

---

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

---

## 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:

- Read an existing grid file into FLUENT.
- Use mixed units to define the geometry and fluid properties.
- Set material properties and boundary conditions for a turbulent forced convection problem.
- Initiate the calculation with residual plotting.
- Calculate a solution using the pressure-based solver.
- Visually examine the flow and temperature fields using FLUENT's postprocessing tools.
- Enable the second-order discretization scheme for improved prediction of the temperature field.
- Adapt the grid based on the temperature gradient to further improve the prediction of the temperature field.

## Prerequisites

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

## Problem Description

The problem to be considered is shown schematically in Figure 1.1. A cold fluid at 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.

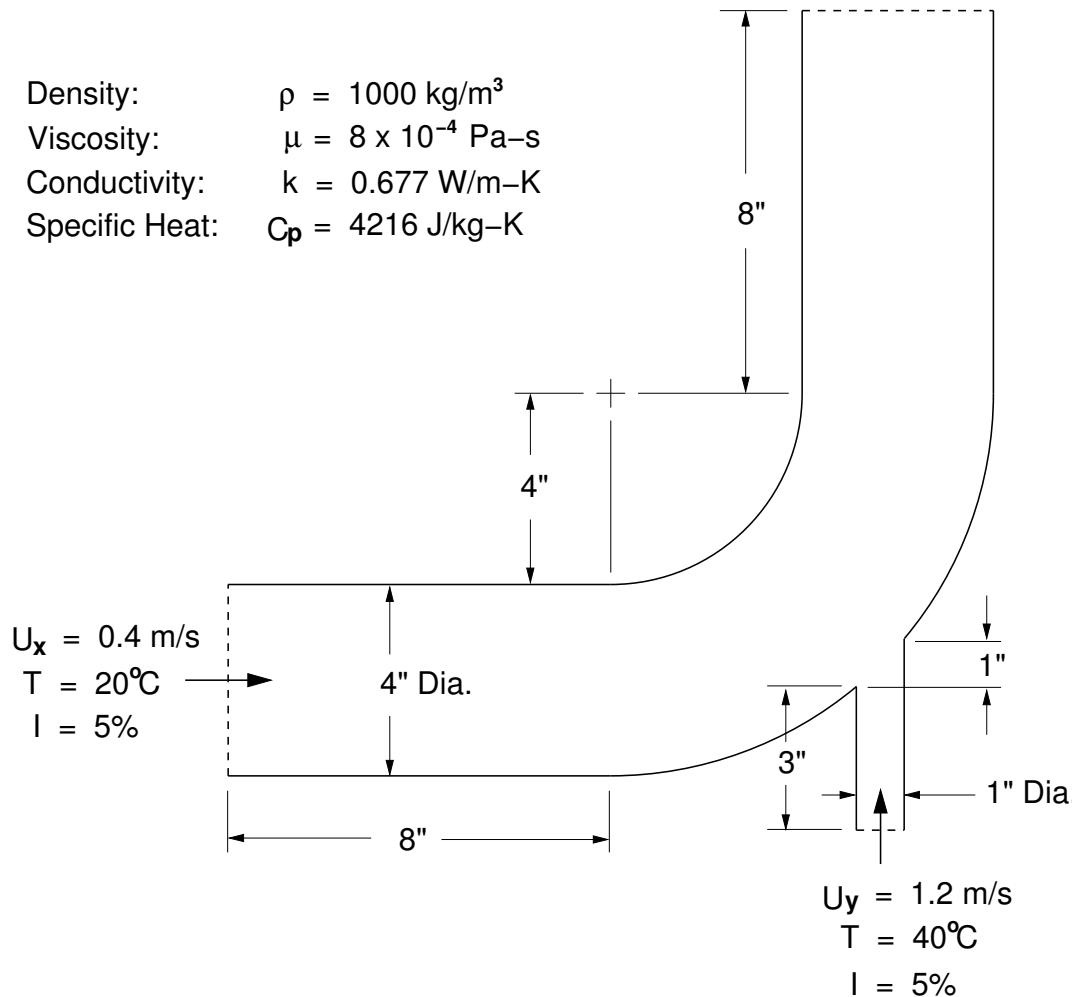


Figure 1.1: Problem Specification

## **Setup and Solution**

### **Preparation**

1. Download **introduction.zip** from the Fluent Inc. User Services Center ([www.fluentusers.com](http://www.fluentusers.com)) to your working folder. This file can be found by using the Documentation link on the **FLUENT** product page.

**OR,**

Copy **introduction.zip** from the **FLUENT** documentation CD to your working folder.

*For Linux / UNIX systems, you can find the file by inserting the CD into your CD-ROM drive and going to the following directory:*

**/cdrom/fluent6.3/help/tutfiles/**

*where **cdrom** must be replaced by the name of your CD-ROM drive.*

*For Windows systems, you can find the file by inserting the CD into your CD-ROM drive and going to the following folder:*

**cdrom:\fluent6.3\help\tutfiles\**

*where **cdrom** must be replaced by the name of your CD-ROM drive (e.g., E).*

2. Unzip **introduction.zip**.

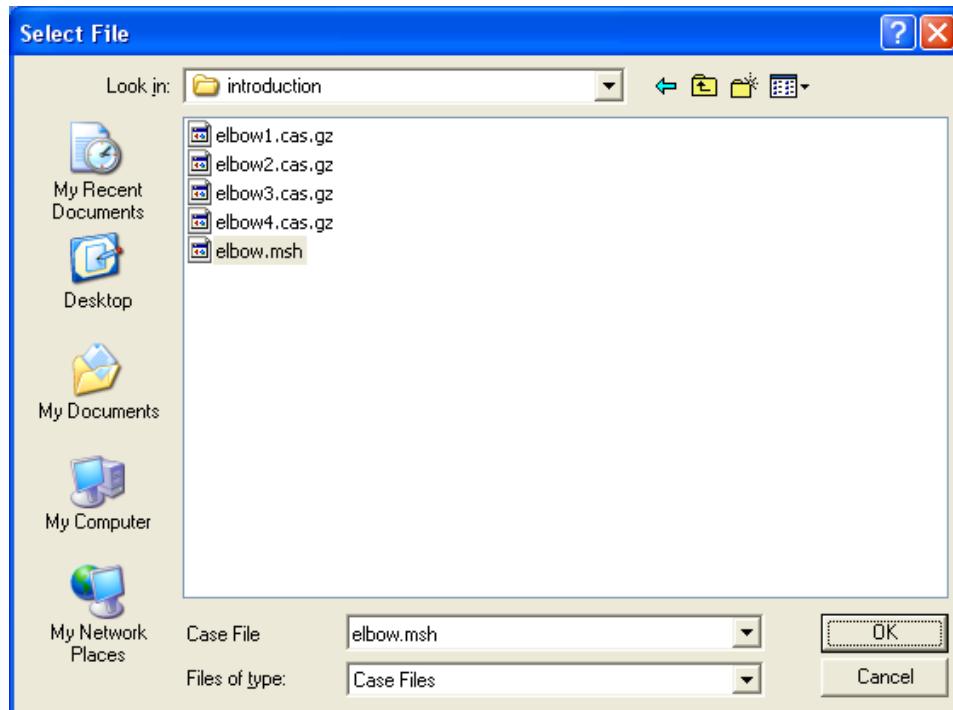
*The file **elbow.msh** can be found in the **introduction** folder created after unzipping the file.*

3. Start the 3D (3d) version of **FLUENT**.

### Step 1: Grid

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

**File** → **Read** → Case...



- Select the grid file by clicking `elbow.msh` in the `introduction` folder created when you unzipped the original file.
- Click **OK** to read the file and close the Select File dialog box.

**Note:** As the grid file is read by FLUENT, messages will appear in the console that report the progress of the conversion. FLUENT will report that 13,852 hexahedral fluid cells have been read, along with a number of boundary faces with different zone identifiers.

2. Check the grid.

**Grid** —→ Check

```
Grid Check

Grid Check

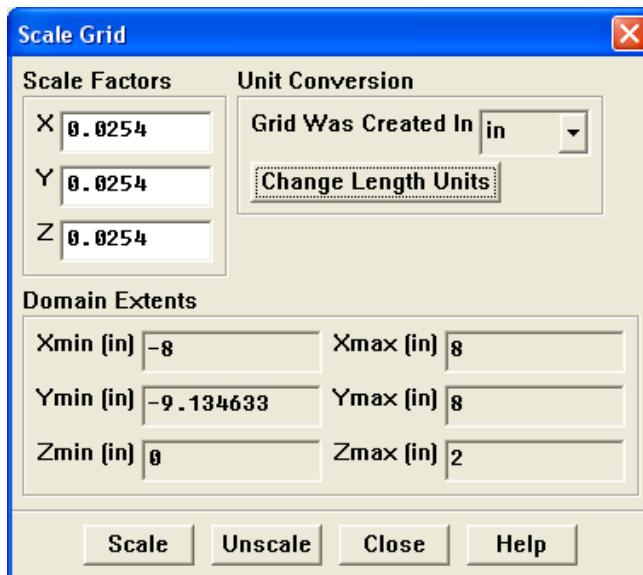
Domain Extents:
  x-coordinate: min (m) = -8.000000e+000, max (m) = 8.000000e+000
  y-coordinate: min (m) = -9.134633e+000, max (m) = 8.000000e+000
  z-coordinate: min (m) = 0.000000e+000, max (m) = 2.000000e+000
Volume statistics:
  minimum volume (m3): 5.098261e-004
  maximum volume (m3): 2.330738e-002
  total volume (m3): 1.607154e+002
Face area statistics:
  minimum face area (m2): 4.865882e-003
  maximum face area (m2): 1.017924e-001
Checking number of nodes per cell.
Checking number of faces per cell.
Checking thread pointers.
Checking number of cells per face.
Checking face cells.
Checking bridge faces.
Checking right-handed cells.
Checking face handedness.
Checking face node order.
Checking element type consistency.
Checking boundary types:
Checking face pairs.
Checking periodic boundaries.
Checking node count.
Checking nosolve cell count.
Checking nosolve face count.
Checking face children.
Checking cell children.
Checking storage.
Done.
```

**Note:** The minimum and maximum values may vary slightly when running on different platforms. The grid check will list the minimum and maximum x and y values from the grid in the default SI unit of meters, and will report a number of other grid features that are checked. Any errors in the grid will be reported at this time. In particular, you should always make sure that the minimum volume is not negative, since FLUENT cannot begin a calculation

when this is the case. In the next step, you will scale the grid so that it is in the correct unit of inches.

3. Scale the grid.

[Grid] → Scale...



- (a) Select inches from the Grid Was Created In drop-down list in the Unit Conversion group box, by first clicking the down-arrow button and then clicking the in item from the list that appears.
- (b) Click Scale to scale the grid.



Be sure to click the Scale button only once.

The reported values of the Domain Extents will be reported in the default SI unit of meters.

- (c) Click the Change Length Units button to set inches as the working unit for length.
- (d) Confirm that the domain extents are as shown in the previous panel.
- (e) Close the Scale Grid panel by clicking Close.

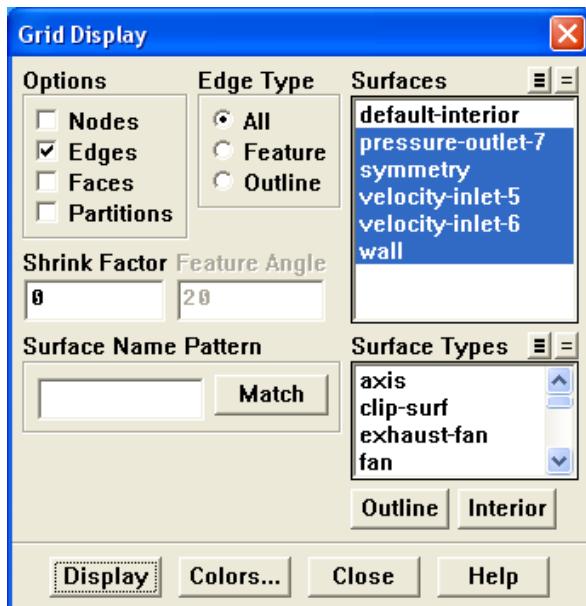
The grid is now sized correctly, and the working unit for length has been set to inches.

**Note:** Because the default SI units will be used for everything except length, there will be 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 wanted the working unit for length to be something other than inches

(e.g., millimeters), you would have to open the Set Units panel from the Define pull-down menu and make the appropriate change.

- Display the grid (Figure 1.2).

Display → Grid...



- Retain the default selection of all the items in the Surfaces selection list except default-interior.

**Note:** A list item is selected if it is highlighted, and deselected if it is not highlighted. You can select and deselect items by clicking on the text.

- Click Display to open a graphics window and display the grid.
- Close the Grid Display panel.

**Extra:** You can use the right mouse button to probe for grid information in the graphics window. If you click the right mouse button on any node in the grid, information will be displayed in the FLUENT console about the associated zone, including the name of the zone. 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. You can perform any of the following actions in the graphics window:

- Rotate the view.

Drag the mouse while pressing the left mouse button. Release the mouse button when the viewing angle is satisfactory.

- *Translate the view.*

*Click the middle mouse button once at any point in the display to center the view at that point.*

- *Zoom in to magnify a portion of the display.*

*Drag the mouse to the right and either up or down while pressing the middle mouse button. This action will cause a white rectangle to appear in the display. When you release the mouse button, a new view will be displayed which consists entirely of the contents of the white rectangle.*

- *Zoom out to reduce the magnification.*

*Drag the mouse to the left and either up or down while pressing the middle mouse button. This action will cause a white rectangle to appear in the display. When you release the mouse button, the magnification of the view will be reduced by an amount that is inversely proportional to the size of the white rectangle. The new view will be centered at the center of the white rectangle.*

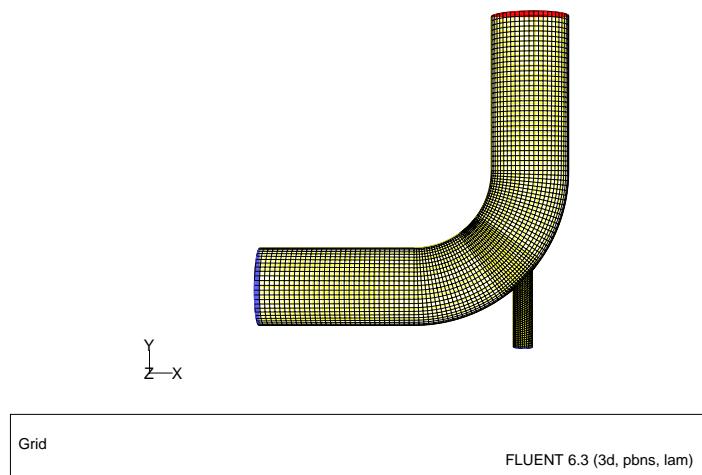
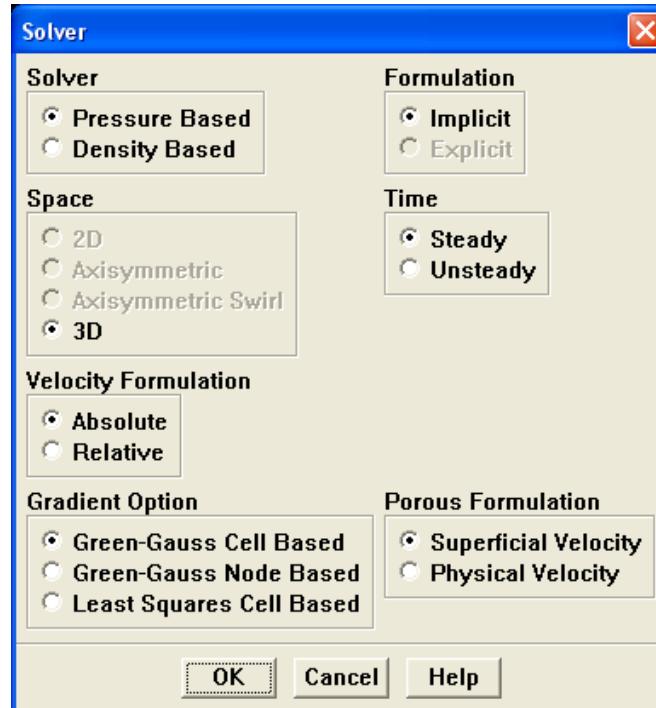


Figure 1.2: The Hexahedral Grid for the Mixing Elbow

## Step 2: Models

1. Retain the default solver settings.

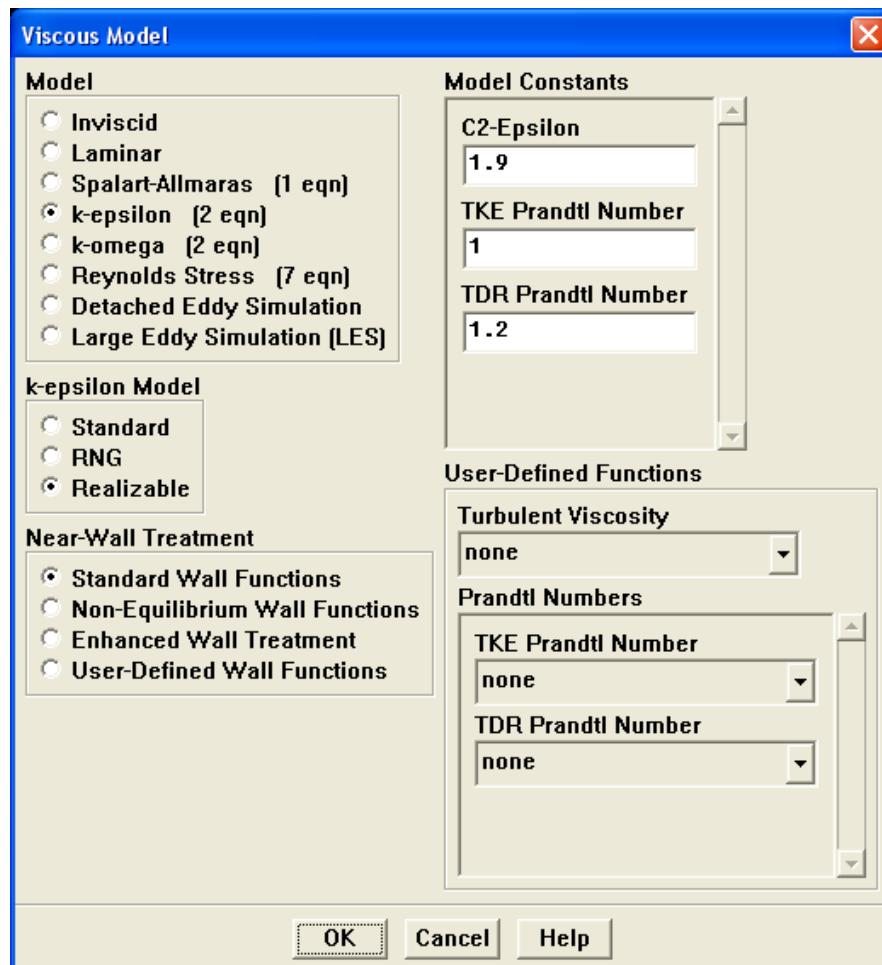
Define → Models → Solver...



- (a) Retain all of the default settings.
- (b) Click OK to close the Solver panel.

2. Turn on the  $k-\epsilon$  turbulence model.

**Define** → **Models** → **Viscous...**



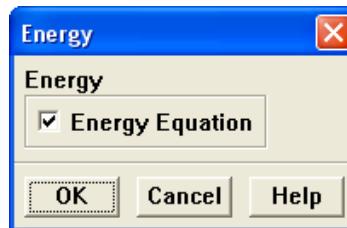
- (a) Select k-epsilon from the Model list by clicking the radio button or the text, so that a black dot appears in the radio button.

*The Viscous Model panel will expand.*

- (b) Select Realizable from the k-epsilon Model list.  
 (c) Click OK to accept the model and close the Viscous Model panel.

3. Enable heat transfer by activating the energy equation.

**Define** → **Models** → **Energy...**



- Enable the **Energy Equation** option by clicking the check box or the text.

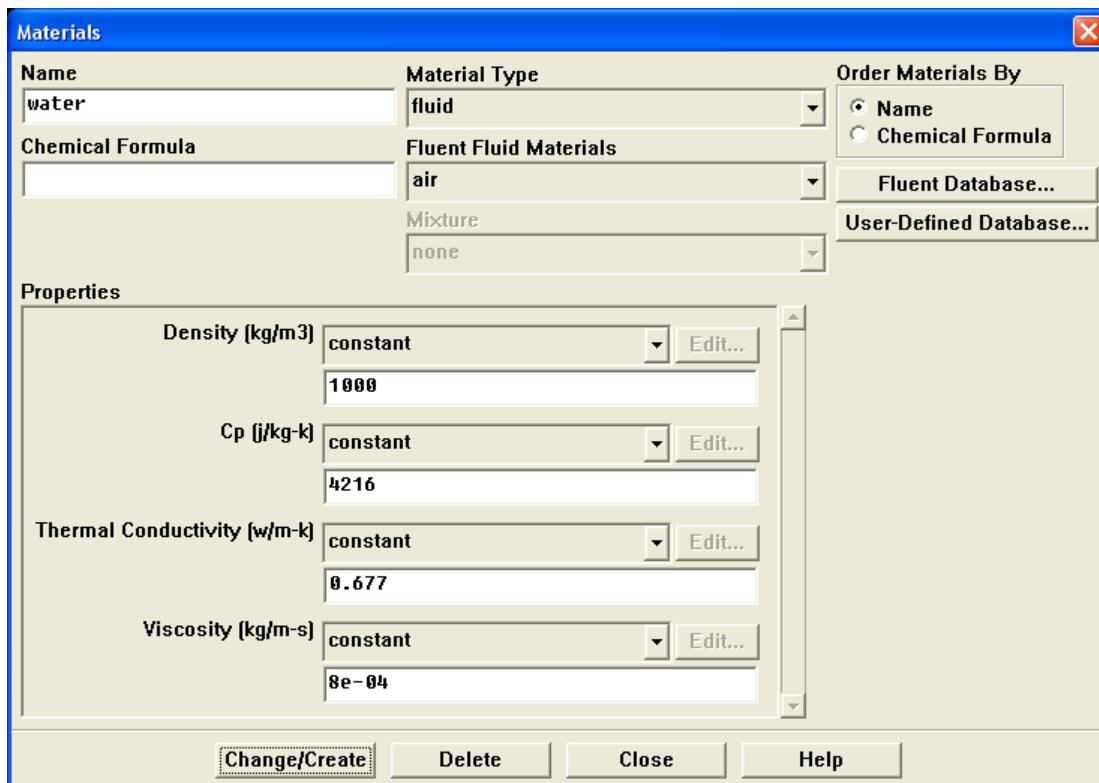
**Note:** An option is enabled when there is a check mark in the check box, and disabled when the check box is empty.

- Click OK to close the Energy panel.

### Step 3: Materials

- Create a new material called water.

**Define** → **Materials...**

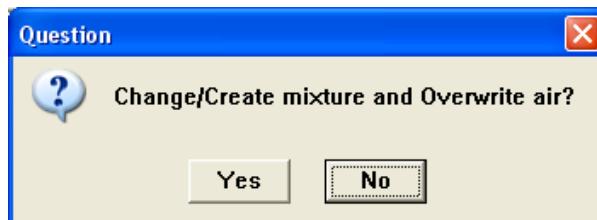


- (a) Enter **water** for Name by double-clicking in the text-entry box under Name and typing with the keyboard.
- (b) Enter the following values in the Properties group box:

Property	Value
Density	1000 kg/m <sup>3</sup>
$C_p$	4216 J/kg – K
Thermal Conductivity	0.677 W/m – K
Viscosity	8e-04 kg/m – s

- (c) Click **Change/Create**.

A Question dialog box will open, asking if you want to overwrite air. Click **No** so that the new material water is added to the list of materials which originally contained only air.

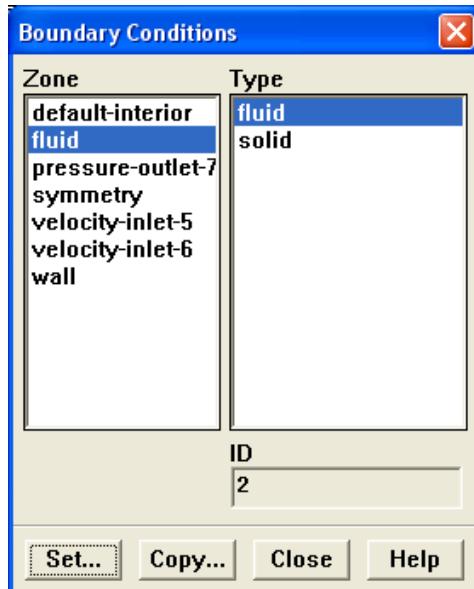


**Extra:** You could have copied the material water-liquid [h2o<1>] from the materials database (accessed by clicking the Fluent Database... button). If the properties in the database are different from those you wish to use, you can edit the values in the Properties group box in the Materials panel and click Change/Create to update your local copy (the database copy will not be affected).

- (d) Make sure that there are now two materials defined locally by examining the Fluent Fluid Materials drop-down list.
- (e) Close the Materials panel.

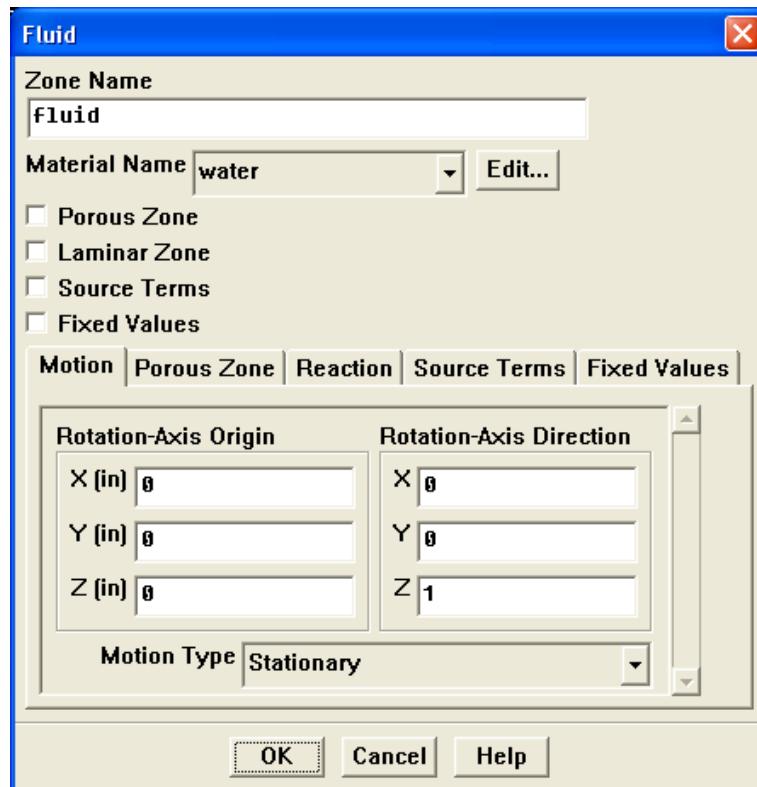
## Step 4: Boundary Conditions

[Define] → Boundary Conditions...



1. Set the boundary conditions for the fluid (fluid).
  - (a) Select fluid from the Zone selection list.

- (b) Click Set... to open the Fluid panel.



- i. Select water from the Material Name drop-down list.
- ii. Click OK to close the Fluid panel.

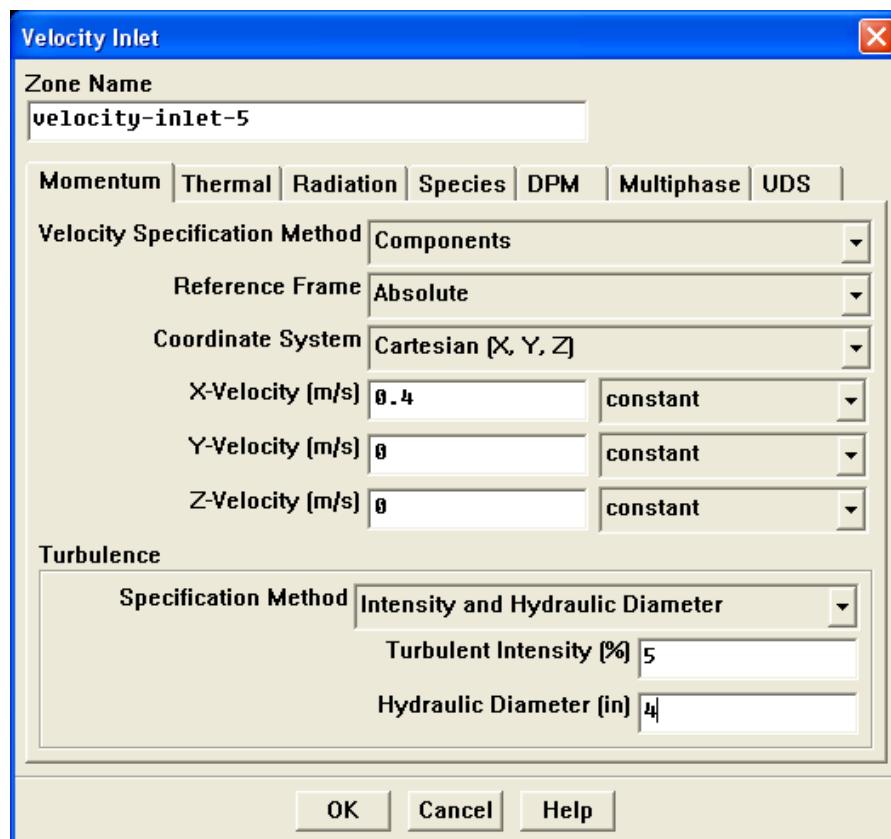
*You have just specified water as the working fluid for this simulation.*

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

**Hint:** If you are unsure of which inlet zone corresponds to the cold inlet, you can probe the grid display with the right mouse button as described in a previous step. Not only will information be displayed in the FLUENT console, but the zone you probed will automatically be selected from the Zone selection list in the Boundary Conditions panel.

- (a) Select velocity-inlet-5 from the Zone selection list.

- (b) Click Set... to open the Velocity Inlet panel.



- i. Select Components from the Velocity Specification Method drop-down list.

*The Velocity Inlet panel will expand.*

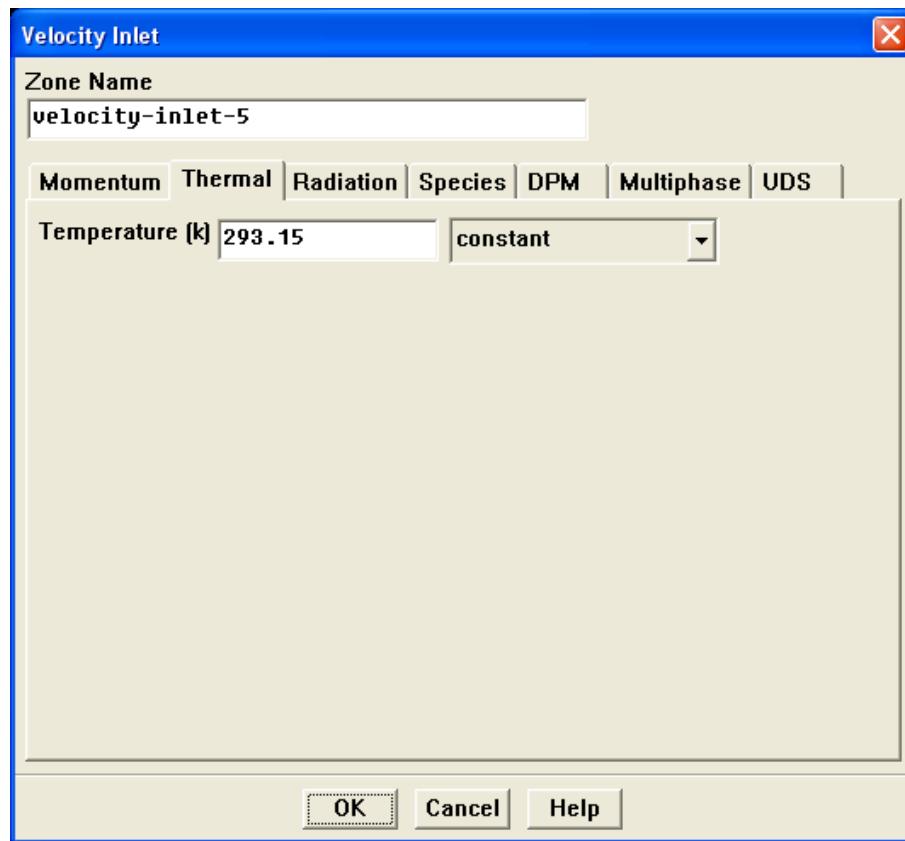
- ii. Enter 0.4 m/s for X-Velocity.
- iii. Retain the default value of 0 m/s for both Y-Velocity and Z-Velocity.
- iv. Select Intensity and Hydraulic Diameter from the Specification Method drop-down list in the Turbulence group box.
- v. Enter 5% for Turbulent Intensity.
- vi. Enter 4 inches for Hydraulic 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.



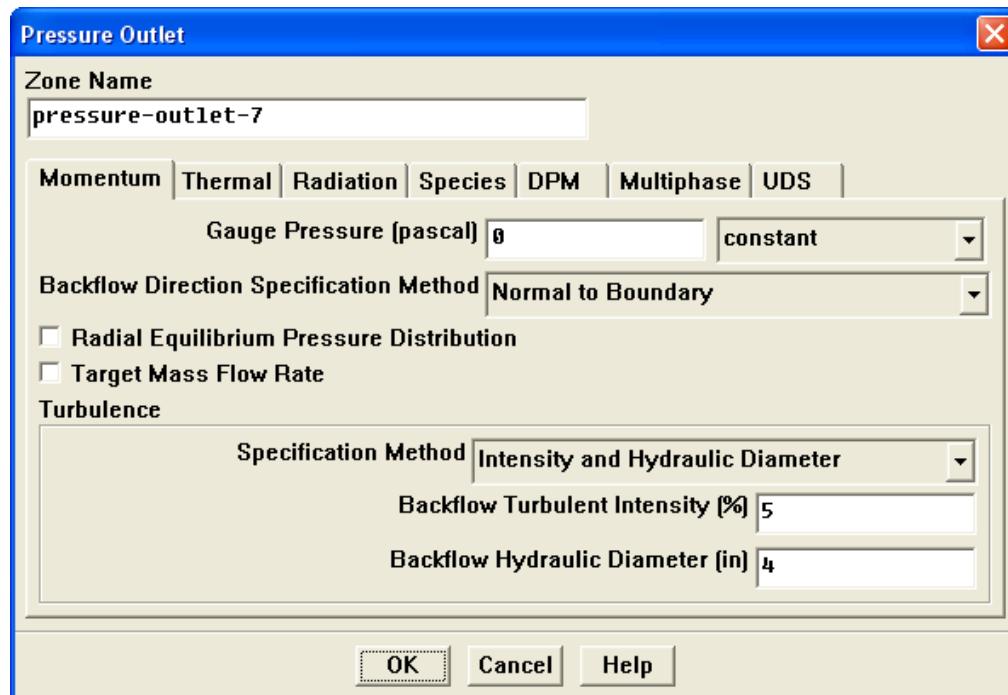
viii. Enter 293.15 K for Temperature.

ix. Click OK to close the Velocity Inlet panel.

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

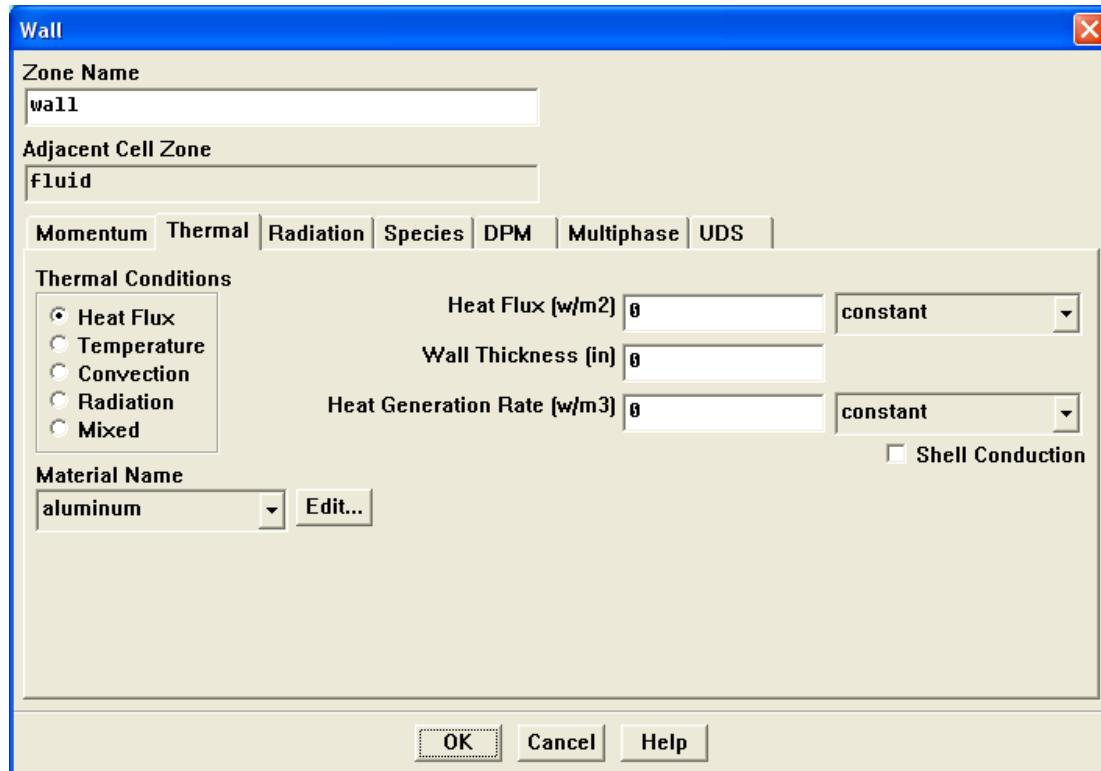
Velocity Specification Method	Components
X-Velocity	0 m/s
Y-Velocity	1.2 m/s
Z-Velocity	0 m/s
Specification Method	Intensity & Hydraulic Diameter
Turbulent Intensity	5%
Hydraulic Diameter	1 inch
Temperature	313.15 K

- Set the boundary conditions at the outlet (pressure-outlet-7), as shown in the following panel.



**Note:** 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.

- For the wall of the pipe (wall), retain the default value of 0 W/m<sup>2</sup> for Heat Flux in the Thermal tab.

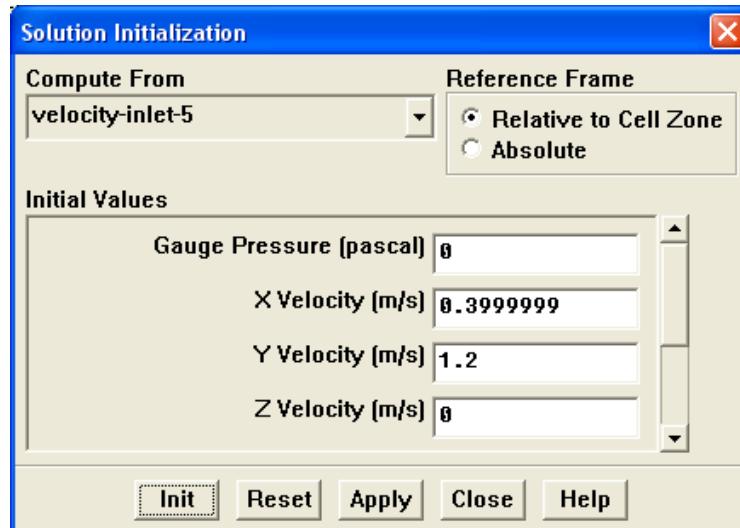


- Close the Boundary Conditions panel.

## Step 5: Solution

1. Initialize the flow field, using the boundary conditions settings at the cold inlet (velocity-inlet-5) as a starting point.

Solve → Initialize → Initialize...



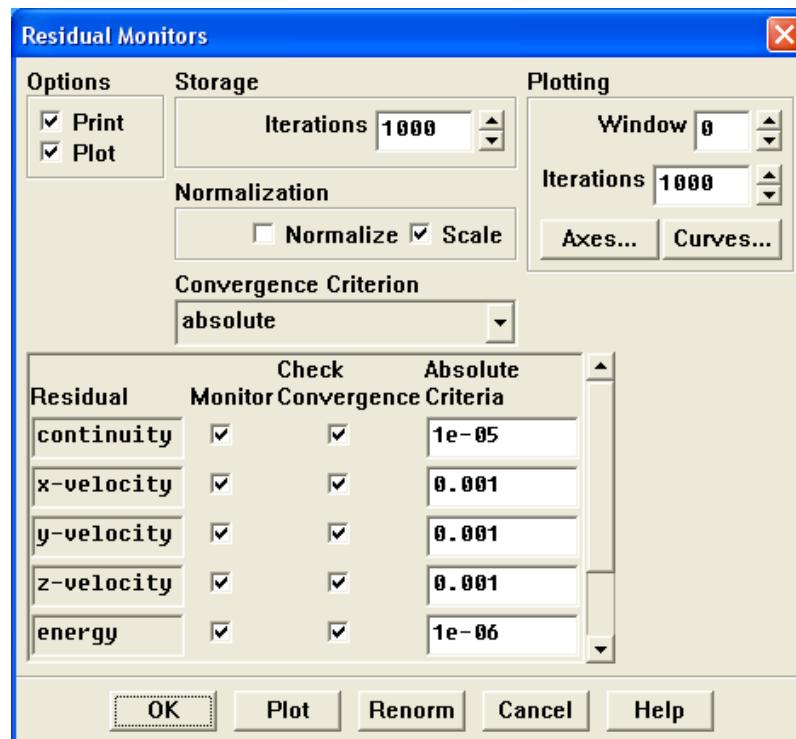
- (a) Select velocity-inlet-5 from the Compute From drop-down list.
- (b) Enter 1.2 m/s for Y Velocity in the Initial Values group box.

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

- (c) Click Init and close the Solution Initialization panel.

2. Enable the plotting of residuals during the calculation.

**Solve** → **Monitors** → **Residual...**

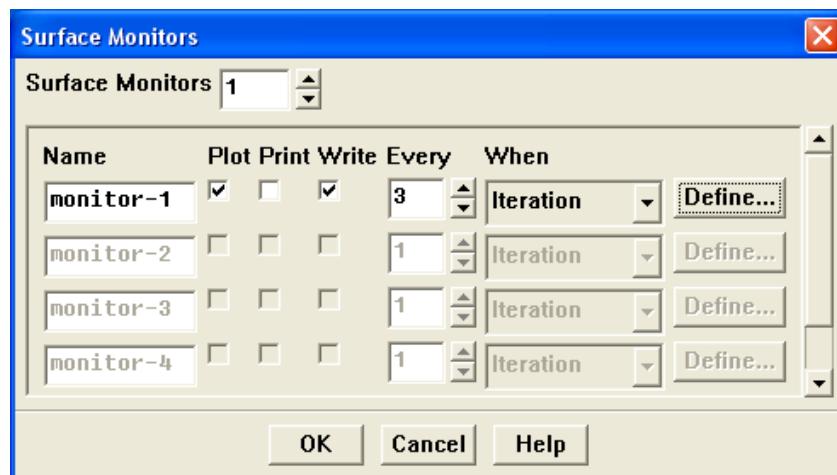


- Enable Plot in the Options group box.
- Enter 1e-05 for the Absolute Criteria of continuity, as shown in the previous panel.
- Click OK to close the Residual Monitors panel.

**Note:** By default, all variables will be monitored and checked by FLUENT as a means to determine the convergence of the solution. Although residuals are useful for checking convergence, a more reliable method is to define a surface monitor. You will do this in the next step.

3. Define a surface monitor at the outlet (pressure-outlet-7).

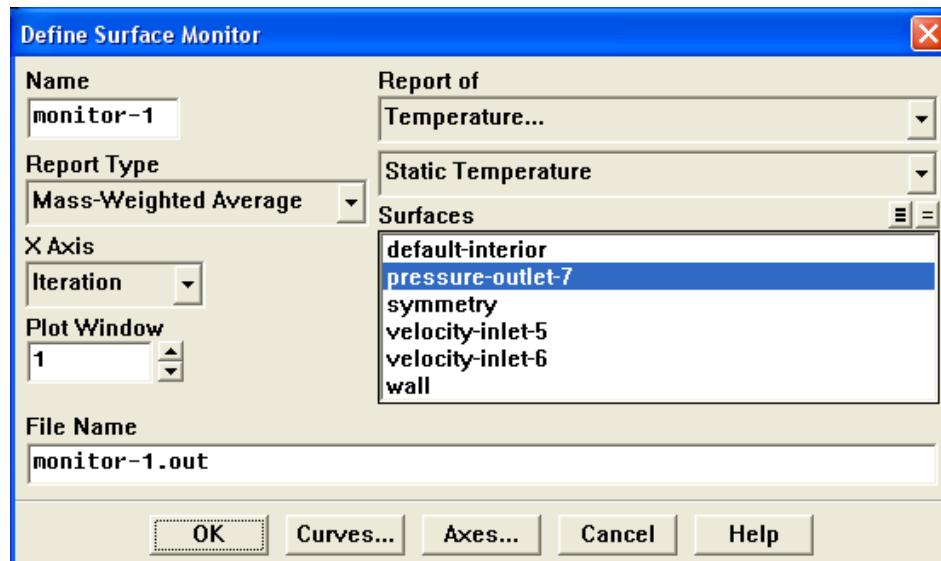
Solve → Monitors → Surface...



- Set Surface Monitors to 1 by clicking once on the up-arrow button.
- Enable the Plot and Write options for monitor-1.
- Set Every to 3 for monitor-1.

*This setting instructs FLUENT to update the plot of the surface monitor and write data to a file after every 3 iterations during the solution.*

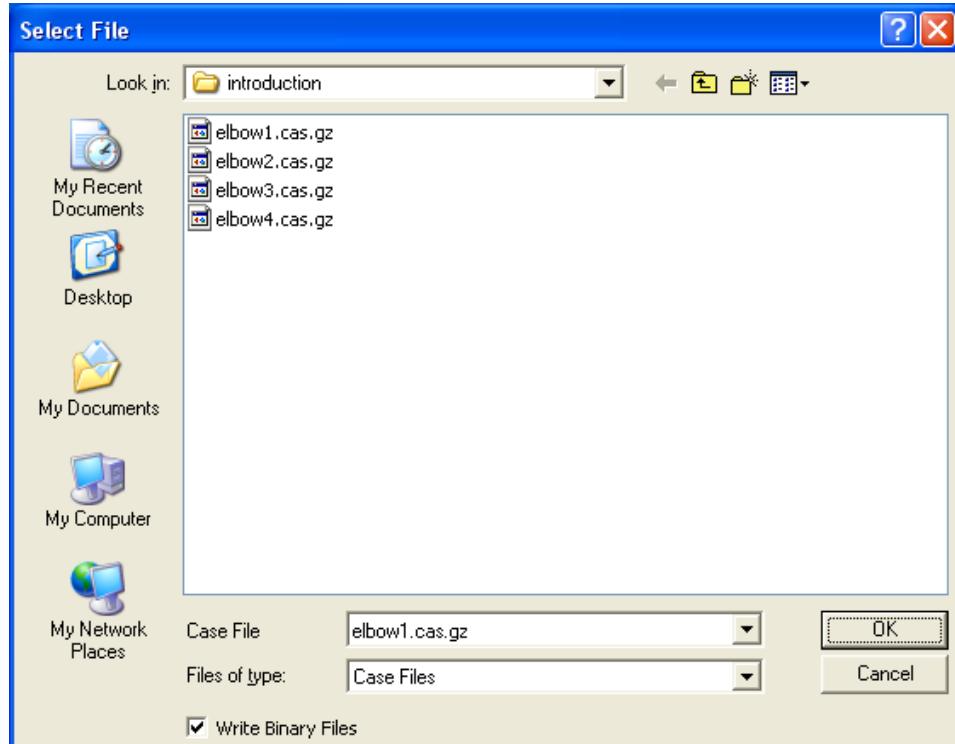
- Click the Define... button to open the Define Surface Monitor panel.



- Select Mass-Weighted Average from the Report Type drop-down list.
- Retain the default entry of monitor-1.out for File Name.

- iii. Select Temperature... and Static Temperature from the Report of drop-down lists.
  - iv. Select pressure-outlet-7 from the Surfaces selection list.
  - v. Click OK to close the Define Surface Monitor panel.
- (e) Click OK to close the Surface Monitors panel.
4. Save the case file (elbow1.cas.gz).

**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 (i.e., the introduction folder). You can indicate a different folder by browsing to it or by creating a new folder.*

- (b) Enter elbow1.cas.gz for Case File.

*Adding the extension .gz to the end of the file name extension instructs FLUENT to save the file in a compressed format. You do not have to include .cas in the extension (e.g., if you enter elbow1.gz, FLUENT will automatically save the file as elbow1.cas.gz). The .gz extension can also be used to save data files in a compressed format.*

- (c) Make sure that the default Write Binary Files option is enabled, so that a binary file will be written.

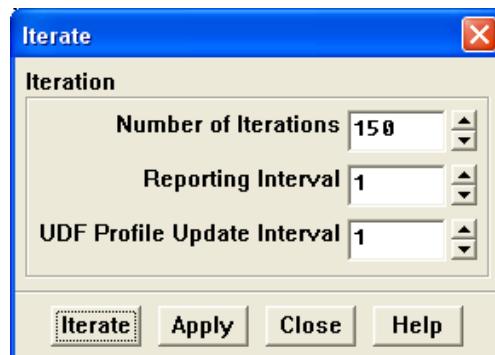
- (d) Click OK to close the Select File dialog box.

**Note:** If you retained the default introduction folder in the Select File dialog box, a Warning dialog box will open to alert you that the file `elbow1.cas.gz` already exists. All of the files you will be instructed to save in this tutorial already exist in the introduction folder, and can be overwritten. Click OK in the Warning dialog box to proceed.



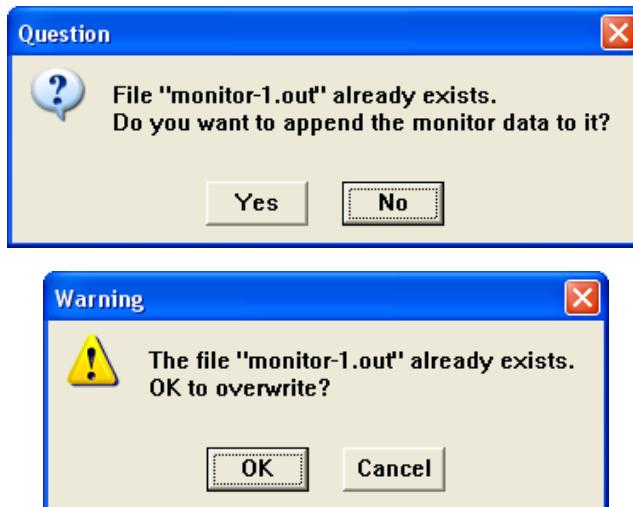
5. Start the calculation by requesting 150 iterations.

**Solve** → Iterate...



- (a) Enter 150 for Number of Iterations.  
(b) Click Iterate.

**Note:** By starting the calculation, you are also starting to save the surface monitor data at the rate specified in the Surface Monitors panel. If a file already exists in your working folder with the name you specified in the Define Surface Monitor panel, then a Question dialog box will open, asking if you would like 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 residuals will be plotted in the graphics window (Figure 1.3). An additional graphics window will open to display the convergence history of the mass-weighted average temperature (Figure 1.4). The solution will reach convergence after approximately 140 iterations.

**Note:** The number of iterations required for convergence varies according to the platform used. Also, since the residual values are different for different computers, the plot that appears on your screen may not be exactly the same as the one shown here.

- (c) Close the Iterate panel when the calculation is complete.

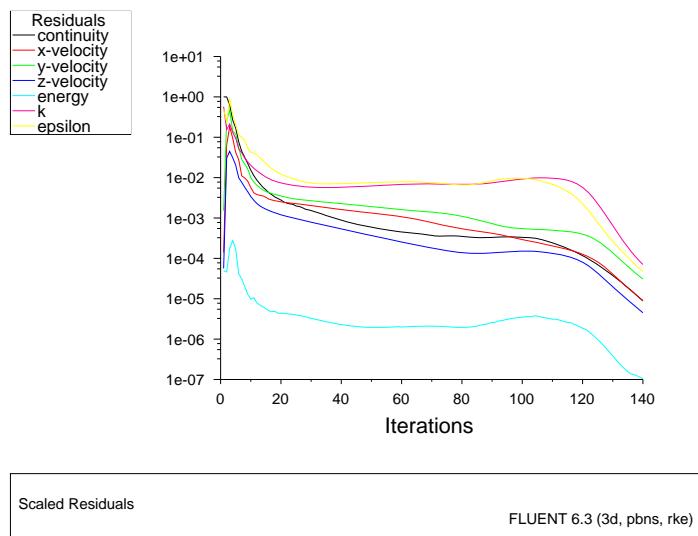


Figure 1.3: Residuals for the First 140 Iterations

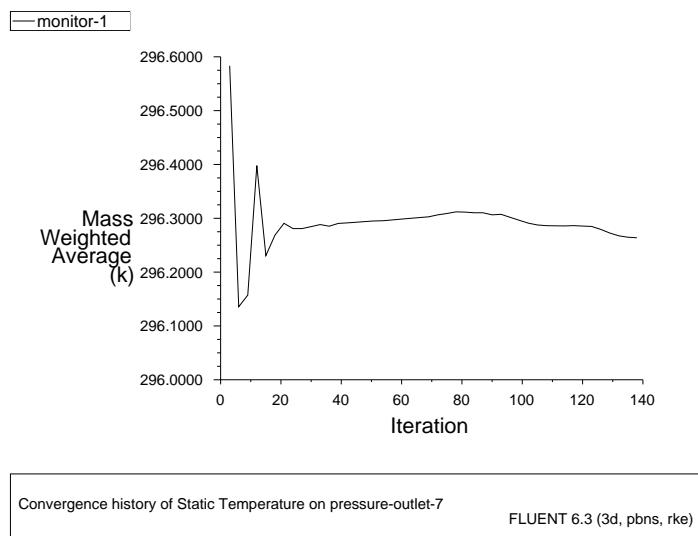


Figure 1.4: Convergence History of the Mass-Weighted Average Temperature

6. Examine the plots for convergence (Figures 1.3 and 1.4).

**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.

When evaluating whether convergence has been reached, there are three indicators:

- 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.

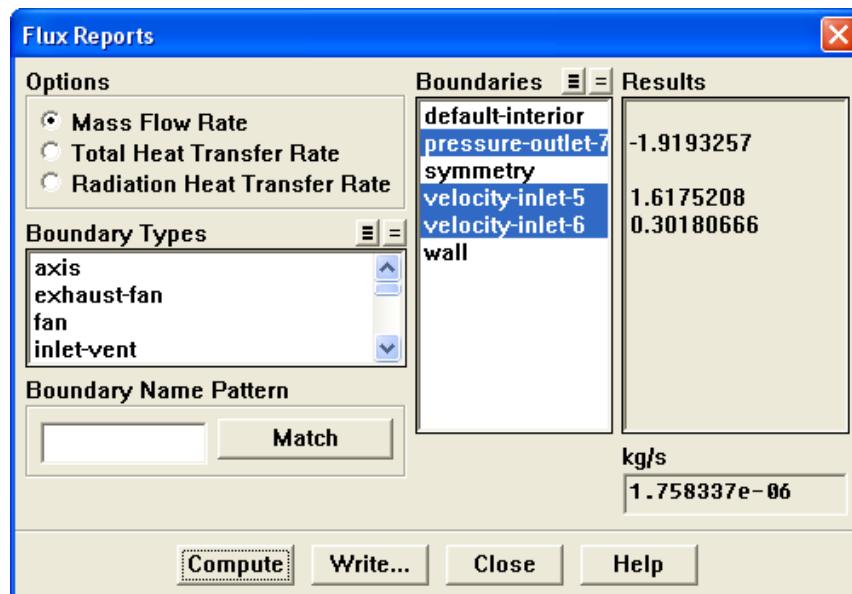
Sometimes the residuals may not fall below the convergence criterion set in the case setup. However, monitoring the representative flow variables through iterations may show that the residuals have stagnated and do not change with further iterations. This could also be considered as convergence.

- 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 panel. 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.

7. Examine the mass flux report for convergence.

**Report** → Fluxes...



- (a) Select pressure-outlet-7, velocity-inlet-5, and velocity-inlet-6 from the Boundaries selection list.
- (b) Click Compute.  
*The sum of the flux for the inlets should be very close to the sum of the flux for the outlets. The difference will be displayed in the lower right field under kg/s, as well as in the console. Note that the imbalance is well below the 0.2% criteria suggested previously.*
- (c) Close the Flux Reports panel.

8. Save the data file (elbow1.dat.gz).

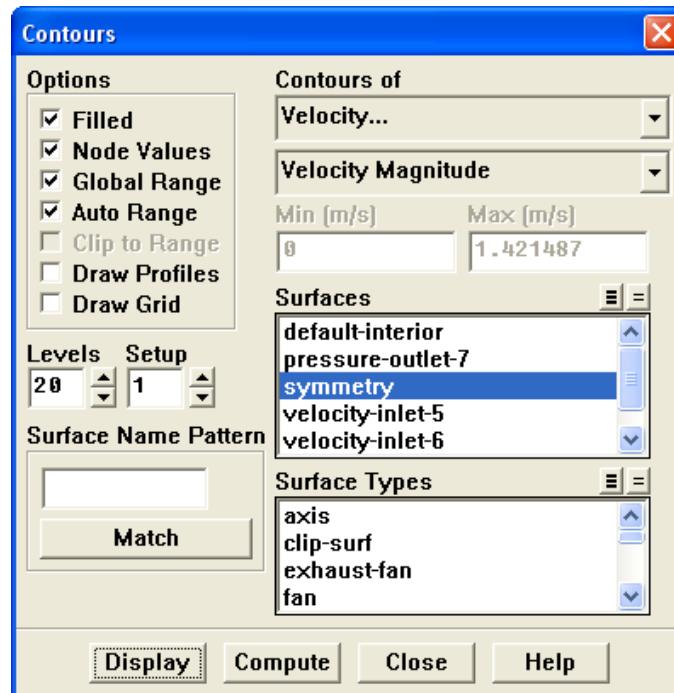
**File** → **Write** → **Data...**

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

## Step 6: Displaying the Preliminary Solution

1. Display filled contours of velocity magnitude on the symmetry plane (Figure 1.5).

**Display** —> Contours...



- Enable Filled in the Options group box.
- Make sure that Node Values is enabled in the Options group box.
- Select Velocity... and Velocity Magnitude from the Contours of drop-down lists.
- Select symmetry from the Surfaces selection list.
- Click Display to display the contours in the graphics window.

**Extra:** Clicking the right mouse button on a point in the displayed domain will cause the value of the corresponding contour to be reported in the console.

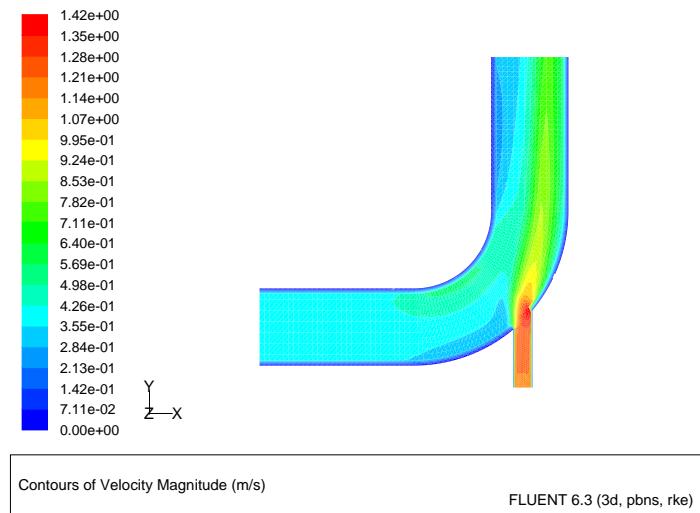
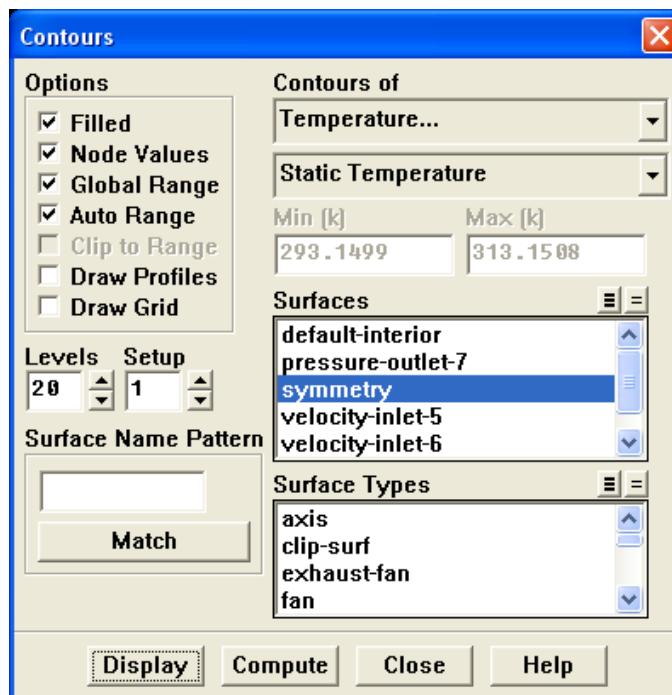


Figure 1.5: Predicted Velocity Distribution after the Initial Calculation

2. Display filled contours of temperature on the symmetry plane (Figure 1.6).

**Display** —> **Contours...**



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

- (b) Click Display and close the Contours panel.

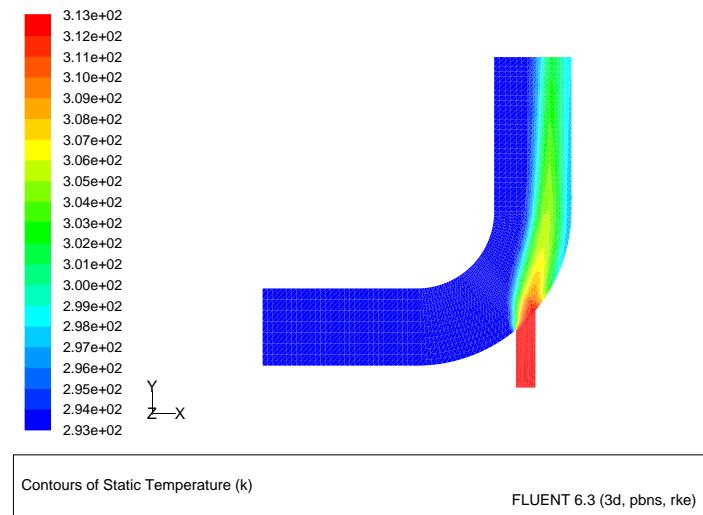
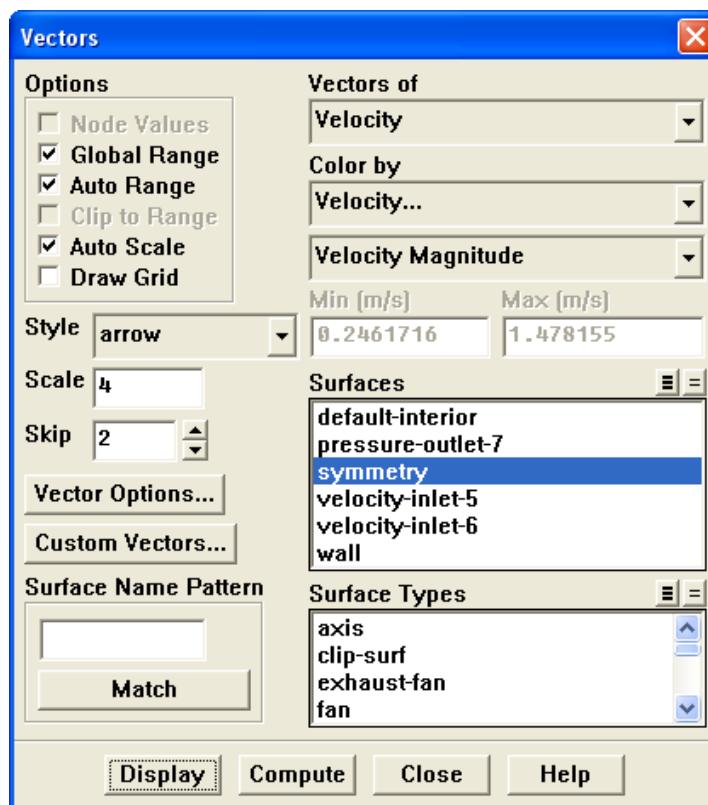


Figure 1.6: Predicted Temperature Distribution after the Initial Calculation

3. Display velocity vectors on the symmetry plane (Figures 1.7 and 1.8).

**Display** —> Vectors...



- Select symmetry from the Surfaces selection list.
- Click Display to plot the velocity vectors.

**Note:** 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.

- Enter 4 for Scale to increase the display size of the vectors.
- Set Skip to 2 to make the individual vectors easier to see.
- Click Display again (Figure 1.7).
- Zoom in on the vectors in the display.

To do this, drag your mouse to the right and either up or down, while pressing the middle mouse button. A rectangle will appear on the screen. Make sure that the rectangle frames the region that you wish to enlarge and let go of the middle mouse button. The image will be redisplayed at a higher magnification (Figure 1.8).

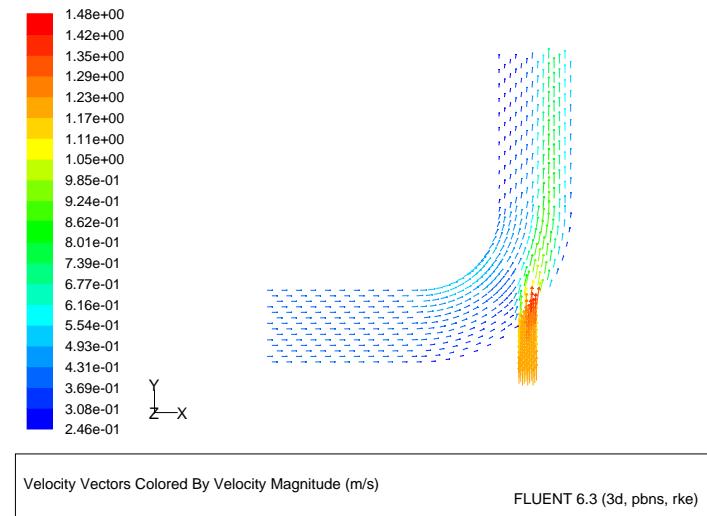


Figure 1.7: Resized Velocity Vectors

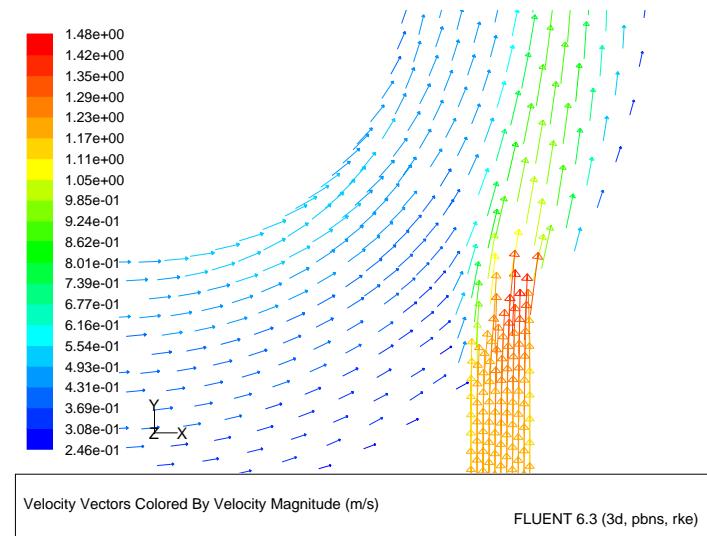


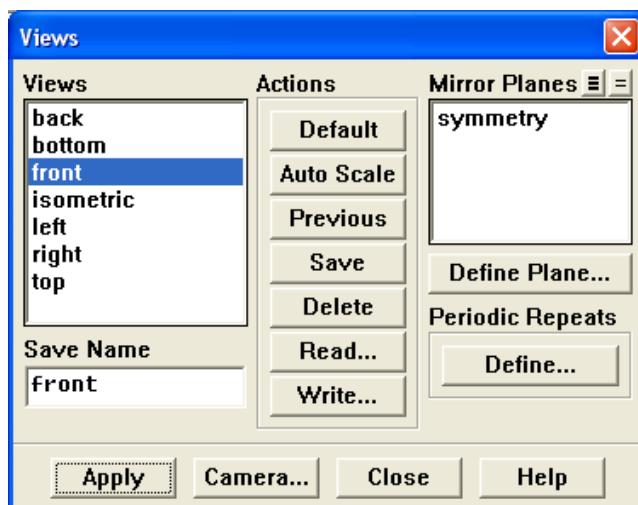
Figure 1.8: Magnified View of Velocity Vectors

- (g) Zoom out to the original view.

*To do this, drag your mouse to the left and either up or down, while pressing the middle mouse button. A rectangle will appear on the screen. Make sure that the rectangle is approximately the same size as the rectangle you made while zooming in, and then let go of the middle mouse button. The image will be redisplayed at a lower magnification (Figure 1.7). If the resulting image is not centered, you can translate the view by clicking once with the middle mouse button near the center of the geometry.*

*Alternatively, you can select the original view in the Views panel. Simply select front from the Views selection list and click Apply, as shown in the following panel.*

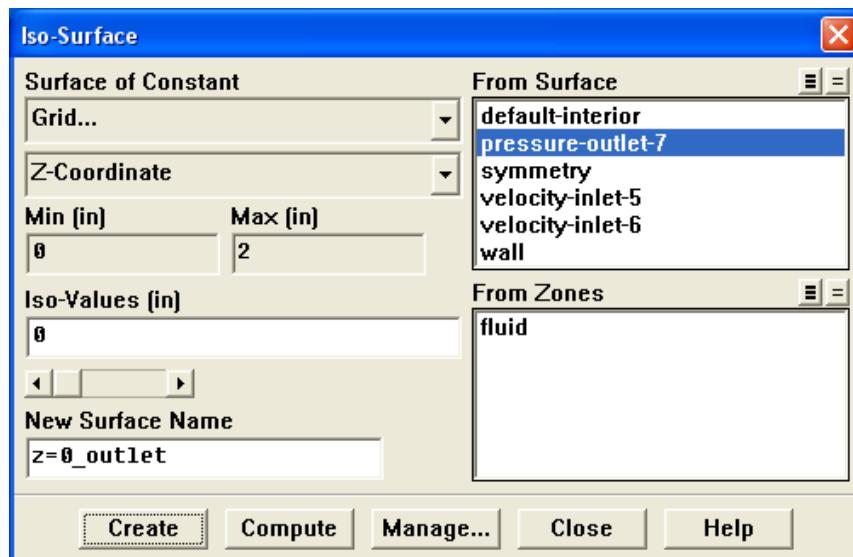
Display → Views...



- (h) Close the Vectors panel.

4. Create a line surface at the centerline of the outlet.

**Surface** → Iso-Surface...



- (a) Select Grid... and Z-Coordinate from the Surface of Constant drop-down lists.
- (b) Click Compute.

*The range of values in the z direction will be displayed in the Min and Max fields.*

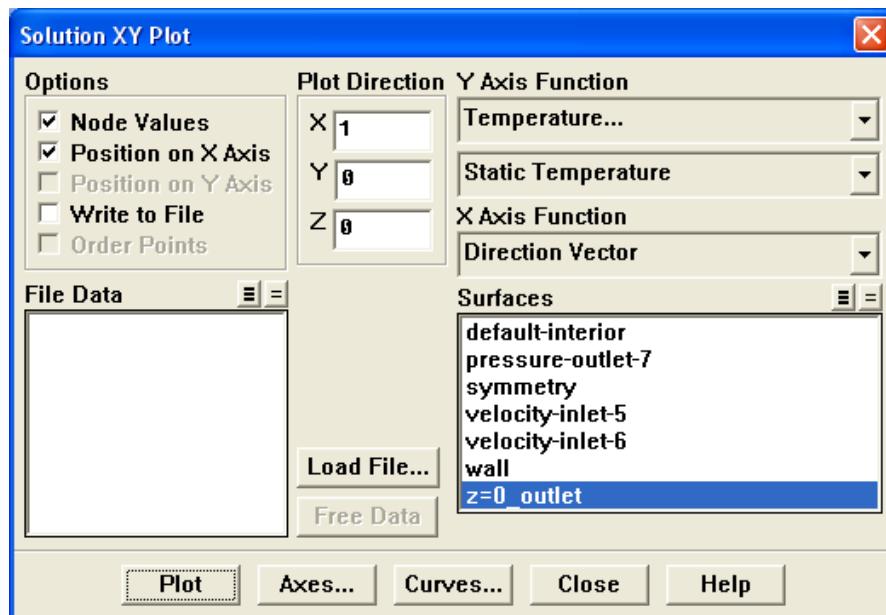
- (c) Retain the default value of 0 inches for Iso-Values.
- (d) Select pressure-outlet-7 from the From Surface selection list.
- (e) Enter z=0\_outlet for New Surface Name.
- (f) Click Create.

*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.*

- (g) Close the Iso-Surface panel.

5. Display and save an XY plot of the temperature profile across the centerline of the outlet for the initial solution (Figure 1.9).

**Plot** → XY Plot...



- Select Temperature... and Static Temperature from the Y Axis Function drop-down lists.
- Select z=0\_outlet from the Surfaces selection list.
- Click Plot.
- Enable Write to File in the Options group box.  
*The button that was originally labeled Plot will change to Write....*
- Click Write... to open 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 panel.

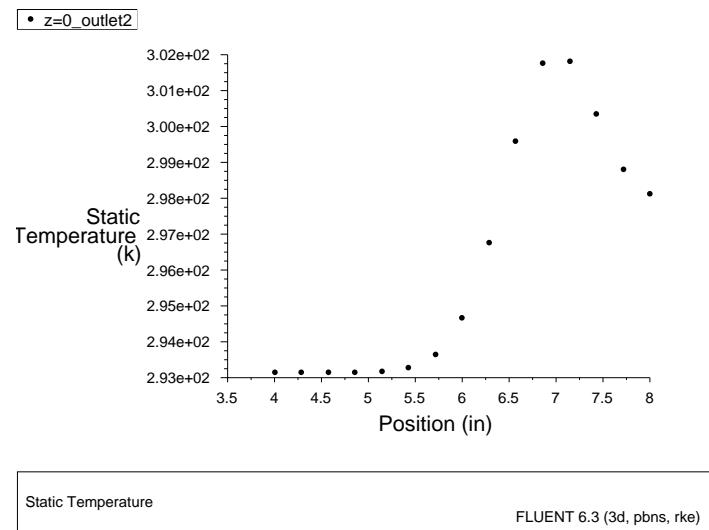
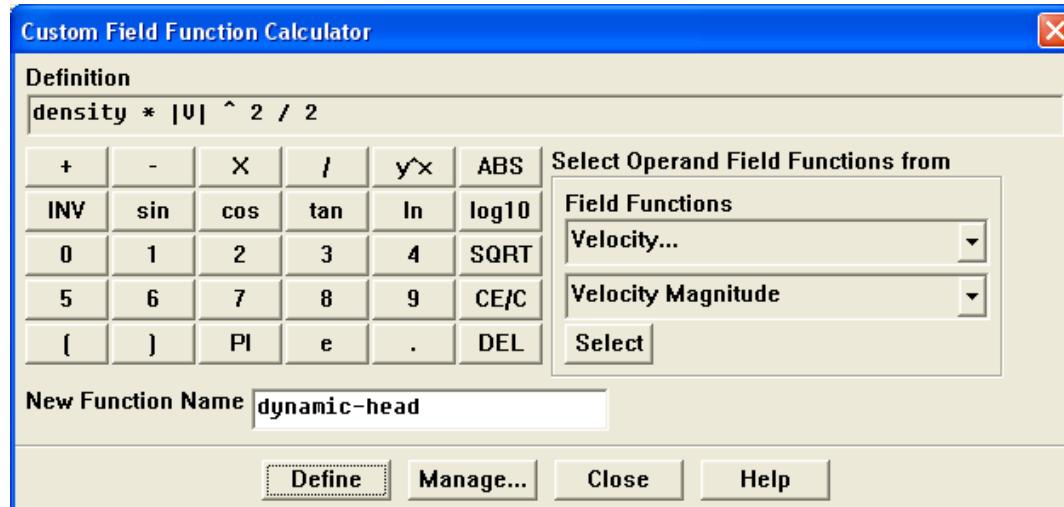


Figure 1.9: Outlet Temperature Profile for the Initial Solution

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

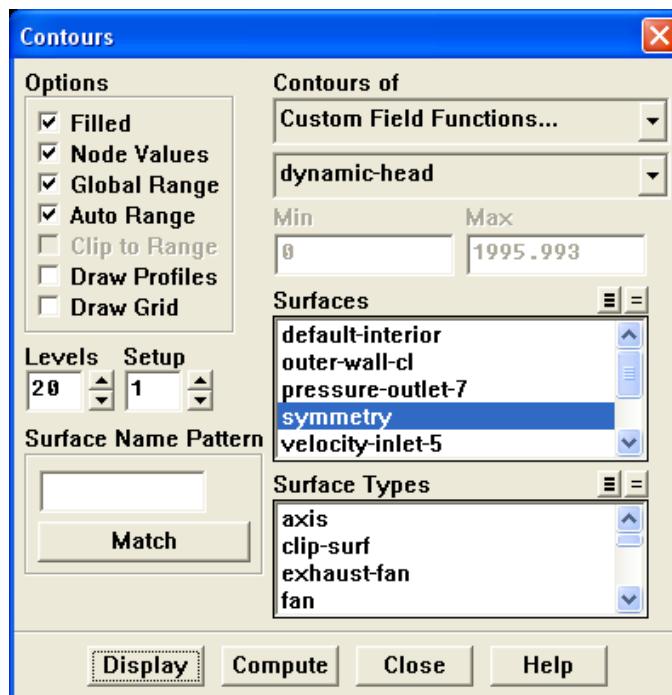
**Define** → Custom Field Functions...



- 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.

- (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 panel.
7. Display filled contours of the custom field function (Figure 1.10).

**Display** —> Contours...



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

**Hint:** Custom Field Functions... is at the top of the upper Contours of drop-down list. After you have opened the drop-down list, scroll up by clicking the up-arrow button on the scroll bar on the right.

- (b) Make sure that symmetry is selected from the Surfaces selection list.
- (c) Click Display and close the Contours panel.

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

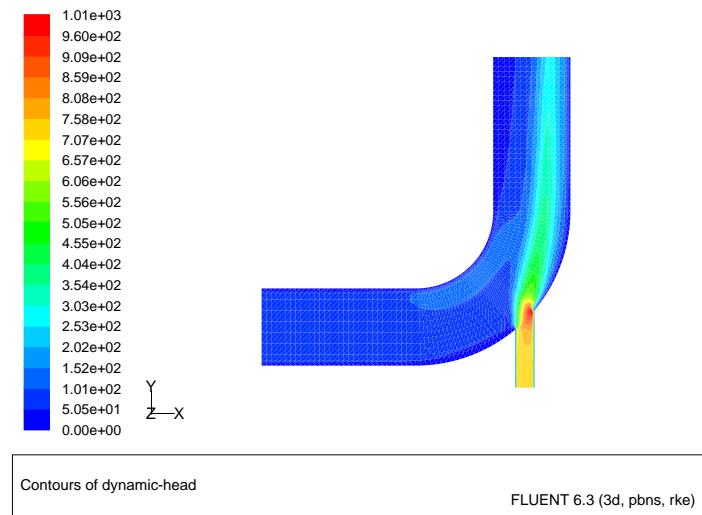


Figure 1.10: Contours of the Dynamic Head Custom Field Function

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

**[File] → [Write] → Case & Data...**

- (a) Make sure that `elbow1.cas.gz` 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` or `.dat` extension, as both will be saved.

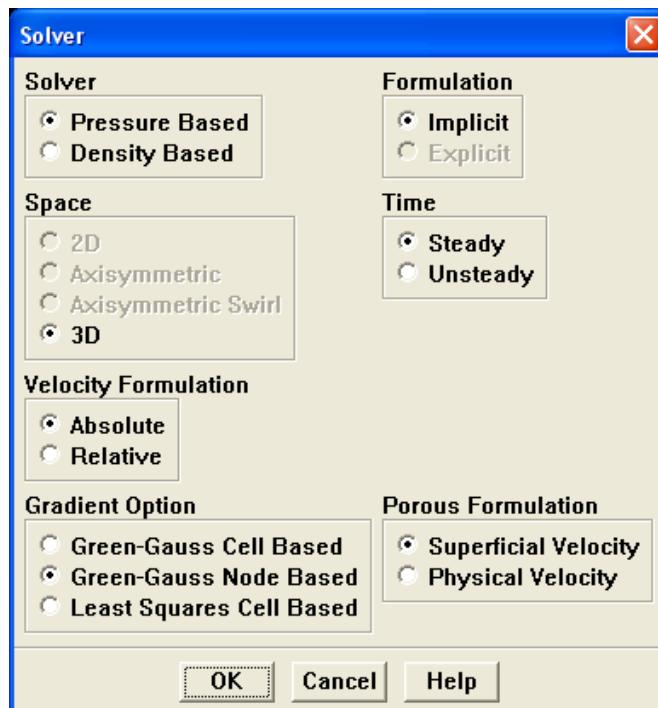
- (b) Click OK to close the Select File dialog box.

## Step 7: Enabling Second-Order Discretization

The elbow solution computed in the first part of this tutorial uses first-order discretization. The resulting solution is very diffusive; mixing is overpredicted, as can be seen in the contour plots of temperature and velocity distribution. You will now change to second-order discretization for all listed equations, in order to improve the accuracy of the solution. With the second-order discretization, you will change the gradient option in the solver from cell-based to node-based in order to optimize energy conservation.

1. Change the solver settings.

**Define** → **Models** → **Solver...**



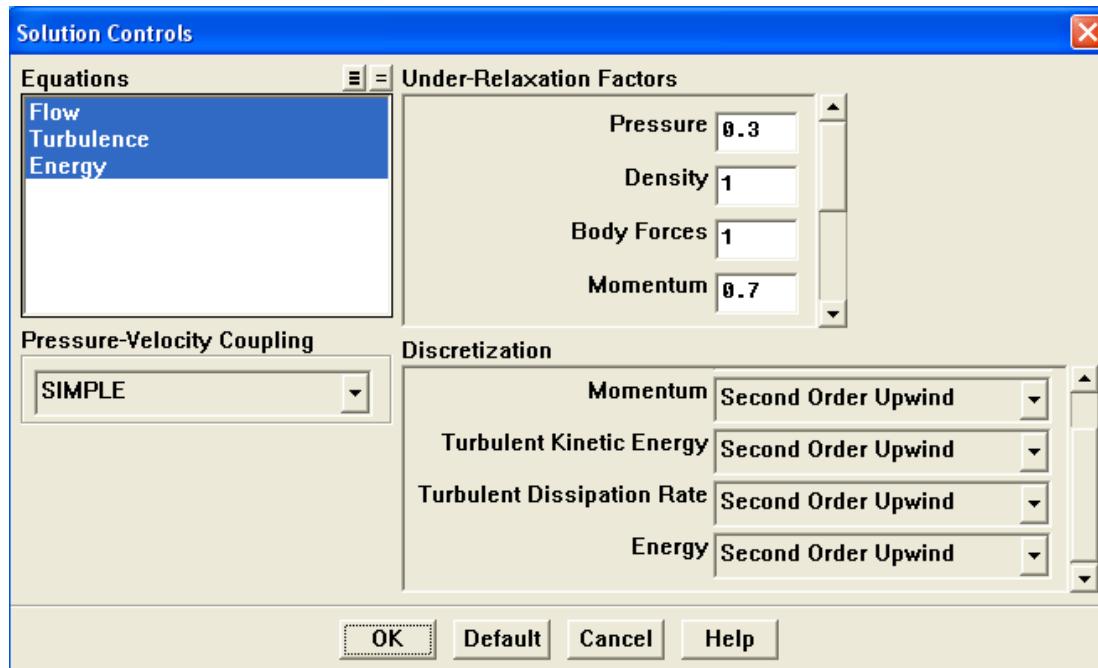
- Select Green-Gauss Node Based from the Gradient Option list.

**Note:** This option is more suitable than the cell-based gradient option for unstructured meshes, as it will ensure better energy conservation.

- Click OK to close the Solver panel.

2. Enable the second-order scheme for the calculation of all the listed equations.

**Solve** → **Controls** → **Solution...**



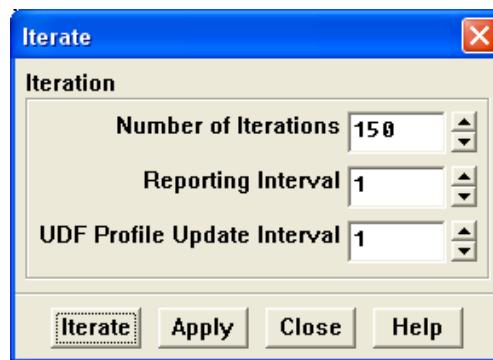
- (a) Retain the default values in the Under-Relaxation Factors group box.
- (b) Select Second Order from the Pressure drop-down list in the Discretization group box.
- (c) Select Second Order Upwind from the Momentum, Turbulent Kinetic Energy, Turbulent Dissipation Rate, and Energy drop-down lists.

**Note:** Scroll down the Discretization group box to find Energy.

- (d) Click OK to close the Solution Controls panel.

3. Continue the calculation by requesting 150 more iterations.

**Solve** → **Iterate...**



**Extra:** To save the convergence history of the surface monitor for this set of iterations as a separate output file, you would need to change the File Name in the Define Surface Monitor to monitor-2.out prior to running the calculation.

- Make sure that 150 is entered for Number of Iterations.
- Click Iterate and close the Iterate panel when the calculation is complete.

The solution will converge in approximately 57 additional iterations (Figure 1.11). The convergence history is shown in Figure 1.12.

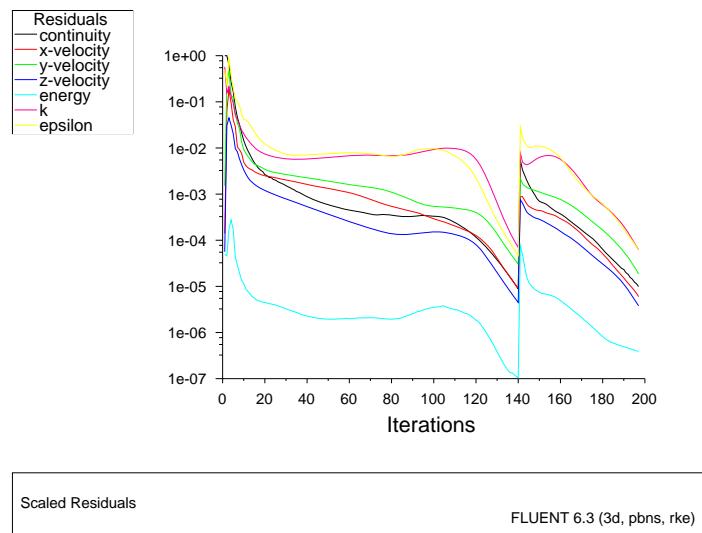


Figure 1.11: Residuals for the Second-Order Energy Calculation

**Note:** You should expect to see the residuals jump whenever you change the solution control parameters.

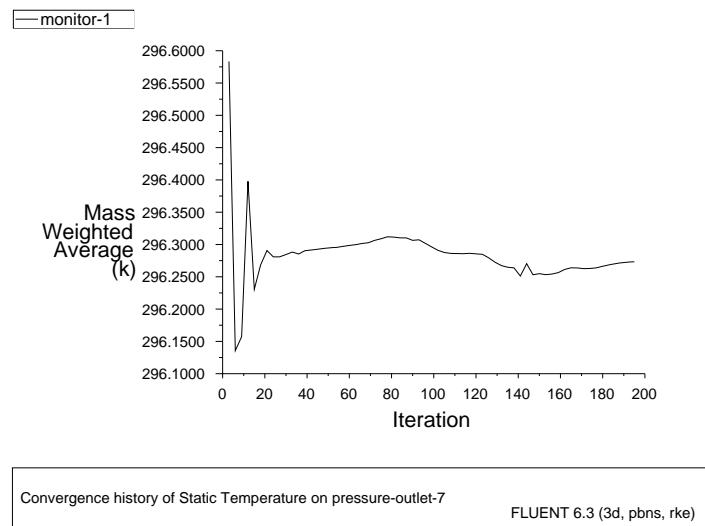


Figure 1.12: Convergence History of Mass-Weighted Average Temperature

4. Save the case and data files for the second-order solution (`elbow2.cas.gz` and `elbow2.dat.gz`).

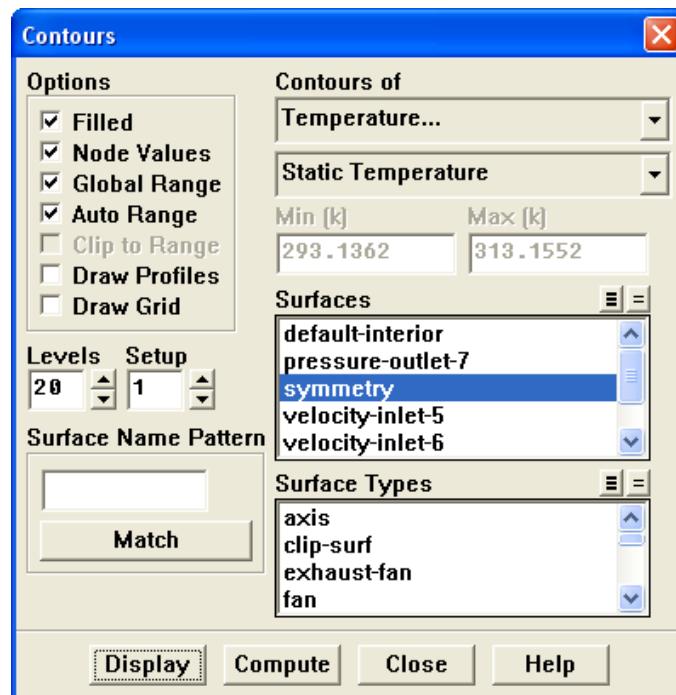
**[File] → [Write] → Case & Data...**

- (a) Enter `elbow2.gz` for Case/Data File.
- (b) Click OK to close the Select File dialog box.

*The files `elbow2.cas.gz` and `elbow2.dat.gz` will be saved in your folder.*

5. Examine the revised temperature distribution (Figure 1.13).

**Display** —> **Contours...**



- Make sure that Filled is enabled in the Options group box.
- Select Temperature... and Static Temperature from the Contours of drop-down lists.
- Make sure that symmetry is selected from the Surfaces selection list.
- Click Display and close the Contours panel.

Figure 1.13 shows the thermal spreading of the warm fluid layer near the outer wall of the bend. Compare Figure 1.13 with Figure 1.6 to see the effects of second-order discretization.

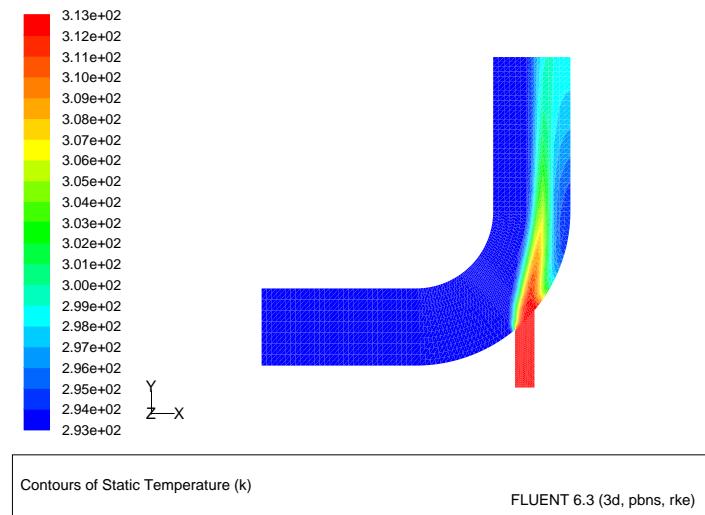
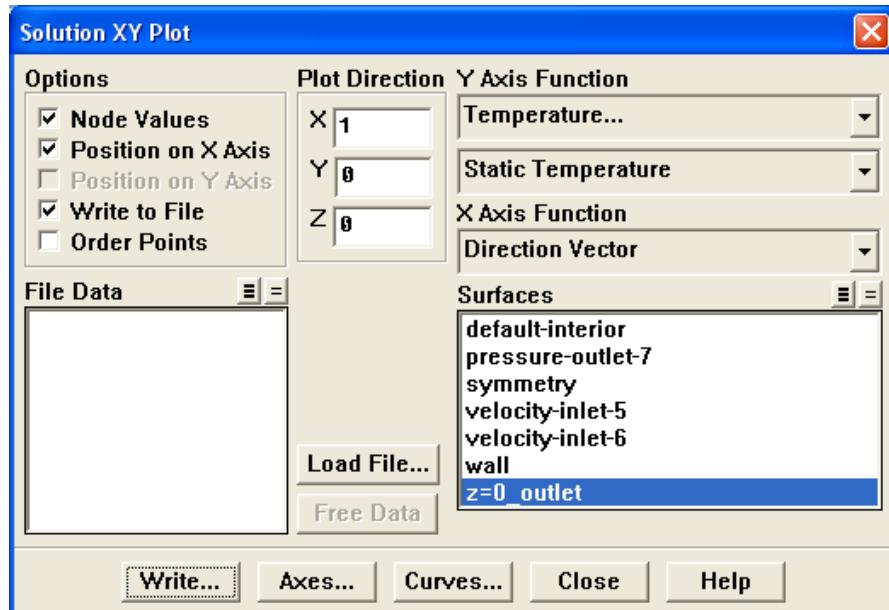


Figure 1.13: Temperature Contours for the Second-Order Solution

6. Display and save an XY plot of the temperature profile across the centerline of the outlet for the second-order solution (Figure 1.14).

**Plot** → XY Plot...



- Disable Write to File in the Options group box by clicking the check box or the text.

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

- (b) Make sure that Temperature... and Static Temperature are selected from the Y Axis Function drop-down lists.
- (c) Make sure that z=0\_outlet is selected from the Surfaces selection list.
- (d) Click Plot.

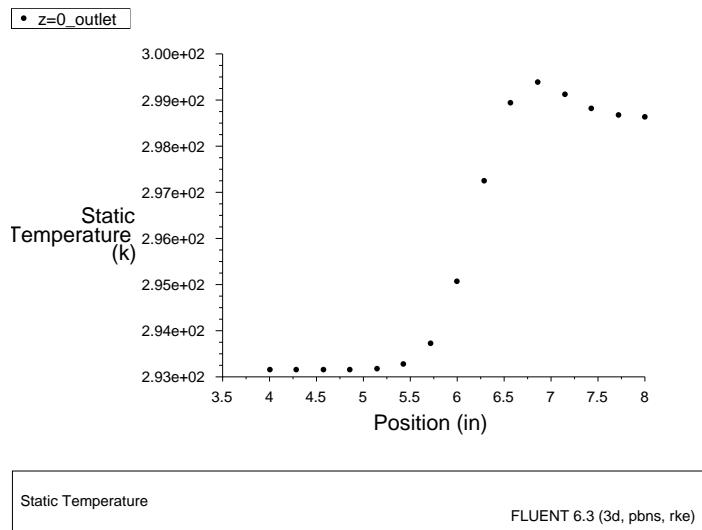


Figure 1.14: Outlet Temperature Profile for the Second-Order Solution

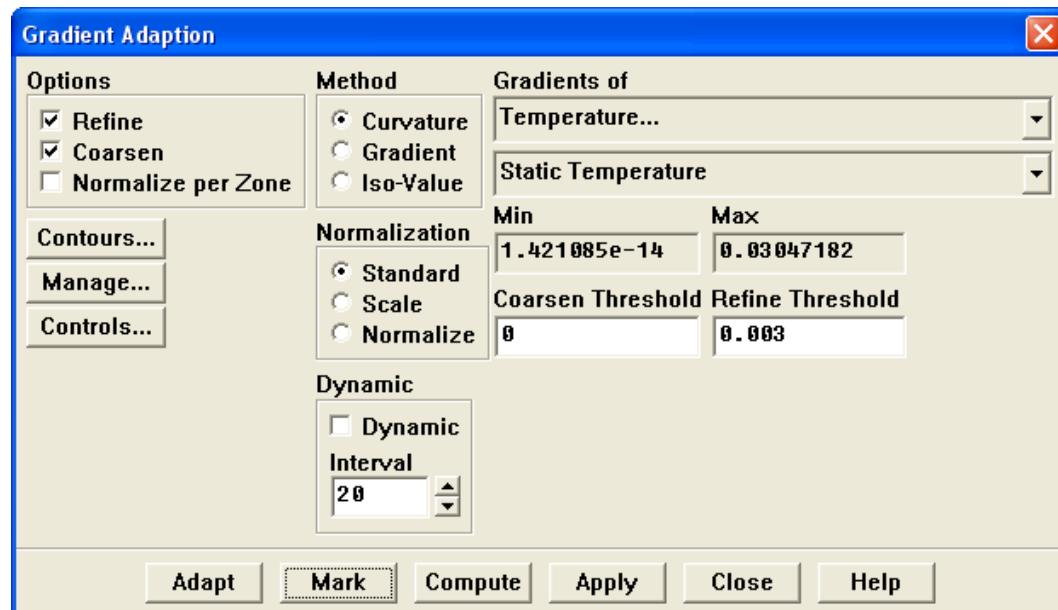
- (e) Enable Write to File in the Options group box.  
*The button that was labeled Plot will change to Write....*
- (f) Click Write... to open the Select File dialog box.
  - i. Enter `outlet_temp2.xy` for XY File.
  - ii. Click OK to save the temperature data.
- (g) Close the Solution XY Plot panel.

## Step 8: Adapting the Grid

The elbow solution can be improved further by refining the grid to better resolve the flow details. In the following steps, you will adapt the grid based on the temperature gradients in the current solution. Once the grid has been refined, you will continue the calculation.

1. Adapt the grid in the regions of high temperature gradient.

**Adapt** → Gradient...



- (a) Make sure that Refine is enabled in the Options group box.

*It is not necessary to deselect Coarsen in this instance, since FLUENT will not coarsen beyond the original mesh for a 3D grid.*

- (b) Select Temperature... and Static Temperature from the Gradients of drop-down lists.

- (c) Click Compute.

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

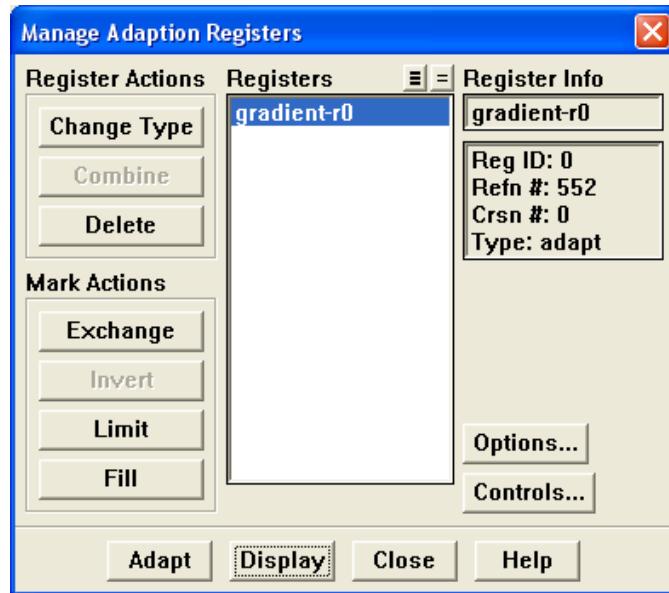
- (d) Enter 0.003 for Refine Threshold.

*It is a good rule of thumb to use 10% of the maximum gradient when setting the value for Refine Threshold.*

- (e) Click Mark.

*FLUENT will report in the console that approximately 1258 cells were marked for adaption.*

- (f) Click the Manage... button to open the Manage Adaption Registers panel.



- i. Click Display.

FLUENT will display the cells marked for adaption in the graphics window (Figure 1.15).

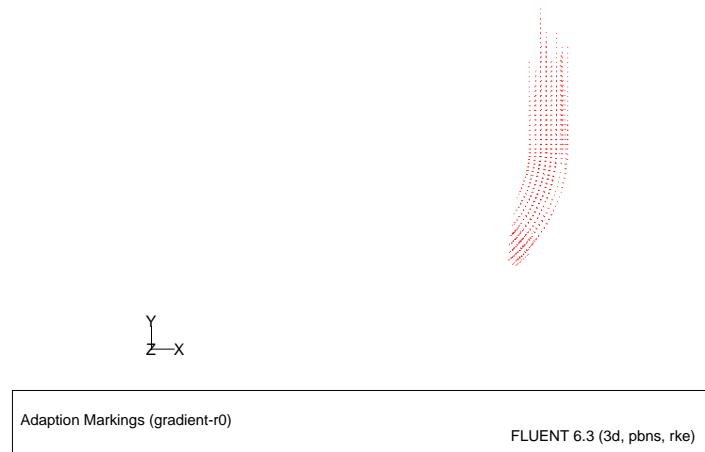
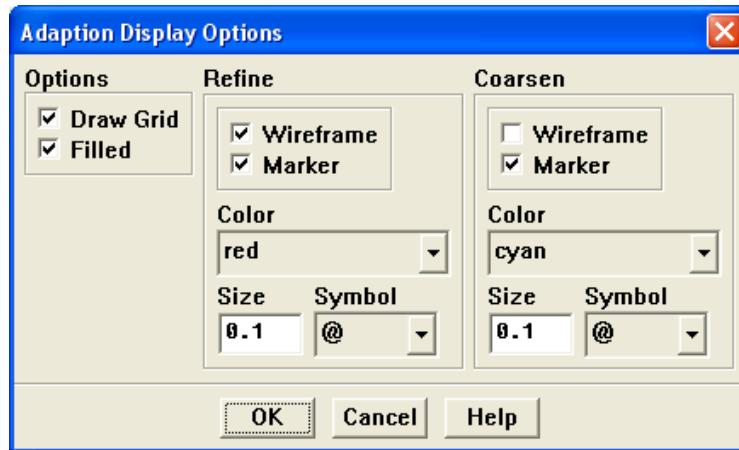


Figure 1.15: Cells Marked for Adaption

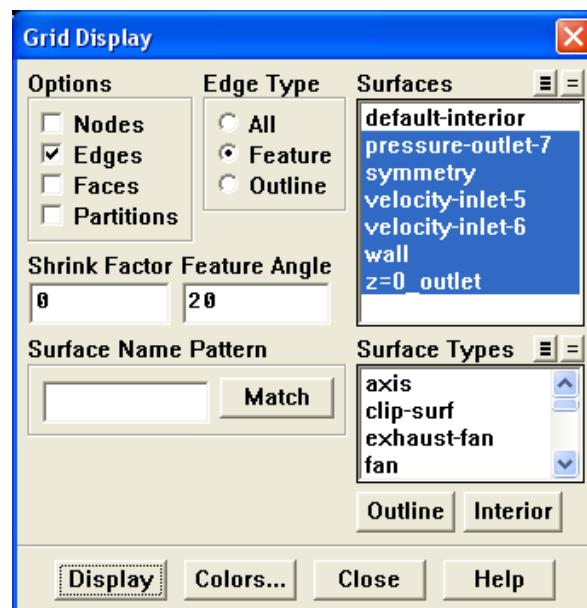
**Extra:** You can change the way FLUENT displays cells marked for adaptation (Figure 1.16) by performing the following steps:

- Click the Options... button in the Manage Adaption Registers panel to open the Adaption Display Options panel.



- Enable Draw Grid in the Options group box.

The Grid Display panel will open.



- Make sure that Edges is the only option enabled in the Options group box.
- Select Feature from the Edge Type list.
- Select all of the items except default-interior from the Surfaces selection list.
- Click Display and close the Grid Display panel.

- G. Enable Filled in the Options group box in the Adaption Display Options panel.
- H. Enable Wireframe in the Refine group box.
- I. Click OK to close the Adaption Display Options panel.
- J. Click Display in the Manage Adaption Registers panel.
- K. Rotate the view and zoom in to get the display shown in Figure 1.16.

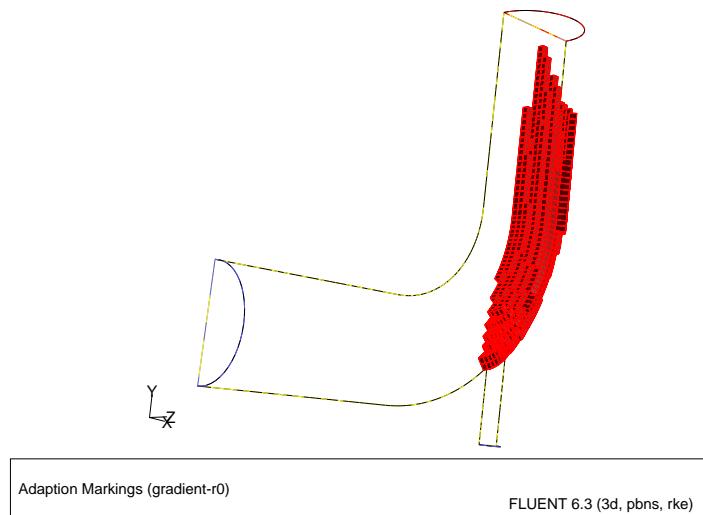
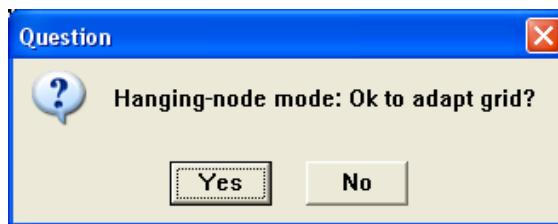


Figure 1.16: Alternate Display of Cells Marked for Adaption

L. After you are finished viewing the marked cells, rotate the view back and zoom out again to return to the angle and magnification shown in Figure 1.13.

- ii. Click Adapt in the Manage Adaption Registers panel.

A Question dialog box will open, asking whether it is acceptable to adapt the grid by creating hanging nodes. Click Yes to proceed.



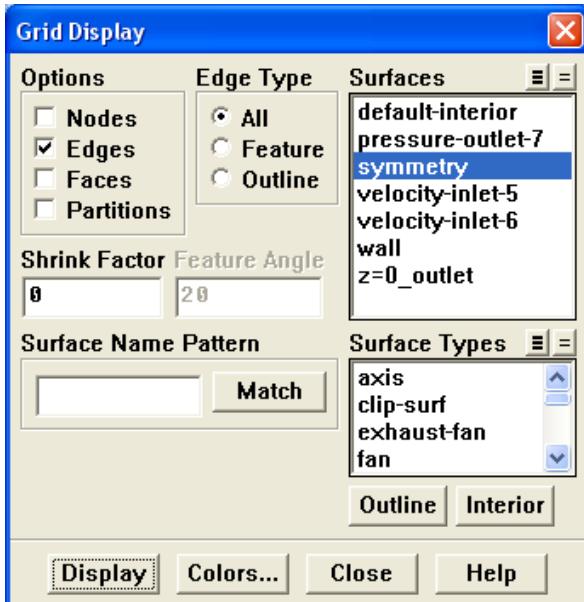
**Note:** There are two different ways to adapt. You can click Adapt in the Manage Adaption Registers panel as was just done, or close this panel and perform the adaption using the Gradient Adaption panel. If

you use the Adapt button in the Gradient Adaption panel, FLUENT will recreate an adaption register. Therefore, once you have the Manage Adaption Registers panel open, it saves time to use the Adapt button there.

- iii. Close the Manage Adaption Registers panel.
- (g) Close the Gradient Adaption panel.

2. Display the adapted grid (Figure 1.17).

**Display** → Grid...



- (a) Make sure that All is selected from the Edge Type list.
- (b) Deselect all of the highlighted items from the Surfaces selection list except for symmetry.
- (c) Click Display and close the Grid Display panel.

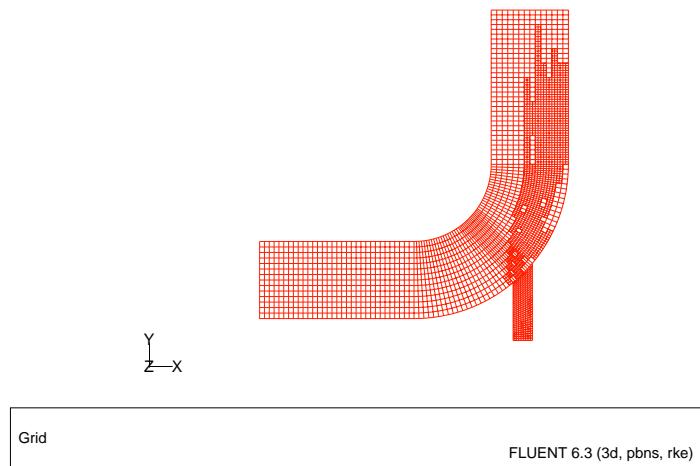
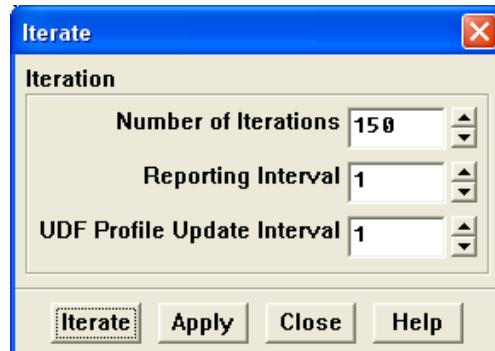


Figure 1.17: The Adapted Grid

3. Request an additional 150 iterations.

**Solve** → Iterate...



*The solution will converge after approximately 100 additional iterations (Figures 1.18 and 1.19).*

4. Save the case and data files for the second-order solution with an adapted grid (`elbow3.cas.gz` and `elbow3.dat.gz`).

**File** → **Write** → Case & Data...

- (a) Enter `elbow3.gz` for Case/Data File.
- (b) Click OK to close the Select File dialog box.

*The files `elbow3.cas.gz` and `elbow3.dat.gz` will be saved in your folder.*

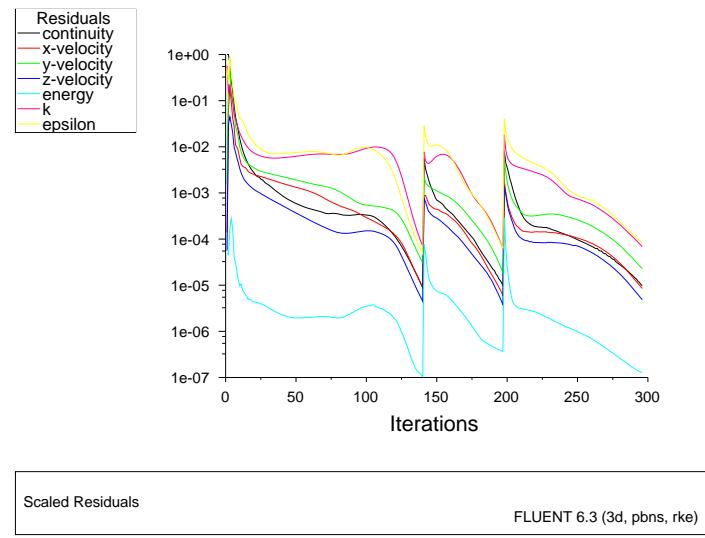


Figure 1.18: The Complete Residual History

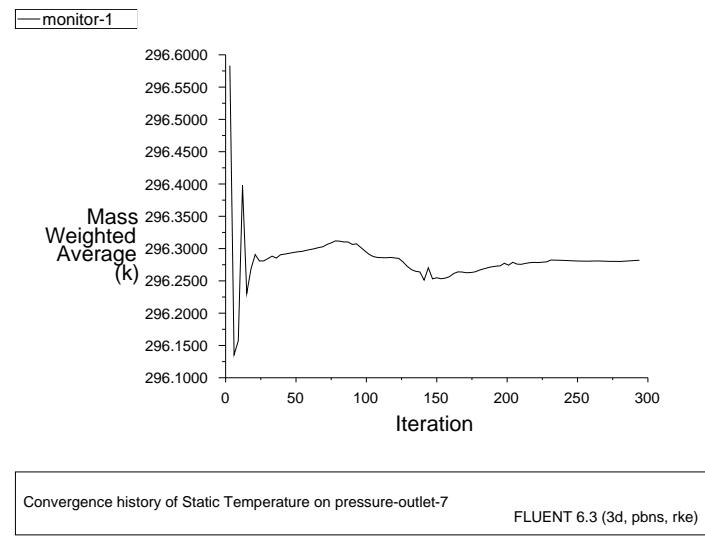


Figure 1.19: Convergence History of Mass-Weighted Average Temperature

5. Examine the filled temperature distribution (using node values) on the revised grid (Figure 1.20).

**Display** —> Contours...

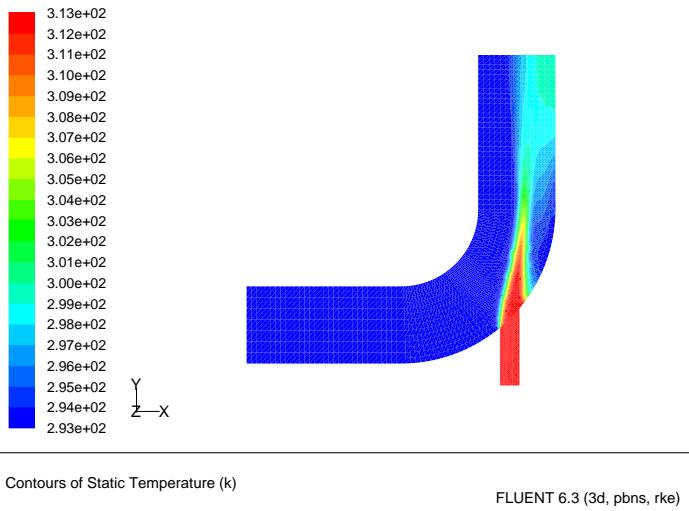
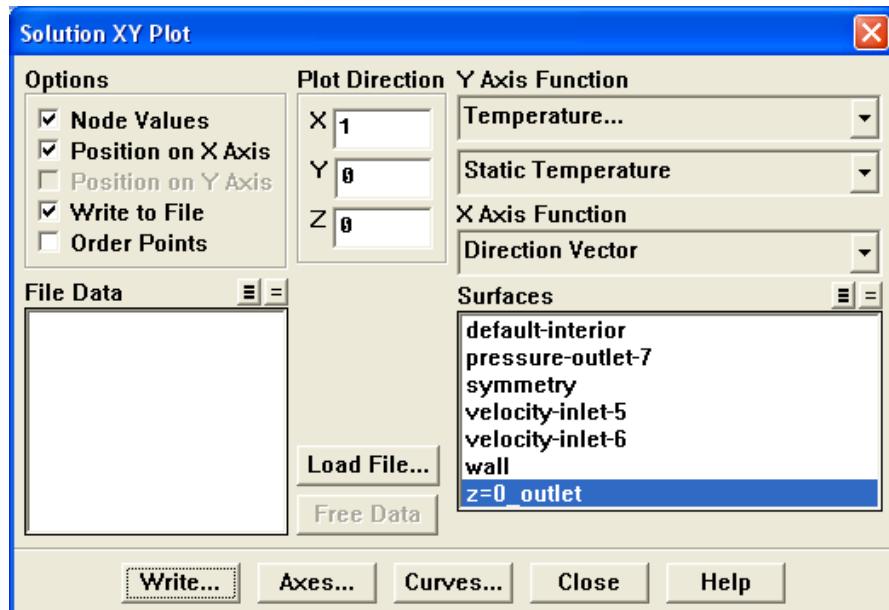


Figure 1.20: Filled Contours of Temperature Using the Adapted Grid

6. Display and save an XY plot of the temperature profile across the centerline of the outlet for the adapted second-order solution (Figure 1.21).

**Plot** —> XY Plot...



- (a) Disable Write to File in the Options group box.

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

- (b) Make sure that Temperature... and Static Temperature are selected from the Y Axis Function drop-down lists.
- (c) Make sure that z=0\_outlet is selected from the Surfaces selection list.
- (d) Click Plot.

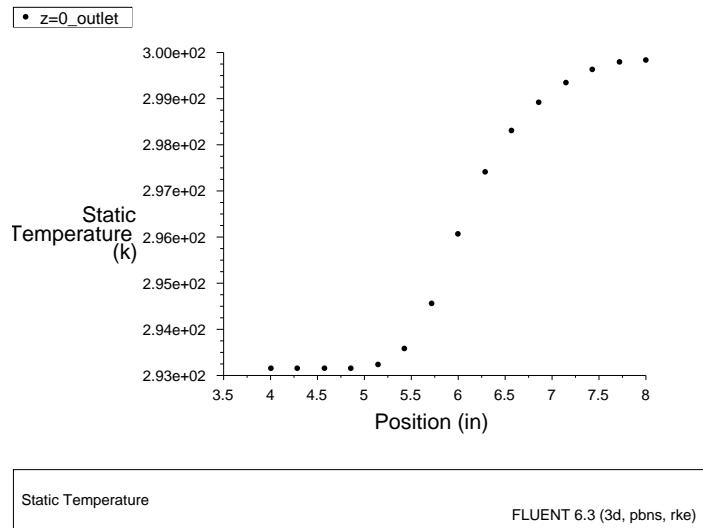


Figure 1.21: Outlet Temperature Profile for the Adapted Second-Order Solution

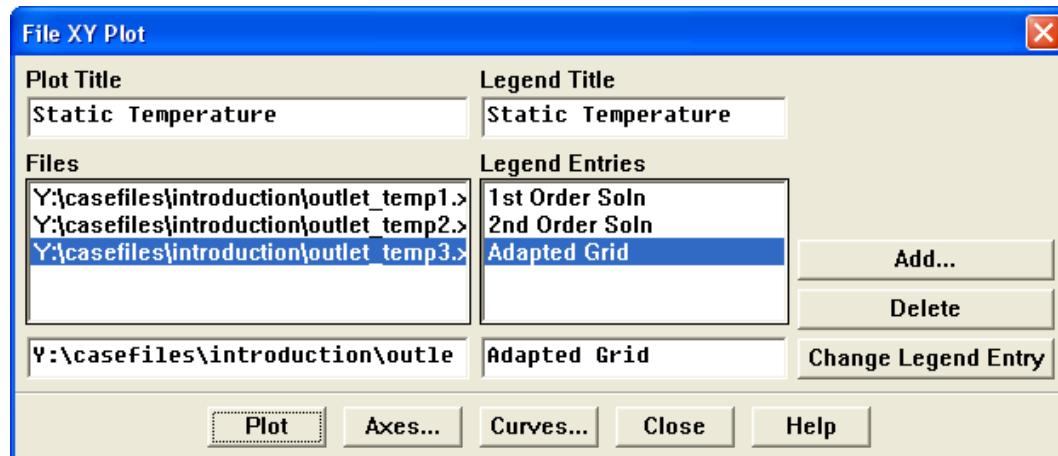
- (e) Enable Write to File in the Options group box.

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

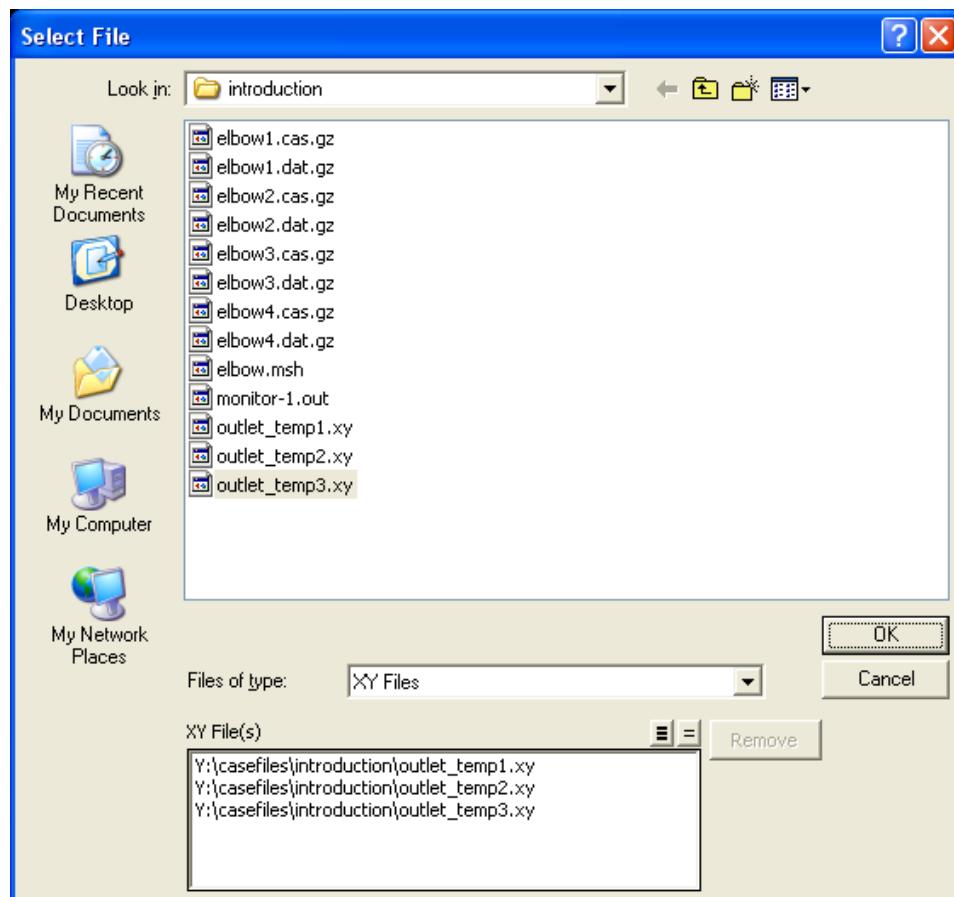
- (f) Click Write... to open the Select File dialog box.
- Enter `outlet_temp3.xy` for XY File.
  - Click OK to save the temperature data.
- (g) Close the Solution XY Plot panel.

7. Display the outlet temperature profiles for each of the three solutions on a single plot (Figure 1.22).

**Plot** —> File...



- (a) Click the Add... button to open the Select File dialog box.



- i. Click once on `outlet_temp1.xy`, `outlet_temp2.xy`, and `outlet_temp3.xy`.

*Each of these files will be listed with their folder in the XY File(s) list to indicate that they have been selected.*

**Hint:** *If you select a file by mistake, simply click the file in the XY File(s) list and then click Remove.*

- ii. Click OK to close the Select File dialog box.
- (b) Select the folder path ending in `outlet_temp1.xy` from the Files selection list.
- (c) Enter **1st Order Soln** in the lowest field on the right (next to the **Change Legend Entry** button).
- (d) Click the **Change Legend Entry** button.  
*The item in the Legend Entries list for `outlet_temp1.xy` will be changed to **1st Order Soln**. This legend entry will be displayed in the upper-left corner of the XY plot generated in a later step.*
- (e) In a similar manner, change the legend entry for the folder path ending in `outlet_temp2.xy` to be **2nd Order Soln**.
- (f) In a similar manner, change the legend entry for the folder path ending in `outlet_temp3.xy` to be **Adapted Grid**.
- (g) Click Plot and close the File XY Plot panel.

*Figure 1.22 shows the three 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.*

**Extra:** You can perform additional grid adaptions based on temperature gradient and run the calculation to see how the temperature profile changes at the outlet. A case and data file (`elbow4.cas.gz` and `elbow4.dat.gz`) has been provided in which the grid has undergone three more levels of adaption, and the resulting temperature profiles have been plotted with `outlet_temp2.xy` and `outlet_temp3.xy` in Figure 1.23.

*It is evident from Figure 1.23 that as the grid is adapted further, the profiles converge on a grid-independent profile. The resulting wall temperature at the outlet is predicted to be around 300.25 K once grid independence is achieved. If the adaption steps had not been performed, the wall temperature would have incorrectly been estimated at around 298.5 K.*

*If computational resources allow, it is always recommended to perform successive adaptions until the solution is independent of the grid (within an acceptable tolerance). Typically, profiles of important variables are examined (in this case, temperature) and compared to determine grid independence.*

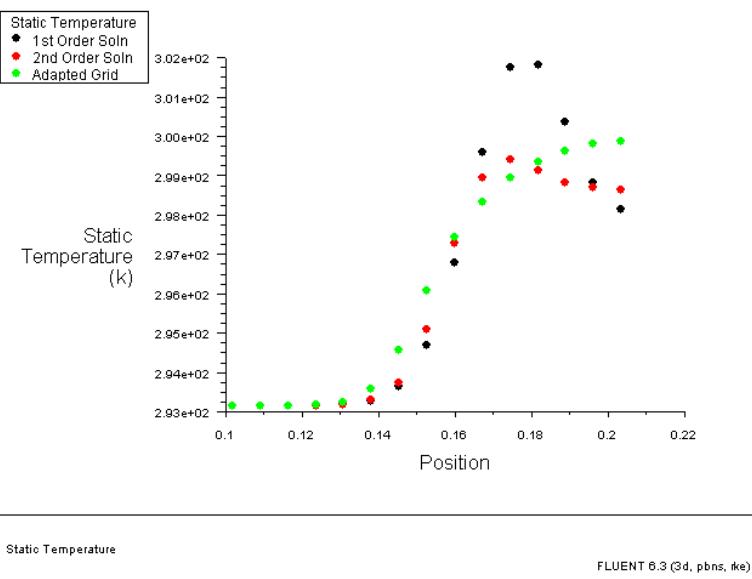


Figure 1.22: Outlet Temperature Profiles for the Three Solutions

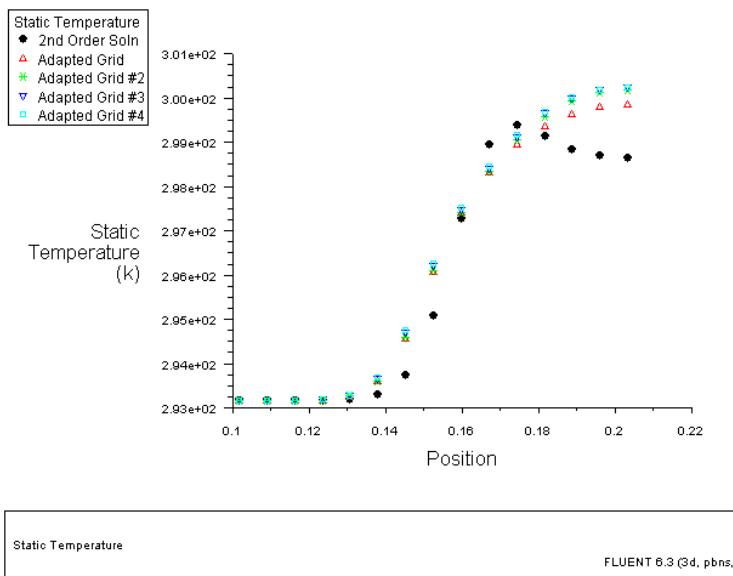


Figure 1.23: Outlet Temperature Profiles for Subsequent Grid Adaption Steps

## **Summary**

Comparison of the filled temperature contours for the first solution (using the original grid and first-order discretization) and the last solution (using an adapted grid and second-order discretization) clearly indicate that the latter is much less diffusive. While first-order discretization is the default scheme in FLUENT, it is good practice to use your first-order solution as a starting guess for a calculation that uses a higher-order discretization scheme and, optionally, an adapted grid.

Note that in this problem, the flow field is decoupled from temperature, since all properties are constant. For such cases, it is more efficient to compute the flow-field solution first (i.e., without solving the energy equation) and then solve for energy (i.e., without solving the flow equations). You will use the **Solution Controls** panel to turn the solution of the equations on and off during this procedure.