables through iterations may show that the residuals have stagnated and do not change with further iterations. This could also be considered as convergence.

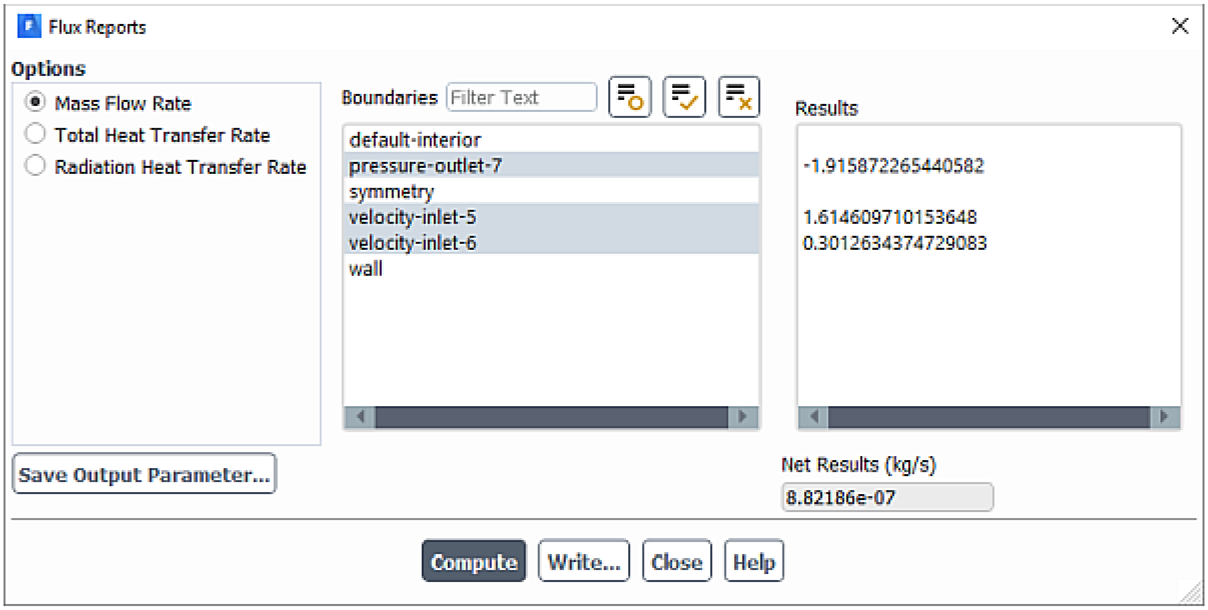
- The overall mass, momentum, energy, and scalar balances are obtained.

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

---

10. Examine the mass flux report for convergence using the **Results** ribbon tab.

 **Results → Reports → Fluxes...**



a. Ensure that **Mass Flow Rate** is selected from the **Options** list.

b. Select **pressure-outlet-7**, **velocity-inlet-5**, and **velocity-inlet-6** from the **Boundaries** selection list.

c. Click **Compute**.

*The individual and net results of the computation will be displayed in the **Results** and **Net Results** boxes, respectively, in the **Flux Reports** dialog box, as well as in the console.*

The sum of the flux for the inlets should be very close to the sum of the flux for the outlets. The net results show that the imbalance in this case is well below the 0.2% criterion suggested previously.

d. Close the **Flux Reports** dialog box.

11. Save the data file (`elbow1.dat.h5`).

File → Write → Data...

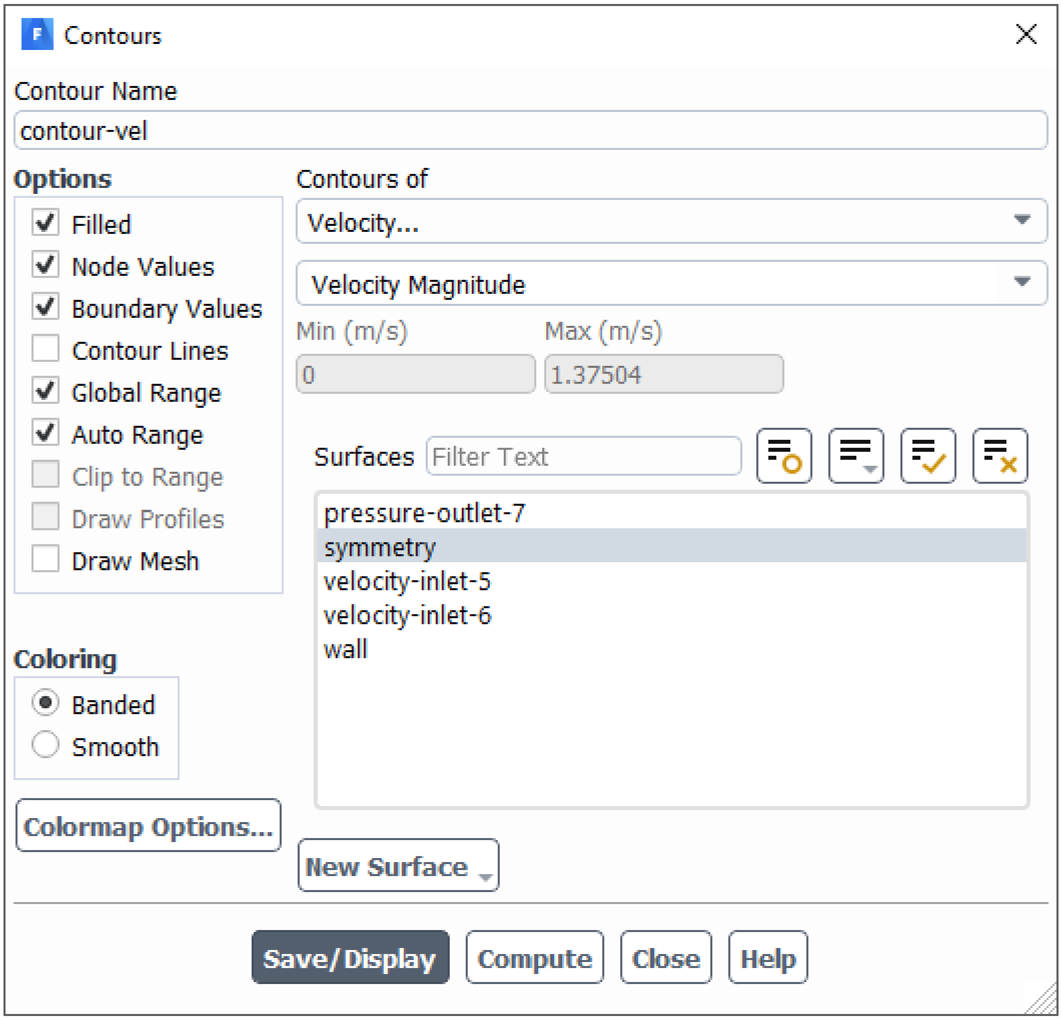
In later steps of this tutorial you will save additional case and data files with different suffixes.

1.4.7. Displaying the Preliminary Solution

In the steps that follow, you will visualize various aspects of the flow for the preliminary solution using the **Results** ribbon tab.

1. Display filled contours of velocity magnitude on the symmetry plane (Figure 1.5: Predicted Velocity Distribution after the Initial Calculation (p. 43)).

Results → Graphics → Contours → New...



a. Enter `contour-vel` for **Contour Name**.

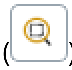
b. Enable **Filled** in the **Options** group box.

c. Ensure that **Node Values** and **Boundary Values** are enabled in the **Options** group box.

d. Select **Banded** in the **Coloring** group box.

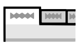
e. Select **Velocity...** and **Velocity Magnitude** from the **Contours of** drop-down lists.

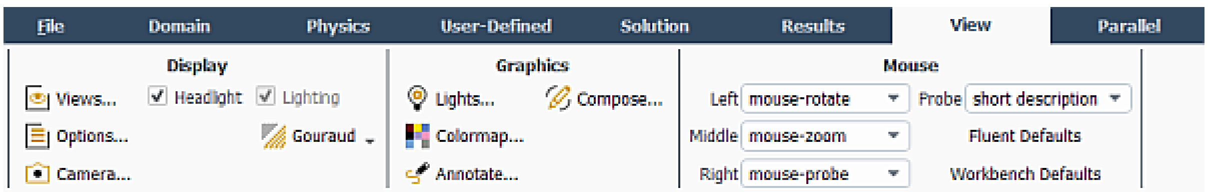
f. Select **symmetry** from the **Surfaces** selection list.

g. Click **Save/Display** to display the contours in the active graphics window. Clicking the **Fit to Window** icon () will cause the object to fit exactly and be centered in the window.

> **Note:**
>
> If you cannot see the velocity contour display, select the appropriate tab in the graphics window.

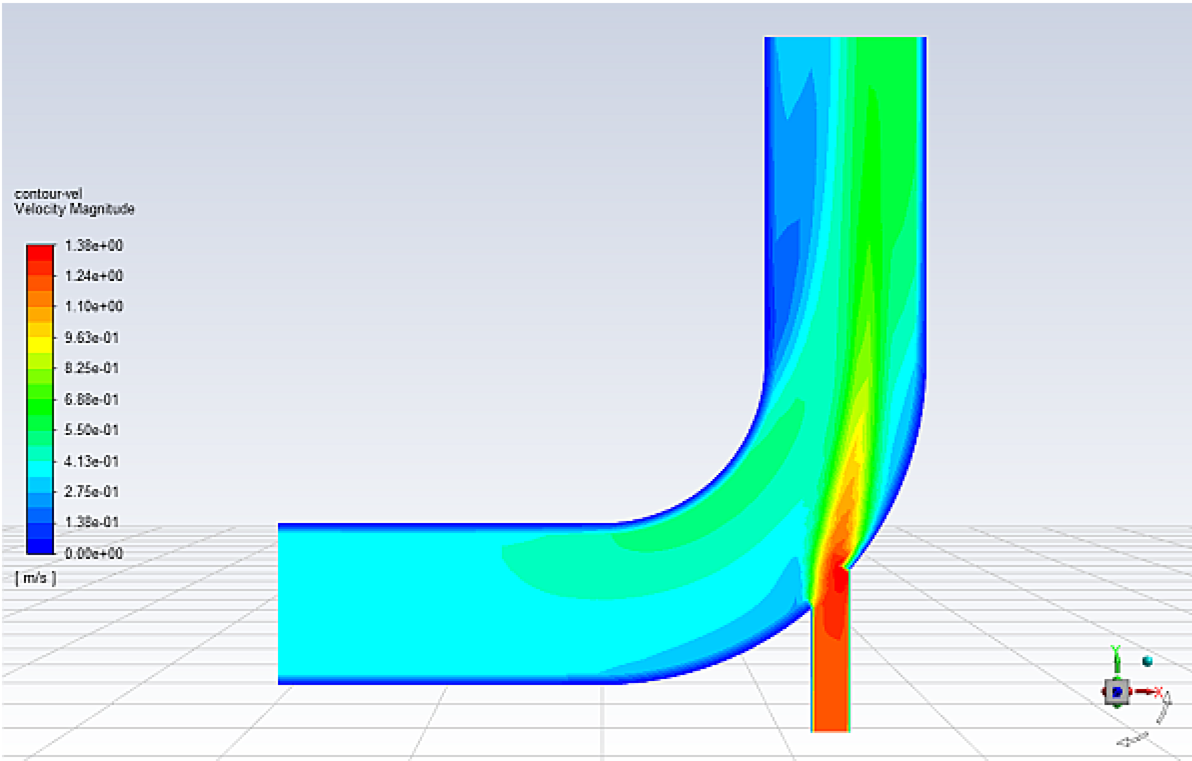
h. Close the **Contours** dialog box.

 **View** → **Display**



Disable the **Headlight** and **Lighting** options.

Figure 1.5: Predicted Velocity Distribution after the Initial Calculation

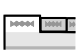


Extra:

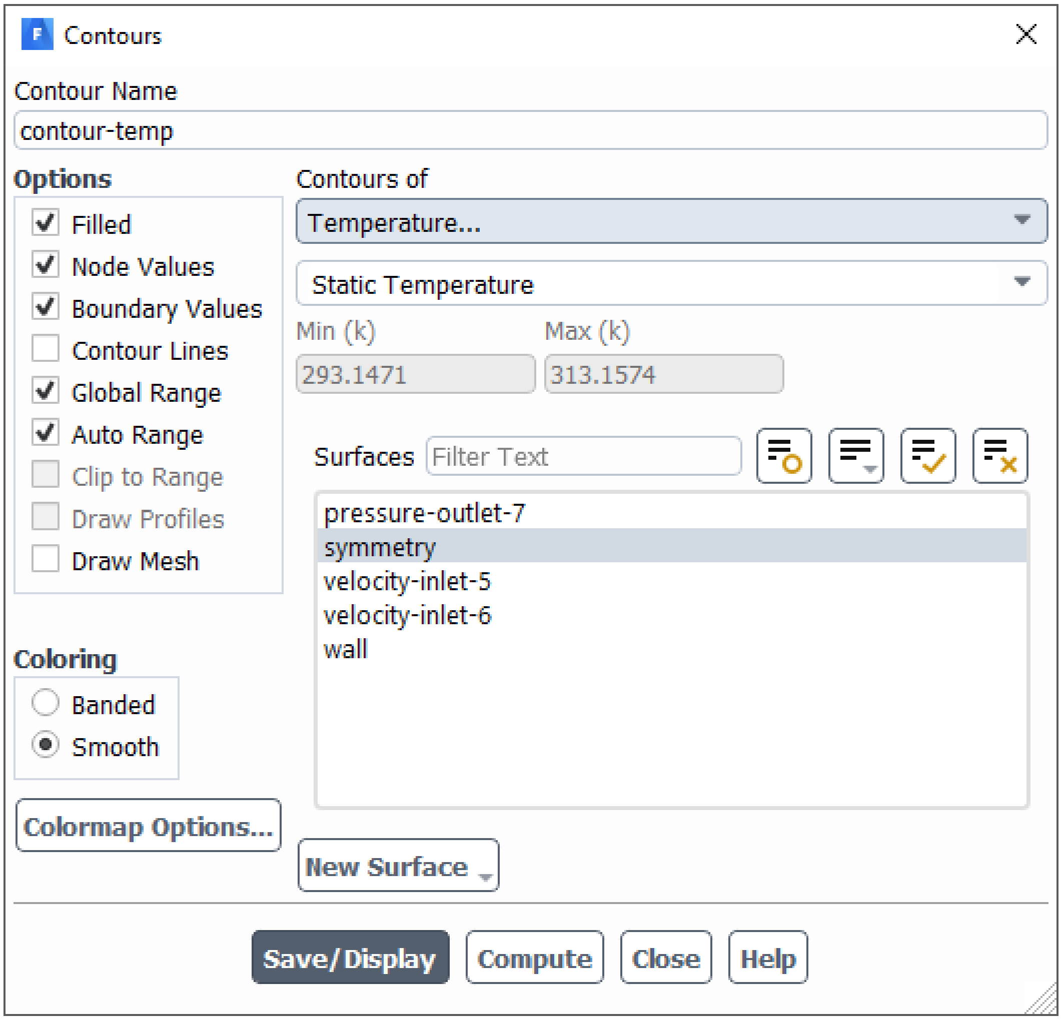
When you probe a point in the displayed domain with the right mouse button or the probe tool, the level of the corresponding contour is highlighted in the colormap in the graphics window, and is also reported in the console.

---

2. Create and display a definition for temperature contours on the symmetry plane (Figure 1.6: Predicted Temperature Distribution after the Initial Calculation (p. 45)).

 **Results** → **Graphics** → **Contours** → **New...**

You can create contour definitions and save them for later use.



a. Enter `contour-temp` for **Contour Name**.

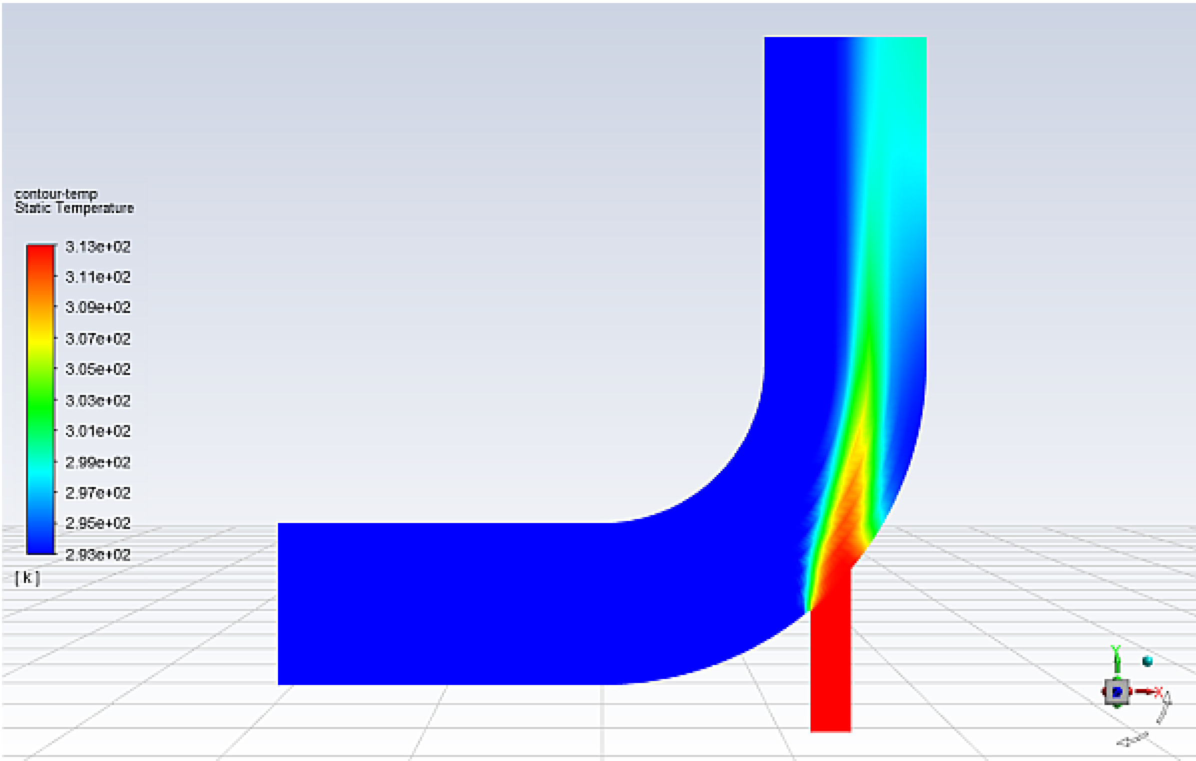
b. Select **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.

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

d. Click **Save/Display** and close the **Contours** dialog box.

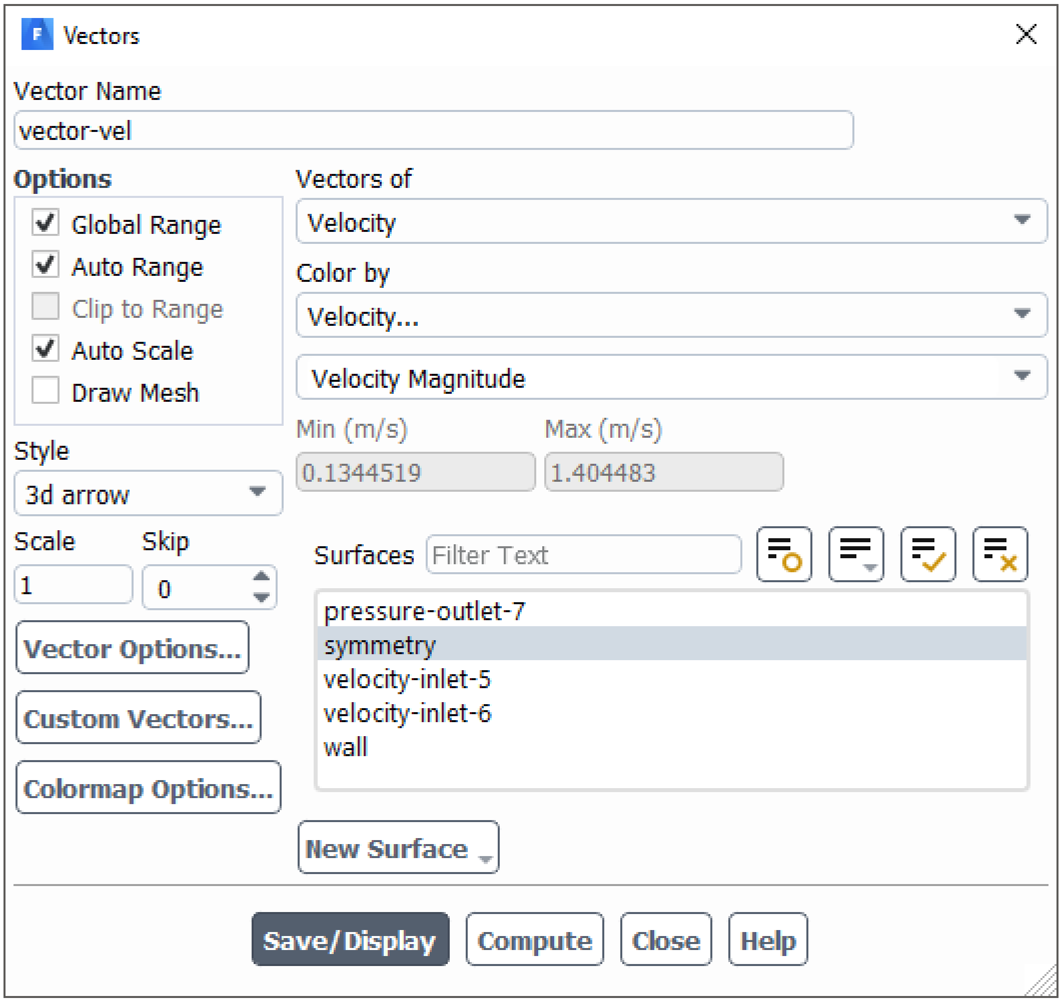
   The new **contour-temp** definition appears under the **Results/Graphics/Contours** tree branch. To edit your contour definition, right-click it and select **Edit...** from the menu that opens.

Figure 1.6: Predicted Temperature Distribution after the Initial Calculation



3. Display velocity vectors on the symmetry plane (Figure 1.9: Magnified View of Resized Velocity Vectors (p. 49)).

   Results → Graphics → Vectors → New...



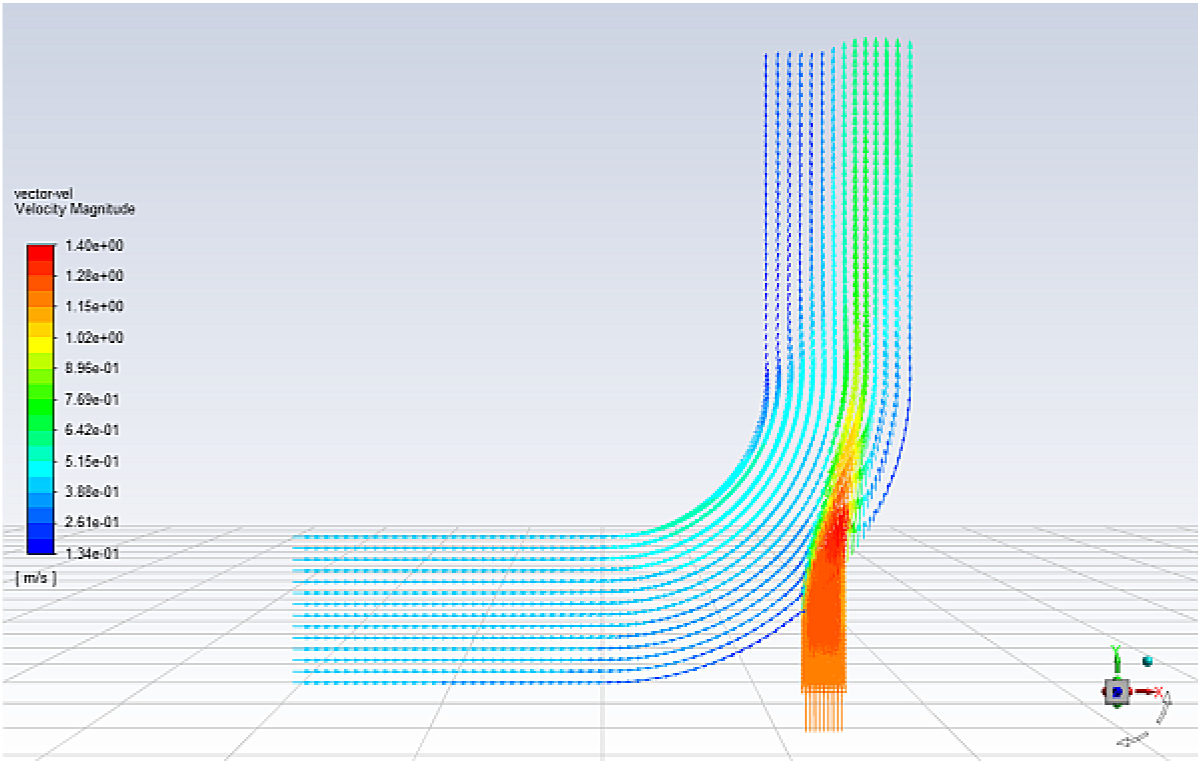
a. Enter `vector-vel` for **Vector Name**.

b. Select **arrow** from the **Style** drop-down box.

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

d. Click **Save/Display** to plot the velocity vectors.

Figure 1.7: Velocity Vectors Colored by Velocity Magnitude



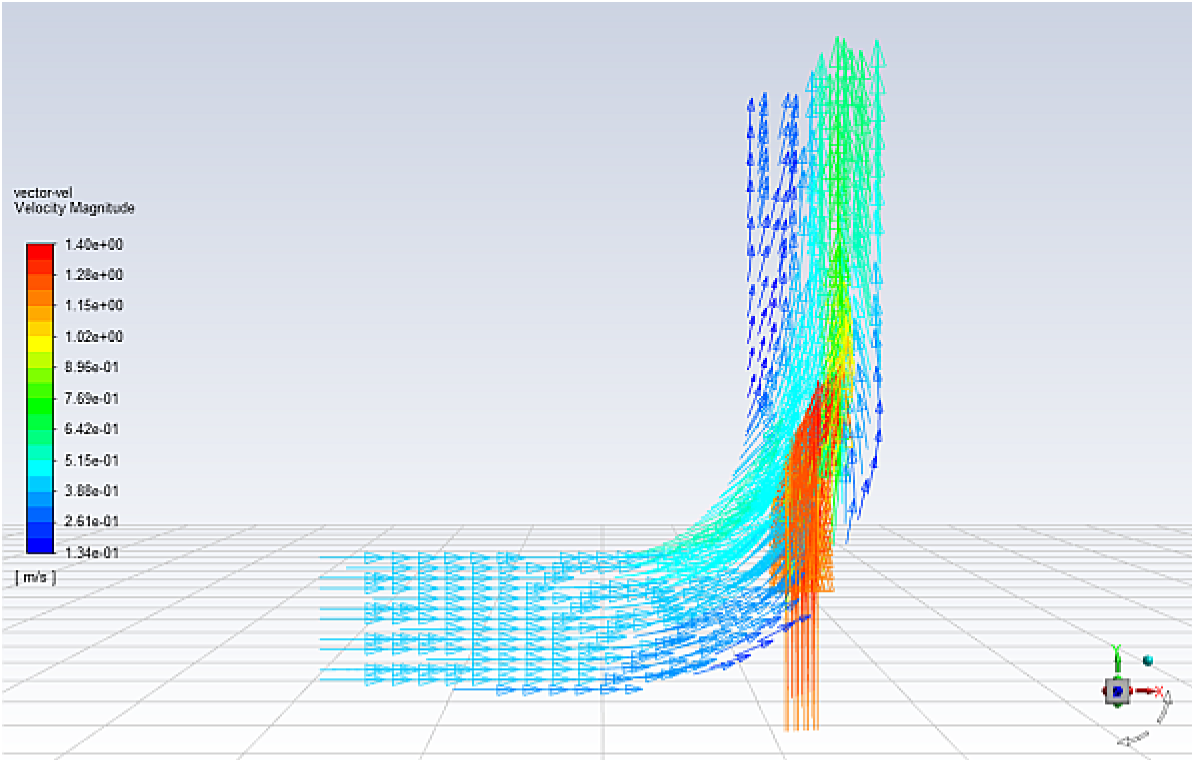
The **Auto Scale** option is enabled by default in the **Options** group box. This scaling sometimes creates vectors that are too small or too large in the majority of the domain. You can improve the clarity by adjusting the **Scale** and **Skip** settings, thereby changing the size and number of the vectors when they are displayed.

e. Enter 4 for **Scale**.

f. Set **Skip** to 2.

g. Click **Save/Display** again to redisplay the vectors.

Figure 1.8: Resized Velocity Vectors

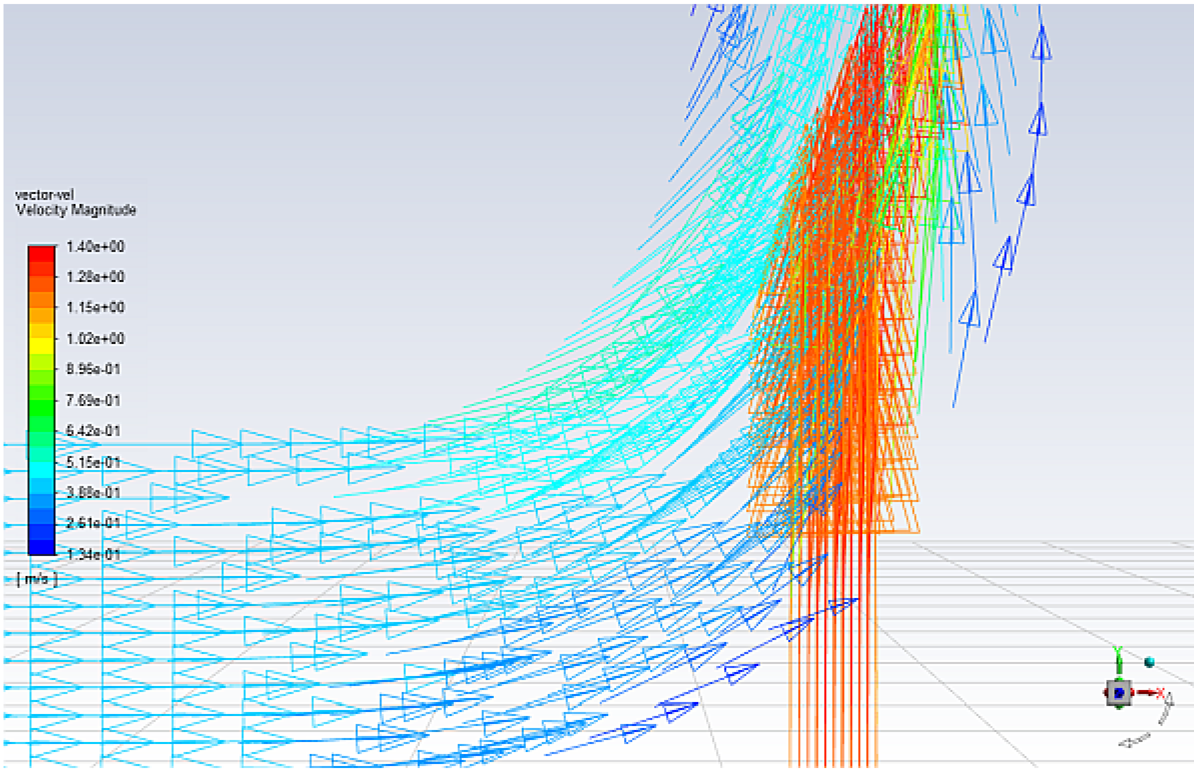


h. Close the **Vectors** dialog box.

i. Zoom in on the vectors in the display.

To manipulate the image, refer to Table 1.1: View Manipulation Instructions (p. 8). The image will be redisplayed at a higher magnification (Figure 1.9: Magnified View of Resized Velocity Vectors (p. 49)).

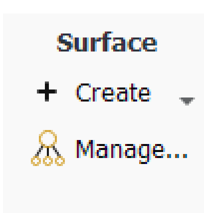
Figure 1.9: Magnified View of Resized Velocity Vectors

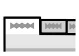


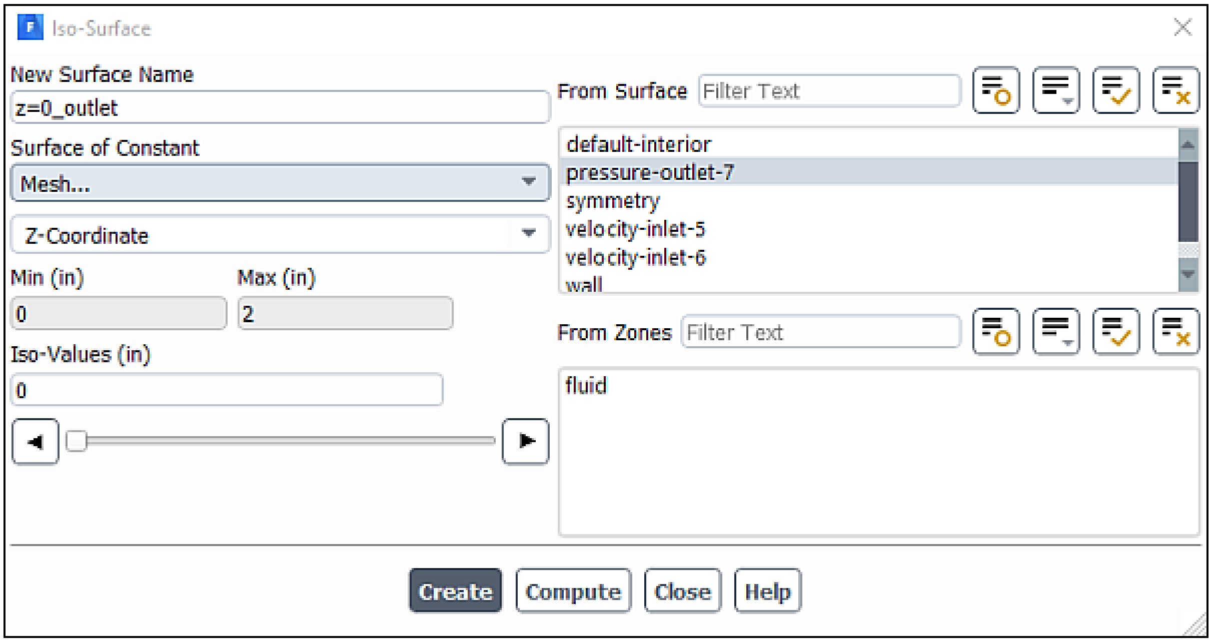
j. Zoom out to the original view.

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

4. Create a line at the centerline of the outlet. For this task, you will use the **Surface** group box of the **Results** tab.



 **Results** → **Surface** → **Create** → **Iso-Surface...**



a. Enter `z=0_outlet` for **New Surface Name**.

b. Select **Mesh...** and **Z-Coordinate** from the **Surface of Constant** drop-down lists.

c. Click **Compute** to obtain the extent of the mesh in the z-direction.

   The range of values in the z-direction is displayed in the **Min** and **Max** fields.

d. Retain the default value of `0` inches for **Iso-Values**.

e. Select `pressure-outlet-7` from the **From Surface** selection list.

f. Click **Create**.

   The new line surface representing the intersection of the plane z=0 and the surface `pressure-outlet-7` is created, and its name **z=0_outlet** appears in the **From Surface** selection list.

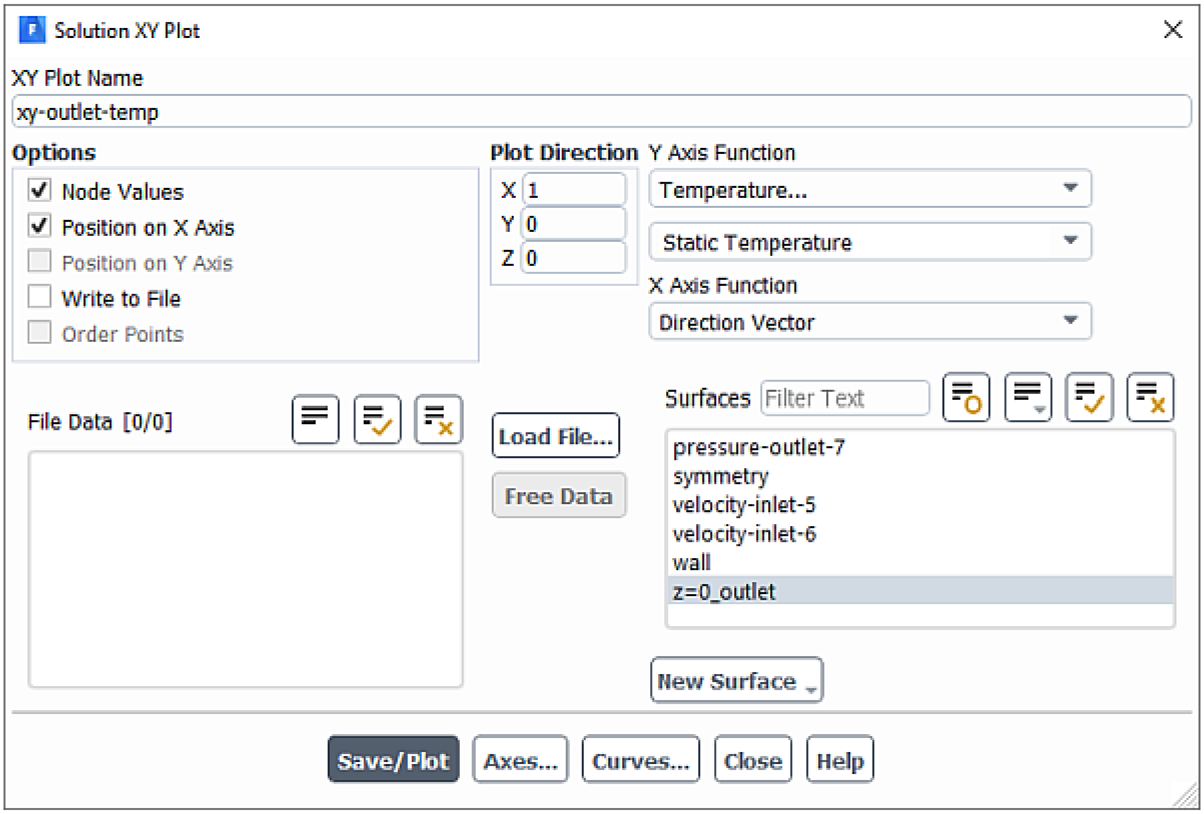
   Note:

   - After the line surface **z=0_outlet** is created, a new entry will automatically be generated for **New Surface Name**, in case you would like to create another surface.
   - If you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the **Surfaces** dialog box.

g. Close the **Iso-Surface** dialog box.

5. Display and save an XY plot of the temperature profile across the centerline of the outlet for the initial solution (Figure 1.10: Outlet Temperature Profile for the Initial Solution (p. 52)).

Results → **Plots** → **XY Plot** → **New...**



a. Enter `xy-outlet-temp` for **XY Plot Name**.

b. Select **Temperature...** and **Static Temperature** from the **Y Axis Function** drop-down lists.

c. Select the **z=0_outlet** surface you just created from the **Surfaces** selection list.

d. Click **Save/Plot**.

e. Enable **Write to File** in the **Options** group box.

   The button that was originally labeled ***Save/Plot*** *will change to* ***Write...****.*

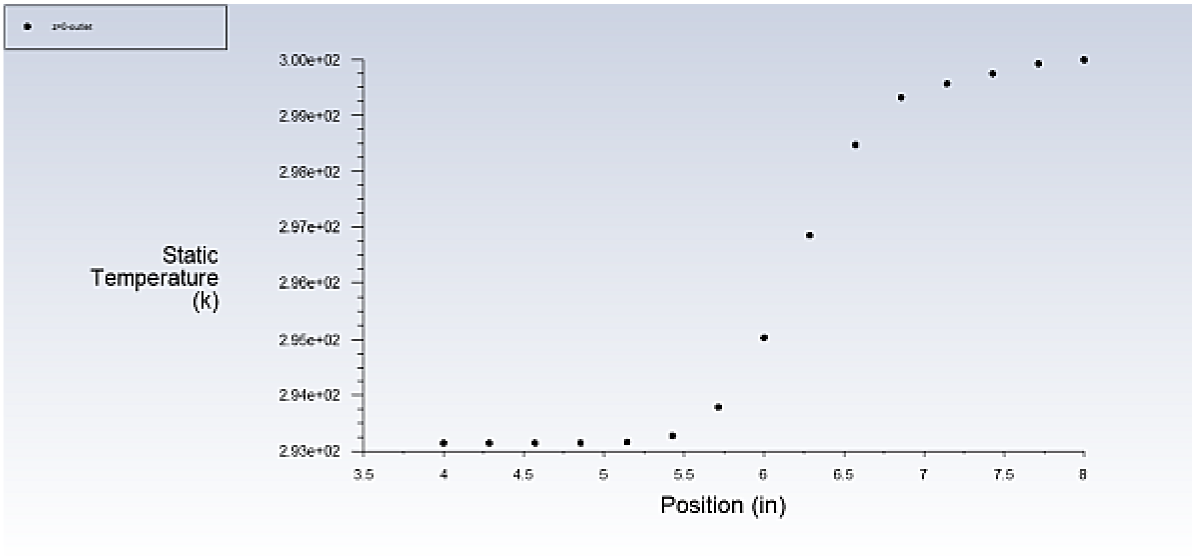
f. Click **Write...**.

   i. In the **Select File** dialog box, enter `outlet_temp1.xy` for **XY File**.

   ii. Click **OK** to save the temperature data and close the **Select File** dialog box.

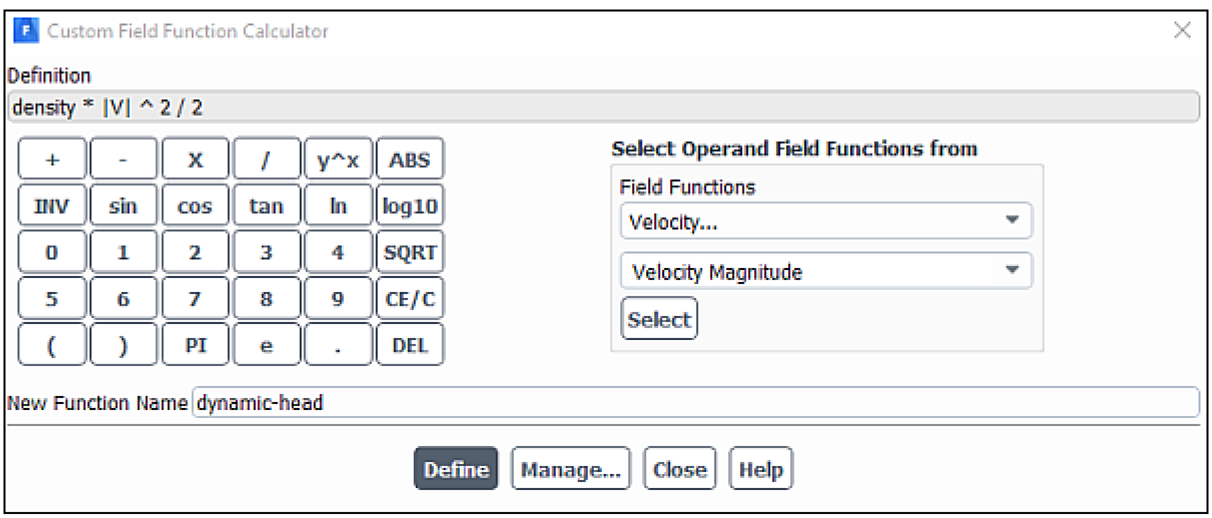
g. Close the **Solution XY Plot** dialog box.

Figure 1.10: Outlet Temperature Profile for the Initial Solution



6. Define a custom field function for the dynamic head formula ($\rho \cdot |V|^2 / 2$).

User-Defined → **Field Functions** → **Custom...**



a. Select **Density...** and **Density** from the **Field Functions** drop-down lists, and click the **Select** button to add **density** to the **Definition** field.

b. Click the **X** button to add the multiplication symbol to the **Definition** field.

c. Select **Velocity...** and **Velocity Magnitude** from the **Field Functions** drop-down lists, and click the **Select** button to add **|V|** to the **Definition** field.

d. Click **y^x** to raise the last entry in the **Definition** field to a power, and click **2** for the power.

e. Click the **/** button to add the division symbol to the **Definition** field, and then click **2**.

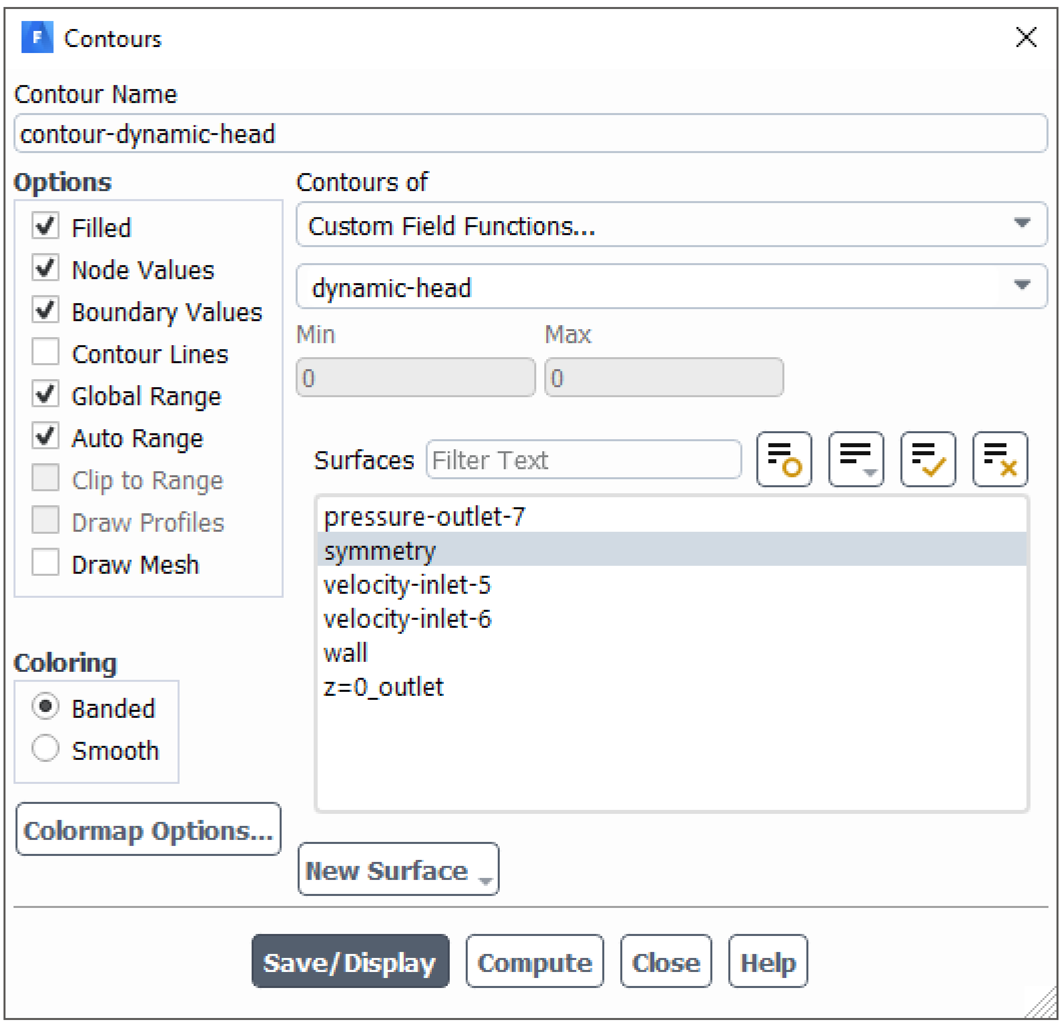
f. Enter `dynamic-head` for **New Function Name**.

g. Click **Define** and close the **Custom Field Function Calculator** dialog box.

The **dynamic-head** tree item will appear under the **Parameters & Customization/Custom Field Functions** tree branch.

7. Display filled contours of the custom field function (Figure 1.11: Contours of the Dynamic Head Custom Field Function (p. 54)).

Results → **Graphics** → **Contours** → **New...**



a. Enter `contour-dynamic-head` for **Contour Name**.

b. Select **Banded** in the **Coloring** group box.

c. Select **Custom Field Functions...** and **dynamic-head** from the **Contours of** drop-down lists.

Tip:

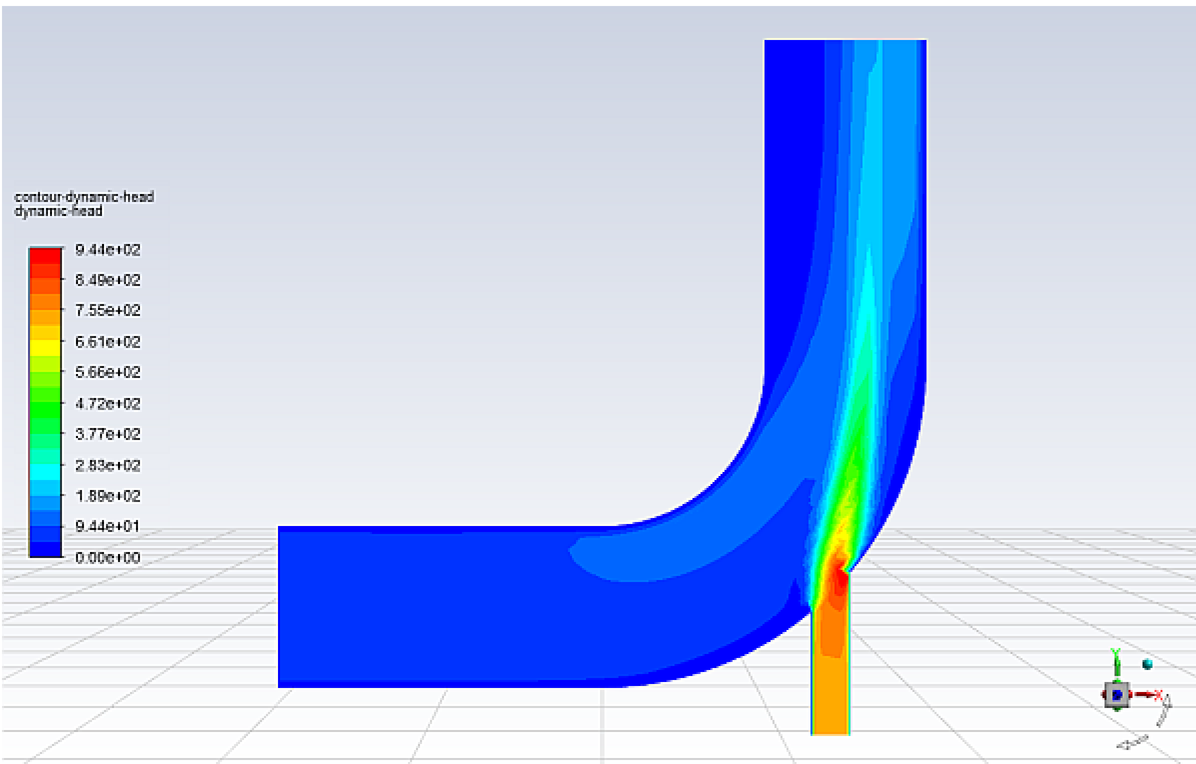
Custom Field Functions... is at the top of the upper **Contours of** drop-down list.

---

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

e. Click **Save/Display** and close the **Contours** dialog box.

Figure 1.11: Contours of the Dynamic Head Custom Field Function



Note:

You may need to change the view by zooming out after the last vector display, if you have not already done so.

---

8. Save the settings for the custom field function by writing the case and data files (`elbow1.cas.h5` and `elbow1.dat.h5`).

File → **Write** → **Case & Data...**

a. Ensure that `elbow1.cas.h5` is entered for **Case/Data File**.

Note:

When you write the case and data file at the same time, it does not matter whether you specify the file name with a `.cas.h5` or `.dat.h5` extension, as both will be saved.

---

b. Click **OK** to save the files and close the **Select File** dialog box.

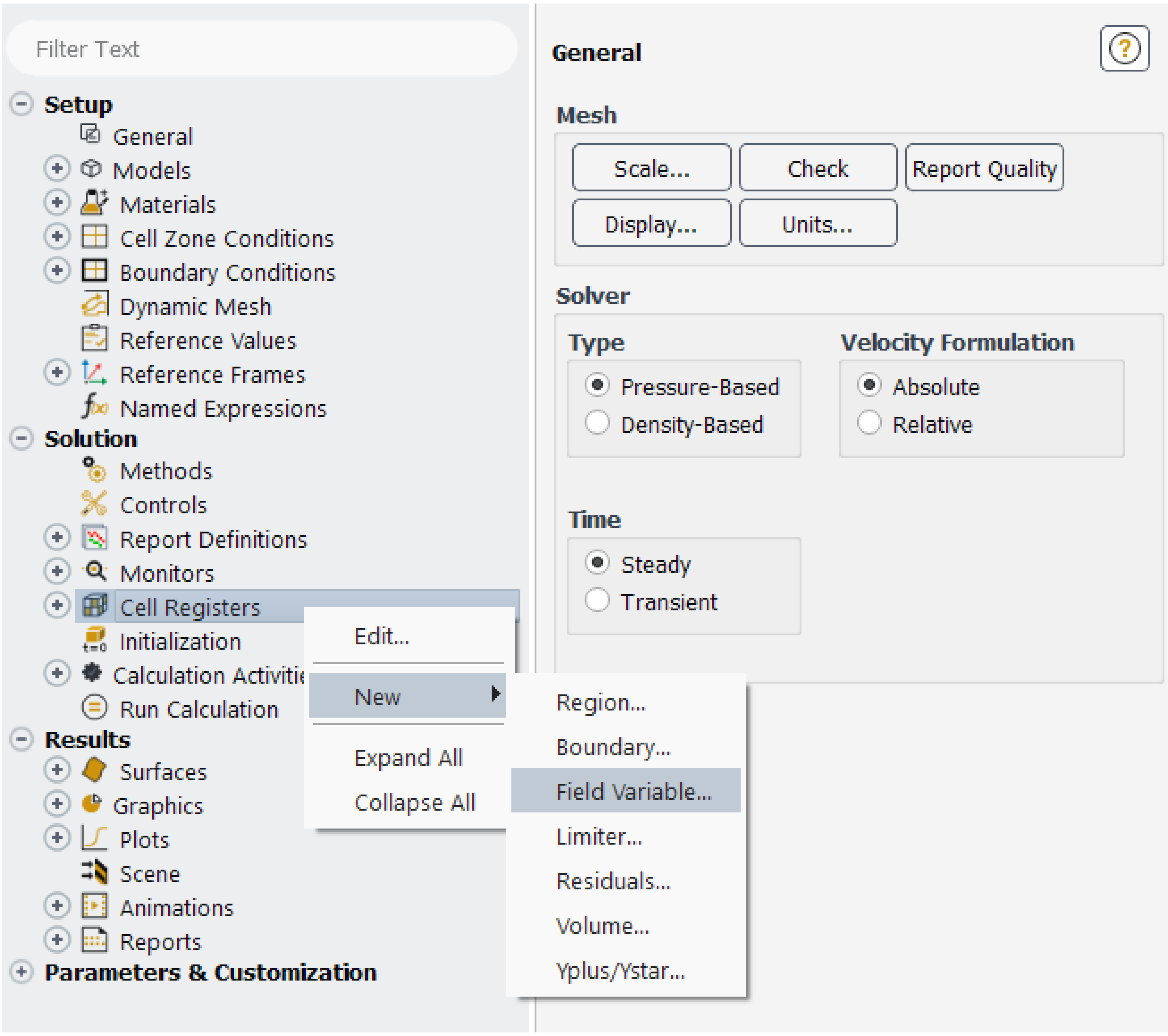
c. Click **OK** to overwrite the files that you had saved earlier.

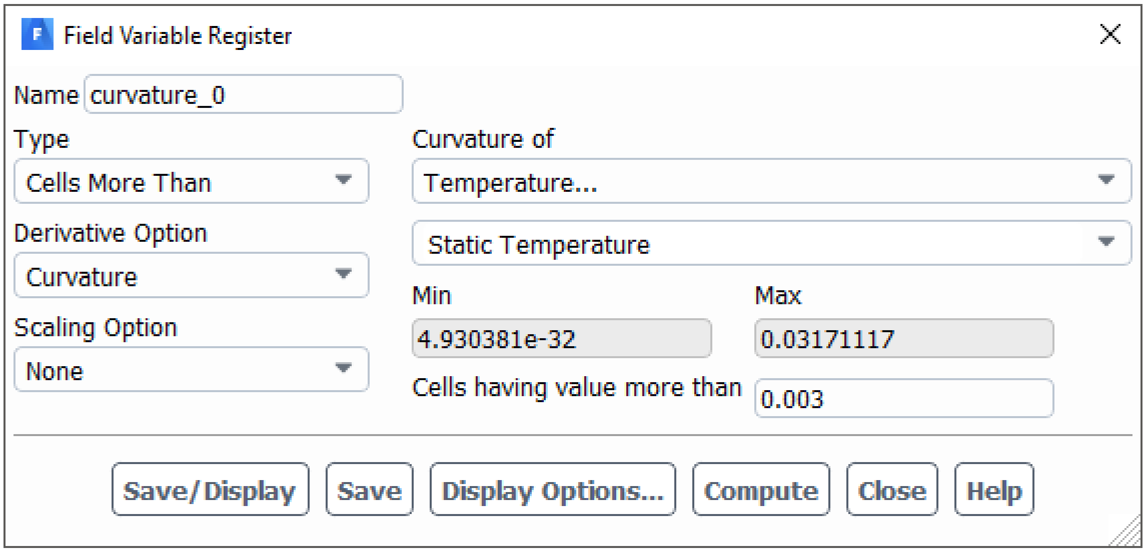
1.4.8. Adapting the Mesh

For the first run of this tutorial, you have solved the elbow problem using a fairly coarse mesh. The elbow solution can be improved further by refining the mesh to better resolve the flow details. ANSYS Fluent provides a built-in capability to easily adapt (locally refine) the mesh according to solution gradients. In the following steps you will adapt the mesh based on the temperature gradients in the current solution and compare the results with the previous results.

1. Define **Cell Registers** to Adapt the mesh in the regions of high temperature gradient.

Solution → **Cell Registers** → **New** → **Field Variable...**


Filter Text
Setup
General
Models
Materials
Cell Zone Conditions
Boundary Conditions
Dynamic Mesh
Reference Values
Reference Frames
Named Expressions
Solution
Methods
Controls
Report Definitions
Monitors
Cell Registers
Initialization
Calculation Activiti
Run Calculation
Results
Surfaces
Graphics
Plots
Scene
Animations
Reports
Parameters & Customization
Edit...
New
Expand All
Collapse All
Region...
Boundary...
Field Variable...
Limiter...
Residuals...
Volume...
Yplus/Ystar...
General
Mesh
Scale...
Check
Report Quality
Display...
Units...
Solver
Type
Pressure-Based
Density-Based
Velocity Formulation
Absolute
Relative
Time
Steady
Transient



a. Select **Cells More Than** from the **Type** drop-down list.

b. Select **Curvature** from the **Derivative Option** drop-down list.

c. Select **Temperature...** and **Static Temperature** from the **Curvature of** drop-down list.

d. Click **Compute**.

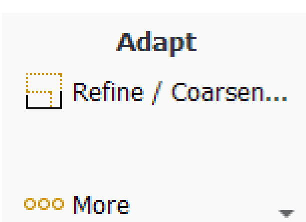
   *ANSYS Fluent will update the **Min** and **Max** values to show the minimum and maximum temperature gradient.*

e. Enter a value of `0.003` for the **Cells having value more than**.

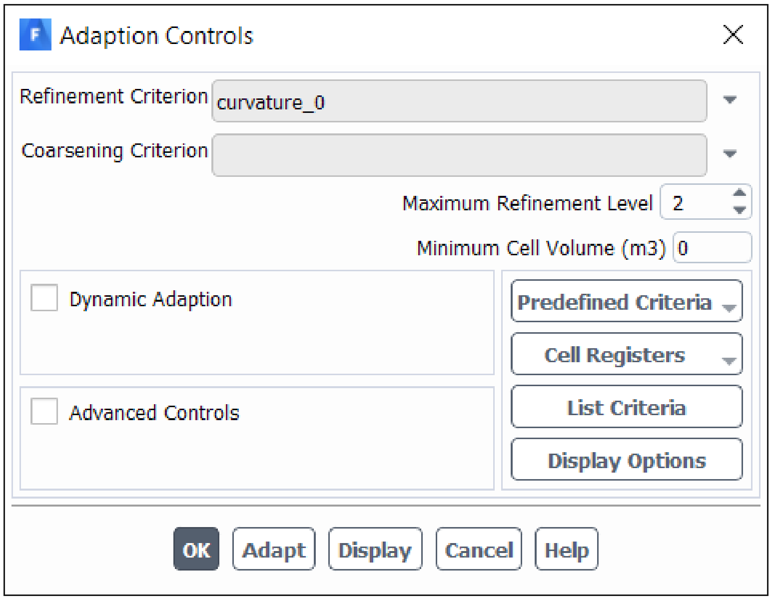
   A general rule is to use 10% of the maximum gradient when setting the value for refinement.

f. Click **Save** and close the **Field Variable Register** daialog box.

2. Setup mesh adaption using the **Cell Registers**. For this task, you will use the **Adapt** group box in the **Domain** ribbon tab.



Domain → **Adapt** → **Refine / Coarsen...**



a. Select the previously defined **curvature_0** cell register from the **Refinement Criterion** drop-down list.

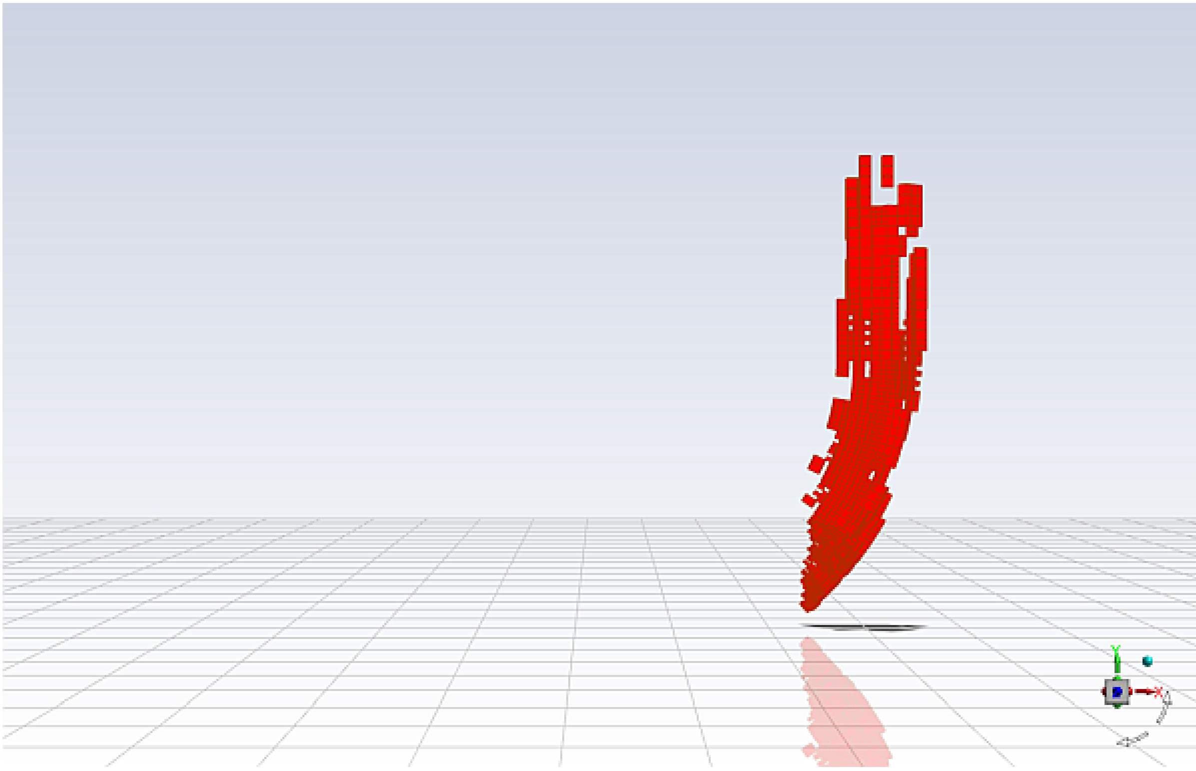
   *ANSYS Fluent will not coarsen beyond the original mesh for a 3D mesh. Hence, it is not necessary to select the **Coarsening Criterion** in this instance.*

b. Click **Adapt**.

c. Click **Display**.

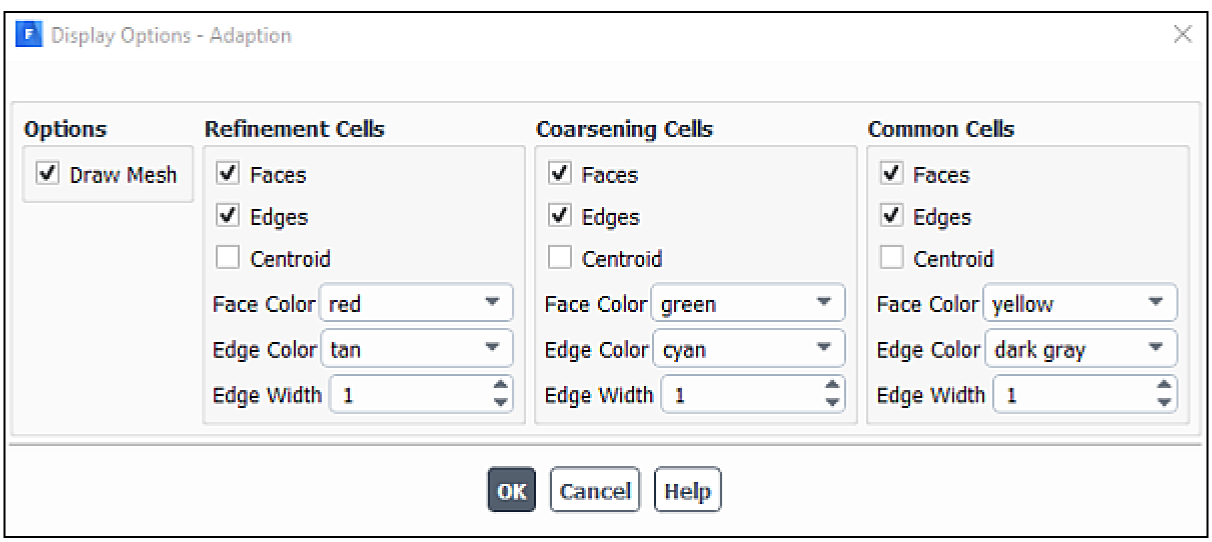
   ANSYS Fluent will display the cells marked for adaption in the graphics window (Figure 1.12: Cells Marked for Adaption (p. 59)).

Figure 1.12: Cells Marked for Adaption



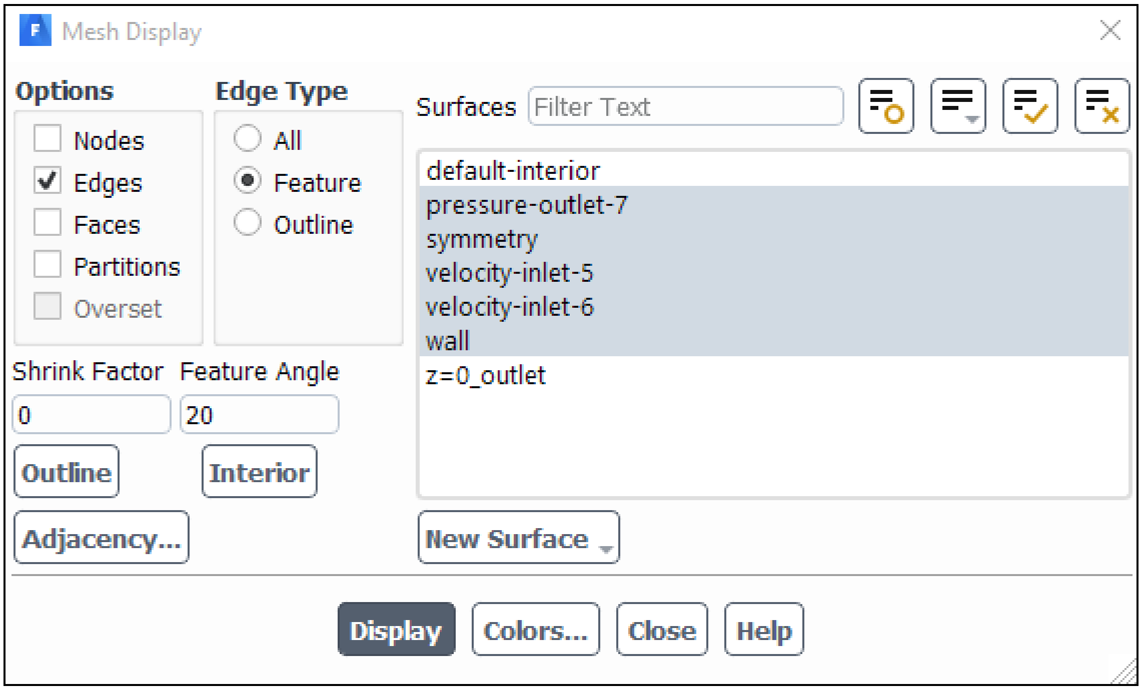
Extra You can change the way ANSYS Fluent displays cells marked for adaption (Figure 1.13: Alternative Display of Cells Marked for Adaption (p. 61)) by performing the following steps:

i. Click **Display Options...** in the **Adaption Controls** dialog box to open the **Display Options - Adaption** dialog box.



ii. Enable **Draw Mesh** in the **Options** group box.

*The **Mesh Display** dialog box will open.*



iii. Ensure that only the **Edges** option is enabled in the **Options** group box.

iv. Select **Feature** from the **Edge Type** list.

v. Select all of the items except **z=0_outlet** from the **Surfaces** selection list.

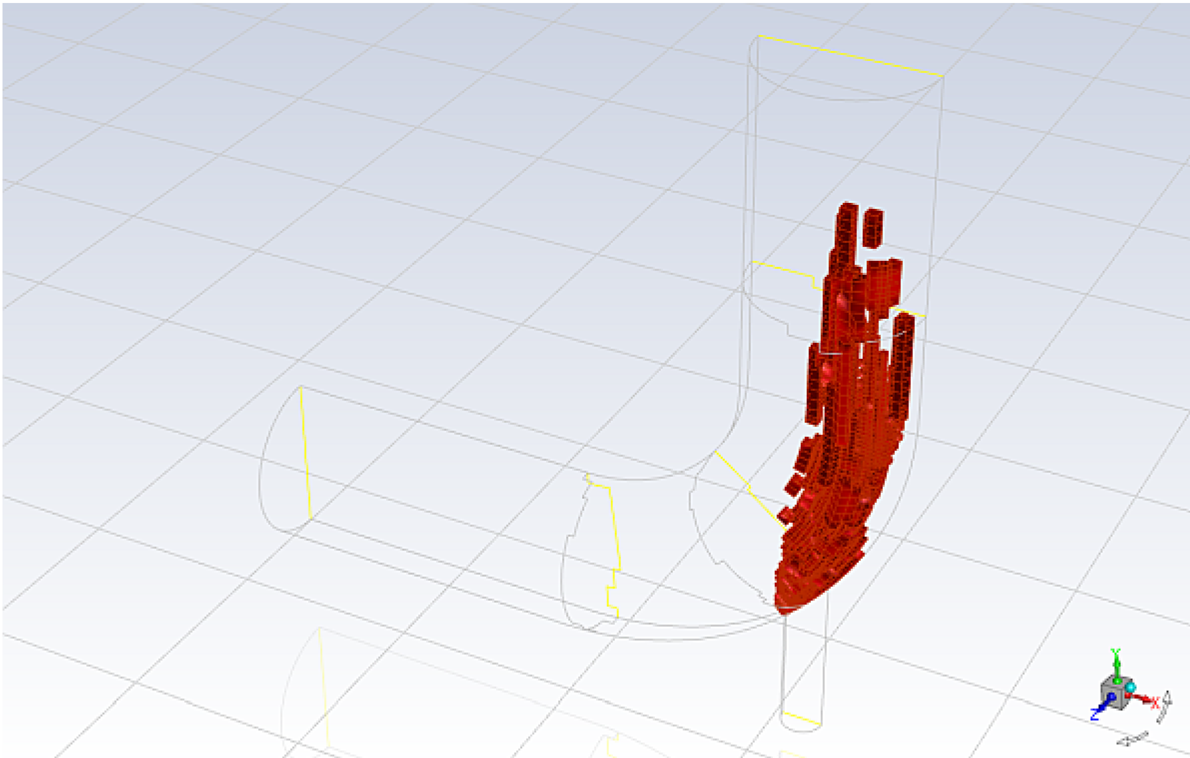
vi. Click **Display** and close the **Mesh Display** dialog box.

vii. Click **OK** to close the **Display Options - Adaption** dialog box.

viii. Click **Display** in the **Adaption Controls** dialog box.

ix. Rotate the view and zoom in to get the display shown in Figure 1.13: Alternative Display of Cells Marked for Adaption (p. 61).

Figure 1.13: Alternative Display of Cells Marked for Adaption

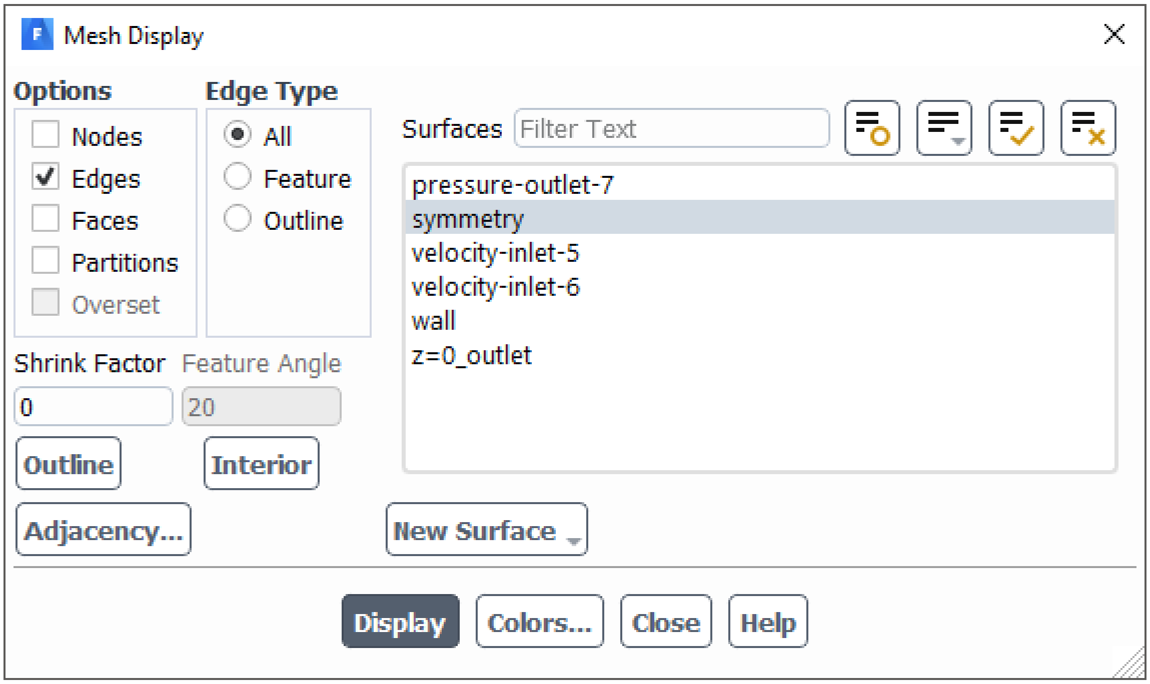


x. After viewing the marked cells, rotate the view back and zoom out again.

xi. Click **OK** to close the **Adaption Controls** dialog box.

3. Display the adapted mesh (Figure 1.14: The Adapted Mesh (p. 63)).

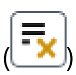
Domain → Mesh → Display...



a. Select **All** from the **Edge Type** list.

b. Deselect all of the highlighted items from the **Surfaces** selection list except for **symmetry**.

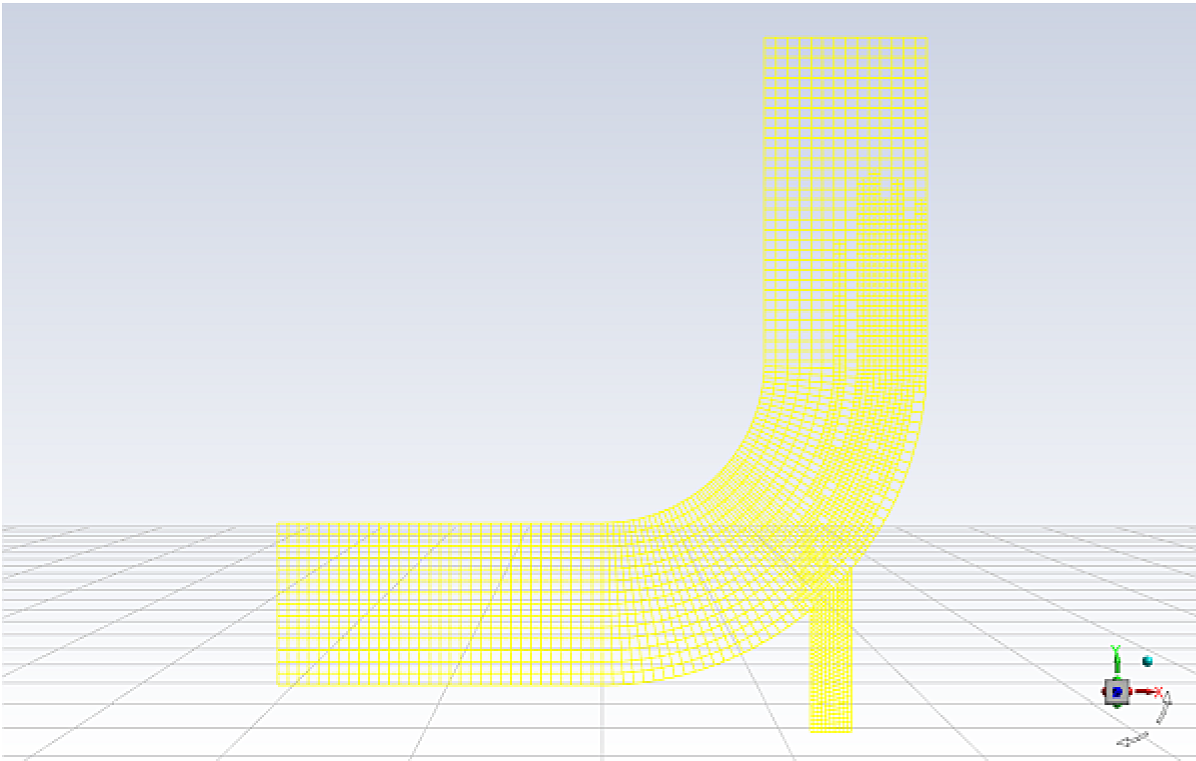
Tip:

To deselect all surfaces, click the **Deselect All Shown** button () at the top of the **Surfaces** selection list. Then select the desired surface from the **Surfaces** selection list.

---

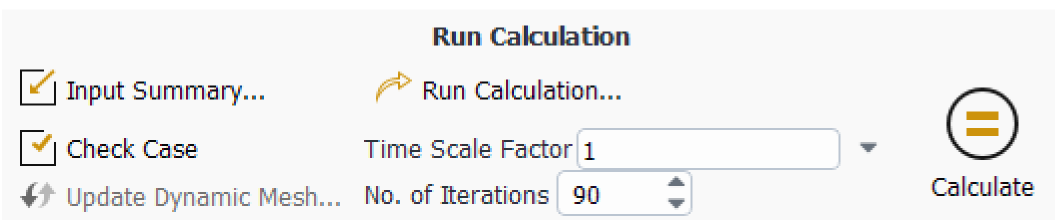
c. Click **Display** and close the **Mesh Display** dialog box.

Figure 1.14: The Adapted Mesh



4. Request an additional 90 iterations.

   Solution → **Run Calculation** → **Calculate**



The solution will converge as shown in Figure 1.15: The Complete Residual History (p. 64) and Figure 1.16: Convergence History of Mass-Weighted Average Temperature (p. 64).

Figure 1.15: The Complete Residual History

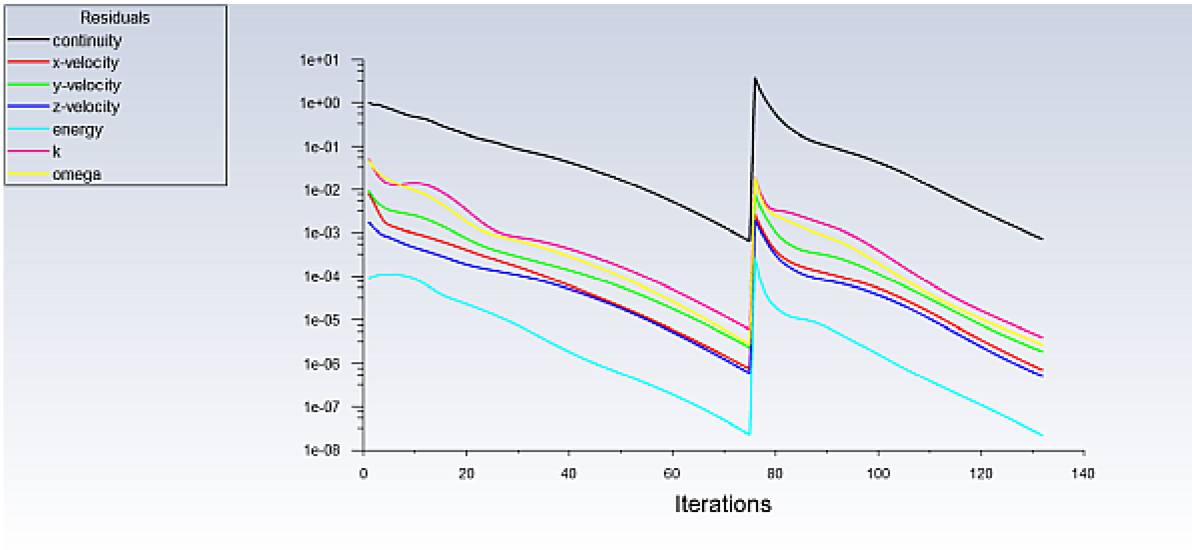
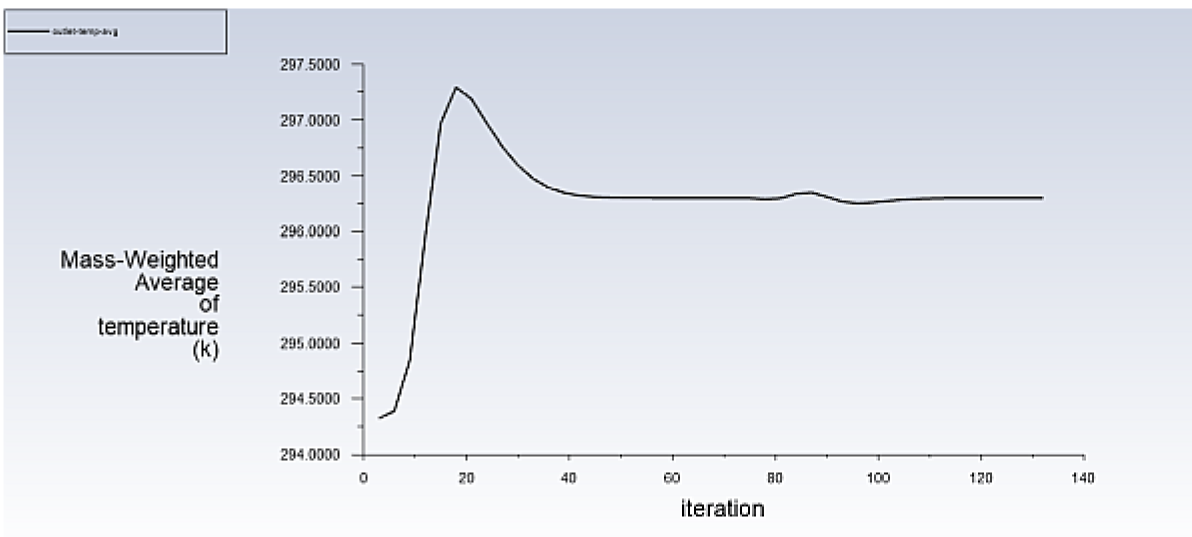


Figure 1.16: Convergence History of Mass-Weighted Average Temperature



5. Save the case and data files for the Coupled solver solution with an adapted mesh (`elbow2.cas.h5` and `elbow2.dat.h5`).

   File → Write → Case & Data...

   a. Enter `elbow2.h5` for **Case/Data File**.

   b. Click **OK** to save the files and close the **Select File** dialog box.

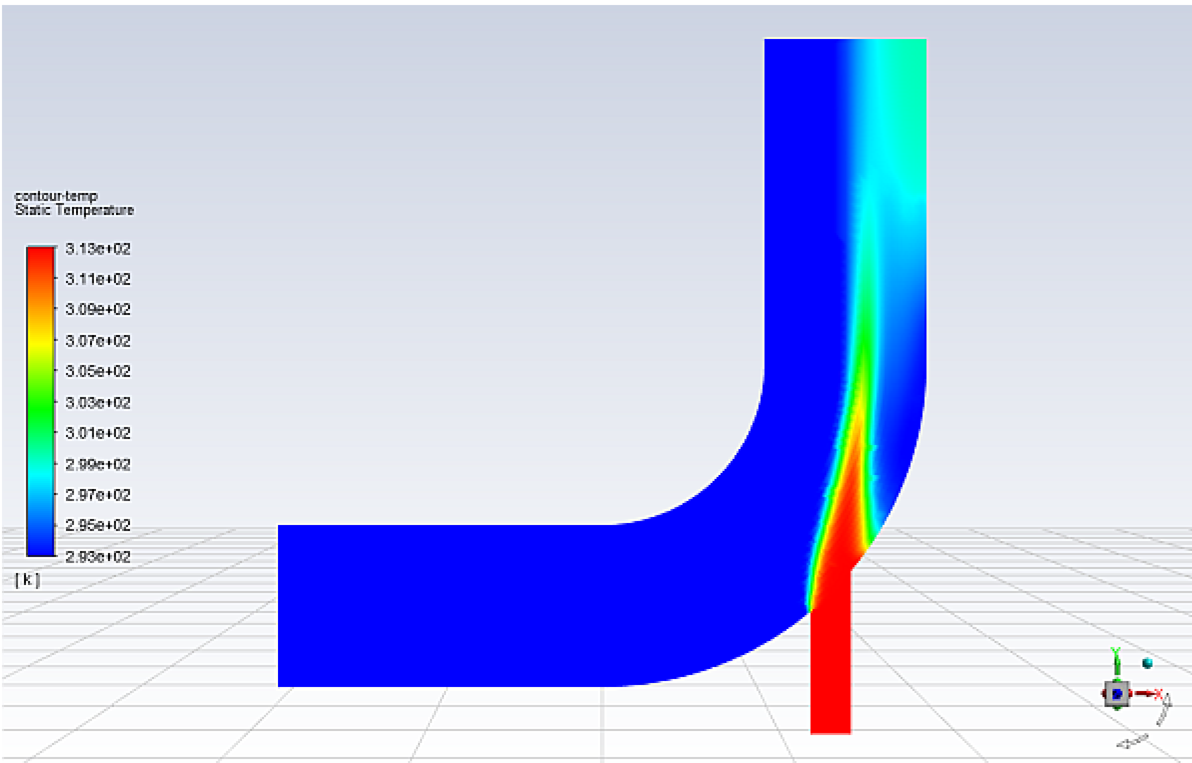
   The files `elbow2.cas.h5` and `elbow2.dat.h5` will be saved in your default folder.

6. Display the temperature distribution (using node values) on the revised mesh using the temperature contours definition that you created earlier (Figure 1.17: Filled Contours of Temperature Using the Adapted Mesh (p. 65)).

   Right-click the **Results/Graphics/Contours/contour-temp** tree item and select **Display** from the menu that opens.

   Results → **Graphics** → **Contours** → **contour-temp** → **Display**

   Figure 1.17: Filled Contours of Temperature Using the Adapted Mesh

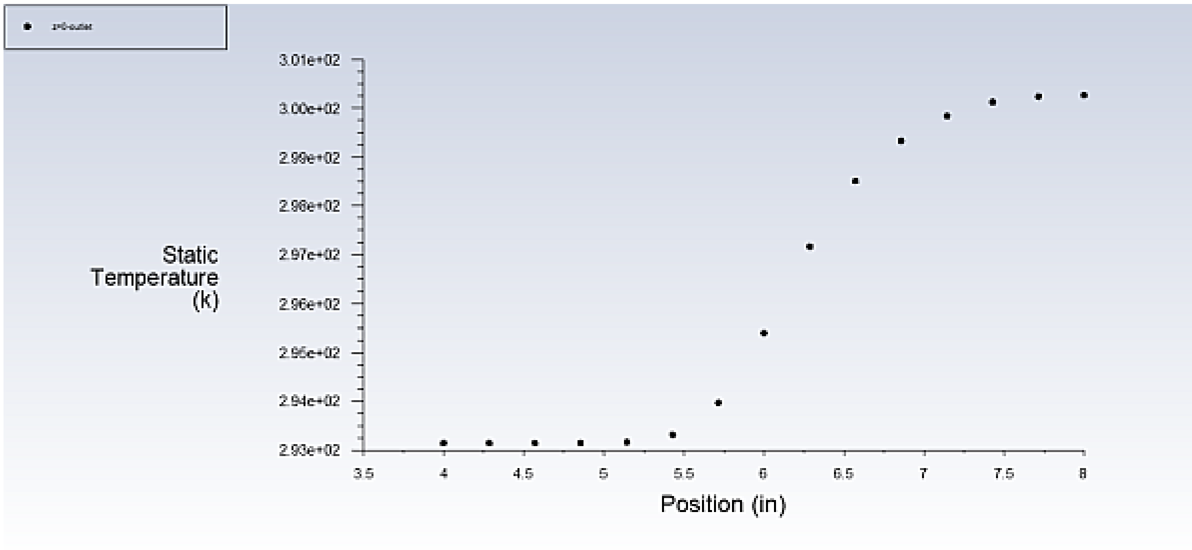


7. Display and save an XY plot of the temperature profile across the centerline of the outlet for the adapted solution (Figure 1.18: Outlet Temperature Profile for the Adapted Coupled Solver Solution (p. 66)).

   Results → **Plots** → **XY Plot** → **xy-outlet-temp** → **Edit...**

   a. Click **Save/Plot** to display the XY plot.

Figure 1.18: Outlet Temperature Profile for the Adapted Coupled Solver Solution



b. Enable **Write to File** in the **Options** group box.

*The button that was originally labeled **Save/Plot** will change to **Write...**.*

c. Click **Write...**.

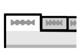
i. In the **Select File** dialog box, enter `outlet_temp2.xy` for **XY File**.

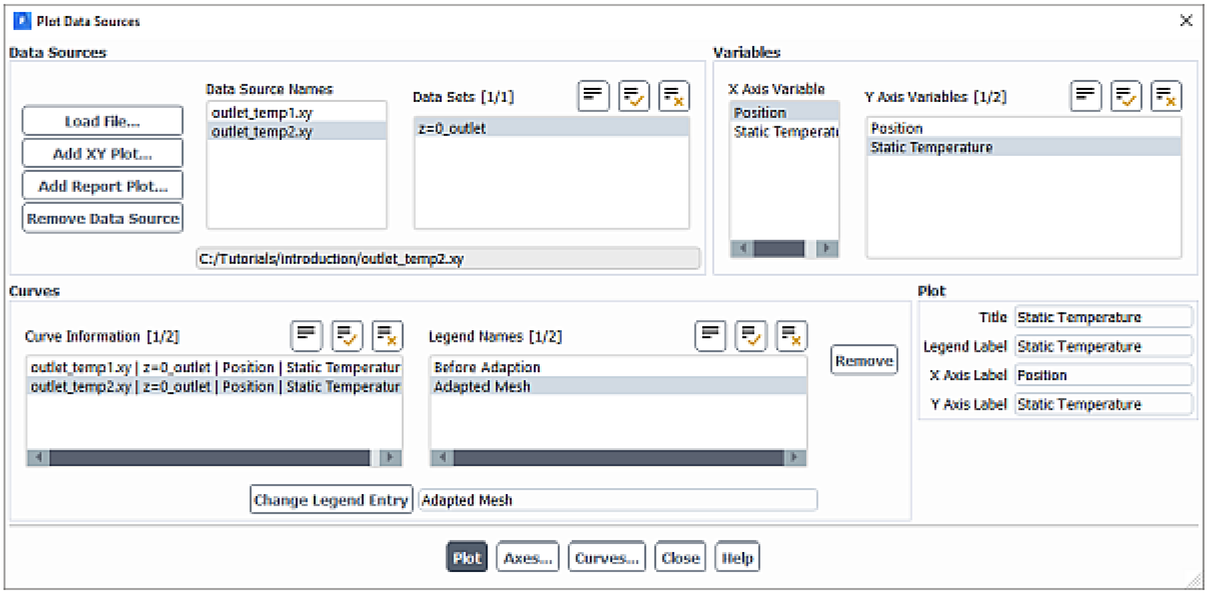
ii. Click **OK** to save the temperature data.

d. Close the **Solution XY Plot** dialog box.

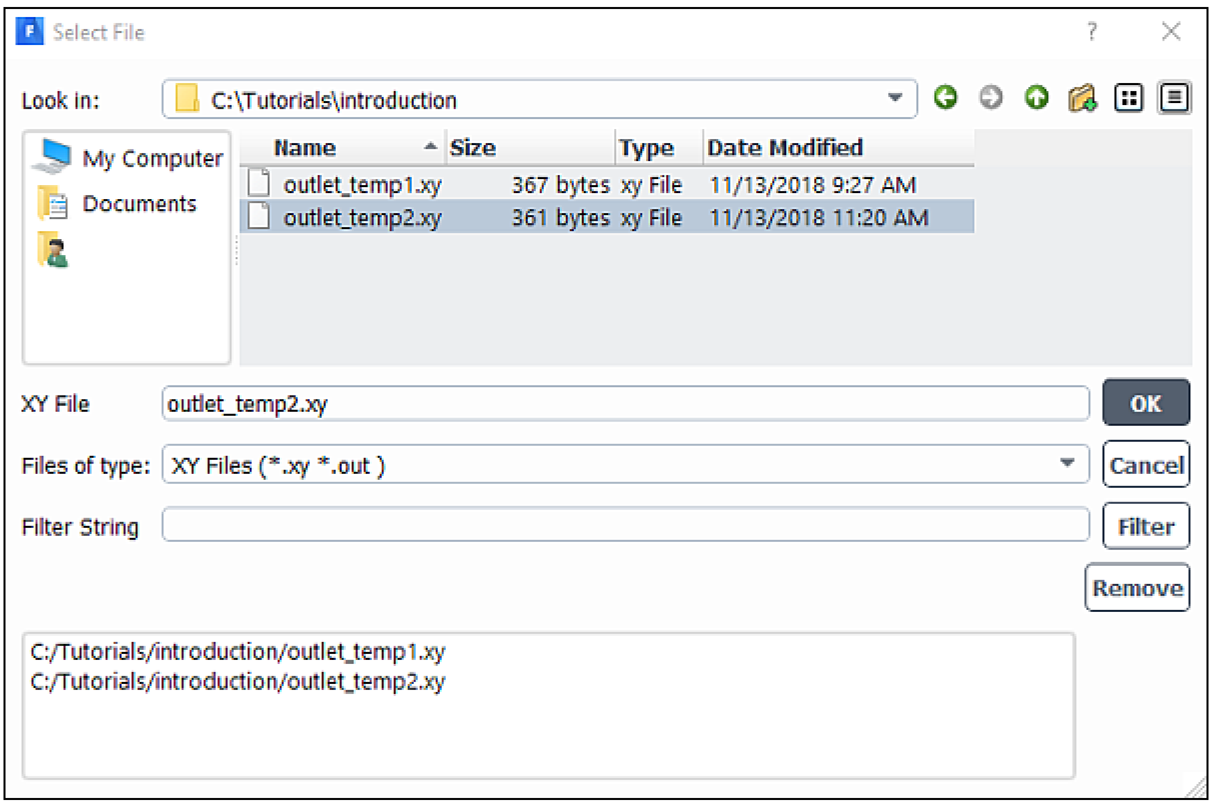
8. Display the outlet temperature profiles for both solutions on a single plot (Figure 1.19: Outlet Temperature Profiles for the Two Solutions (p. 68)).

a. Open the **Plot Data Sources** dialog box.

 **Results → Plots → Data Sources...**



b. Click the **Load File...** button to open the **Select File** dialog box.



i. Select **outlet_temp1.xy** and **outlet_temp2.xy**.

Each of these files will be listed with their folder path in the bottom list to indicate that they have been selected.

Tip:

If you select a file by mistake, simply click the file in the bottom list and then click **Remove**.

ii. Click **OK** to save the files and close the **Select File** dialog box.

c. Select the folder path ending in **outlet_temp1.xy** from the **Curve Information** selection list (**Curves** group box).

d. Enter `Before Adaption` in the lower-right text-entry box.

e. Click the **Change Legend Entry** button.

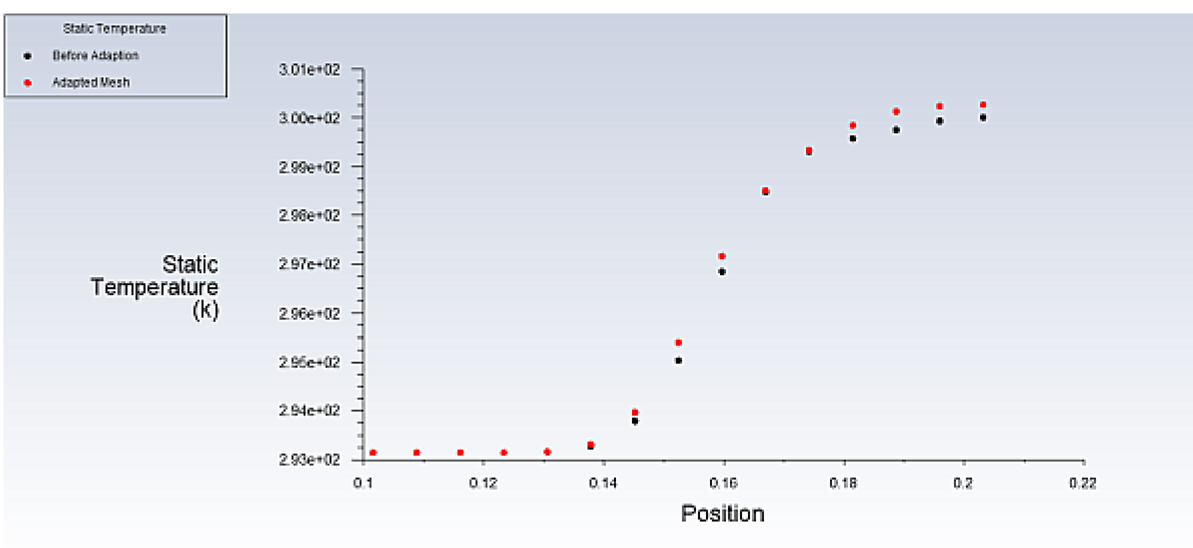
*The item in the **Legend Entries** list for **outlet_temp1.xy** will be changed to **Before Adaption**. This legend entry will be displayed in the upper-left corner of the XY plot generated in a later step.*

f. In a similar manner, change the legend entry for the folder path ending in **outlet_temp2.xy** to be `Adapted Mesh`.

g. Click **Plot** and close the **Plot Data Sources** dialog box.

Figure 1.19: Outlet Temperature Profiles for the Two Solutions (p. 68) shows the two temperature profiles at the centerline of the outlet. It is apparent by comparing both the shape of the profiles and the predicted outer wall temperature that the solution is highly dependent on the mesh and solution options. Specifically, further mesh adaption should be used in order to obtain a solution that is independent of the mesh.

Figure 1.19: Outlet Temperature Profiles for the Two Solutions



1.5. Summary

The solution results are changed by the adaption of the mesh, which indicates that a sufficiently refined mesh is required to obtain a mesh independent solution.