Динамика течения и теплообмен в смешивающихся потоках воздуха (Пример из официальной документации)

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.

1.3. Problem Description

The problem to be considered is shown schematically in Figure 1.1: Problem Specification. 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.

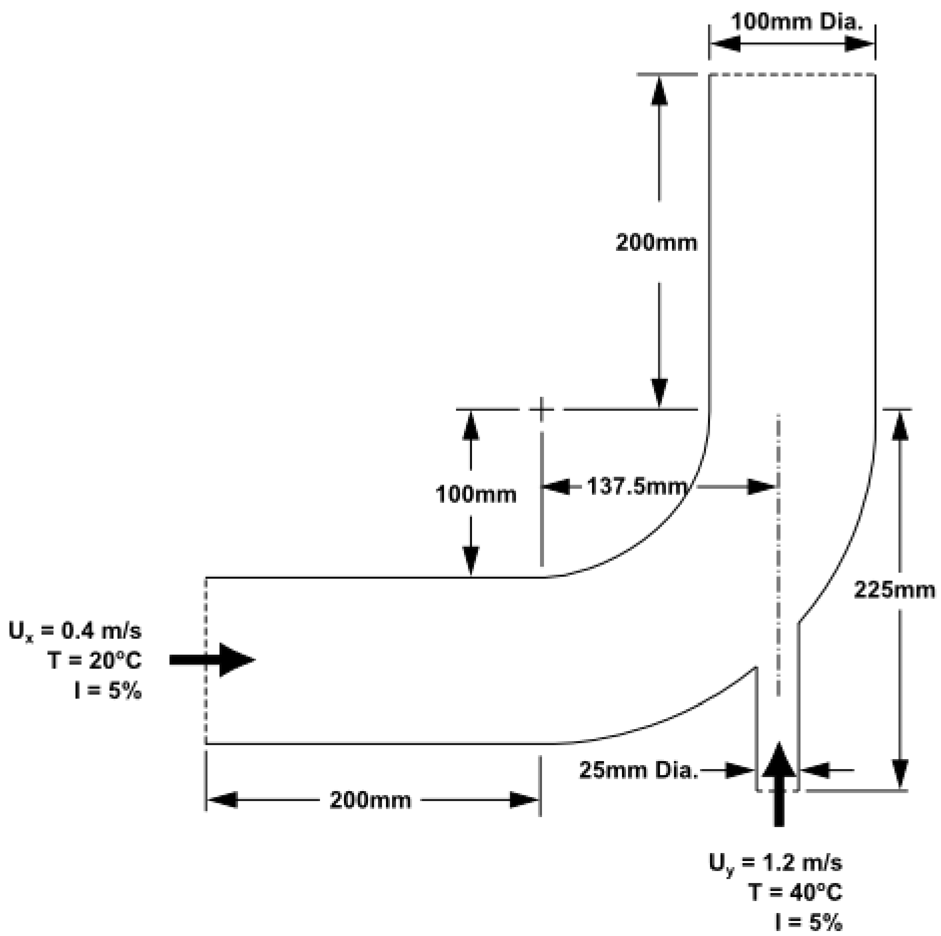


Figure 1.1: Problem Specification

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 for detailed information.

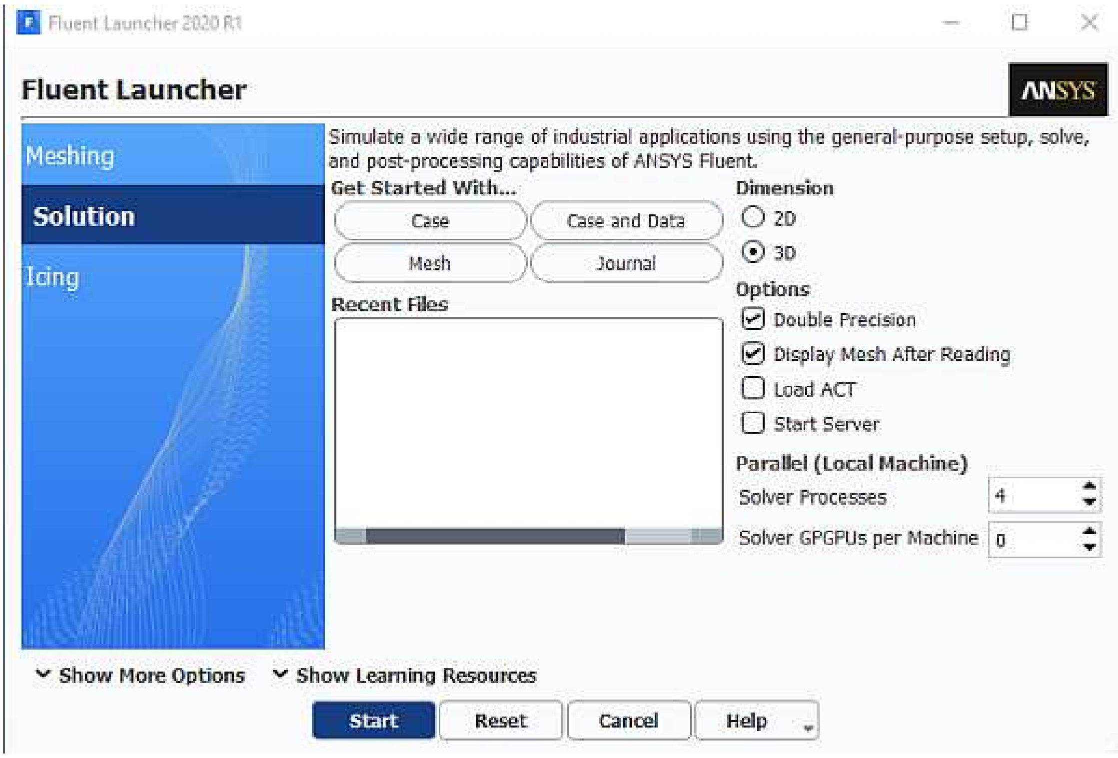
1.4.1. Preparation

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.

1.4.2. Launching ANSYS Fluent

- 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.



Ensure that the proper options are enabled.

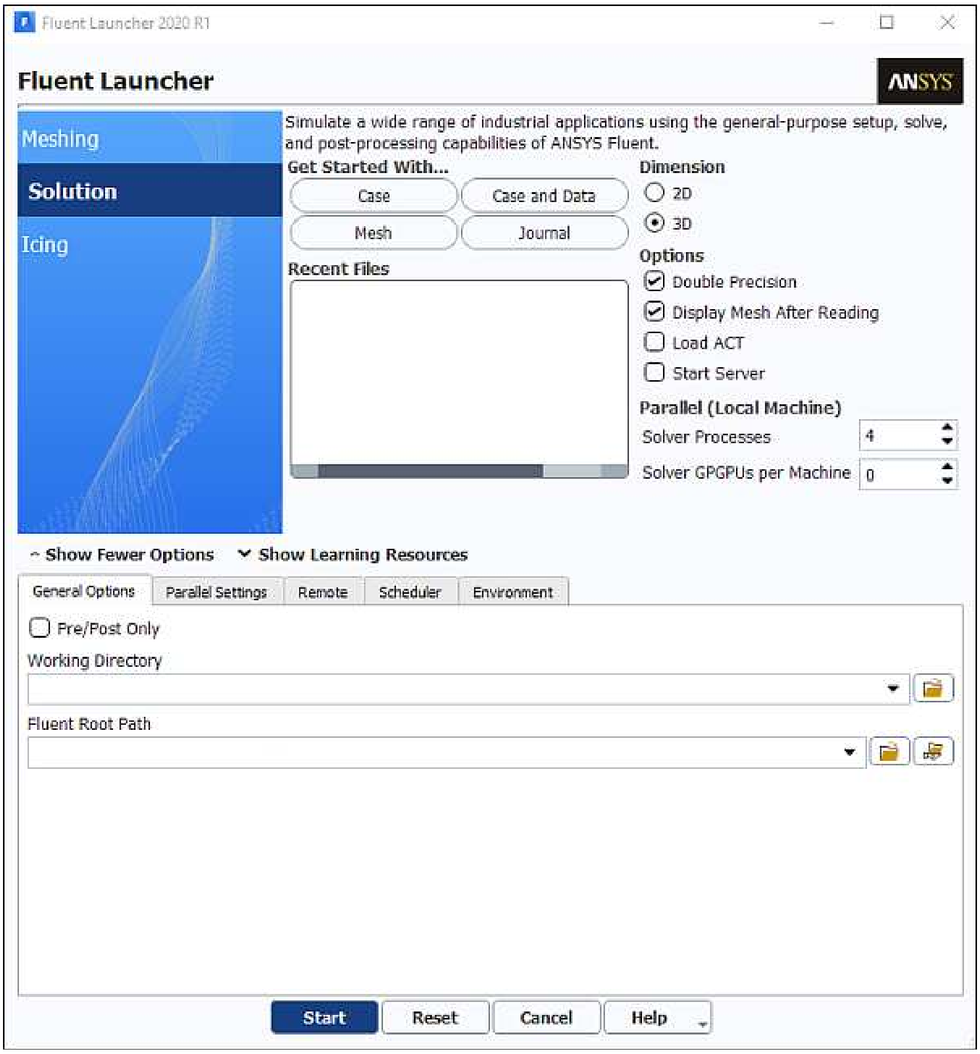
- Select **3D** from the **Dimension** list by clicking the radio button or the text.
- Ensure that the **Double Precision** option is selected.
- Ensure that the **Display Mesh After Reading** option is enabled.
- Set Processes to `4` under the Parallel (local Machine).

Note: Fluent will retain your preferences for future sessions.

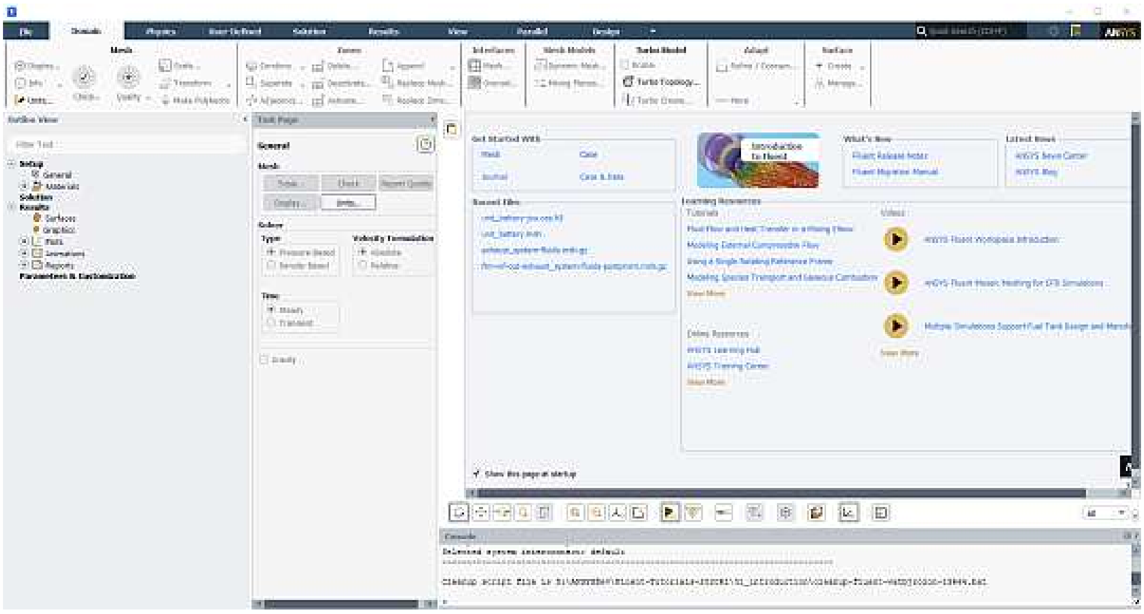
Set the working folder to the one created when you unzipped `introduction.zip`.

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



Click **OK** to launch ANSYS Fluent.



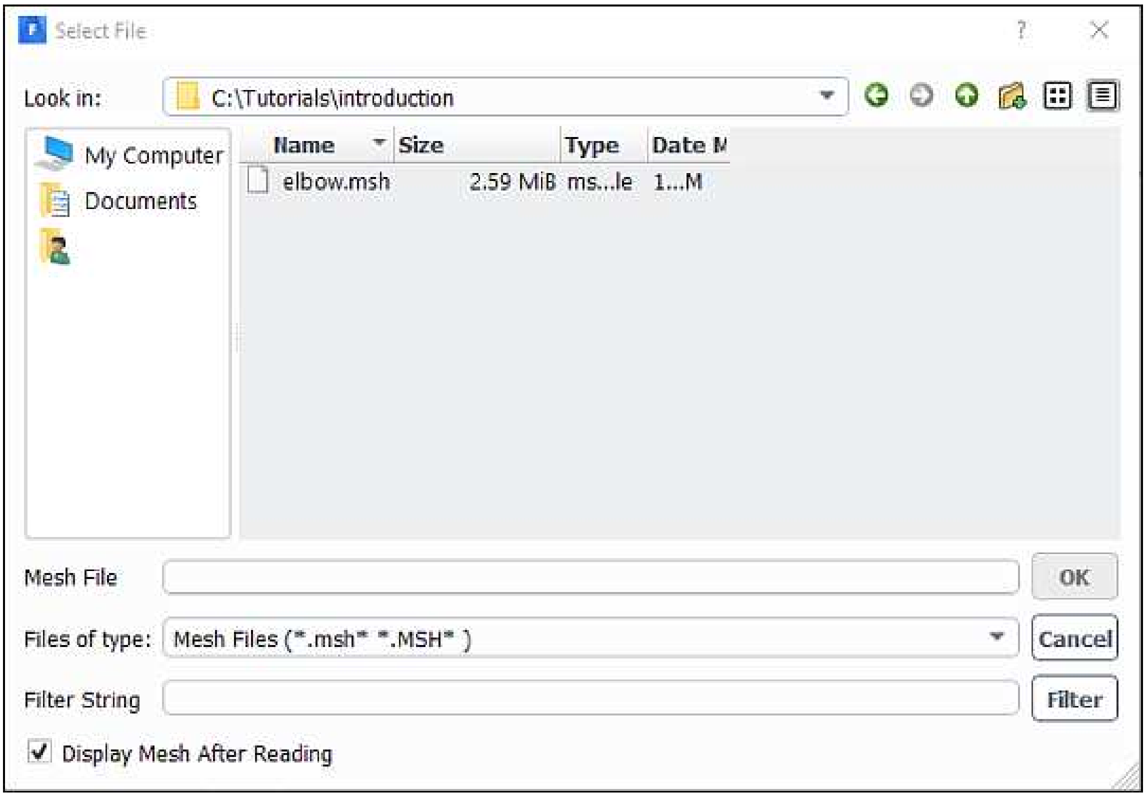
For more information about the Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

1.4.3. Reading the Mesh

- 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...



- Select the mesh file by clicking **elbow.msh** in the **introduction** folder created when you unzipped the original file.
- Enable the Display Mesh After Reading in the Select File dialog box.
- 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.

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.

1. Click the z-axis.
2. Clicking the **Fit to Window** icon, , will cause the object to fit exactly and be centered in the window.
3. Figure 1.2: The Hexahedral Mesh for the Mixing Elbow

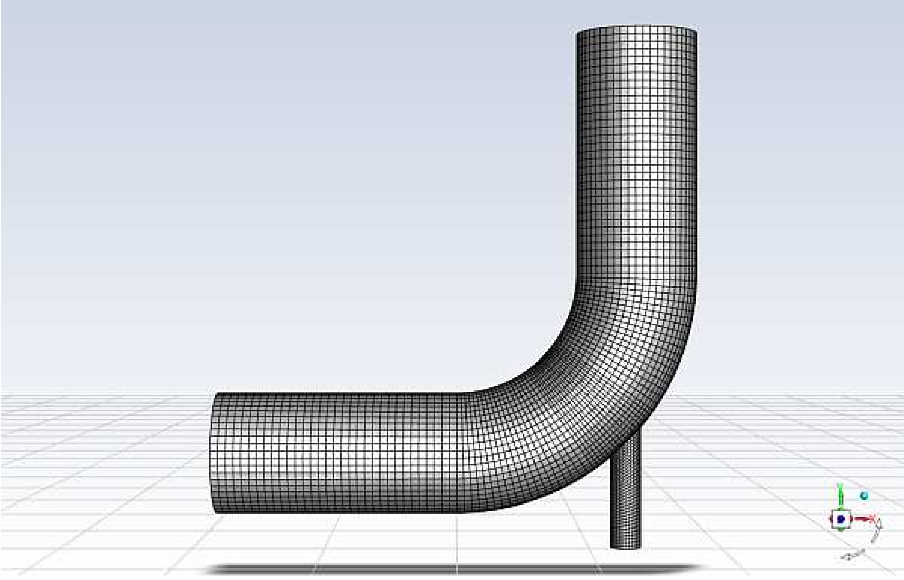


Figure 1.2: The Hexahedral Mesh for the Mixing Elbow

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
```

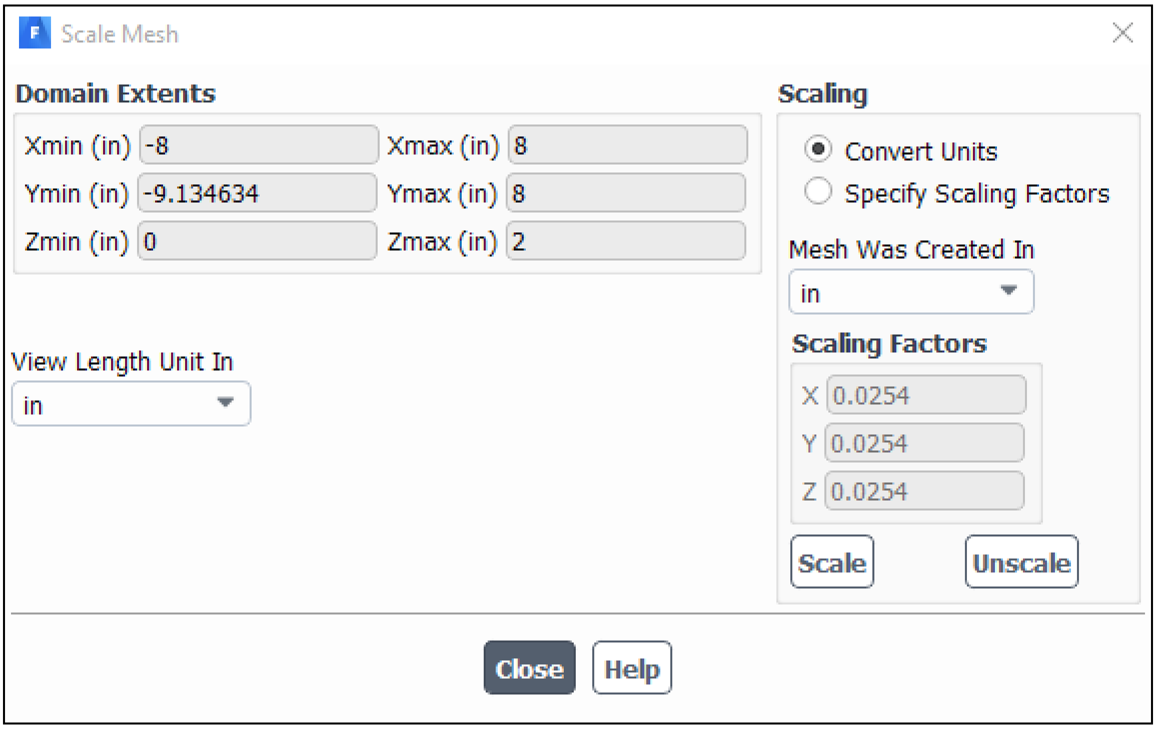
```
   maximum face area (m2): 1.017924e-01
 Checking mesh.................................
Done.
```

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...**



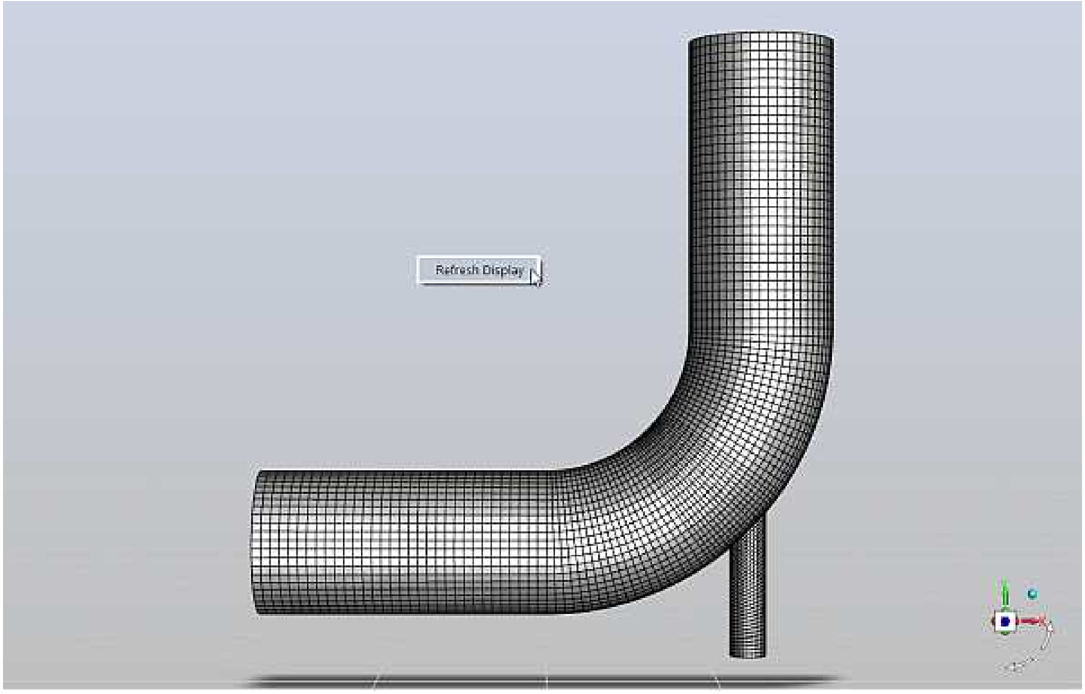
Ensure that **Convert Units** is selected in the **Scaling** group box.

1. 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.
2. Click **Scale** to scale the mesh.
3. **Warning:** Be sure to click the **Scale** button only once.

Domain Extents will continue to be reported in the default SI unit of meters.

4. Select **in** from the **View Length Unit In** drop-down list to set inches as the working unit for length.
5. Confirm that the domain extents are as shown in the previous dialog box.
6. Close the **Scale Mesh** dialog box.

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



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

1. Check the mesh.

Domain → Mesh → Check → Perform Mesh Check

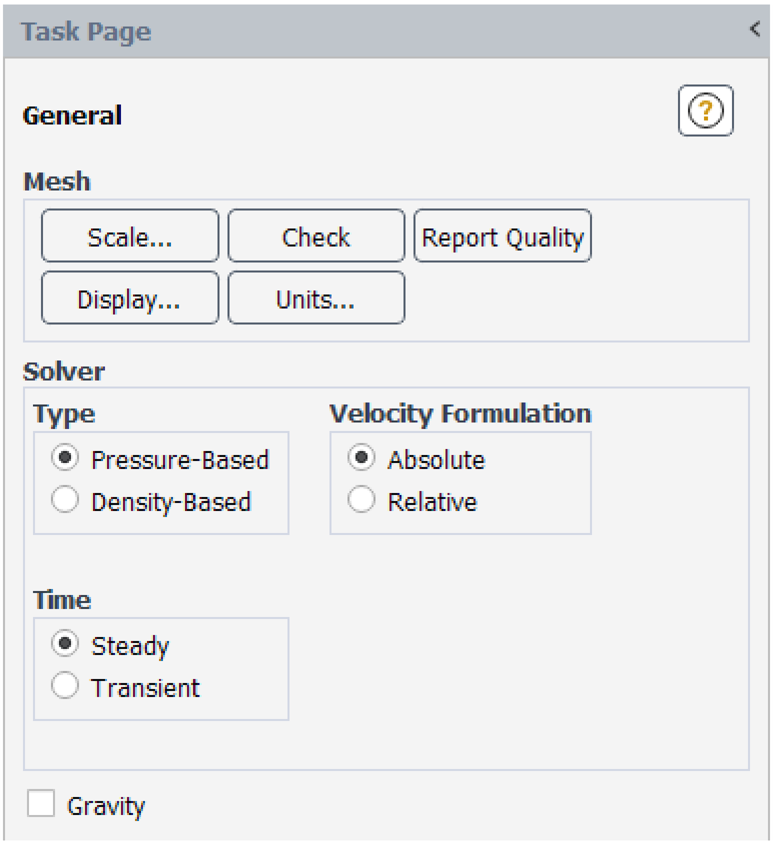
Note: It is a good idea to check the mesh after you manipulate it (that is, scale, convert to polyhedra, merge, separate, fuse, add zones, or smooth and swap). This will ensure that the quality of the mesh has not been compromised.

1.4.5. Setting Up Physics

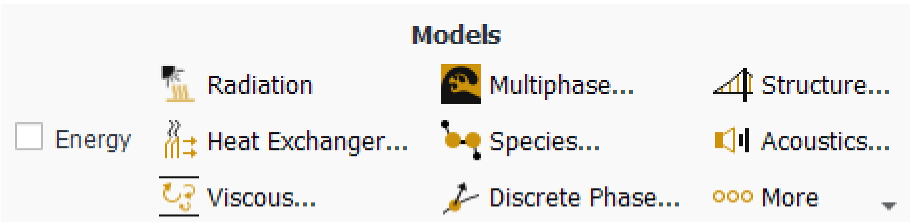
In the steps that follow, you will select a solver and specify physical models, material properties, and zone conditions for your simulation using the **Physics** ribbon tab.

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

Physics → Solver → General



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.

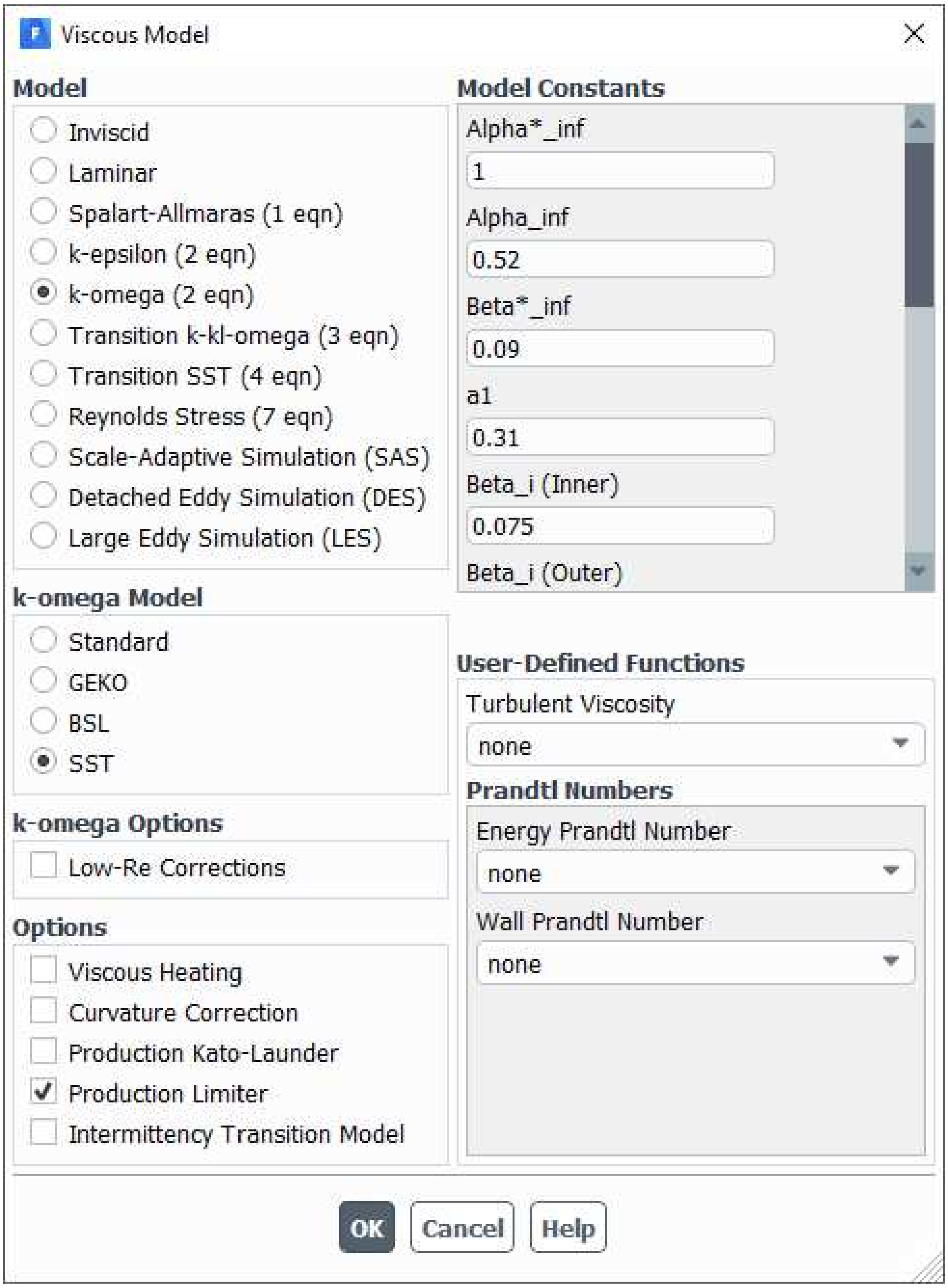
2. Enable heat transfer by activating the energy equation.

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

Physics → Models → Energy

3. Enable the k-ω turbulence model.

Physics → Models → Viscous...



4. Retain the default selection of **k-omega** from the **Model** list.
5. Retain the default selection of **SST** in the **k-omega Model** group box.
6. Click **OK** to accept all the other default settings and close the **Viscous Model** dialog box.
7. 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.

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

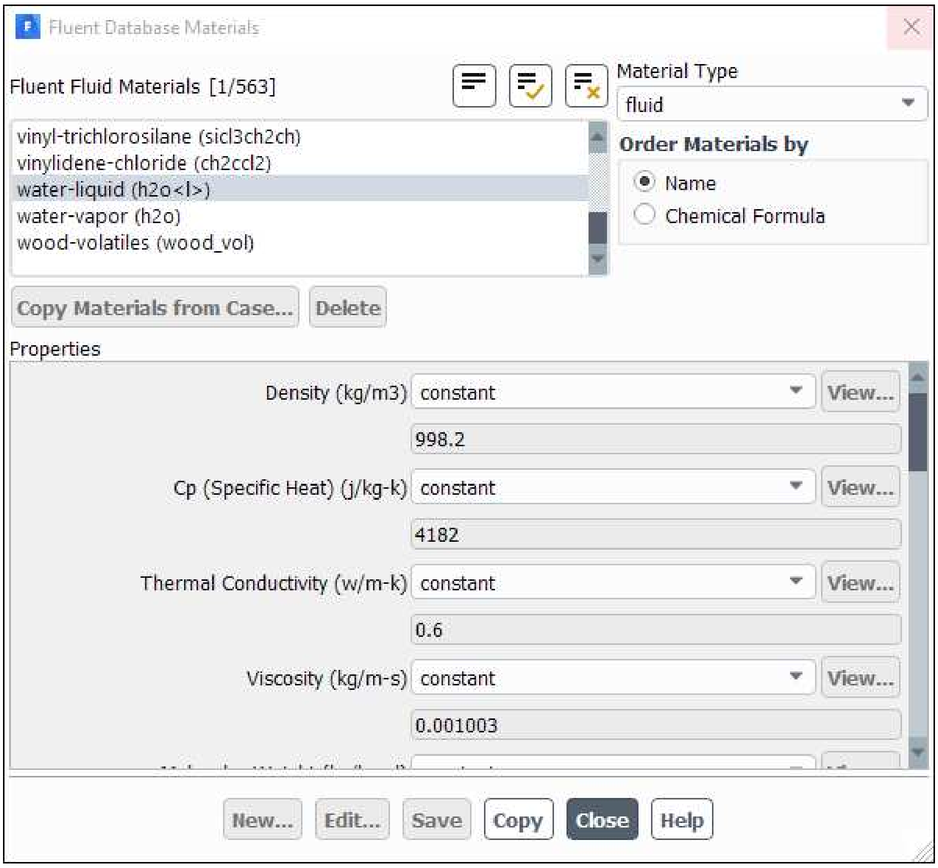
Create a new material called **water** using the **Create/Edit Materials** dialog box.

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

Physics → Materials → Create/Edit...

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

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

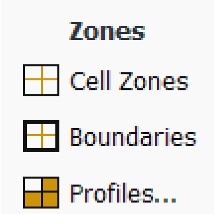


Ensure that there are now two materials (water-liquid and air) defined locally by examining the **Fluent Fluid Materials** drop-down list.

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

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

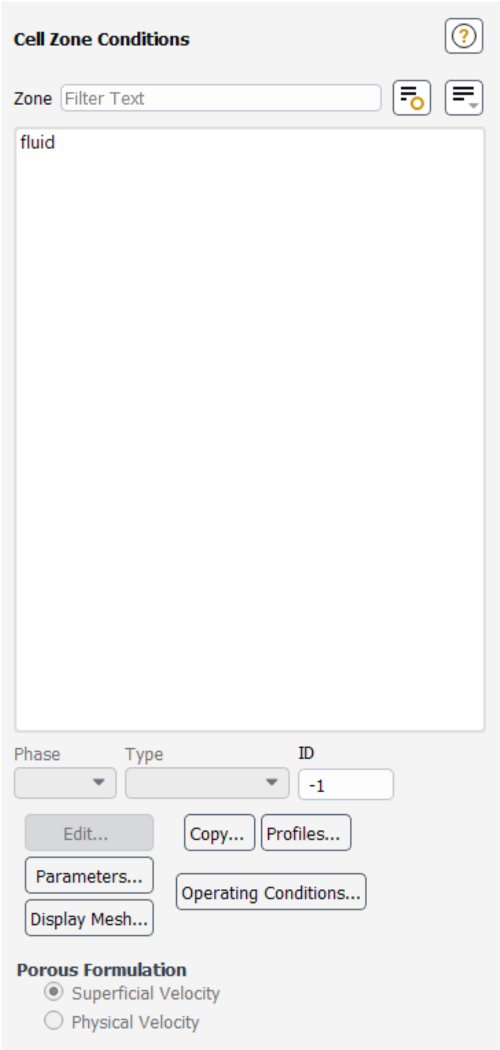
Set up the cell zone conditions for the fluid zone (**fluid**) using the **Zones** group box of the **Physics** ribbon tab.



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

Physics → **Zones** → **Cell Zones**

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



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

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

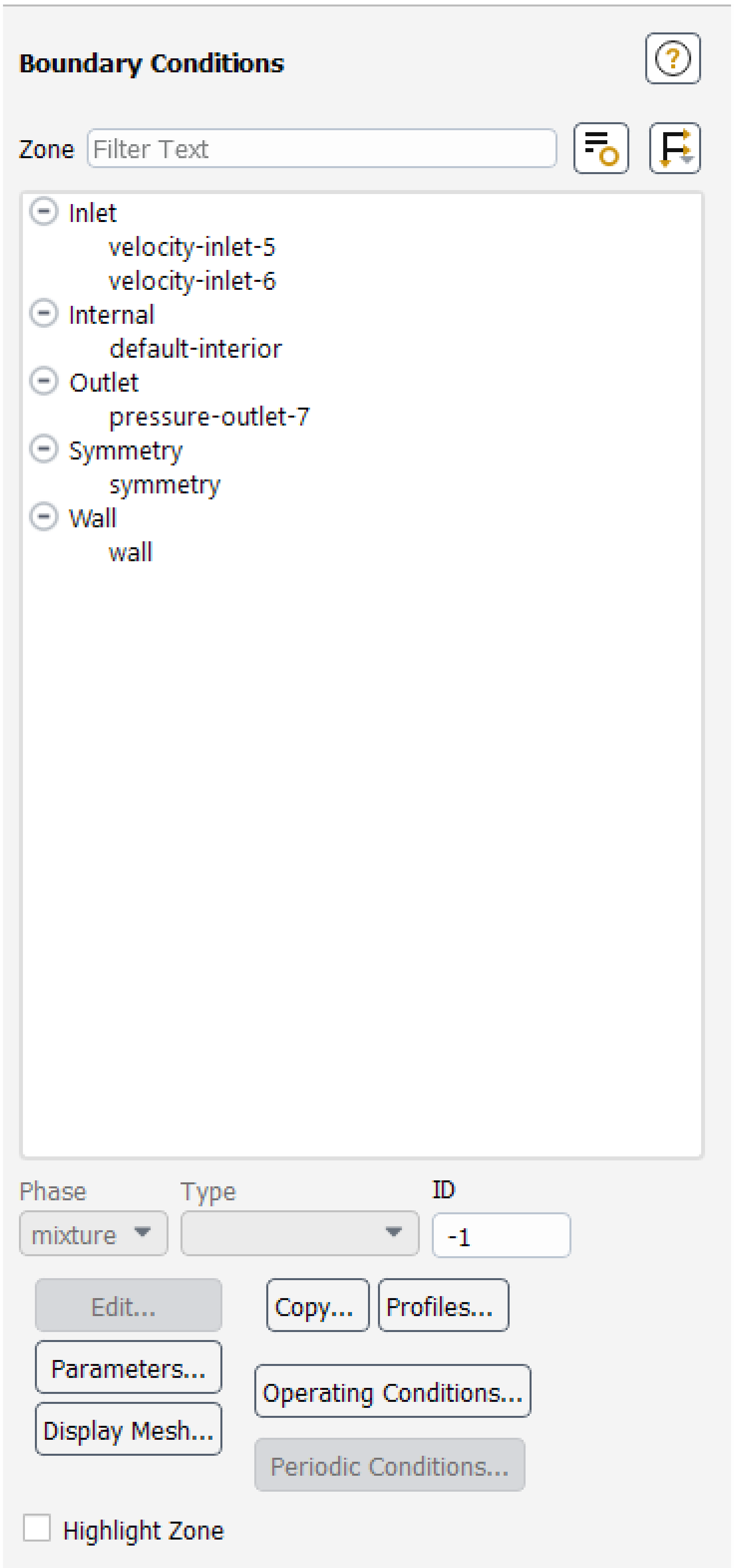
Click **OK** to close the **Fluid** dialog box.

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.

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

Physics → Zones → Boundaries

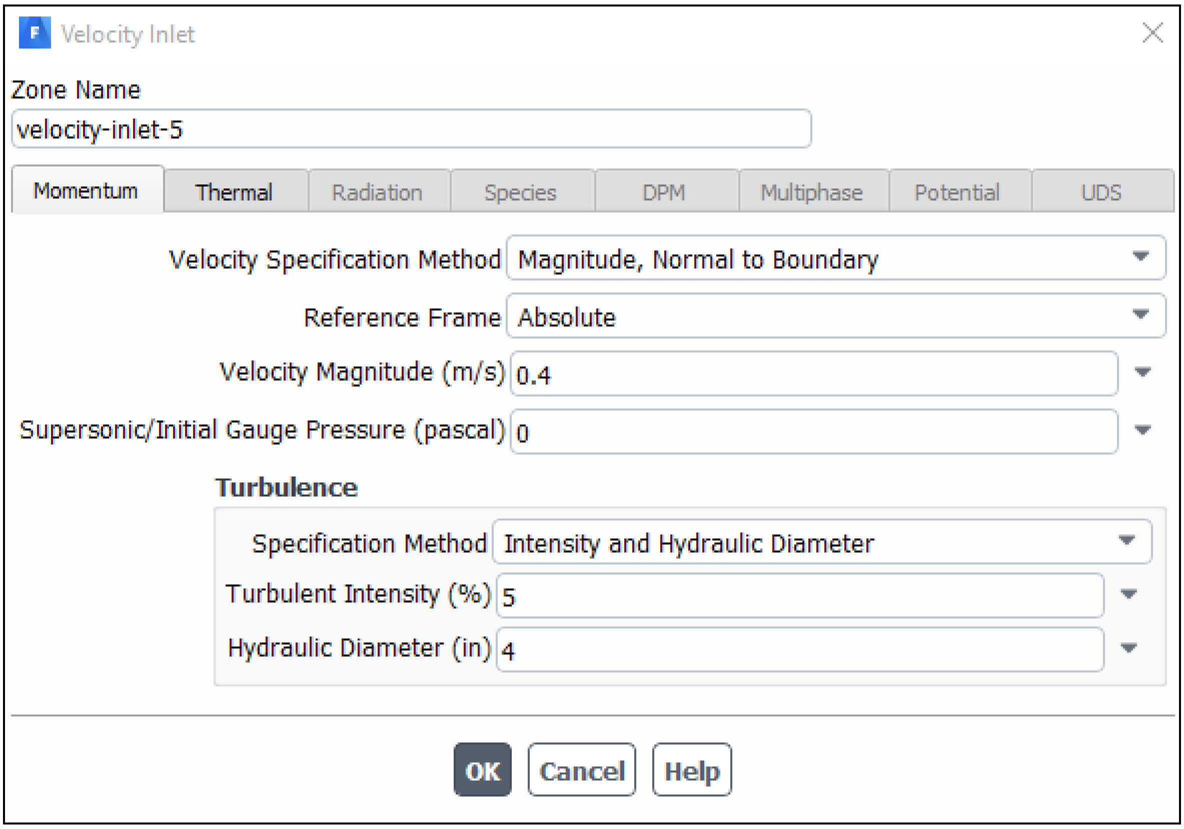
This opens the **Boundary Conditions** task page where the boundaries defined in your simulation are displayed in the **Zone** selection list.



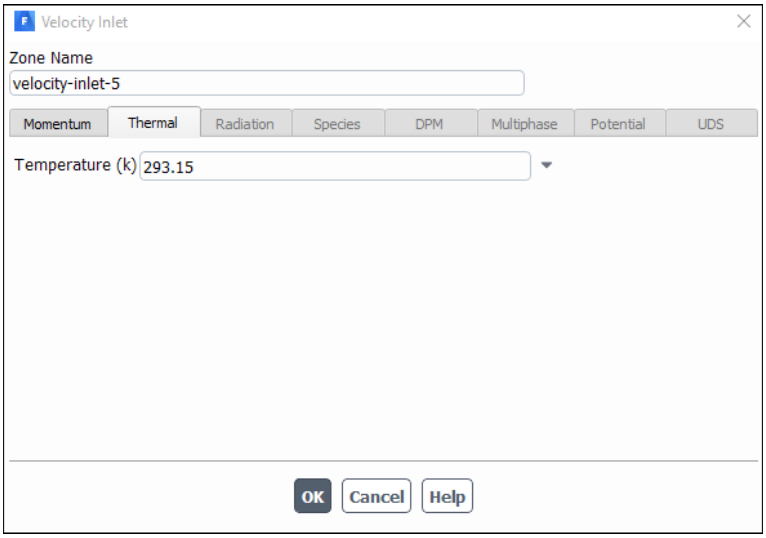
Here the zones have names with numerical identifying tags. It is good practice to give boundaries meaningful names in a meshing application to help when you set up the model. You can also change boundary names in Fluent by simply editing the boundary and making revisions in the **Zone Name** text box.

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

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



- Retain the default selection of **Magnitude, Normal to Boundary** from the **Velocity Specification Method** drop-down list.
- Enter `0.4` [m/s] for **Velocity Magnitude**.
- In the Turbulence group box, select Intensity and Hydraulic Diameter from the Specification Method drop-down list.
- Retain the default value of `5` [%] for **Turbulent Intensity**.
- Enter `4` [inches] for **Hydraulic Diameter**.
- Click the **Thermal** tab.

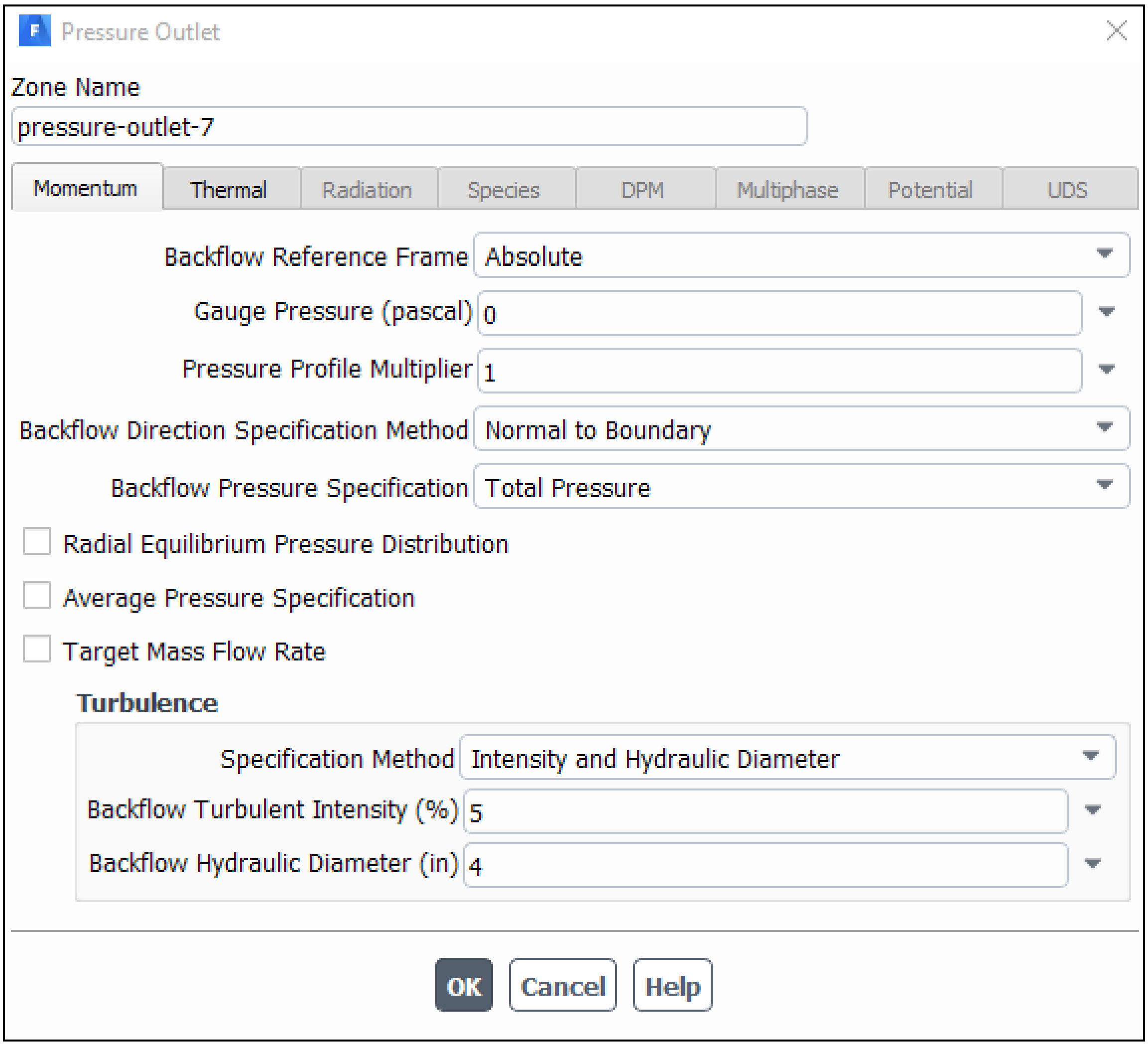


- Enter `293.15` [K] for **Temperature**.
- Click **OK** to close the **Velocity Inlet** dialog box.

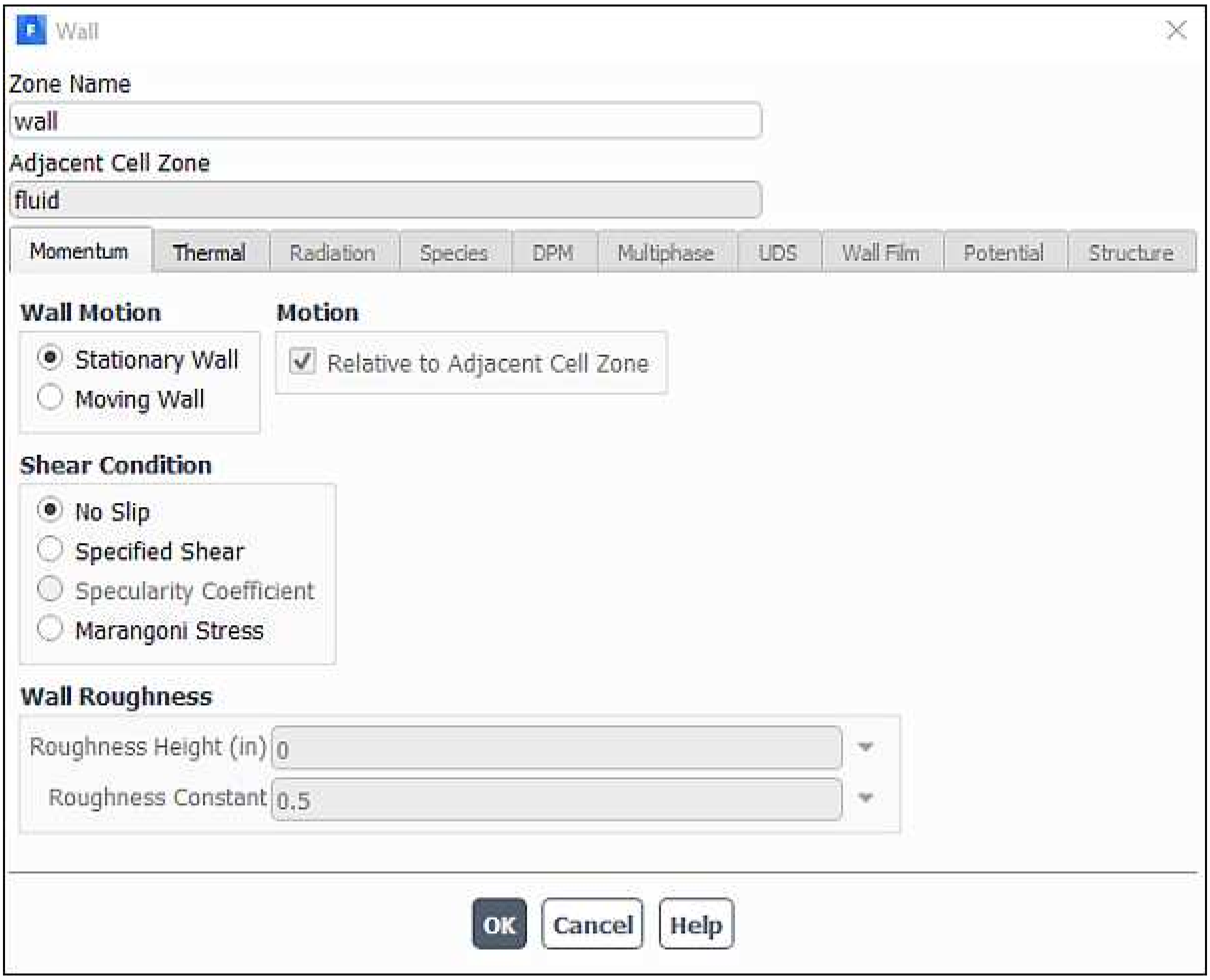
In a similar manner, set the boundary conditions at the hot inlet (**velocity-inlet-6**), using the values in the following table:

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]

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



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

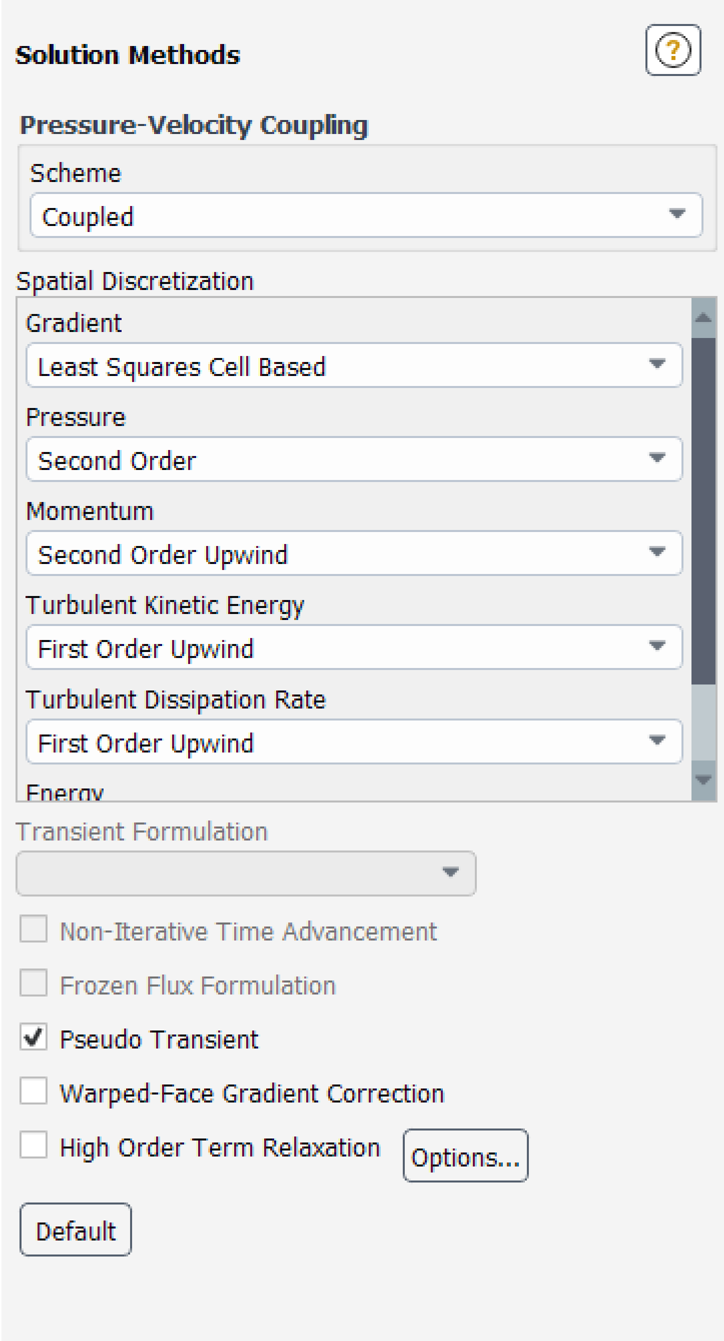


1.4.6. Solving

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

- Select a solver scheme.
- In the **Solution** ribbon tab, click **Methods...** (**Solution** group box).

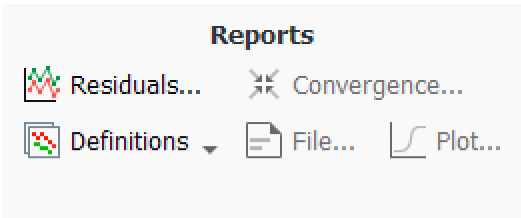
Solution → Solution → Methods...



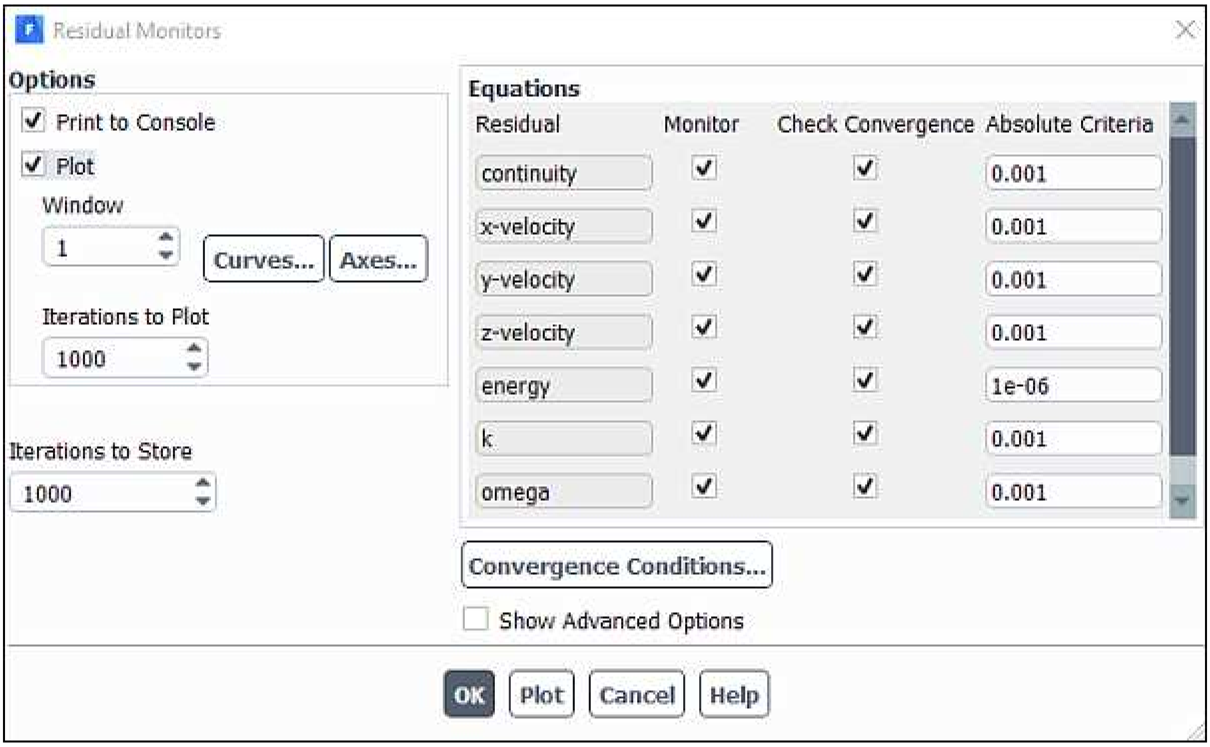
Retain the default selections.

Enable the plotting of residuals during the calculation.

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



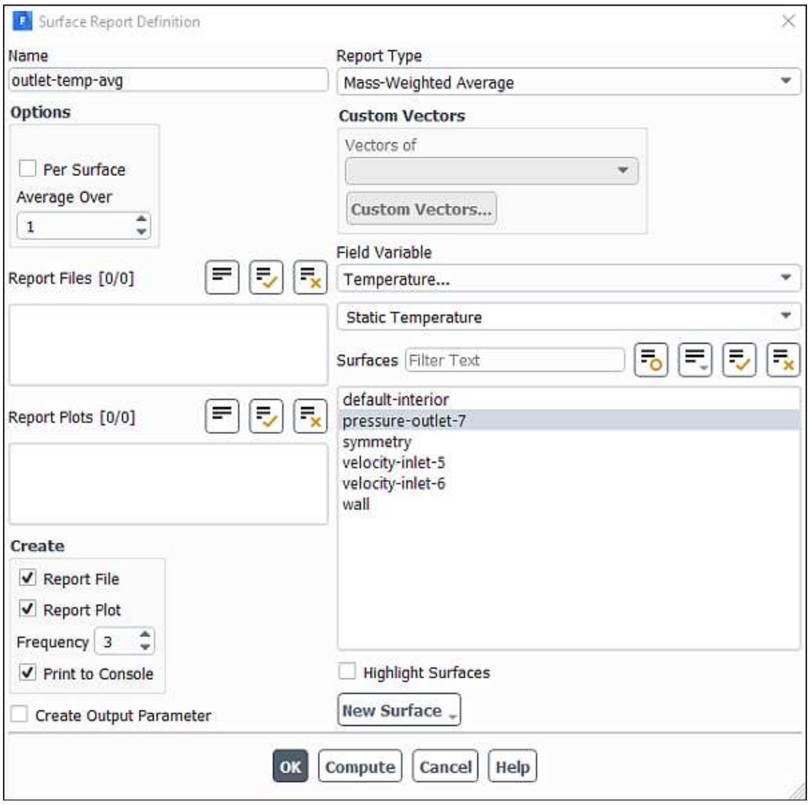
Solution → Reports → Residuals...



- Ensure that **Plot** is enabled in the **Options** group box.
- Retain the default value of `0.001` for the **Absolute Criteria** of **continuity**.
- Click **OK** to close the **Residual Monitors** dialog box.

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

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



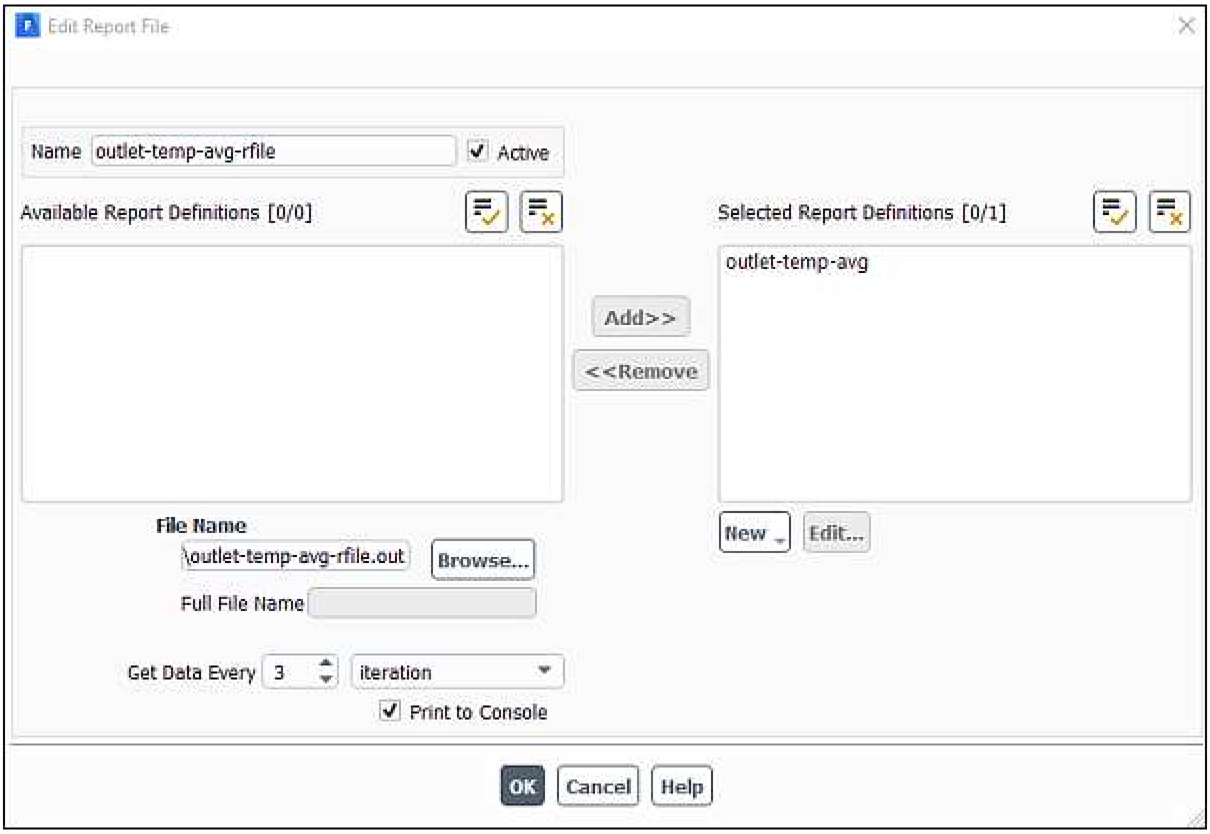
- Enter `outlet-temp-avg` for the **Name** of the report definition.
- Enable Report File, Report Plot, and Print to Console in the Create group box.

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.

- 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.

- Select Temperature... and Static Temperature from the Field Variable drop-down lists.
- Select **pressure-outlet-7** from the **Surfaces** selection list.
- 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)
- 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.

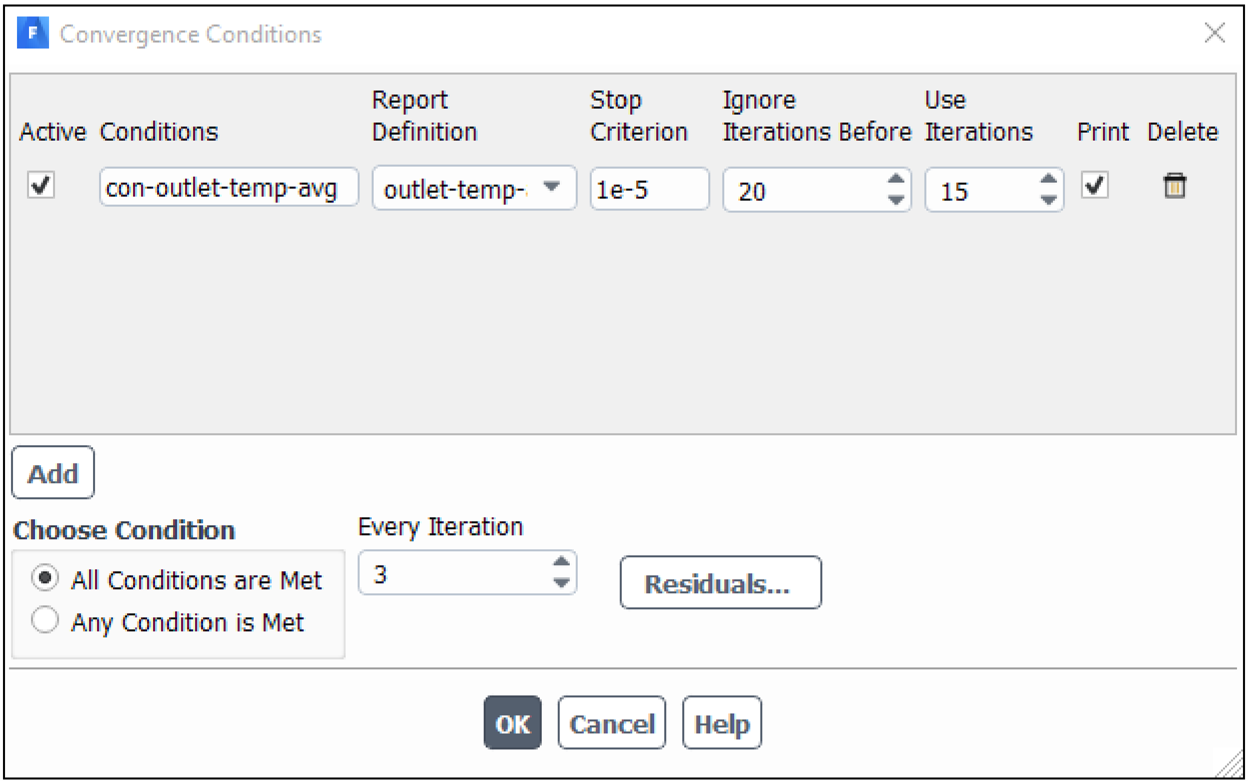
Verify that outlet-temp-avg is in the Selected Report Definitions list.

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.

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

- Click **OK** to close the **Edit Report File** dialog box.
- Create a convergence condition for **outlet-temp-avg**.

Solution → Reports → Convergence...

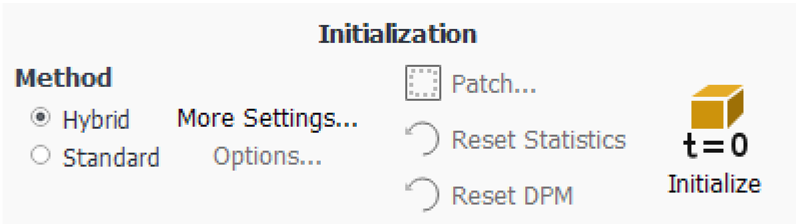


- Click the **Add** button.
- Enter `con-outlet-temp-avg` for **Conditions**.
- Select **outlet-temp-avg** from the **Report Definition** drop-down list.
- Enter `1e-5` for **Stop Criterion**.
- Enter `20` for Ignore Iterations Before.
- Enter `15` for **Use Iterations**.
- Enable **Print**.
- Set Every Iteration to `3`.
- 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.

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

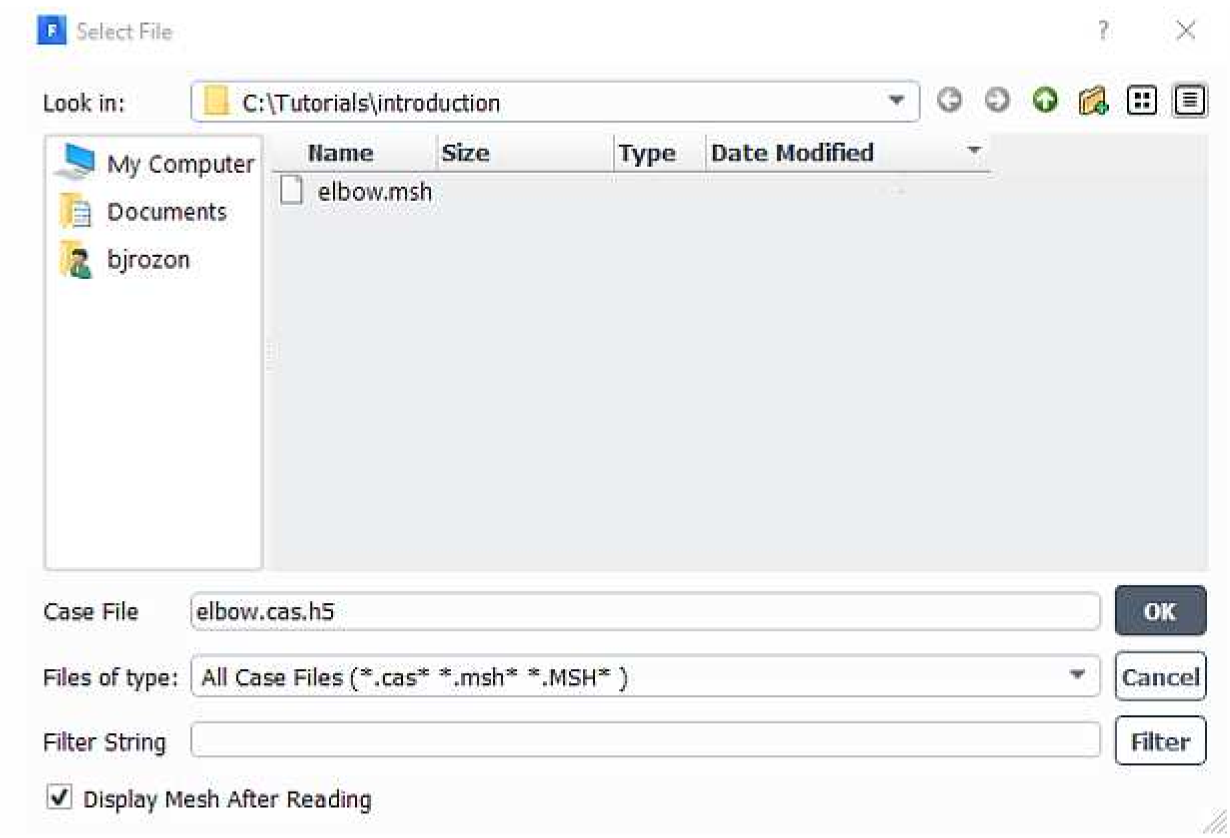
Solution → Initialization



- Retain the default selection of **Hybrid** from the **Method** list.
- Click **Initialize**.

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

File → Write → Case...

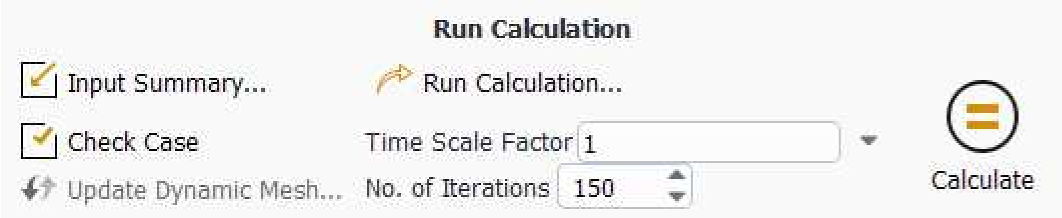


- (optional) Indicate the folder in which you would like the file to be saved.

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.

- Enter `elbow1.cas.h5` for **Case File**.
- Ensure that the default **Write Binary Files** option is enabled, so that a binary file will be written.
- Click **OK** to save the case file and close the **Select File** dialog box.
- Start the calculation by requesting 150 iterations in the **Solution** ribbon tab (**Run Calculation** group box).

Solution → Run Calculation



- Enter `150` for No. of Iterations.
- Click Calculate.

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).

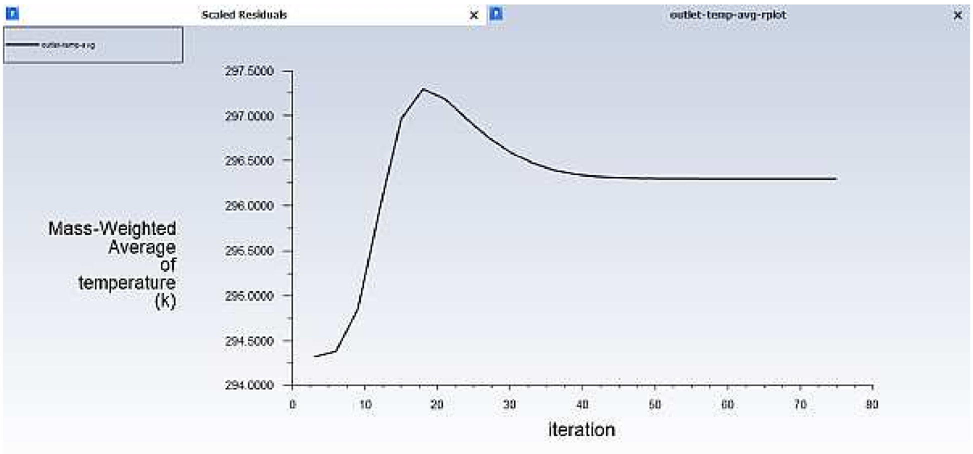


Figure 1.3: Convergence History of the Mass-Weighted Average Temperature

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

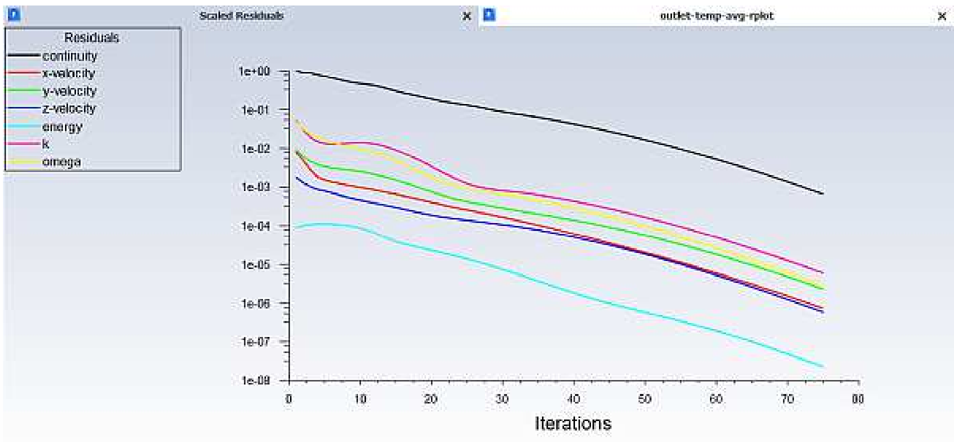


Figure 1.4: Residuals

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.

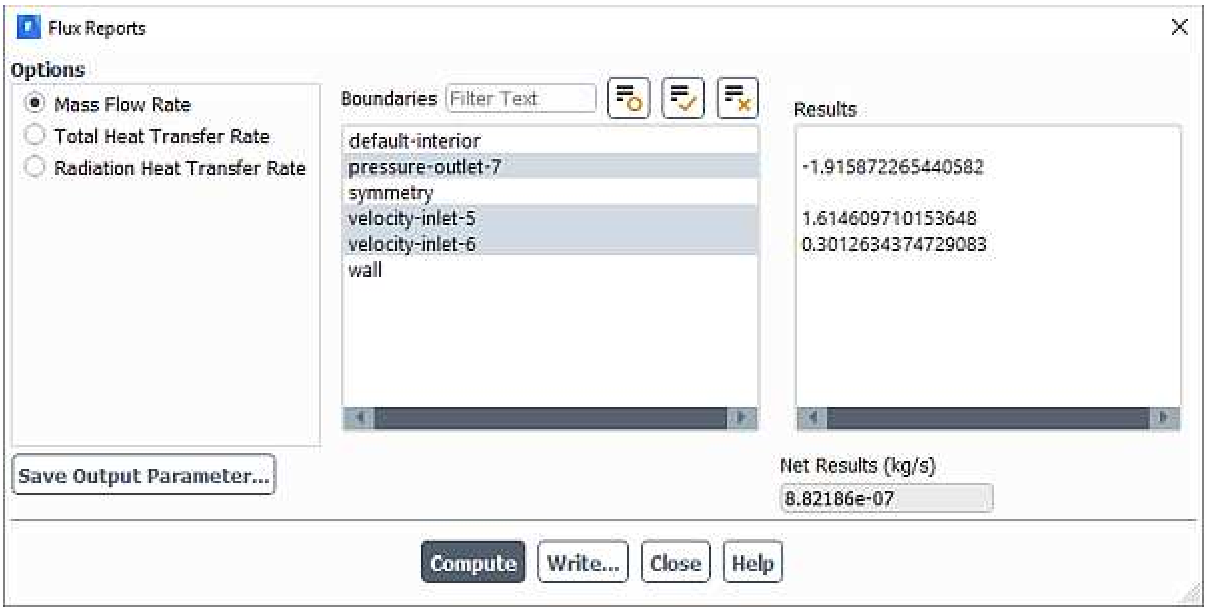
The solution will be stopped by ANSYS Fluent when any of the following occur:

- 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.

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

Results → Reports → Fluxes...



- Ensure that **Mass Flow Rate** is selected from the **Options** list.
- Select pressure-outlet-7, velocity-inlet-5, and velocity-inlet-6 from the Boundaries selection list.
- 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.

Close the **Flux Reports** dialog box.

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

File → Write → Data...

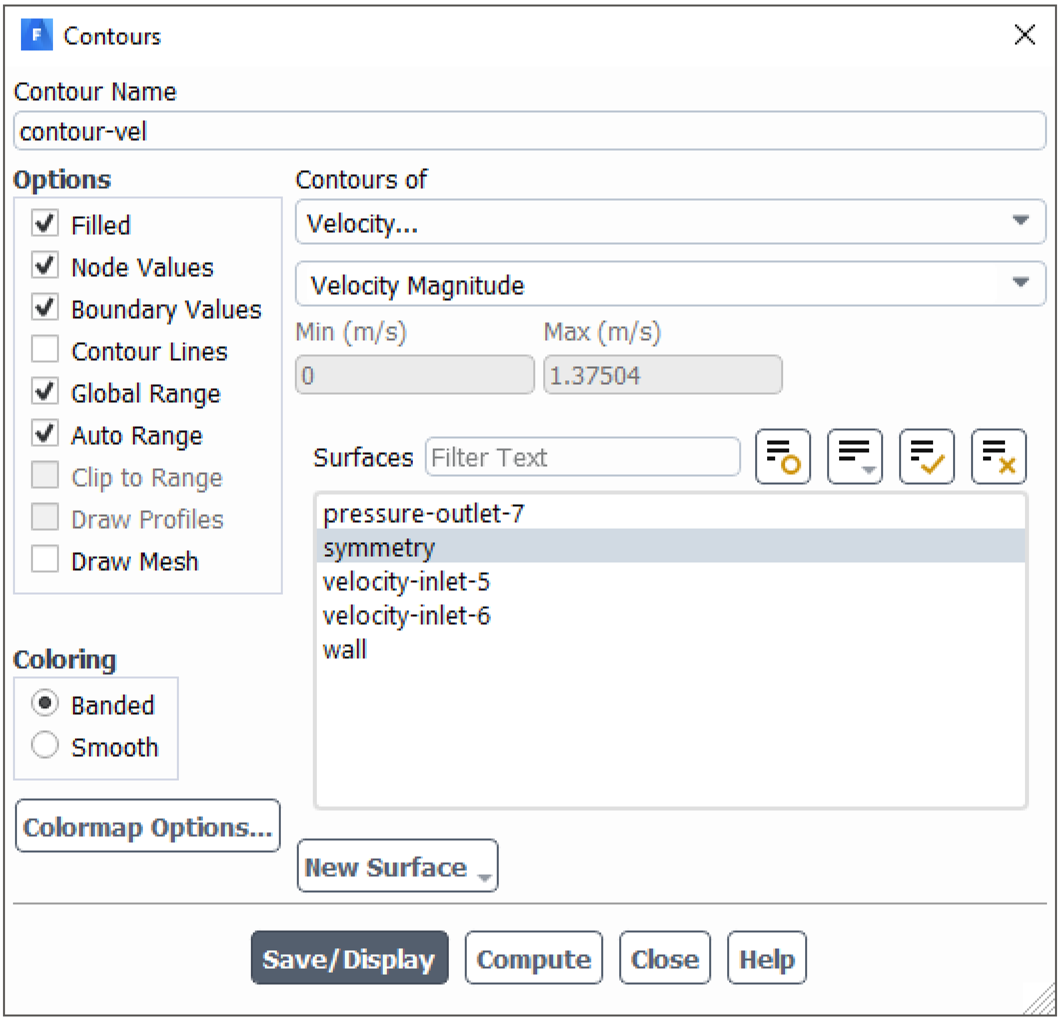
In later steps of this tutorial you will save additional case and data files with different suffixes.

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.

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

Results → Graphics → Contours → New...



- Enter `contour-vel` for **Contour Name**.
- Enable **Filled** in the **Options** group box.
- Ensure that **Node Values** and **Boundary Values** are enabled in the **Options** group box.
- Select **Banded** in the **Coloring** group box.
- Select **Velocity...** and **Velocity Magnitude** from the **Contours of** drop-down lists.
- Select **symmetry** from the **Surfaces** selection list.
- 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.

Close the **Contours** dialog box.

View → Display

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

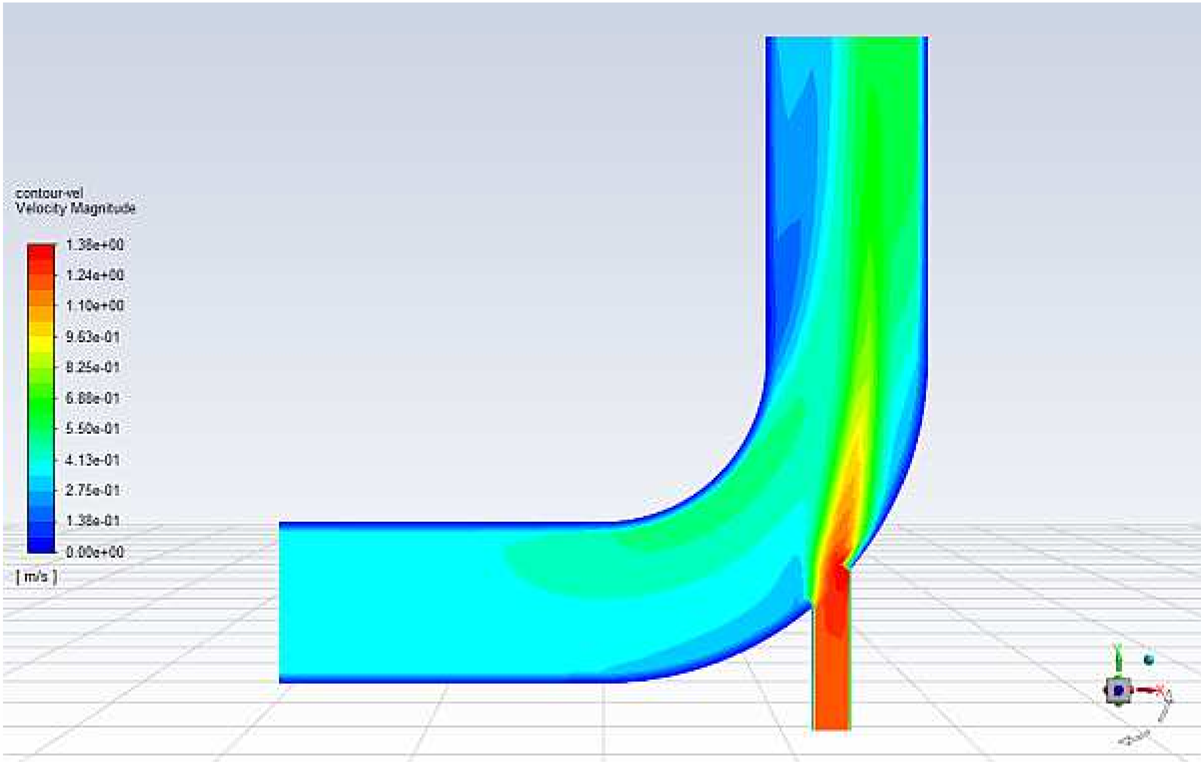
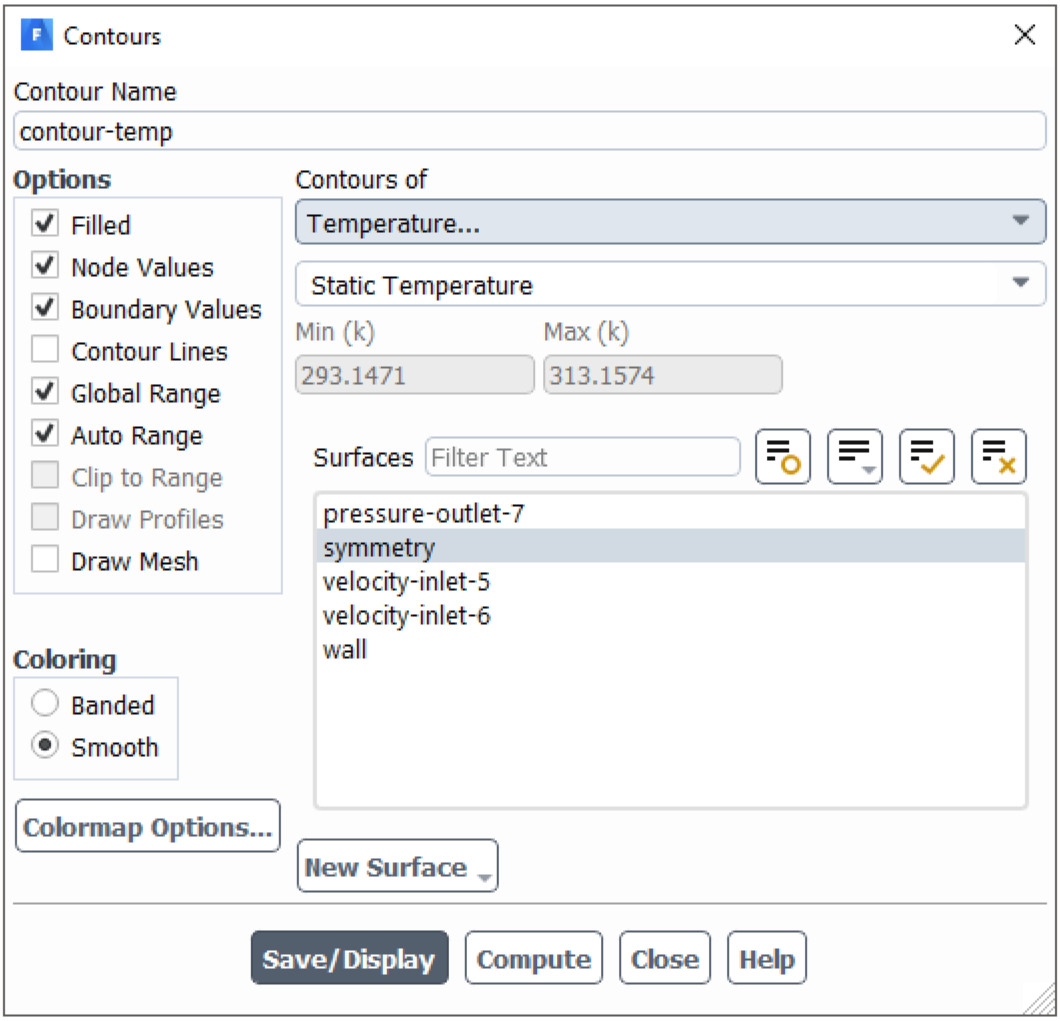


Figure 1.5: Predicted Velocity Distribution after the Initial Calculation

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

Results → Graphics → Contours → New...

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



- Enter `contour-temp` for **Contour Name**.
- Select **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.
- Select **symmetry** from the **Surfaces** selection list.
- 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.

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

Results → Graphics → Vectors → New...

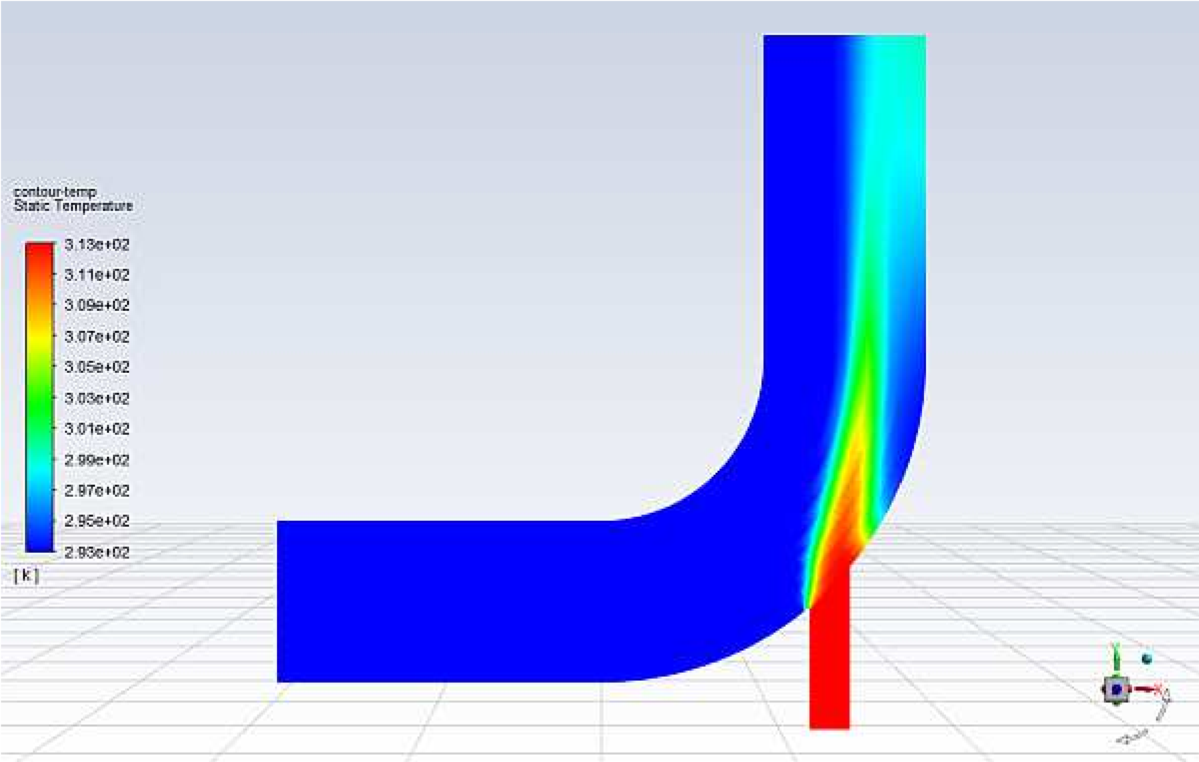
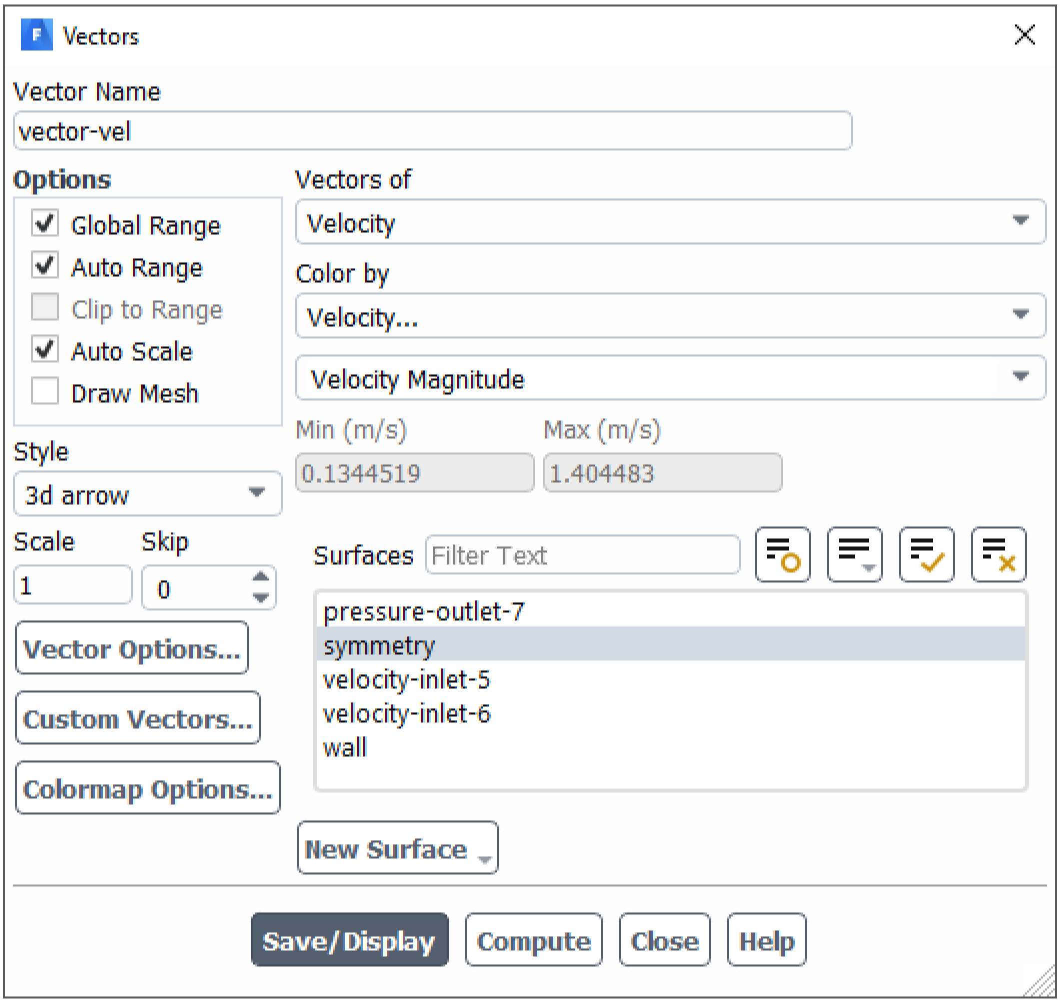


Figure 1.6: Predicted Temperature Distribution after the Initial Calculation



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

- Select **arrow** from the **Style** drop-down box.
- Select **symmetry** from the **Surfaces** selection list.
- Click **Save/Display** to plot the velocity vectors.

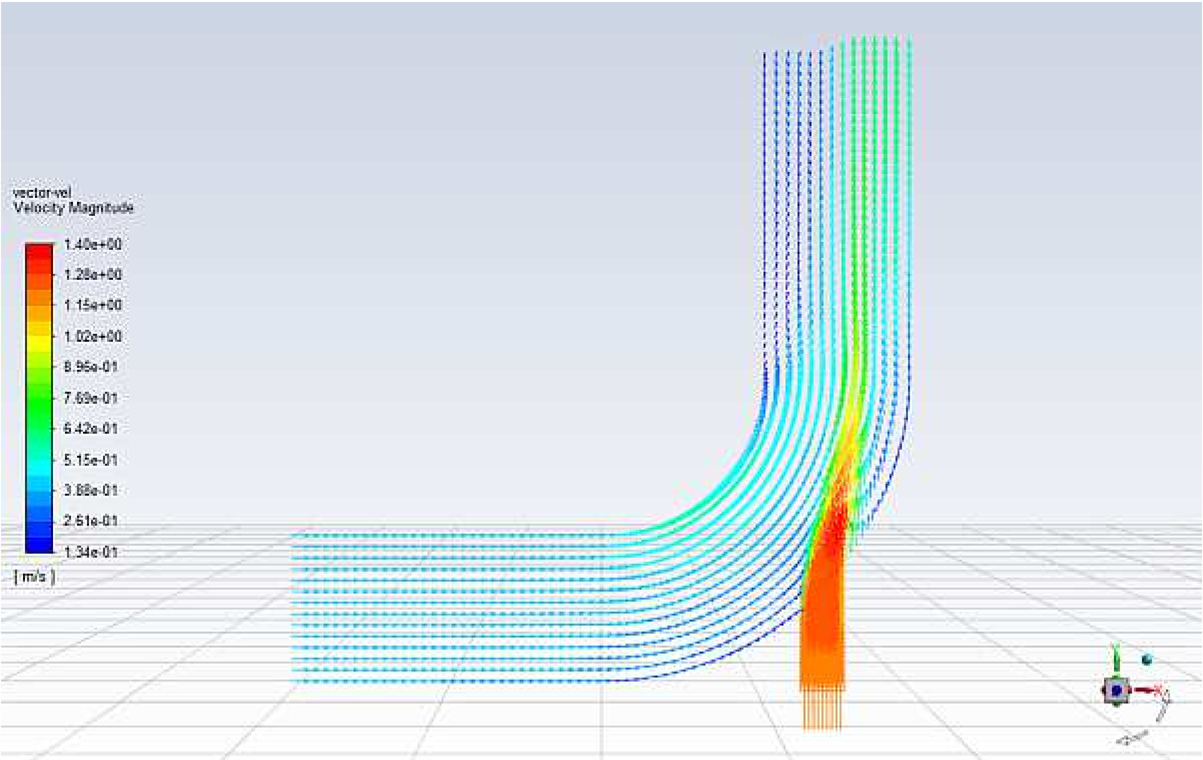


Figure 1.7: Velocity Vectors Colored by Velocity Magnitude

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.

- Enter `4` for **Scale**.
- Set **Skip** to `2`.
- Click **Save/Display** again to redisplay the vectors.

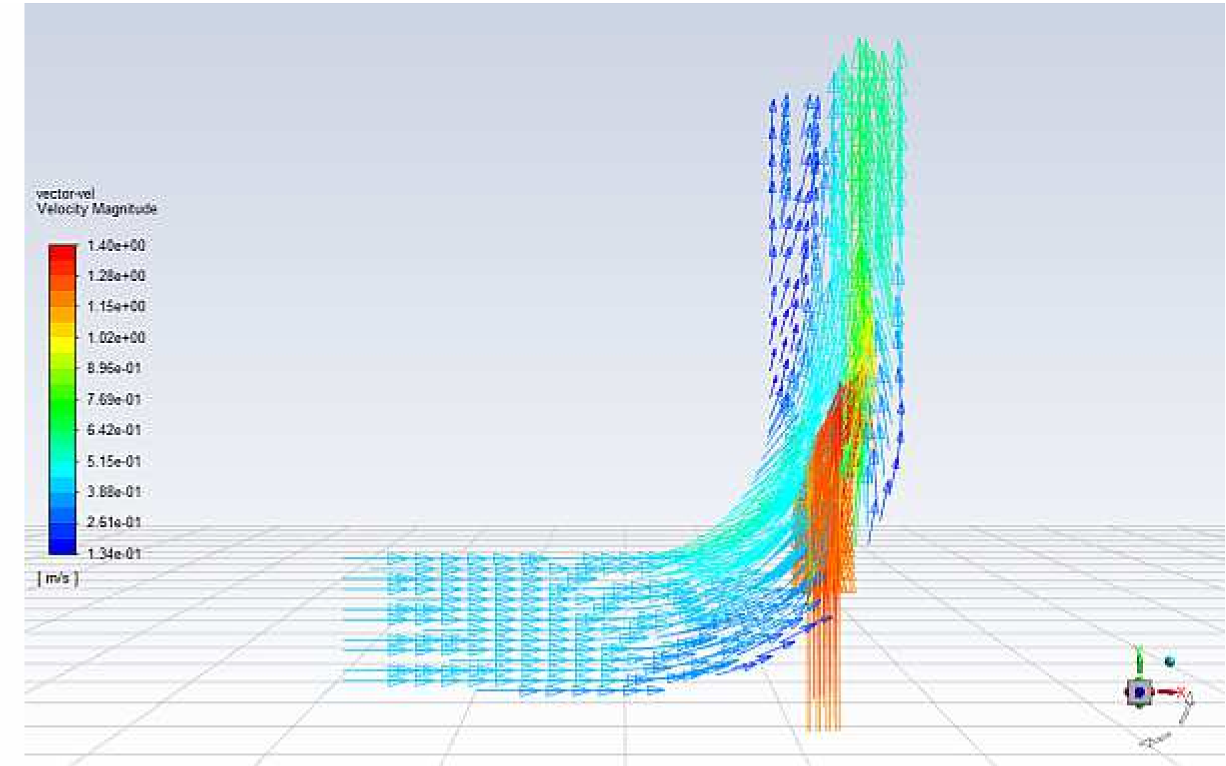


Figure 1.8: Resized Velocity Vectors

- Close the **Vectors** dialog box.
- Zoom in on the vectors in the display.

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

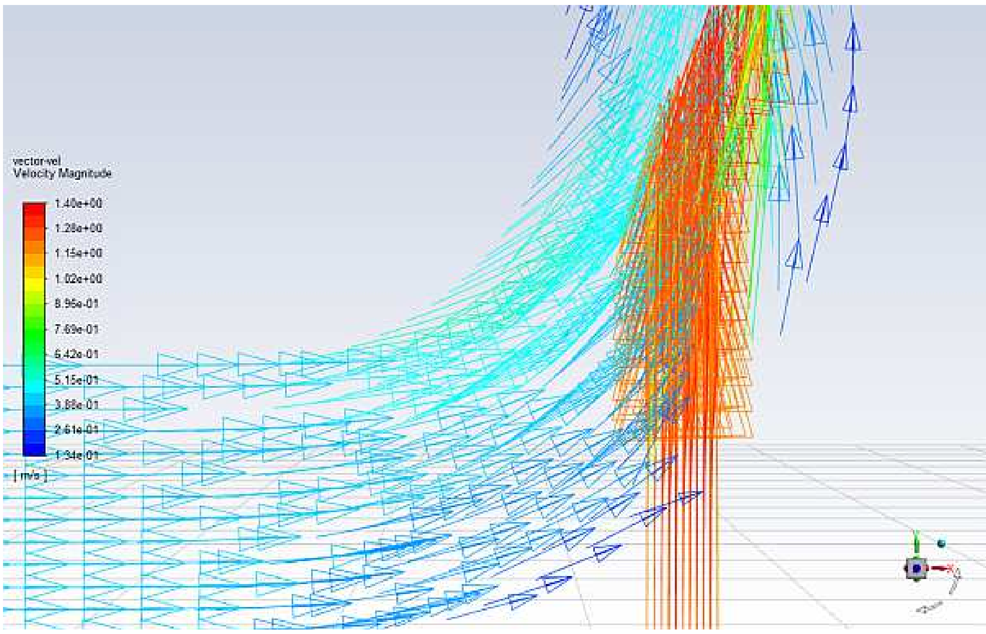
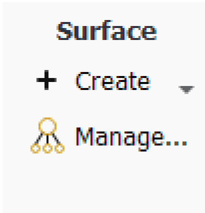


Figure 1.9: Magnified View of Resized Velocity Vectors

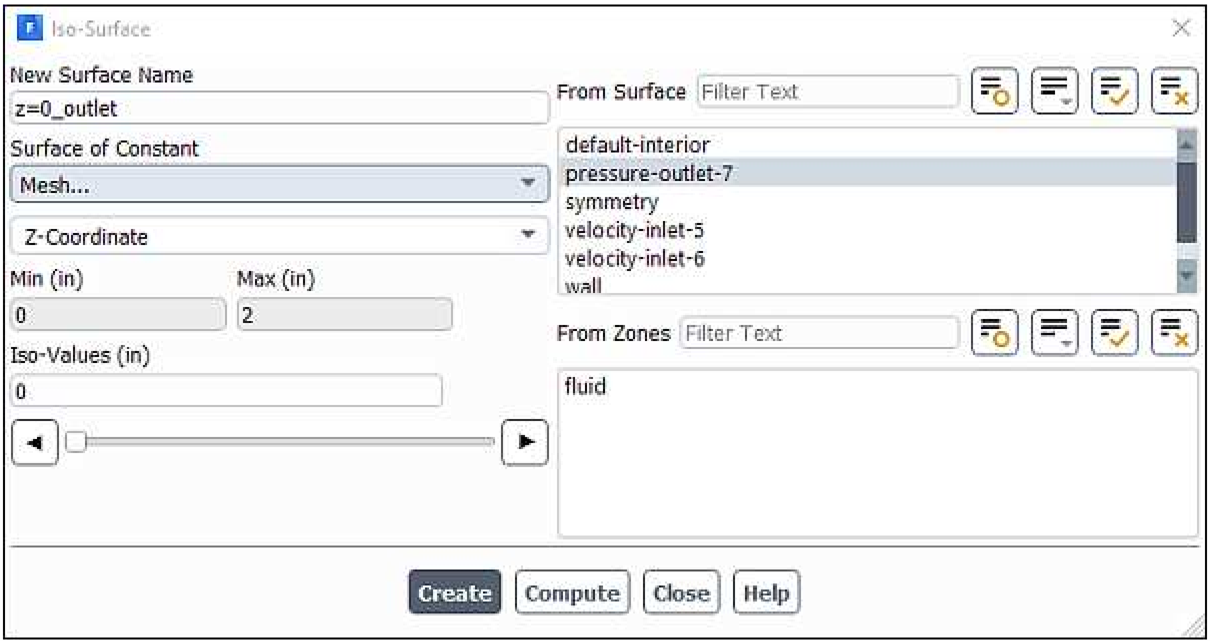
- Zoom out to the original view.

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

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...



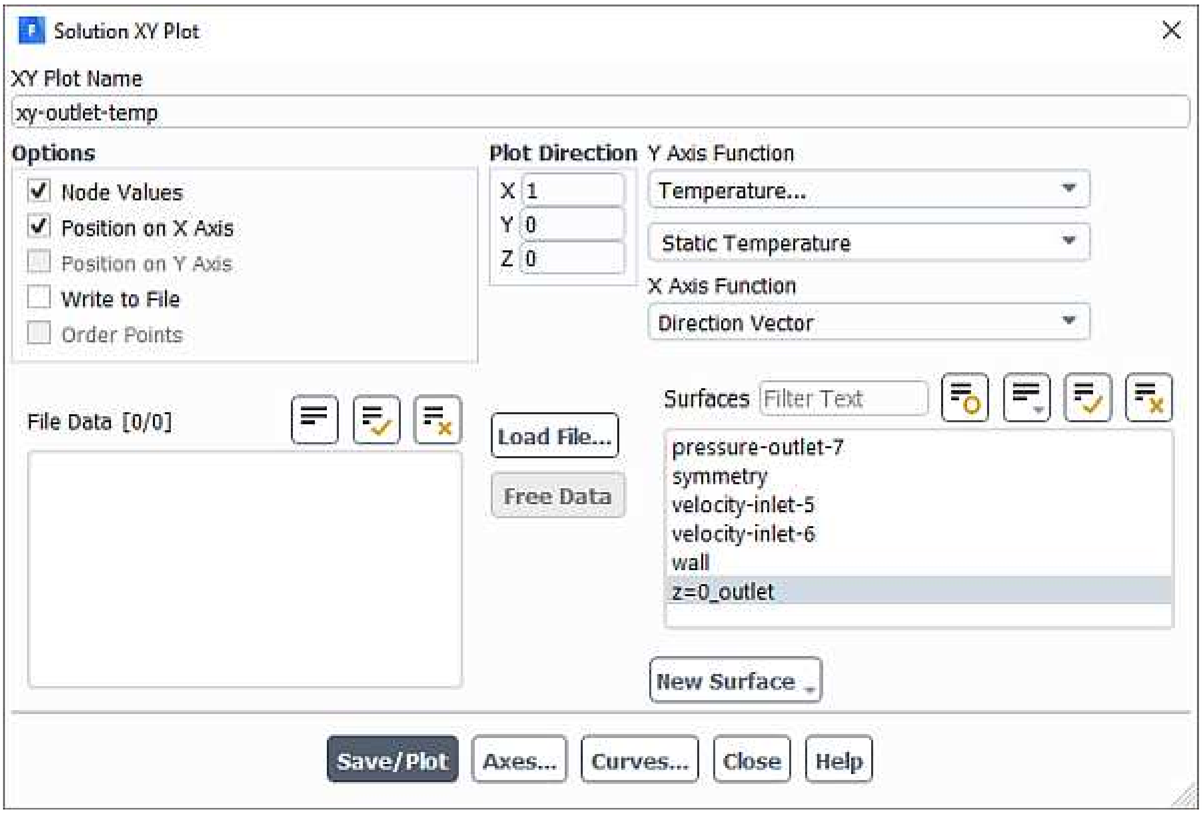
- Enter `z=0_outlet` for **New Surface Name**.
- Select **Mesh...** and **Z-Coordinate** from the **Surface of Constant** drop-down lists.
- Click **Compute** to obtain the extent of the mesh in the z-direction.
- The range of values in the z-direction is displayed in the **Min** and **Max** fields.
- Retain the default value of `0` inches for **Iso-Values**.
- Select `pressure-outlet-7` from the **From Surface** selection list.
- 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.

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

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).

Results → Plots → XY Plot → New...



- Enter `xy-outlet-temp` for **XY Plot Name**.
- Select **Temperature...** and **Static Temperature** from the **Y Axis Function** drop-down lists.
- Select the **z=0_outlet** surface you just created from the **Surfaces** selection list.
- Click **Save/Plot**.
- Enable **Write to File** in the **Options** group box.

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

- Click Write....
- In the **Select File** dialog box, enter `outlet_temp1.xy` for **XY File**.
- Click **OK** to save the temperature data and close the **Select File** dialog box.
- Close the **Solution XY Plot** dialog box.

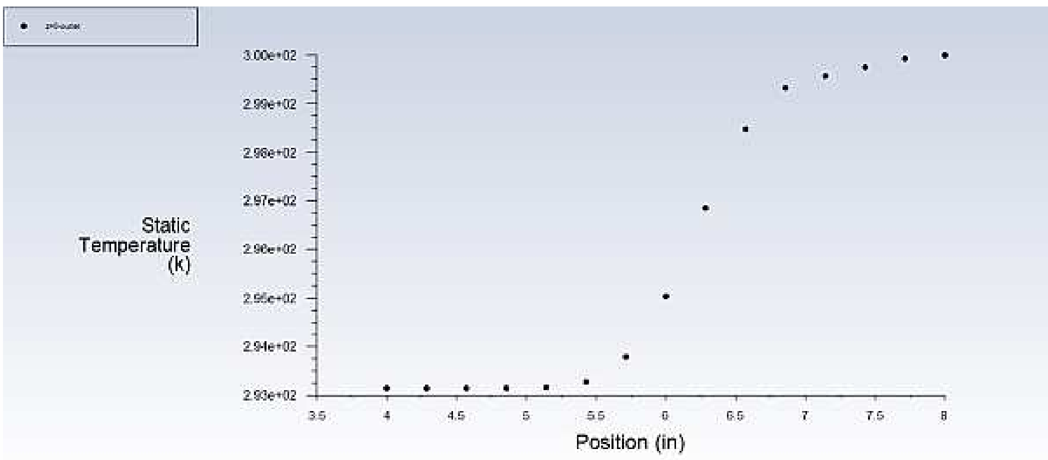
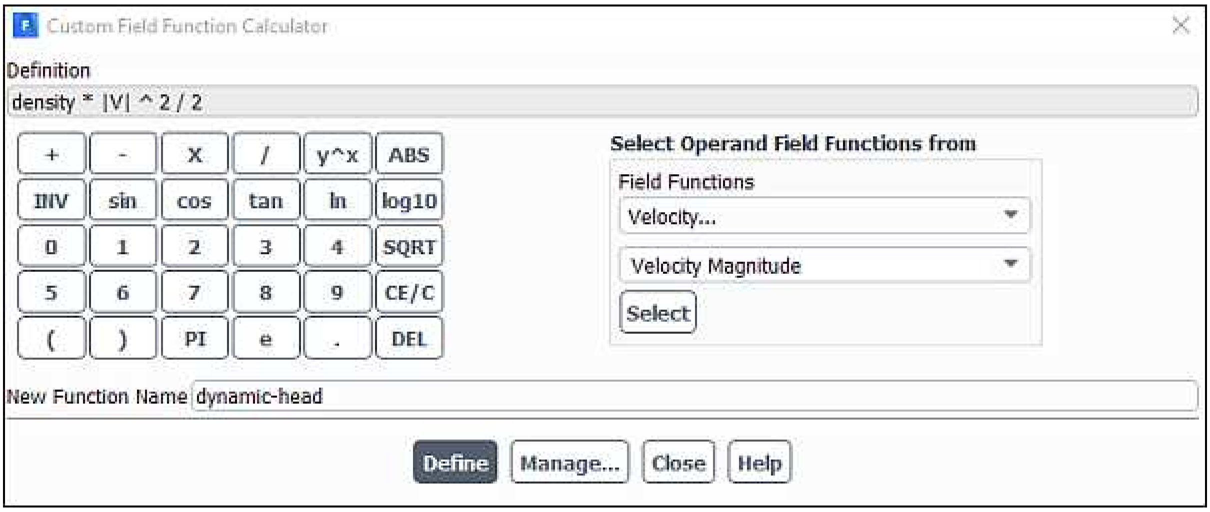


Figure 1.10: Outlet Temperature Profile for the Initial Solution

Define a custom field function for the dynamic head formula.

User-Defined → Field Functions → Custom...



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

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

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

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

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

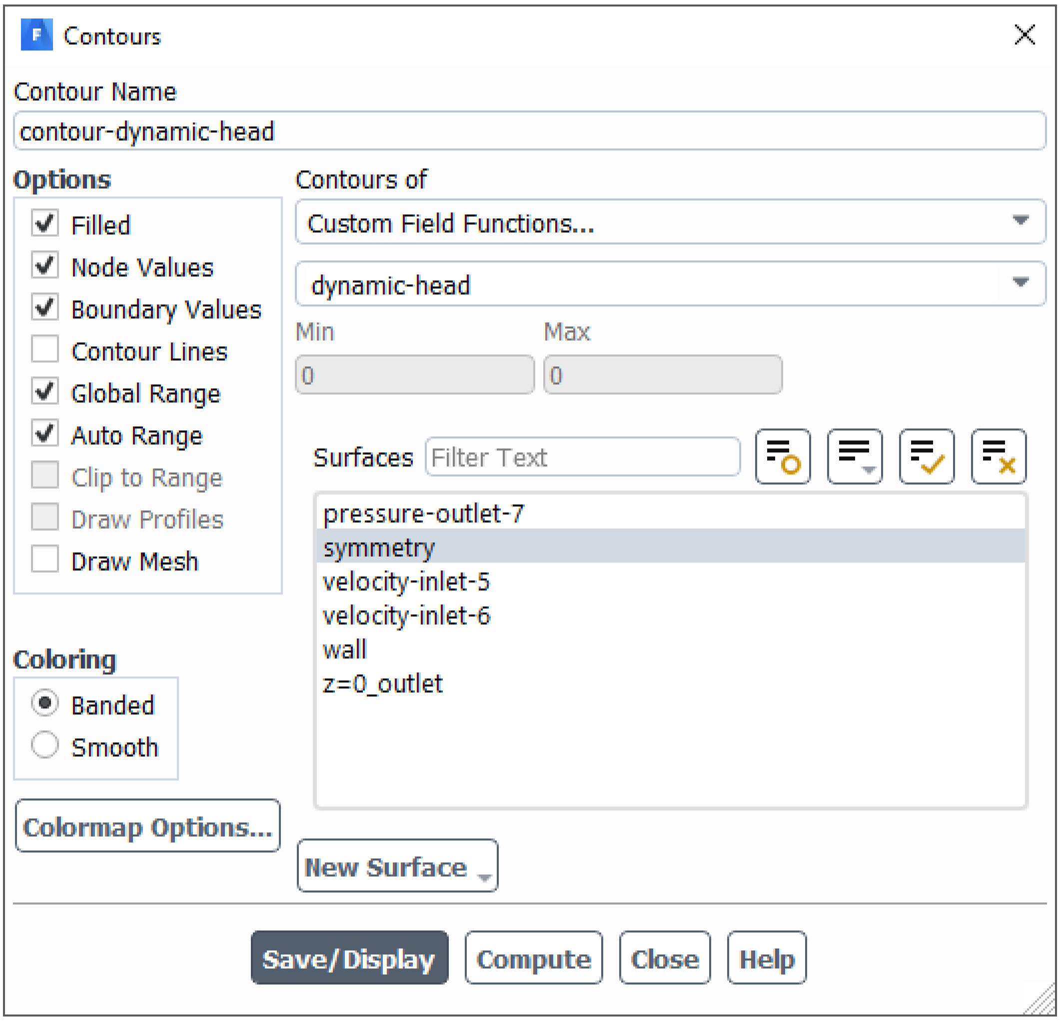
Enter `dynamic-head` for **New Function Name**.

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.

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

Results → Graphics → Contours → New...



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

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

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

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

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

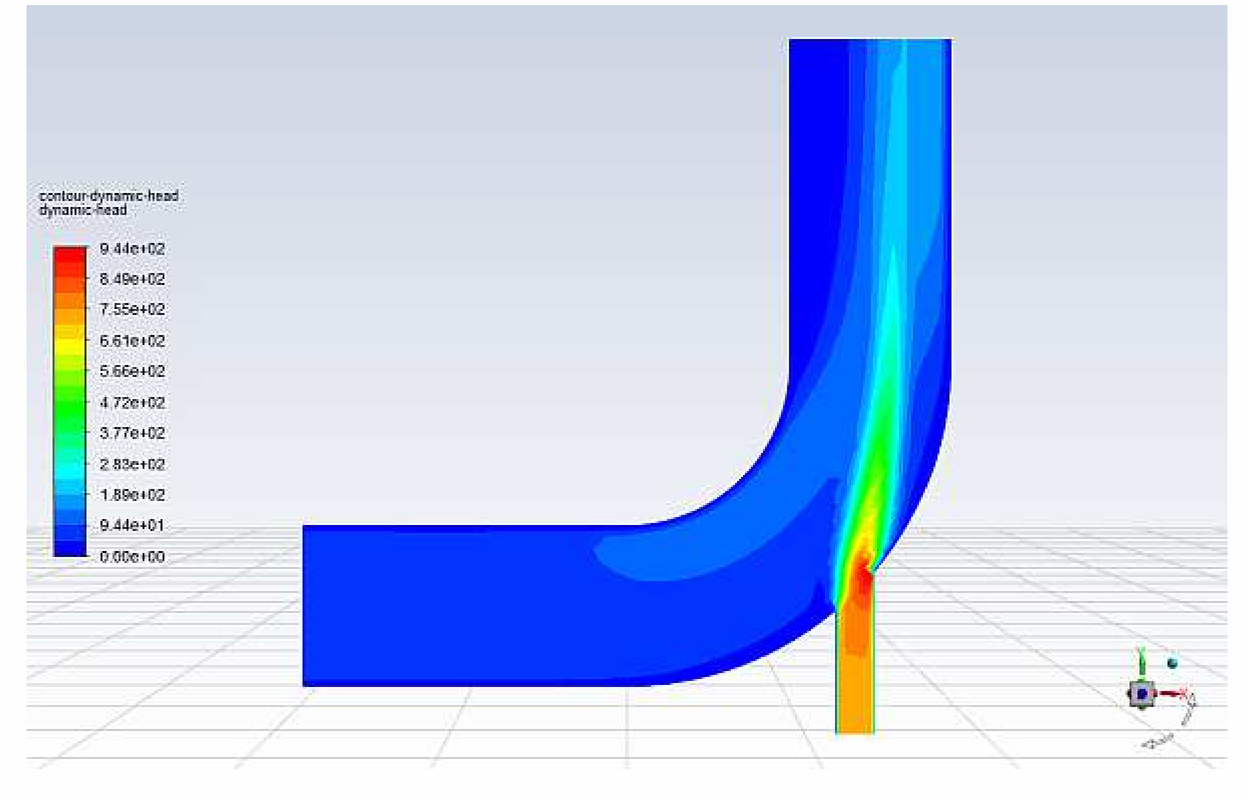


Figure 1.11: Contours of the Dynamic Head Custom Field Function

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...

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.

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

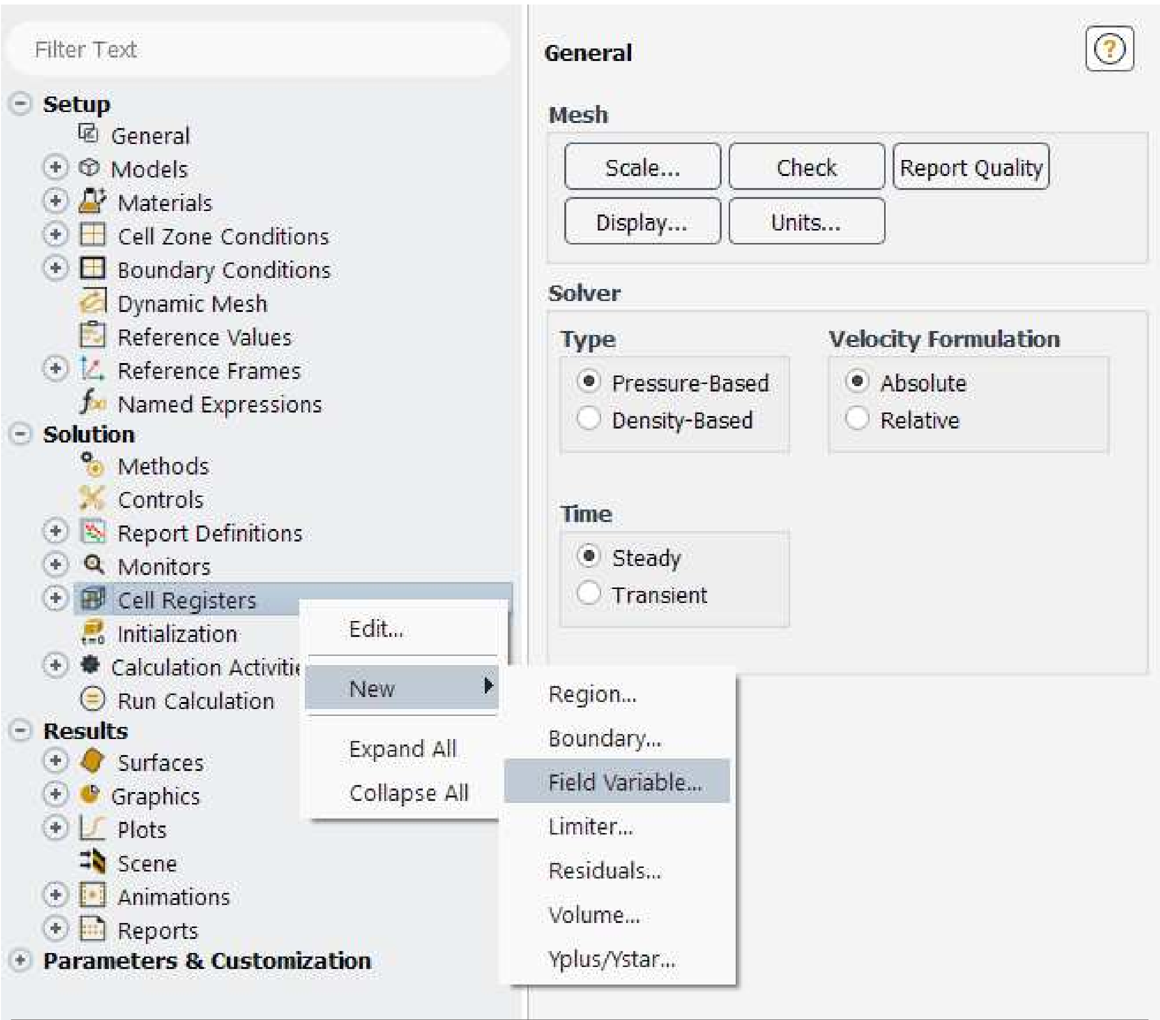
Click **OK** to overwrite the files that you had saved earlier.

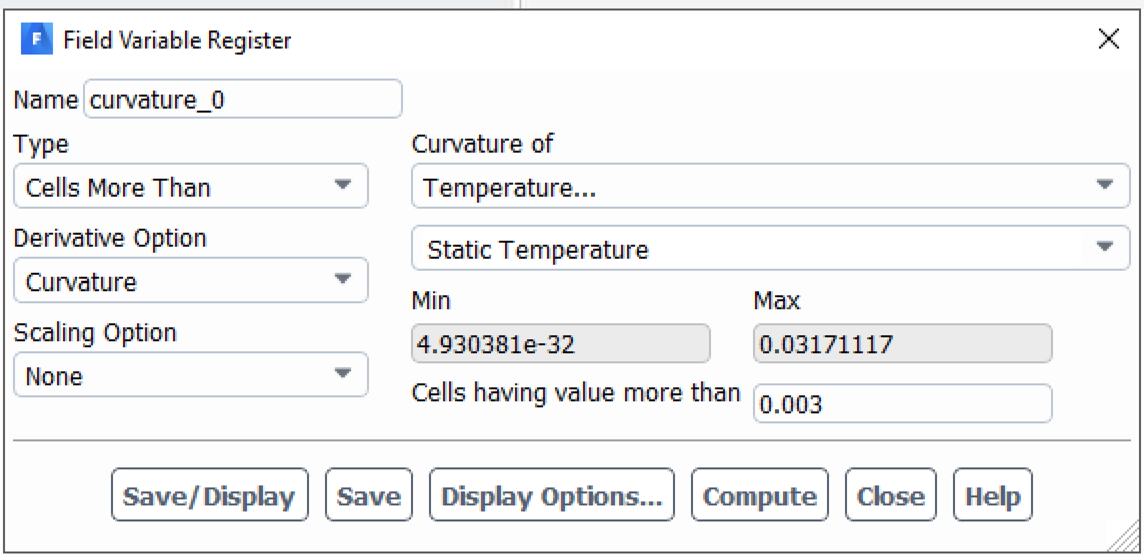
1.4.8. Adapting the Mesh

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.

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

Solution → Cell Registers New → Field Variable...





- Select **Cells More Than** from the **Type** drop-down list.
- Select **Curvature** from the **Derivative Option** drop-down list.
- Select **Temperature...** and **Static Temperature** from the **Curvature of** drop-down list.
- Click **Compute**.

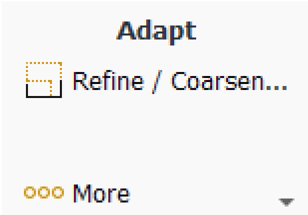
ANSYS Fluent will update the **Min** and **Max** values to show the minimum and maximum temperature gradient.

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

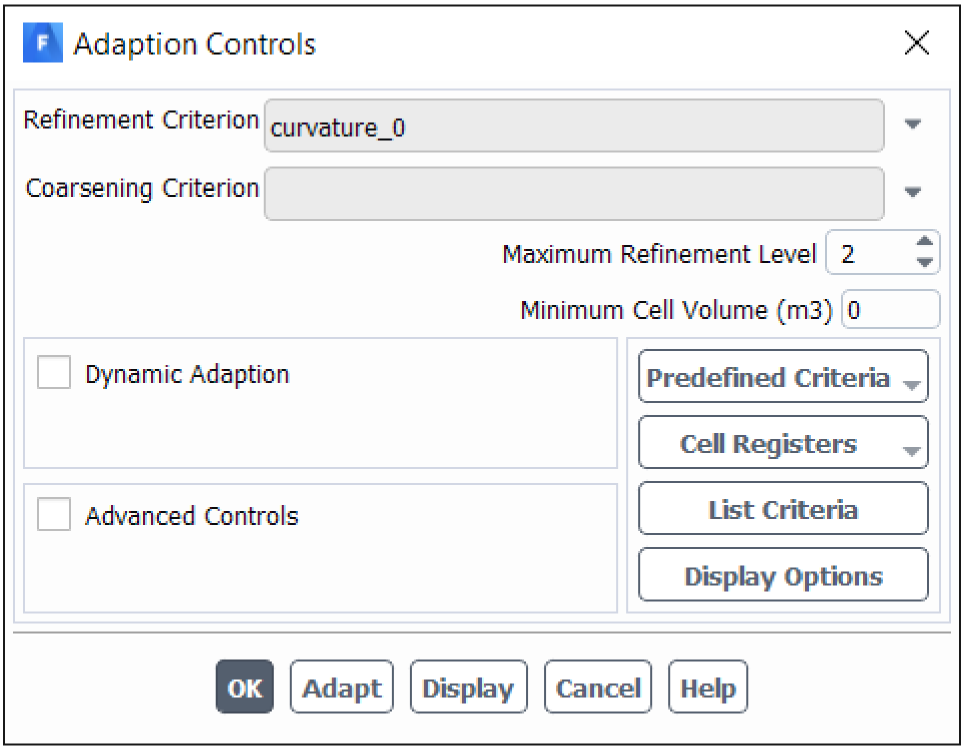
A general rule is to use 10% of the maximum gradient when setting the value for refinement.

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

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...



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.

Click **Adapt**.

Click **Display**.

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

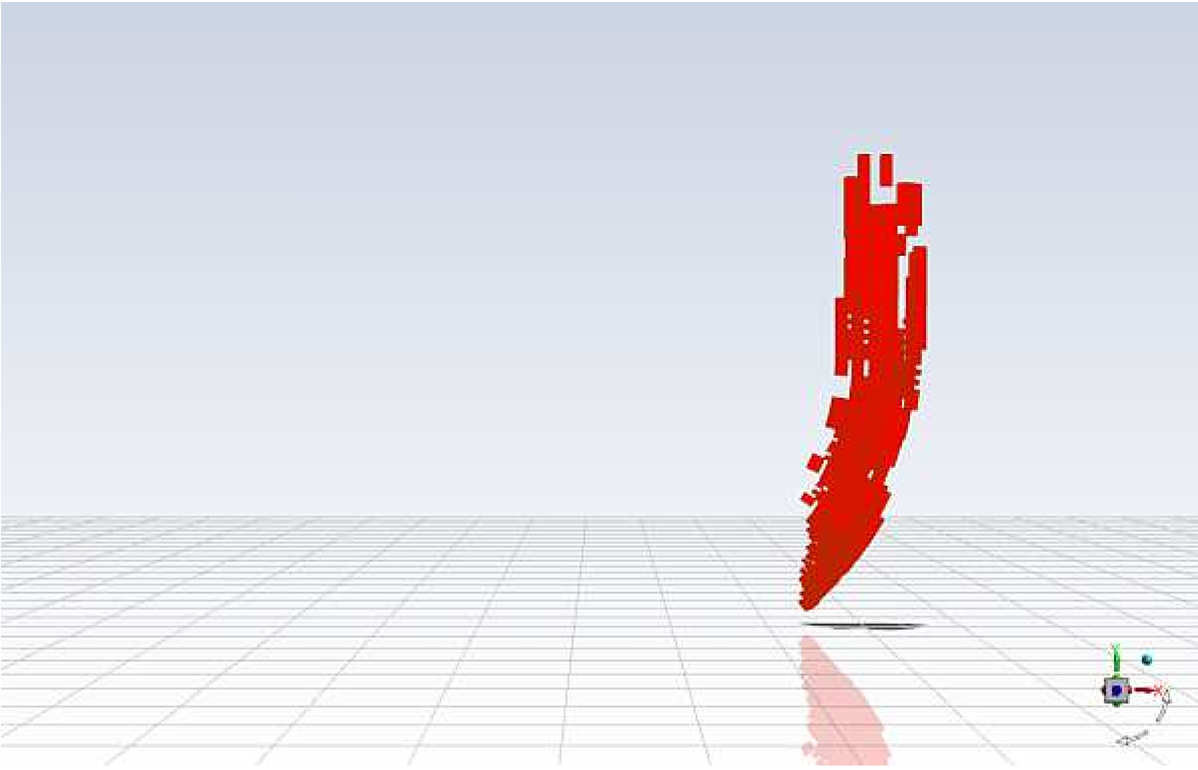
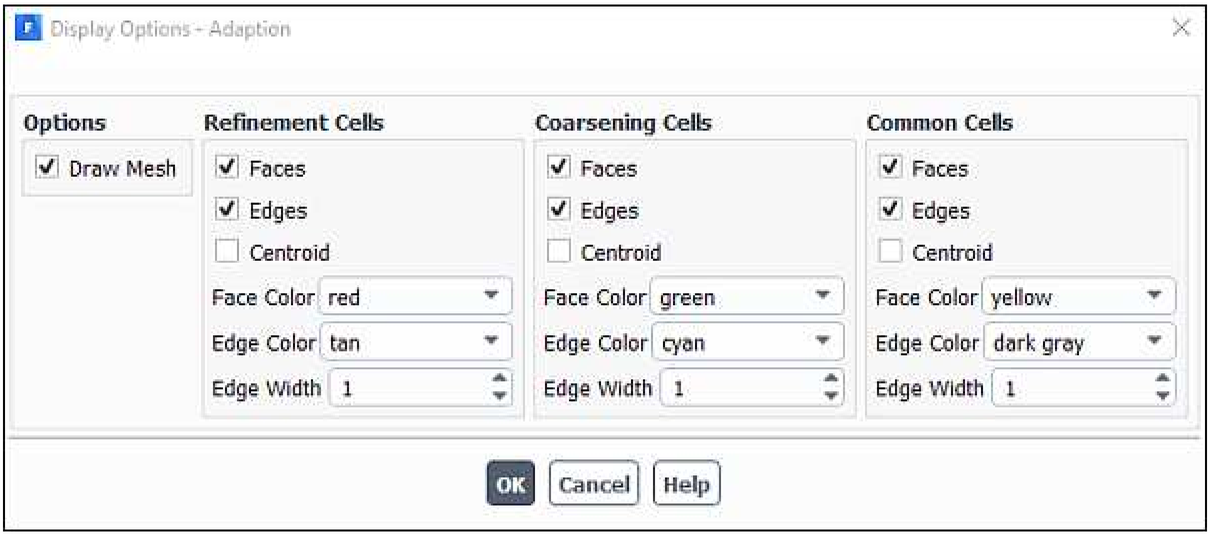


Figure 1.12: Cells Marked for Adaption

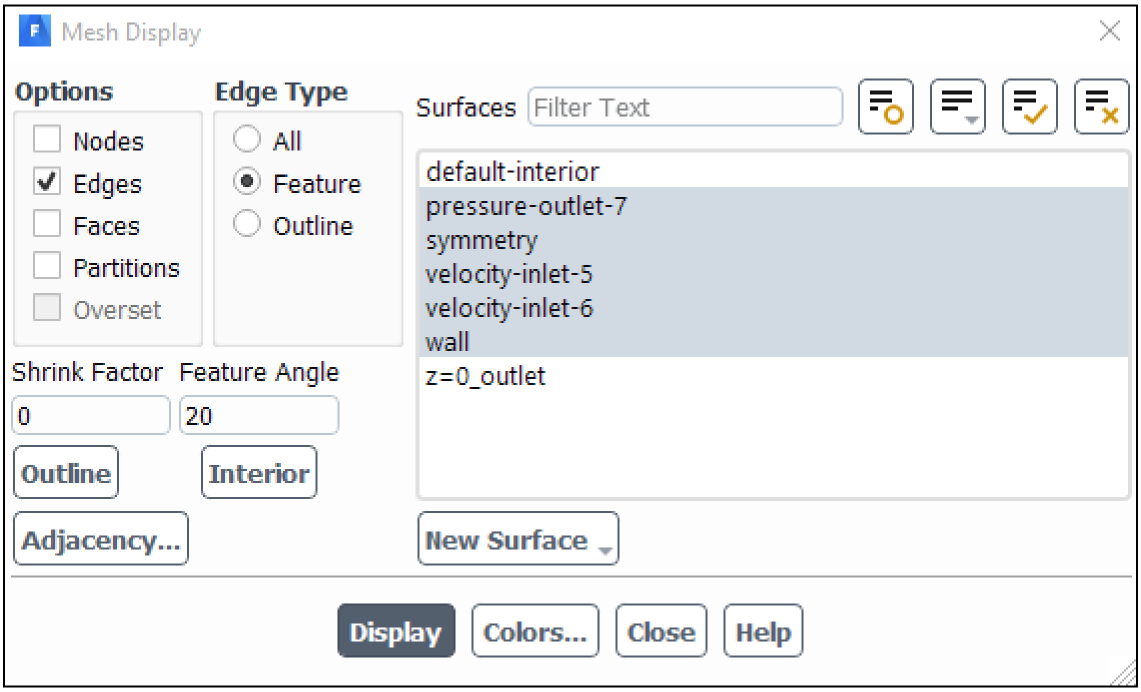
Extra — You can change the way ANSYS Fluent displays cells marked for adaption (Figure 1.13: Alternative Display of Cells Marked for Adaption) by performing the following steps:

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



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

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



- Ensure that only the **Edges** option is enabled in the **Options** group box.
- Select **Feature** from the **Edge Type** list.
- Select all of the items except **z=0_outlet** from the **Surfaces** selection list.
- Click **Display** and close the **Mesh Display** dialog box.
- Click **OK** to close the **Display Options - Adaption** dialog box.
- Click **Display** in the **Adaption Controls** dialog box.
- Rotate the view and zoom in to get the display shown in Figure 1.13: Alternative Display of Cells Marked for Adaption.

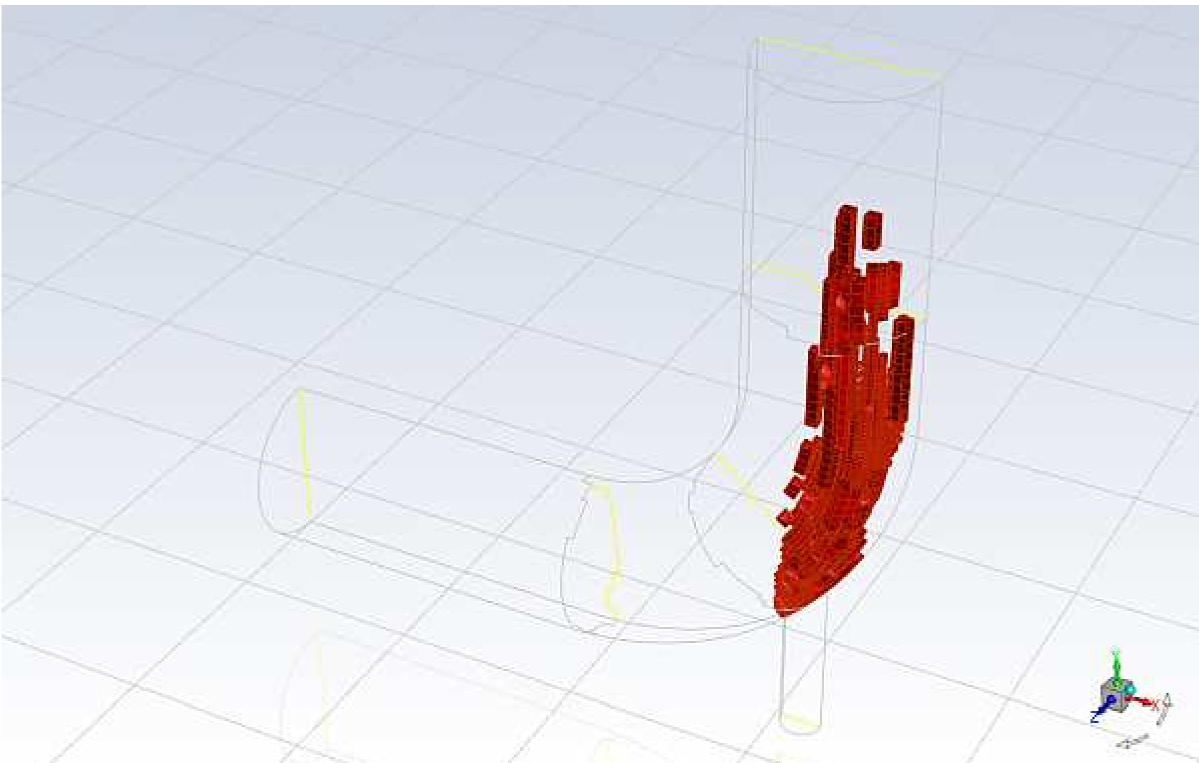


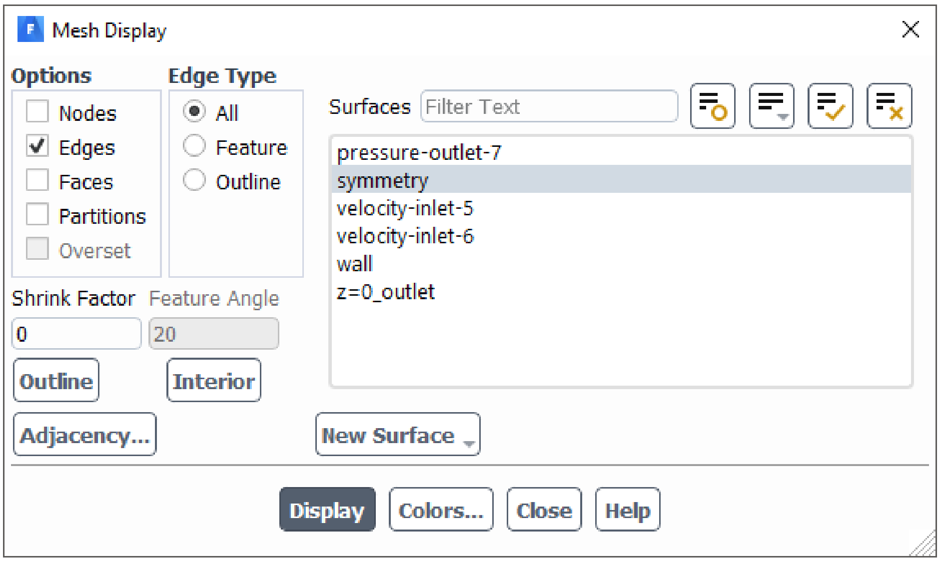
Figure 1.13: Alternative Display of Cells Marked for Adaption

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

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

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

Domain → Mesh → Display...



- Select **All** from the **Edge Type** list.
- Deselect all of the highlighted items from the **Surfaces** selection list except for **symmetry**.

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

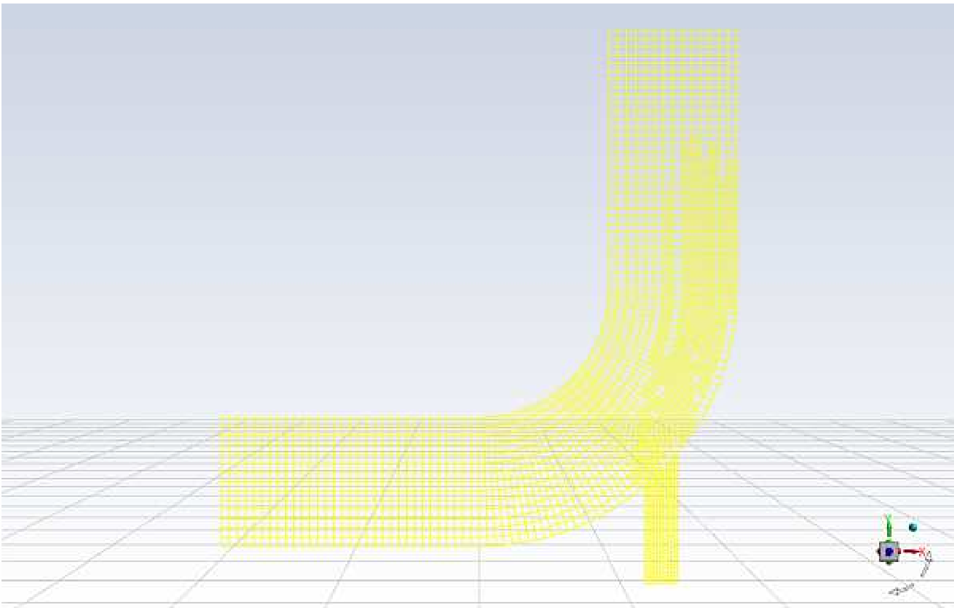
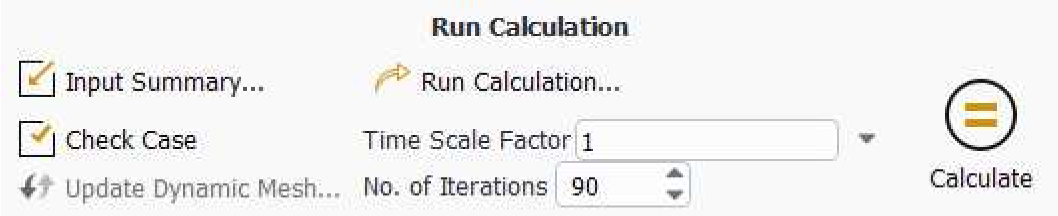


Figure 1.14: The Adapted Mesh

Request an additional 90 iterations.

Solution → Run Calculation → Calculate



The solution will converge as shown in [Figure 1.15: The Complete Residual History] and [Figure 1.16: Convergence History of Mass-Weighted Average Temperature].

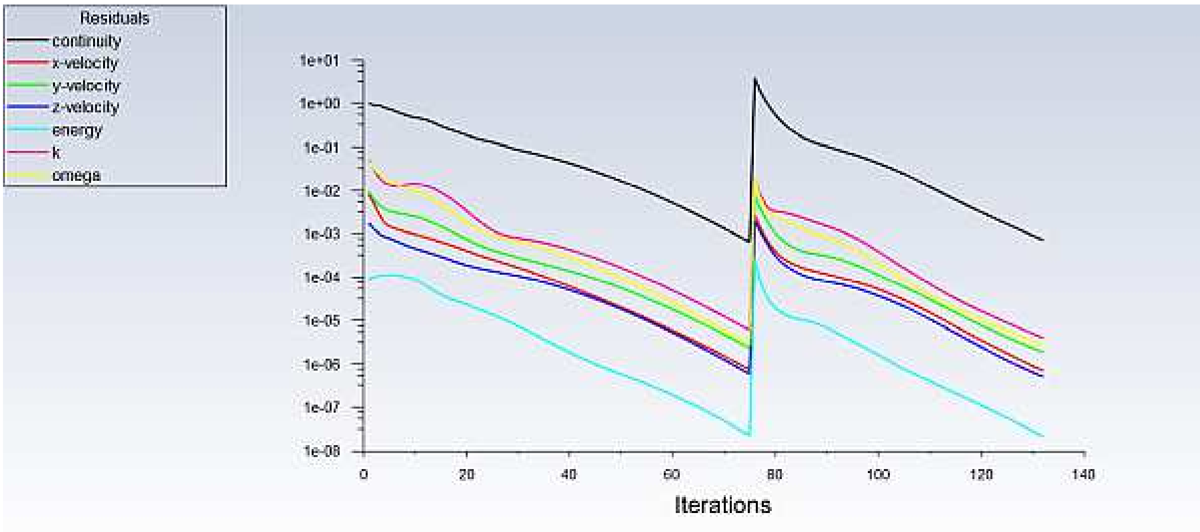


Figure 1.15: The Complete Residual History

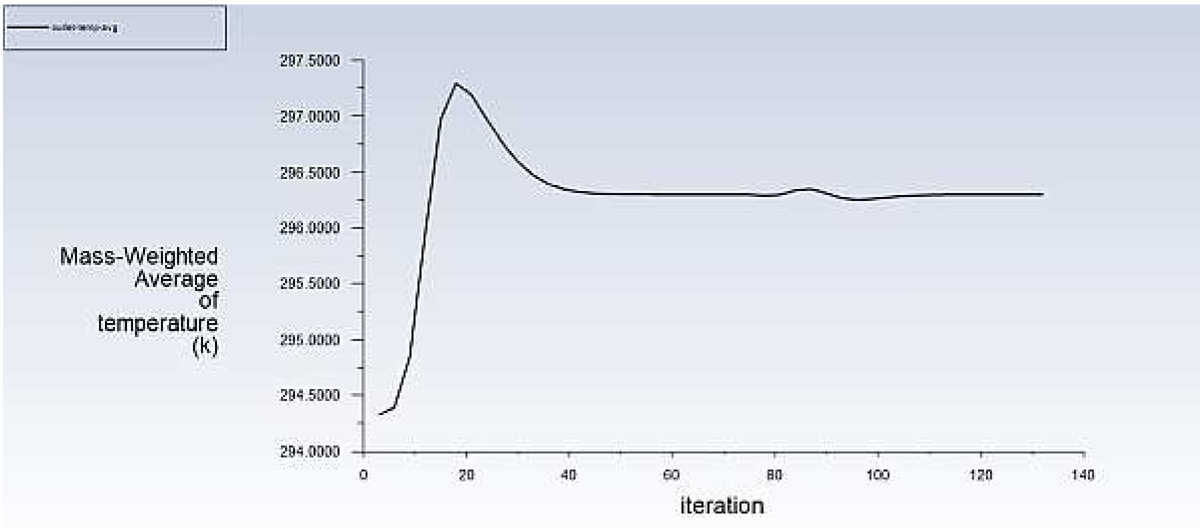


Figure 1.16: Convergence History of Mass-Weighted Average Temperature

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...

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

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.

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).

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

Results → Graphics → Contours → contour-temp Display

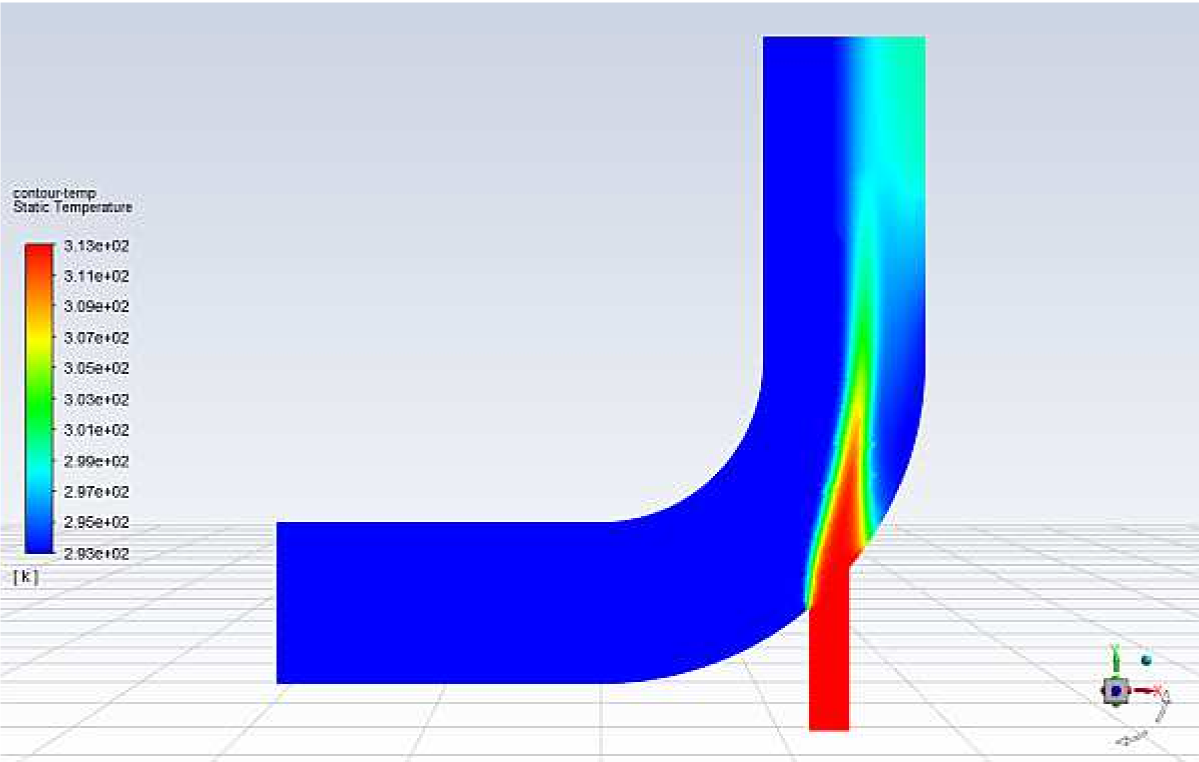


Figure 1.17: Filled Contours of Temperature Using the Adapted Mesh

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).

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

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

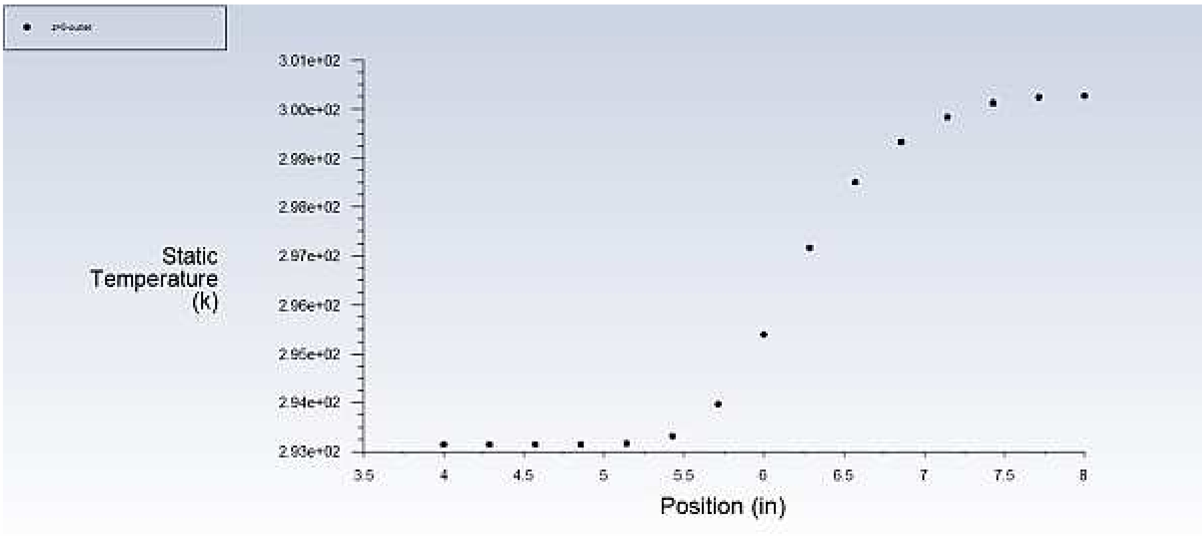


Figure 1.18: Outlet Temperature Profile for the Adapted Coupled Solver Solution

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

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

Click Write....

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

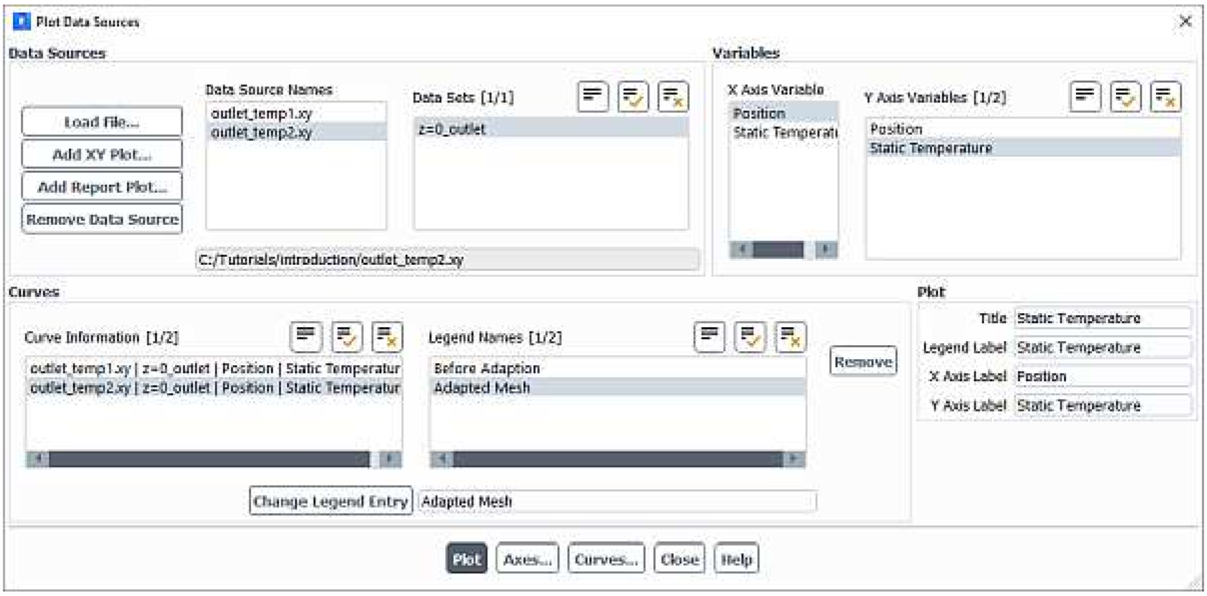
Click **OK** to save the temperature data.

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

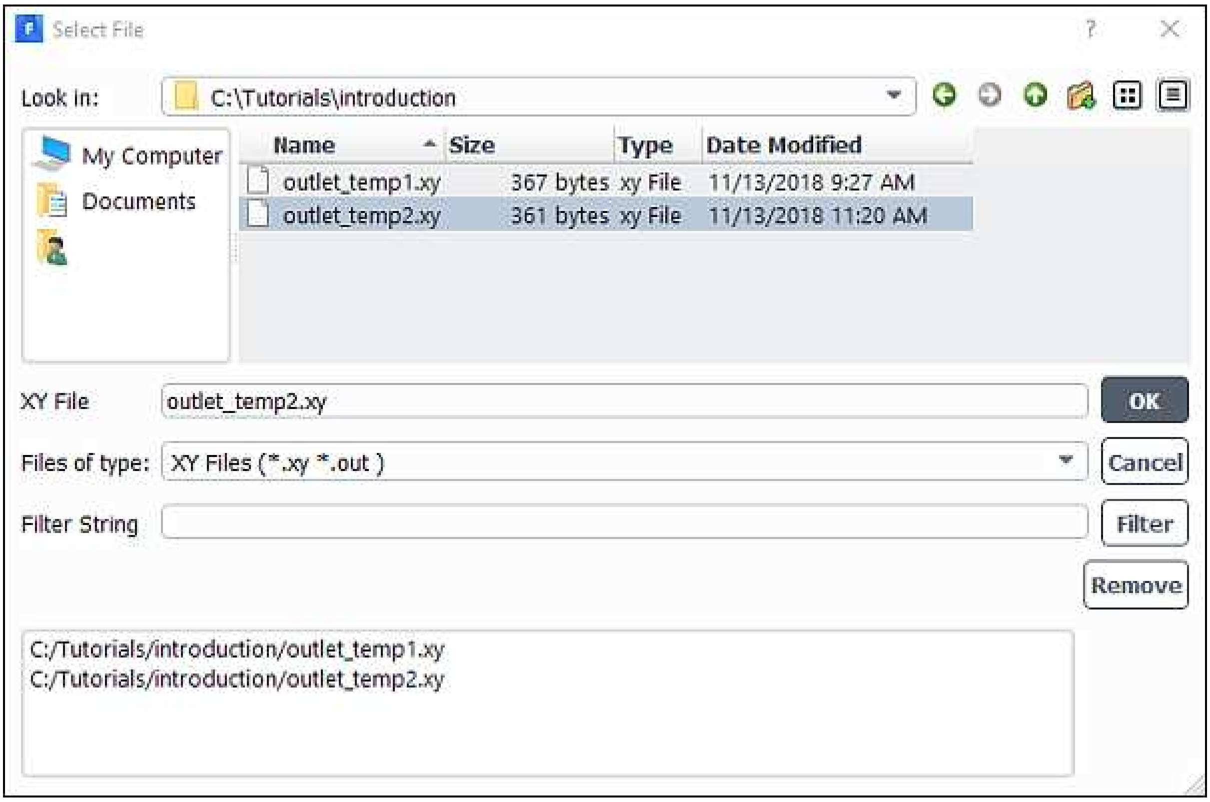
Display the outlet temperature profiles for both solutions on a single plot (Figure 1.19: Outlet Temperature Profiles for the Two Solutions).

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

Results → Plots → Data Sources...



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



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.

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

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

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

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.

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

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

Figure 1.19: Outlet Temperature Profiles for the Two Solutions 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.

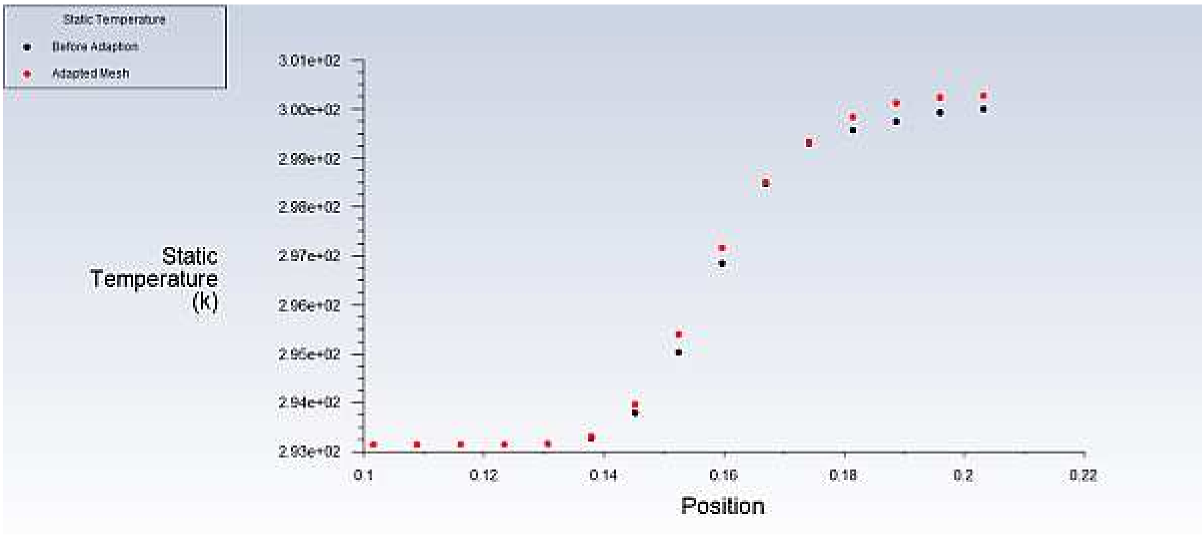


Figure 1.19: Outlet Temperature Profiles for the Two Solutions

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.