

---

# Tutorial 1. Introduction to Using FLUENT

---

**Introduction:** This tutorial illustrates the setup and solution of the two-dimensional turbulent fluid flow and heat transfer in a mixing junction. 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 neighborhood of the mixing region in order to properly design the location of inlet pipes.

In this tutorial you will learn how to:

- 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 segregated solver
- Examine the flow and temperature fields using graphics
- Enable the second-order discretization scheme for improved prediction of temperature
- Adapt the grid based on the temperature gradient to further improve the prediction of temperature

**Prerequisites:** This tutorial assumes that you have little experience with FLUENT, but that you are generally familiar with the interface. If you are not, please review the sample session in Chapter 1 of the User's Guide.

**Problem Description:** The problem to be considered is shown schematically in Figure 1.1. A cold fluid at 26°C enters through the large pipe and mixes with a warmer fluid at 40°C in the elbow. The pipe dimensions are in inches, and the fluid properties and boundary conditions are given in SI units. The Reynolds number at the main inlet is  $2.03 \times 10^5$ , so that a turbulent model will be necessary.

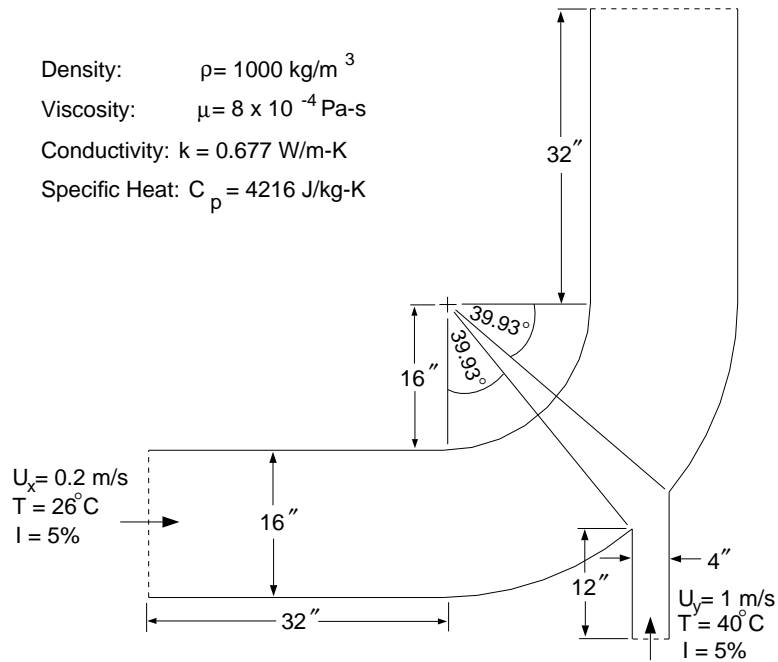


Figure 1.1: Problem Specification

## Preparation

1. Copy the file `elbow/elbow.msh` from the FLUENT documentation CD to your working directory.

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

`/cdrom/fluent6.1/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 directory:*

`cdrom:\fluent6.1\help\tutfiles\`

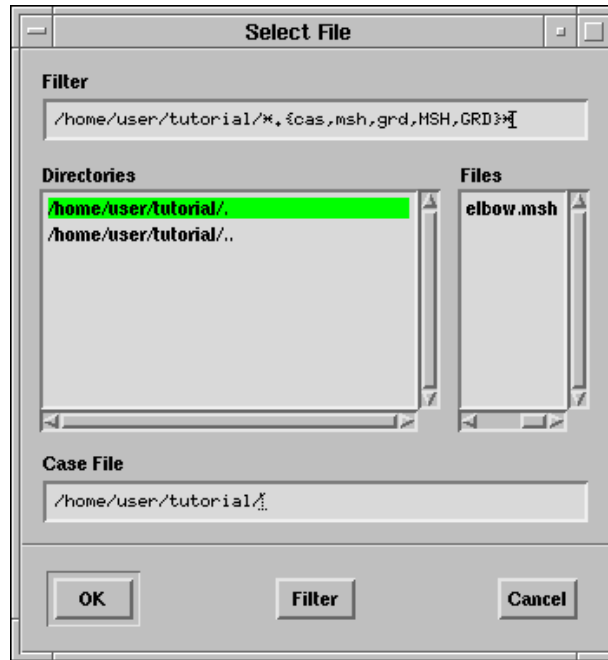
*where cdrom must be replaced by the name of your CD-ROM drive (e.g., E).*

2. Start the 2D version of FLUENT.

## Step 1: Grid

1. Read the grid file elbow.msh.

File → Read → Case...



- (a) Select the file elbow.msh by clicking on it under Files and then clicking on OK.

**Note:** As this grid is read by FLUENT, messages will appear in the console window reporting the progress of the conversion. After reading the grid file, FLUENT will report that 918 triangular fluid cells have been read, along with a number of boundary faces with different zone identifiers.

2. Check the grid.

Grid → Check

## Grid Check

## Domain Extents:

x-coordinate: min (m) = 0.000000e+00, max (m) = 6.400001e+01

y-coordinate: min (m) = -4.538534e+00, max (m) = 6.400000e+01

## Volume statistics:

minimum volume (m3): 2.782193e-01

maximum volume (m3): 3.926232e+00

total volume (m3): 1.682930e+03

## Face area statistics:

minimum face area (m2): 8.015718e-01

maximum face area (m2): 4.118252e+00

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 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.

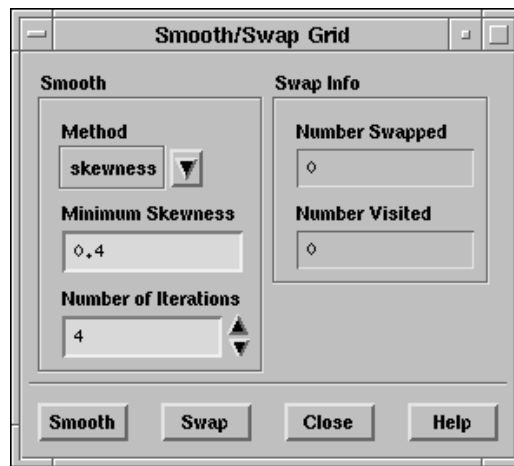
Checking storage.

Done.

**Note:** *The grid check lists the minimum and maximum x and y values from the grid, in the default SI units of meters, and reports on a number of other grid features that are checked. Any errors in the grid would 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 if this is the case. To scale the grid to the correct units of inches, the Scale Grid panel will be used.*

3. Smooth (and swap) the grid.

**Grid** → Smooth/Swap...



*To ensure the best possible grid quality for the calculation, it is good practice to smooth a triangular or tetrahedral grid after you read it into FLUENT.*

- (a) Click the **Smooth** button and then click **Swap** repeatedly until FLUENT reports that zero faces were swapped.

*If FLUENT cannot improve the grid by swapping, no faces will be swapped.*

- (b) Close the panel.

4. Scale the grid.

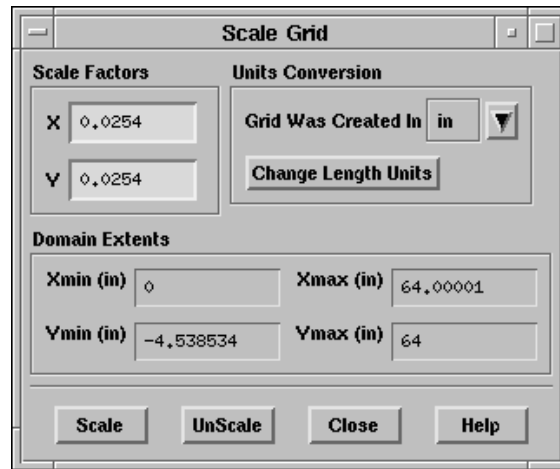
**Grid** → Scale...

- (a) Under **Units Conversion**, select in from the drop-down list to complete the phrase **Grid Was Created In** in (inches).
- (b) Click **Scale** to scale the grid.

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

- (c) Click **Change Length Units** to set inches as the working units for length.

*Confirm that the maximum  $x$  and  $y$  values are 64 inches (see Figure 1.1).*

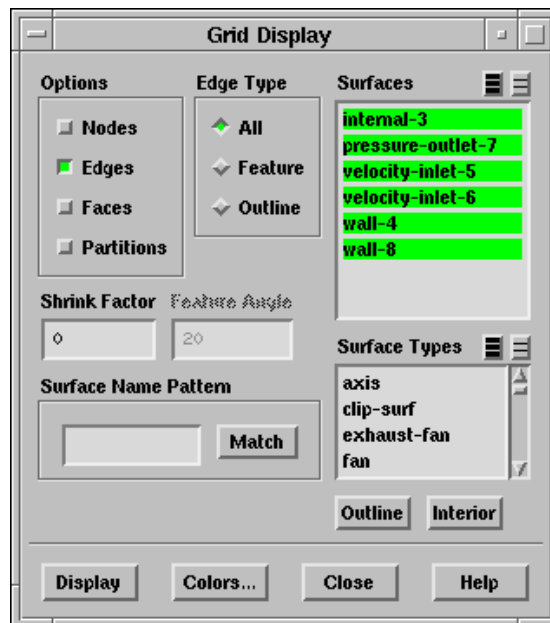


- (d) The grid is now sized correctly, and the working units for length have been set to inches. Close the panel.

**Note:** Because the default SI units will be used for everything but the 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 want to change the working units for length to something other than inches, say, mm, you would have to visit the Set Units panel in the Define pull-down menu.

5. Display the grid (Figure 1.2).

Display → Grid...



- (a) Make sure that all of the surfaces are selected and click Display.

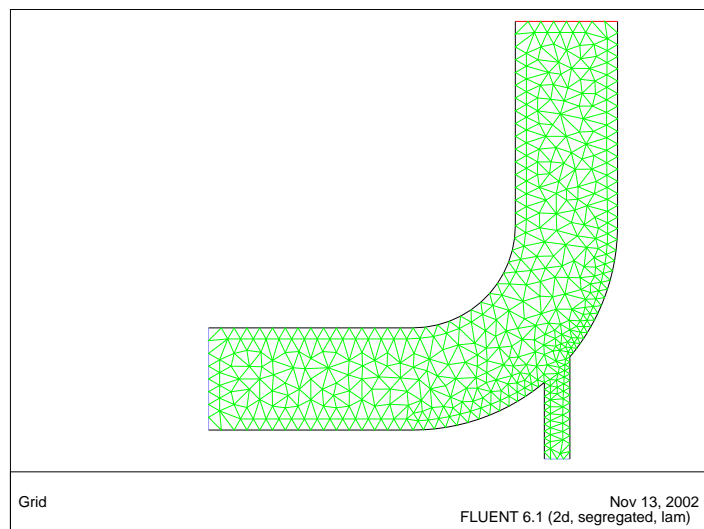


Figure 1.2: The Triangular Grid for the Mixing Elbow

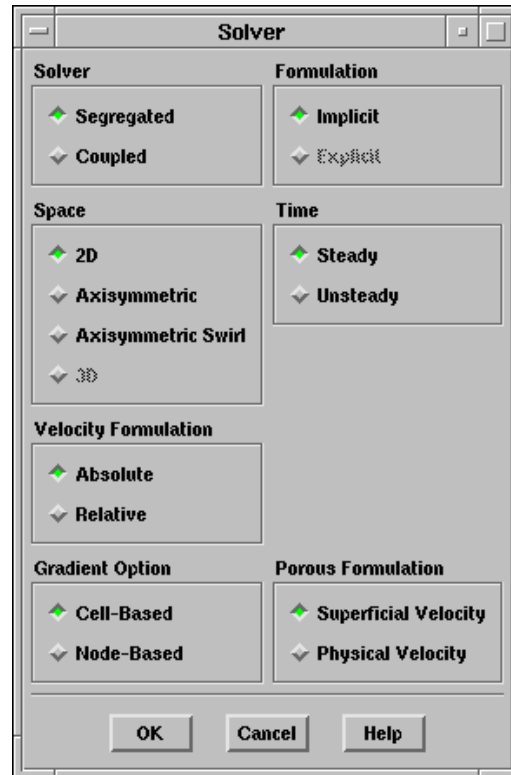
**Extra:** *You can use the right mouse button to check which zone number corresponds to each boundary. If you click the right mouse button on one of the boundaries in the graphics window, its zone number, name, and type will be printed in the FLUENT console window. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly.*



## Step 2: Models

1. Keep the default solver settings.

Define → Models → Solver...



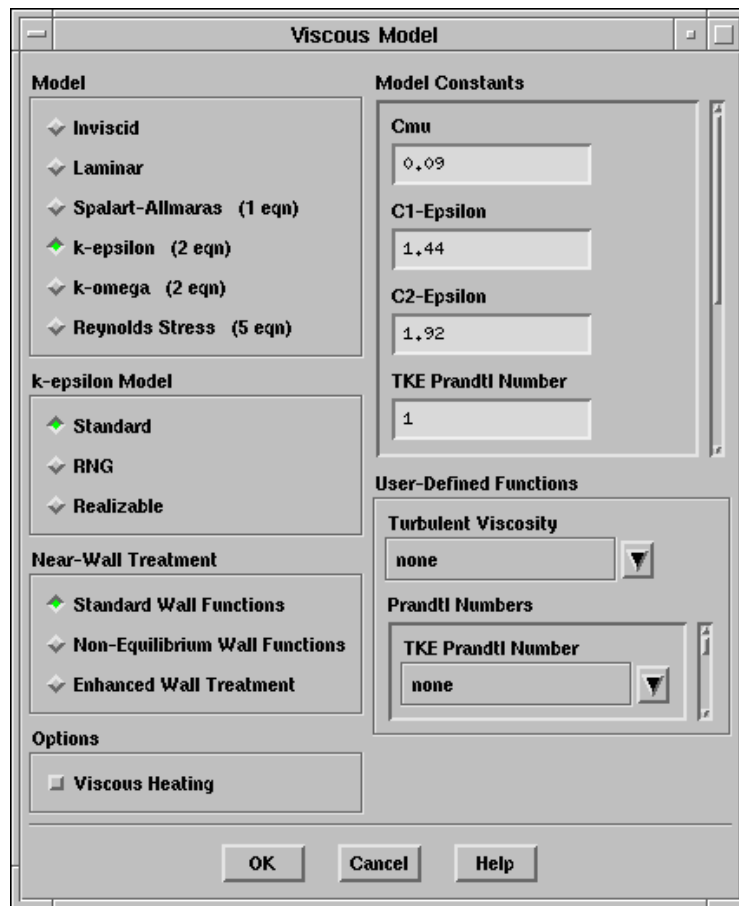
2. Turn on the standard  $k$ - $\epsilon$  turbulence model.

Define → Models → Viscous...

- (a) Select k-epsilon in the Model list.

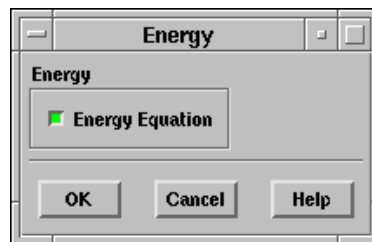
*The original Viscous Model panel will expand when you do so.*

- (b) Accept the default Standard model by clicking OK.



3. Enable heat transfer by activating the energy equation.

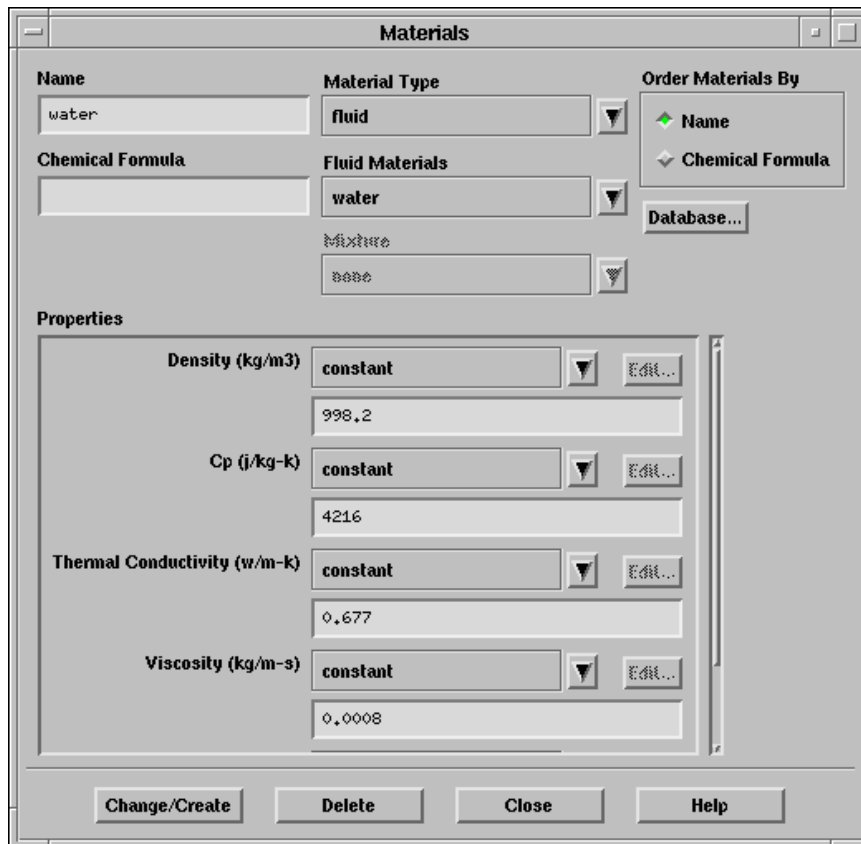
Define → Models → Energy...



## Step 3: Materials

1. Create a new material called water.

Define → Materials...



- (a) Type the name **water** in the Name text-entry box.
- (b) Enter the values shown in the table below under Properties:

Property	Value
density	1000 kg/m <sup>3</sup>
$c_p$	4216 J/kg-K
thermal conductivity	0.677 W/m-K
viscosity	$8 \times 10^{-4}$ kg/m-s

- (c) Click **Change/Create**.
- (d) Click **No** when FLUENT asks if you want to overwrite air.

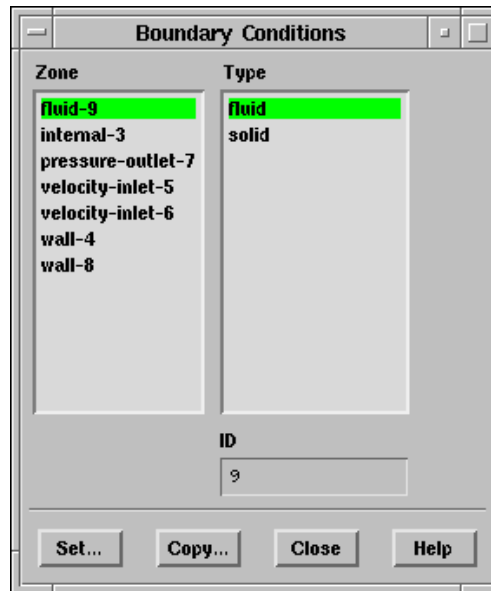
*The material water will be added to the list of materials which originally contained only air. You can confirm that there are now two materials defined by examining the drop-down list under Fluid Materials.*

**Extra:** *You could have copied the material **water** from the materials database (accessed by clicking on the **Database...** button). If the properties in the database are different from those you wish to use, you can still edit the values under **Properties** and click the **Change/Create** button to update your local copy. (The database will not be affected.)*

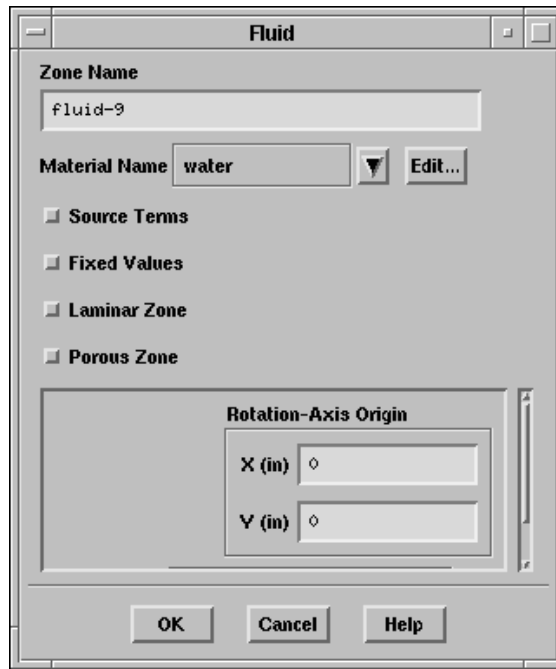
- (e) Close the **Materials** panel.

## Step 4: Boundary Conditions

Define → Boundary Conditions...

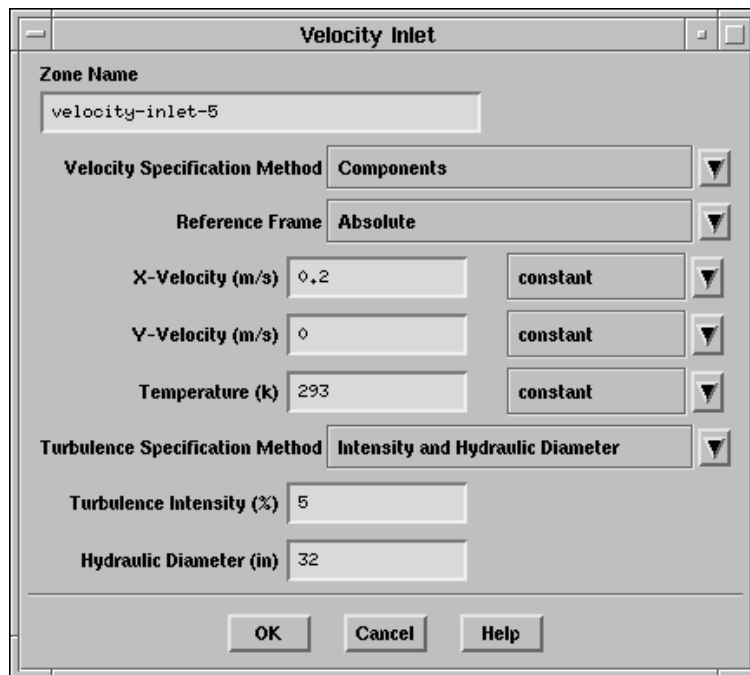


1. Set the conditions for the fluid.
  - (a) Select fluid-9 under Zone.  
*The Type will be reported as fluid.*
  - (b) Click Set... to open the Fluid panel.
  - (c) Specify water as the fluid material by selecting water in the Material Name drop-down list. Click on OK.



2. Set the boundary conditions at the main inlet.
  - (a) Select velocity-inlet-5 under Zone and click Set....

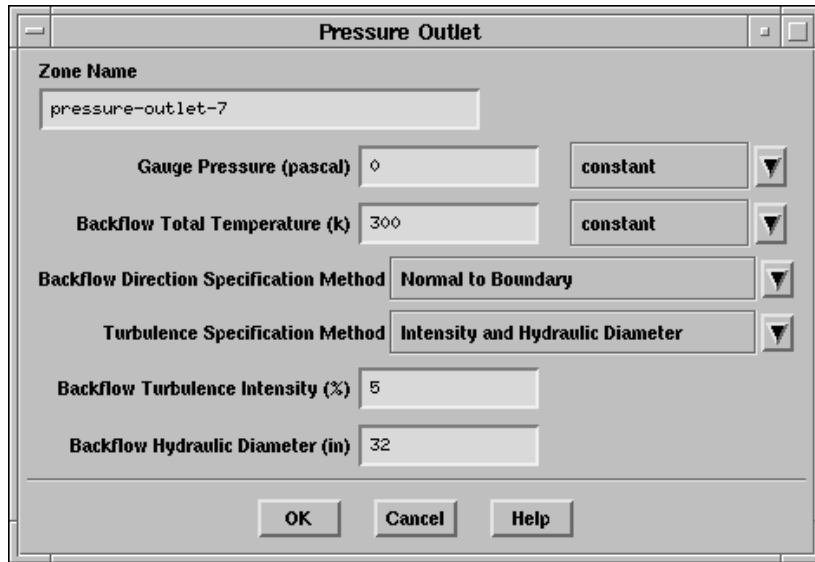
**Hint:** *If you are unsure of which inlet zone corresponds to the main inlet, you can probe the grid display with the right mouse button and the zone ID will be displayed in the FLUENT console window. In the Boundary Conditions panel, the zone that you probed will automatically be selected in the Zone list. In 2D simulations, it may be helpful to return to the Grid Display panel and deselect the display of the fluid and interior zones (in this case, fluid-9 and internal-3) before probing with the mouse button for zone names.*



- (b) Choose Components as the Velocity Specification Method.
  - (c) Set an X-Velocity of 0.2 m/s.
  - (d) Set a Temperature of 293 K.
  - (e) Select Intensity and Hydraulic Diameter as the Turbulence Specification Method.
  - (f) Enter a Turbulence Intensity of 5%, and a Hydraulic Diameter of 32 in.
3. Repeat this operation for velocity-inlet-6, using the values in the following table:

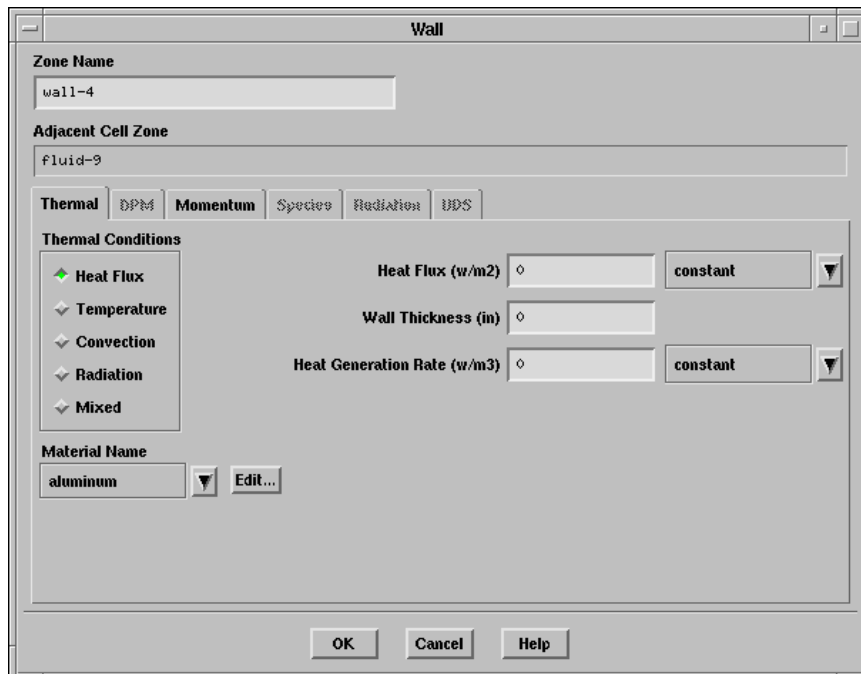
velocity specification method	components
$y$ velocity	1.0 m/s
temperature	313 K
turbulence specification method	intensity & hydraulic diameter
turbulence intensity	5%
hydraulic diameter	8 in

4. Set the boundary conditions for pressure-outlet-7, as shown in the panel below.



*These values will be used in the event that flow enters the domain through this boundary.*

5. For wall-4, keep the default settings for a Heat Flux of 0.





6. For wall-8, you will also keep the default settings.

**Note:** *If you probe your display of the grid (without the interior cells) you will see that wall-8 is the wall on the outside of the bend just after the junction. This separate wall zone has been created for the purpose of doing certain postprocessing tasks, to be discussed later in this tutorial.*

## Step 5: Solution

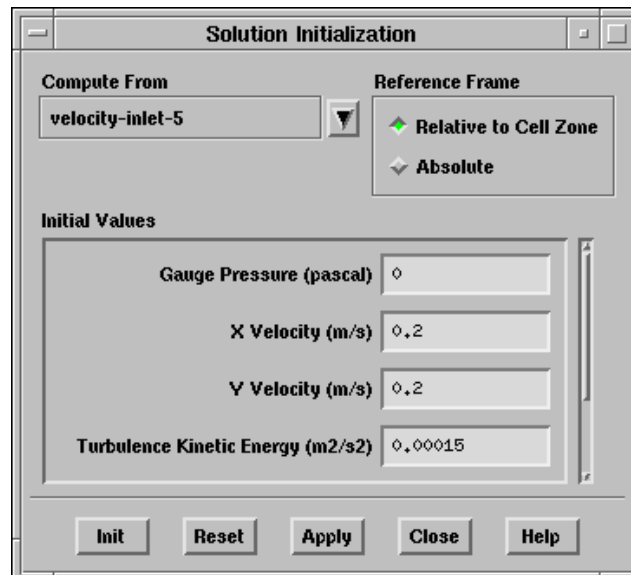
1. Initialize the flow field using the boundary conditions set at velocity-inlet-5.

**Solve** → **Initialize** → Initialize...

- (a) Choose velocity-inlet-5 from the Compute From list.
- (b) Add a Y Velocity value of 0.2 m/sec throughout the domain.

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

- (c) Click Init and Close the panel.



2. Enable the plotting of residuals during the calculation.

Solve → Monitors → Residual...

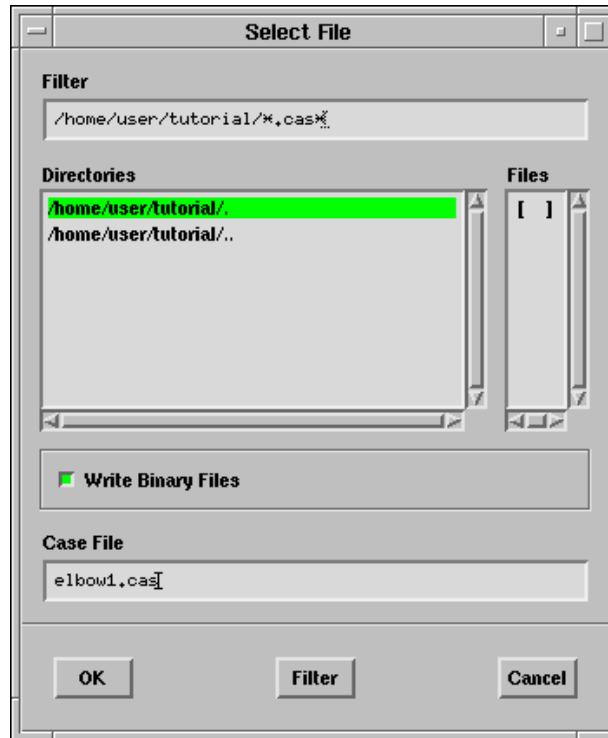


- (a) Select Plot under Options, and click OK.

**Note:** *By default, all variables will be monitored and checked for determining the convergence of the solution.*

3. Save the case file (elbow1.cas).

File → Write → Case...

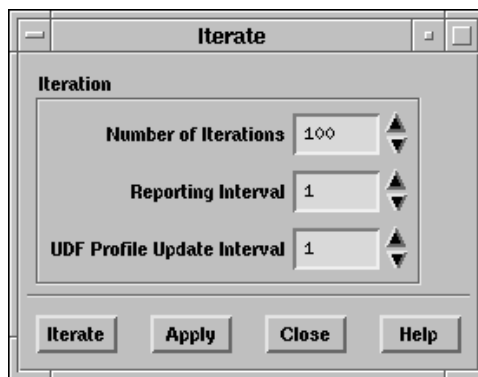


Keep the Write Binary Files (default) option on so that a binary file will be written.

4. Start the calculation by requesting 100 iterations.

Solve → Iterate...

- (a) Input 100 for the Number of Iterations and click Iterate.



*The solution reaches convergence after approximately 60 iterations. The residual plot is shown in Figure 1.3. Note that 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.*

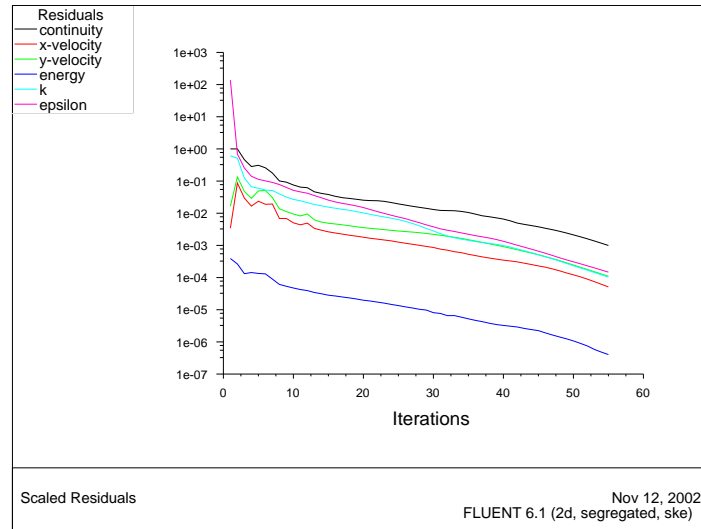


Figure 1.3: Residuals for the First 60 Iterations

#### 5. Check for convergence.

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.

The three methods to check for convergence are:

- Monitoring the residuals.

Convergence will occur 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  $10^{-6}$ .

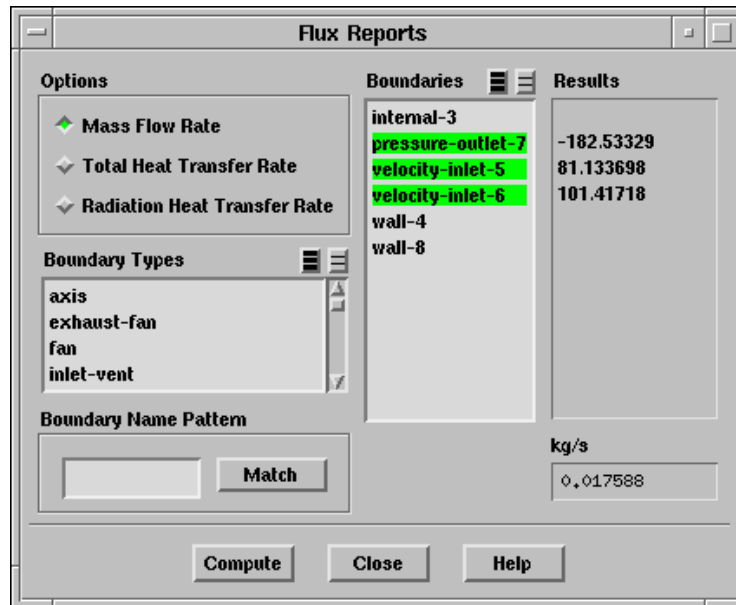
- Solution no longer changes with more iterations.

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.

- Overall mass, momentum, energy and scalar balances are obtained.

Check the overall mass, momentum, energy and scalar balances in the Flux Reports panel. The net imbalance should be less than 0.1% of the net flux through the domain.

Report → Fluxes



6. Save the data file (elbow1.dat).

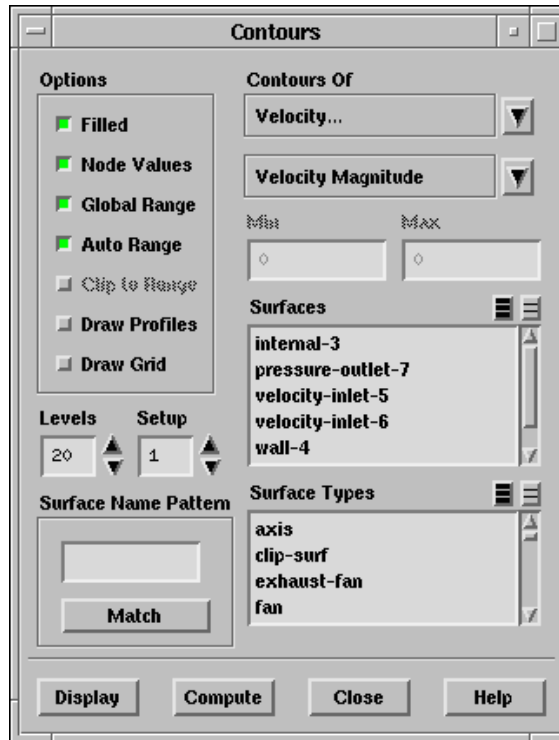
Use the same prefix (*elbow1*) that you used when you saved the case file earlier. Note that additional case and data files will be written later in this session.

File → Write → Data...

## Step 6: Displaying the Preliminary Solution

1. Display filled contours of velocity magnitude (Figure 1.4).

Display → Contours...



- (a) Select Velocity... and then Velocity Magnitude from the drop-down lists under Contours Of.
- (b) Select Filled under Options.
- (c) Click Display.

**Note:** *Right-clicking on a point in the domain will cause the value of the corresponding contour to be displayed in the console window.*

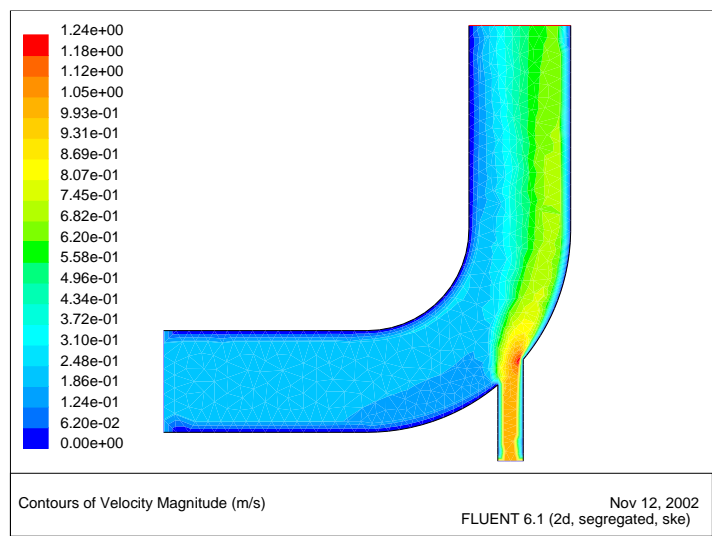
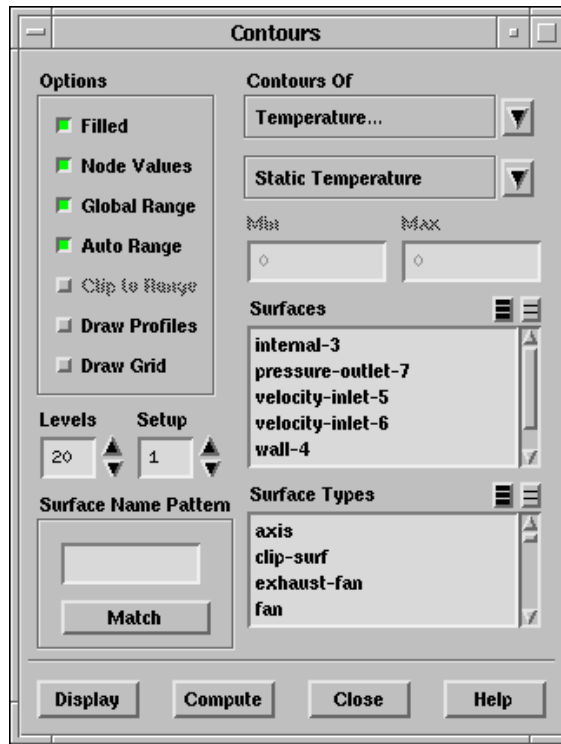


Figure 1.4: Predicted Velocity Distribution After the Initial Calculation



2. Display filled contours of temperature (Figure 1.5).



- (a) Select Temperature... and Static Temperature in the drop-down lists under Contours Of.
- (b) Click Display.

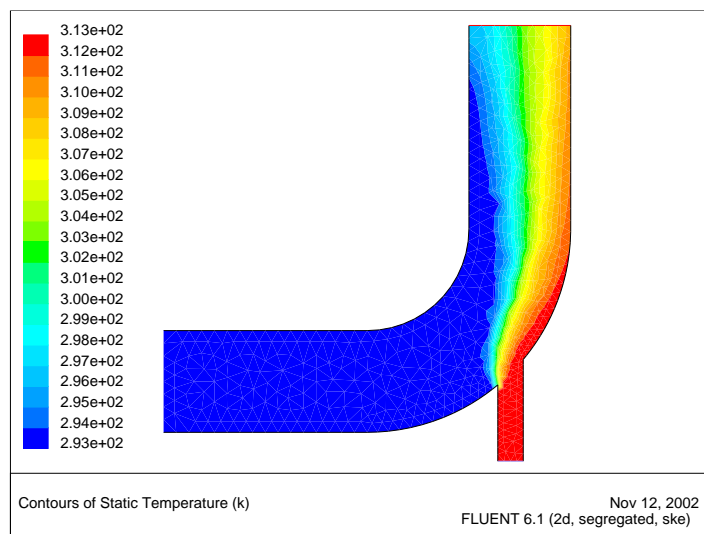


Figure 1.5: Predicted Temperature Distribution After the Initial Calculation

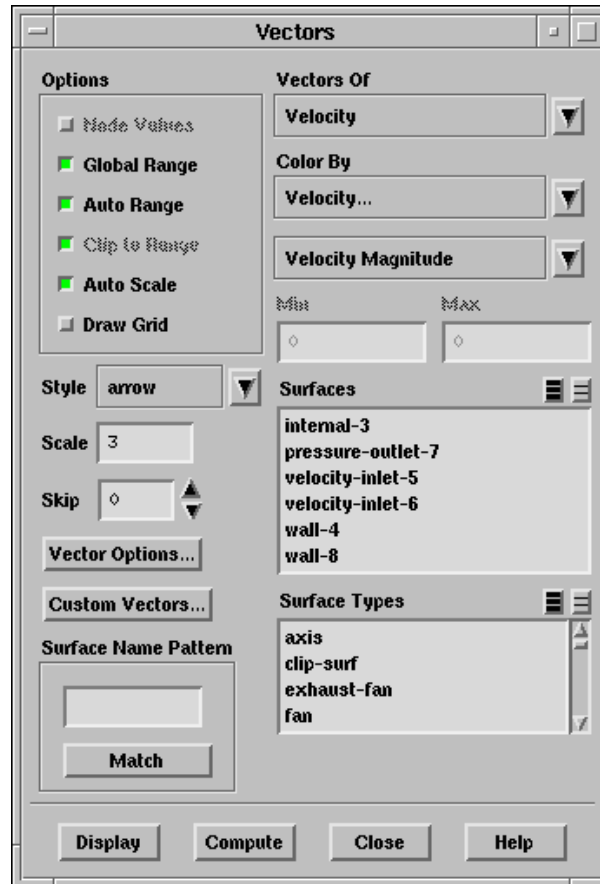
## 3. Display velocity vectors (Figure 1.6).

**Display** → Vectors...

- (a) Click Display to plot the velocity vectors.

**Note:** *The Auto Scale button is on by default under Options. This scaling sometimes creates vectors that are too small or too large in the majority of the domain.*

- (b) Resize the vectors by increasing the Scale factor to 3.



- (c) Display the vectors once again.
- (d) Use the middle mouse button to zoom the view. To do this, hold down the button and drag your mouse to the right and either up or down to construct a rectangle on the screen. The rectangle should be a frame around the region that you wish to enlarge. Let go of the mouse button and the image will be redisplayed (Figure 1.7).
- (e) Un-zoom the view by holding down the middle mouse button and dragging it to the left to create a rectangle. When you let go, the image will be redrawn. If the resulting image is not centered, you can use the left mouse button to translate it on your screen.

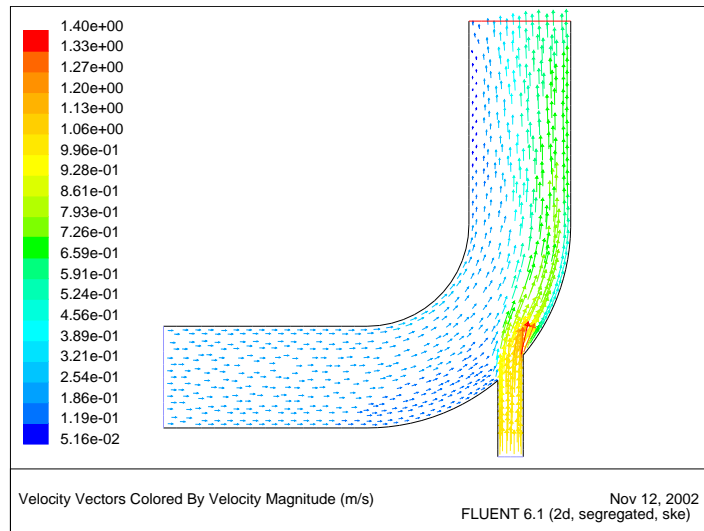


Figure 1.6: Resized Velocity Vectors

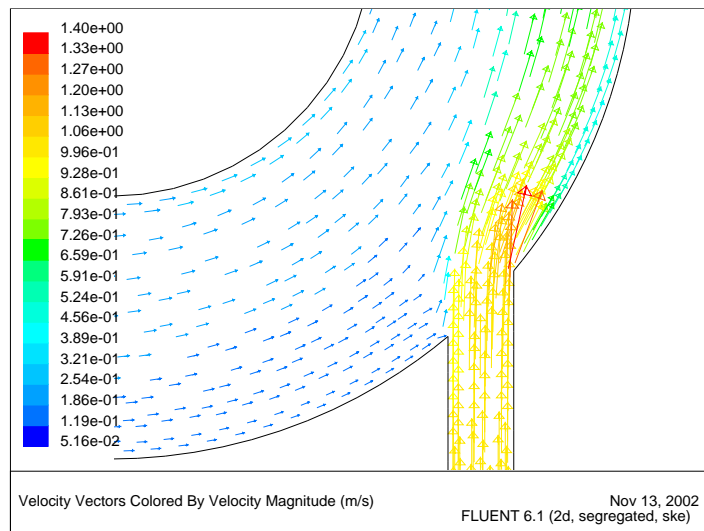
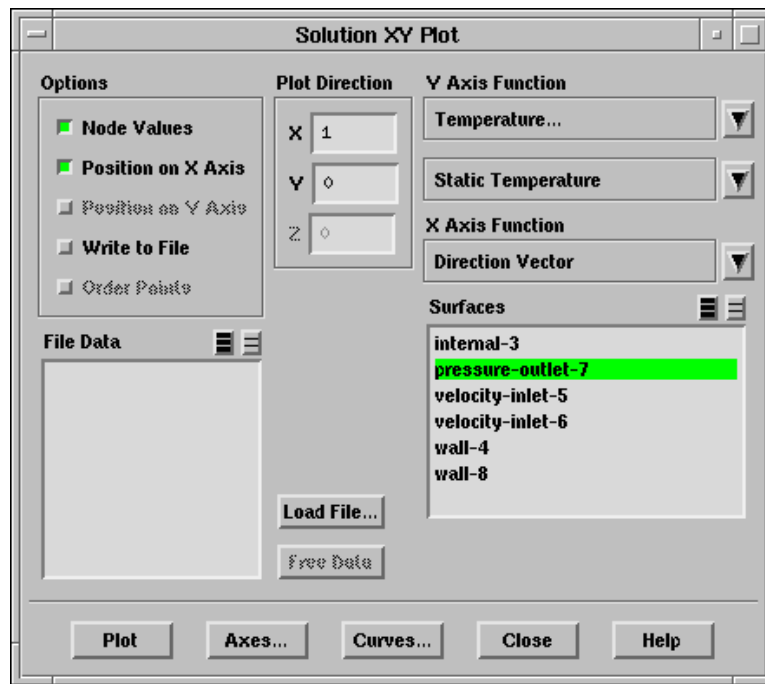


Figure 1.7: Magnified View of Velocity Vectors

4. Create an XY plot of temperature across the exit (Figure 1.8).

**Plot** → XY Plot...



- Select Temperature... and Static Temperature in the drop-down lists under the Y Axis Function.
- Select pressure-outlet-7 from the Surfaces list.
- Click Plot.

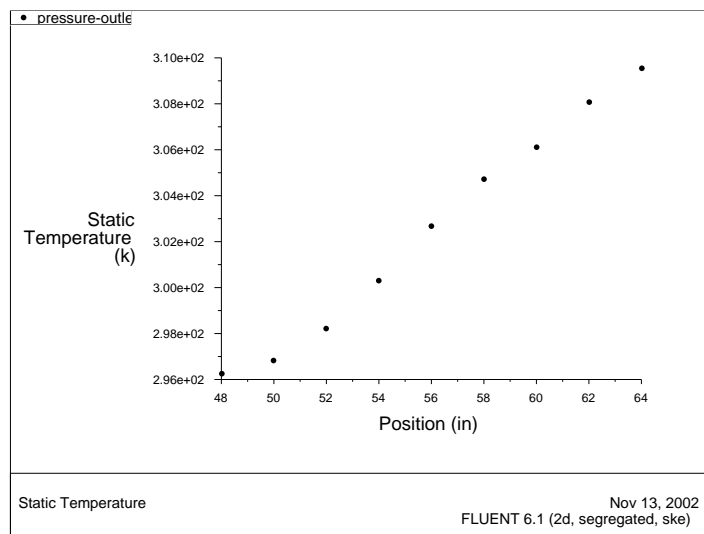
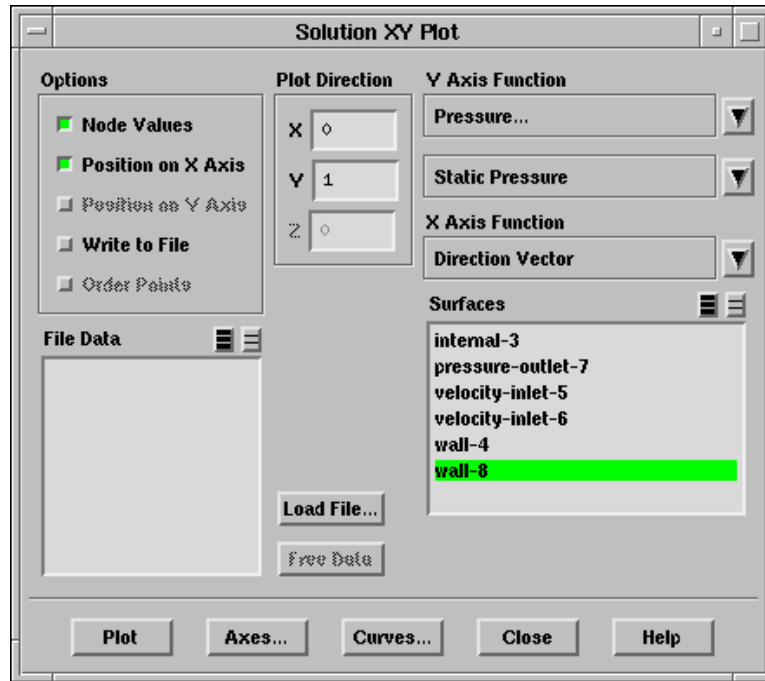


Figure 1.8: Temperature Distribution at the Outlet

5. Make an XY plot of the static pressure on the outer wall of the large pipe, wall-8 (Figure 1.9).



- (a) Choose Pressure... and Static Pressure from the Y Axis Function drop-down lists.
- (b) Deselect pressure-outlet-7 and select wall-8 from the Surfaces list.
- (c) Change the Plot Direction for X to 0, and the Plot Direction for Y to 1.
- With a Plot Direction vector of  $(0, 1)$ , FLUENT will plot static pressure at the cells of wall-8 as a function of  $y$ .*
- (d) Click Plot.

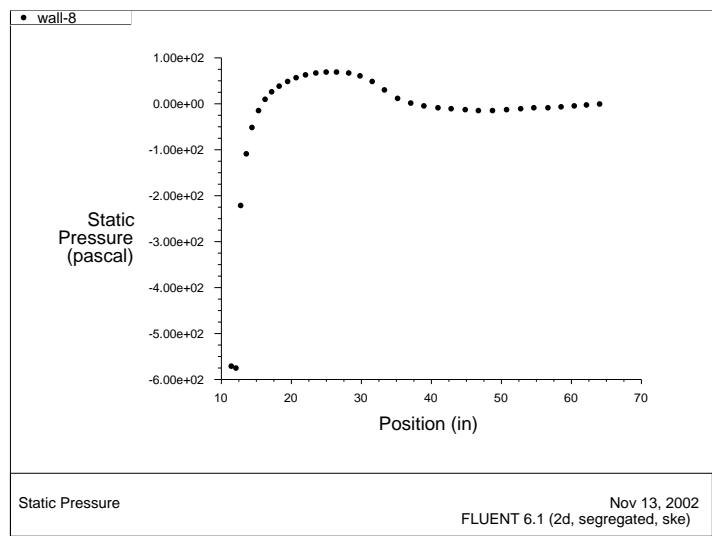
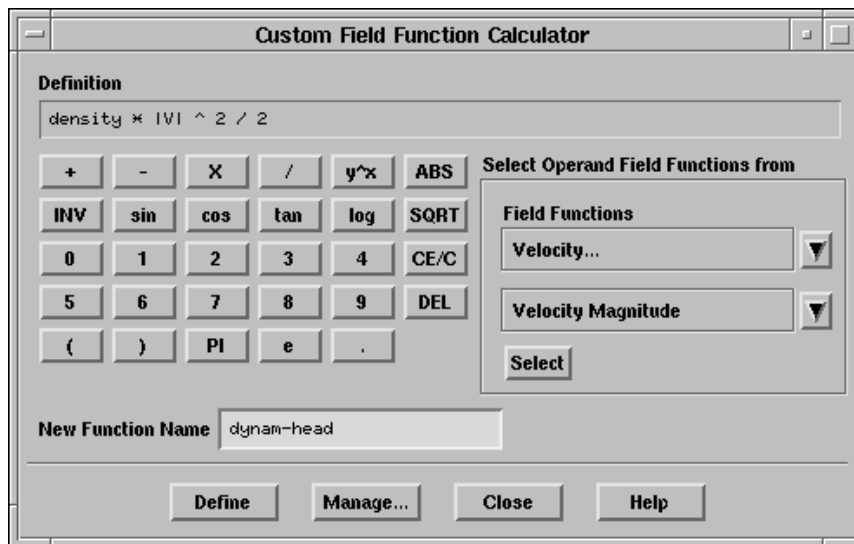


Figure 1.9: Pressure Distribution along the Outside Wall of the Bend



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

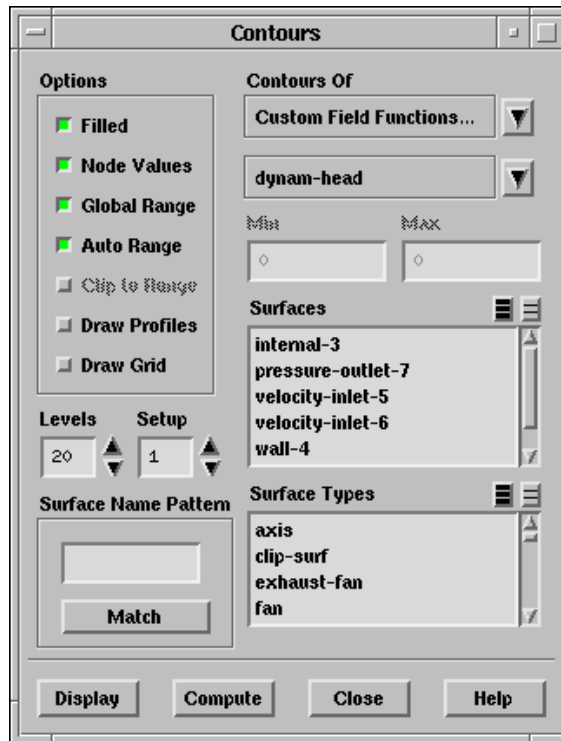
**Define** → Custom Field Functions...



- In the Field Functions drop-down list, select Density and click the Select button.
- Click the multiplication button, X.
- In the Field Functions drop-down list, select Velocity and Velocity Magnitude and click Select.
- Click  $y^x$  to raise the last entry to a power, and click 2 for the power.
- Click the divide button, /, and then click 2.
- Enter the name dynam-head in the New Function Name text entry box.
- Click Define, and then Close the panel.

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

Display → Contours...



- (a) Select Custom Field Functions... in the drop-down list under Contours Of.  
*The function you created, dynam-head, will be shown in the lower drop-down list.*
- (b) Click Display, and then Close the panel.

**Note:** *You may need to un-zoom your view after the last vector display, if you have not already done so.*

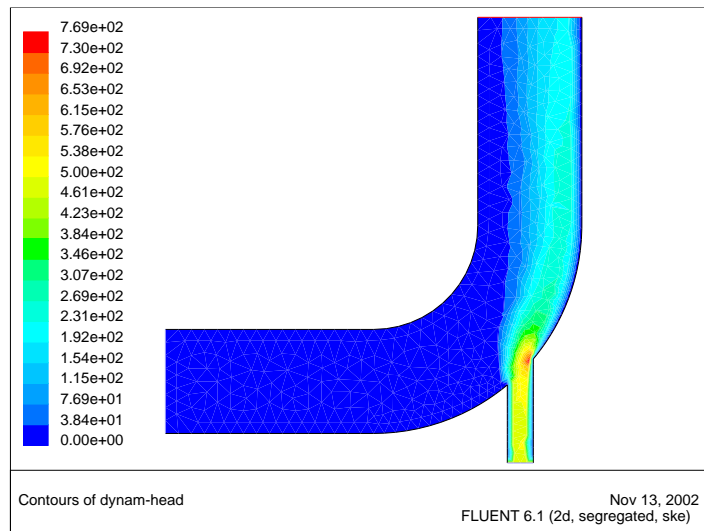


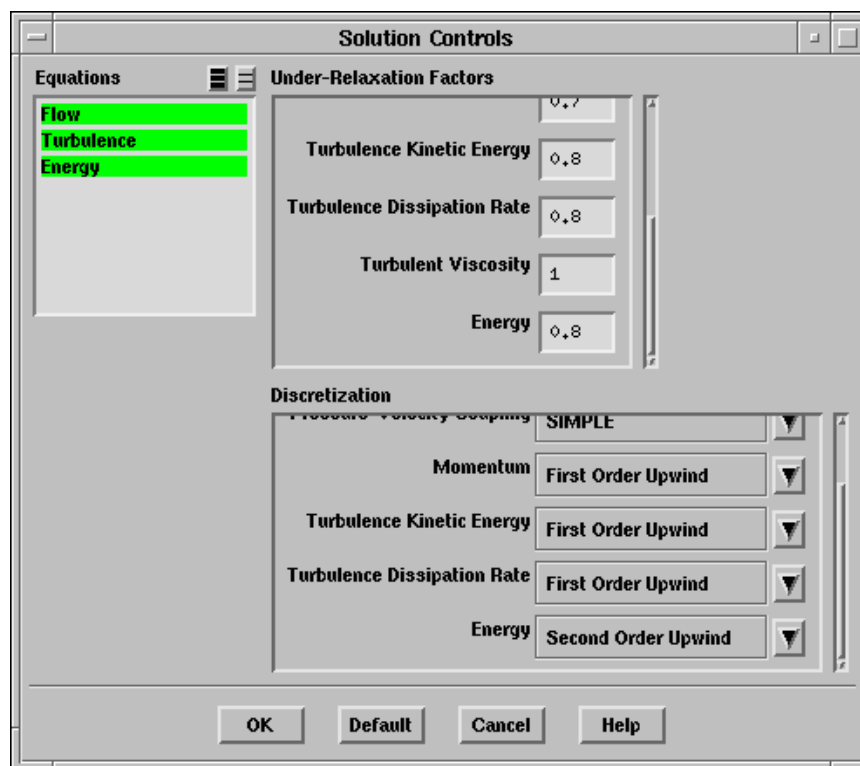
Figure 1.10: Contours of the Custom Field Function, Dynamic Head

## 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 the energy equation in order to improve the accuracy of the solution. With the second-order discretization, you will need to use a less aggressive (lower) value for the energy under-relaxation to ensure convergence.

1. Enable the second-order scheme for the calculation of energy and decrease the energy under-relaxation factor.

Solve → Controls → Solution...

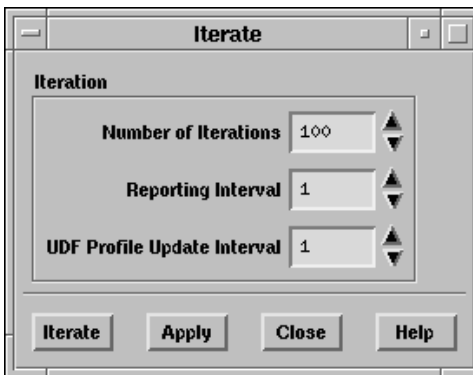


- (a) Under Discretization, select Second Order Upwind for Energy.
- (b) Under Under-Relaxation Factors, set the Energy under-relaxation factor to 0.8.

**Note:** You will have to scroll down both the Discretization and Under-Relaxation Factors lists to see the Energy options.

2. Continue the calculation by requesting 100 more iterations.

**Solve** → Iterate...



*The solution converges in approximately 35 additional iterations.*

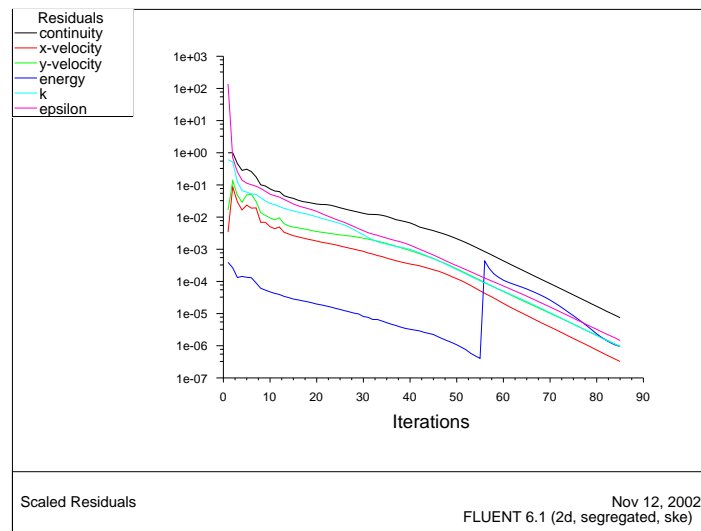


Figure 1.11: Residuals for the Second-Order Energy Calculation

**Note:** *Whenever you change the solution control parameters, it is natural to see the residuals jump.*

3. Write the case and data files for the second-order solution (`elbow2.cas` and `elbow2.dat`).

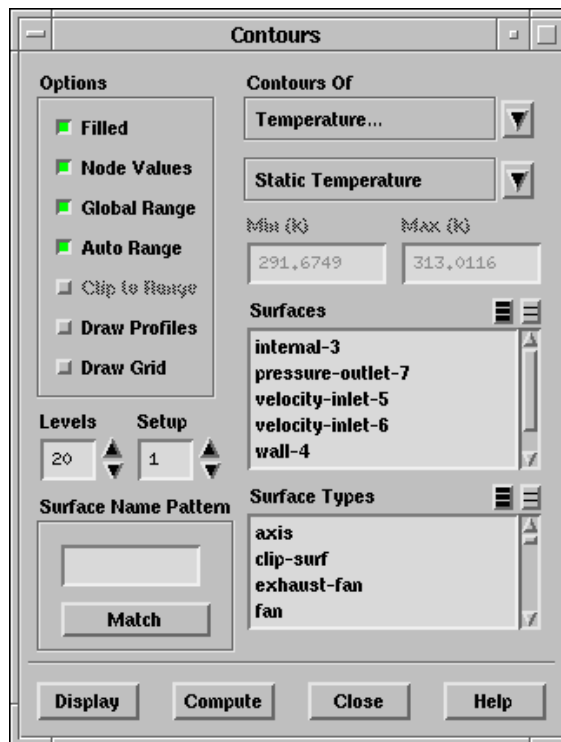
**File** → **Write** → Case & Data...

- (a) Enter the name `elbow2` in the Case/Data File box.
- (b) Click OK.

*The files `elbow2.cas` and `elbow2.dat` will be created in your directory.*

4. Examine the revised temperature distribution (Figure 1.12).

**Display** → Contours...



*The thermal spreading after the elbow has been reduced from the earlier prediction (Figure 1.5).*

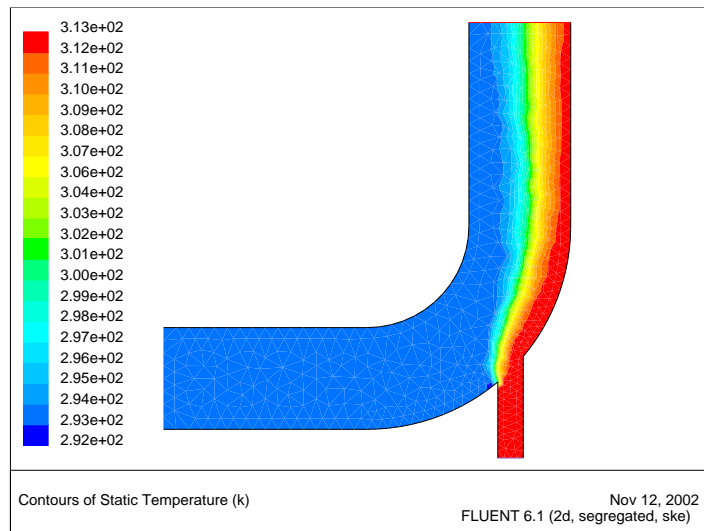


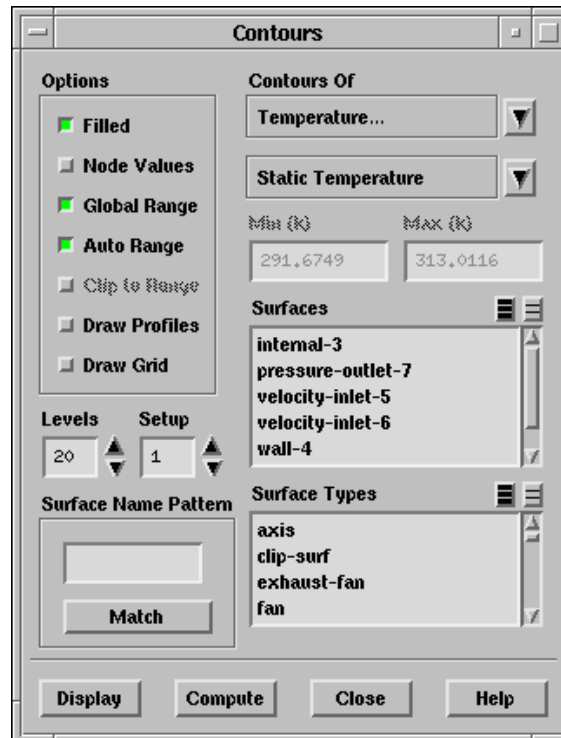
Figure 1.12: Temperature Contours for the Second-Order Solution

## Step 8: Adapting the Grid

The elbow solution can be improved further by refining the grid to better resolve the flow details. In this step, you will adapt the grid based on the temperature gradients in the current solution. Before adapting the grid, you will first determine an acceptable range of temperature gradients over which to adapt. Once the grid has been refined, you will continue the calculation.

1. Plot filled contours of temperature on a cell-by-cell basis (Figure 1.13).

Display → Contours...

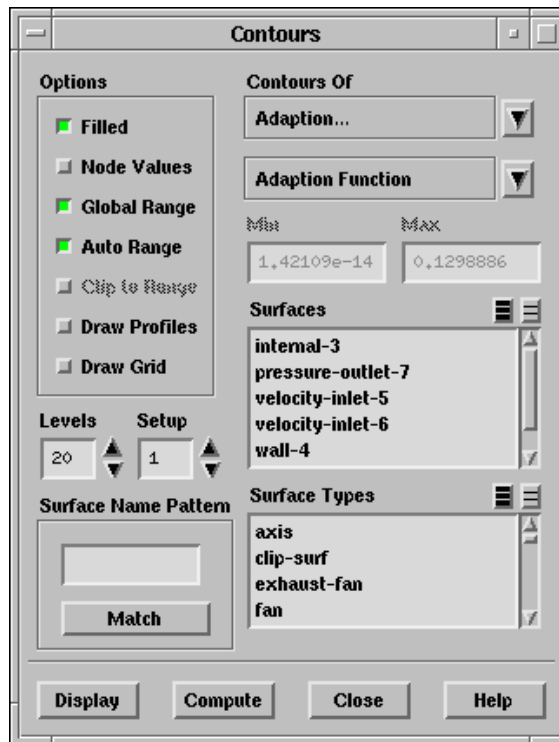


- (a) Select Temperature... and Static Temperature in the Contours Of drop-down lists.
- (b) Deselect Node Values under Options and click Display.

**Note:** When the contours are displayed you will see the cell values of temperature instead of the smooth-looking node values. Node values are obtained by averaging the values at all of the cells that share the node. Cell values are the values that are stored at each cell center and are displayed throughout the cell. Examining the cell-by-cell values is helpful when you are preparing to do an adaptation of the grid because it indicates the region(s) where the adaptation will take place.



2. Plot the temperature gradients that will be used for adaption (Figure 1.14).



- Select Adaption... and Adaption Function in the Contours Of drop-down lists.
- Click Display to see the gradients of temperature, displayed on a cell-by-cell basis.

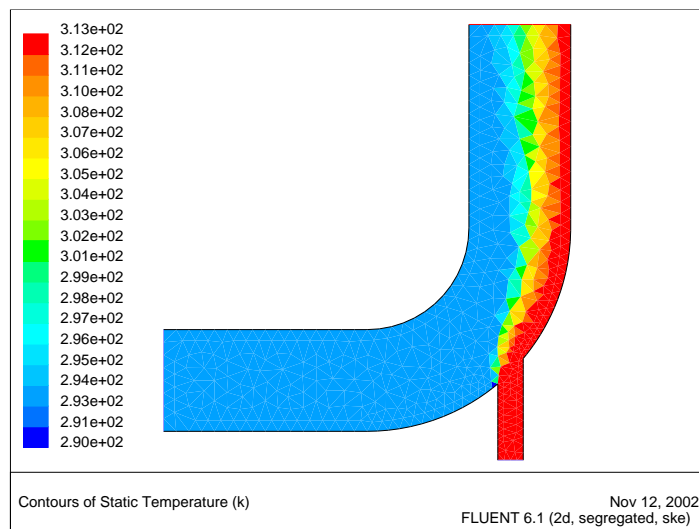


Figure 1.13: Temperature Contours for the Second-Order Solution: Cell Values

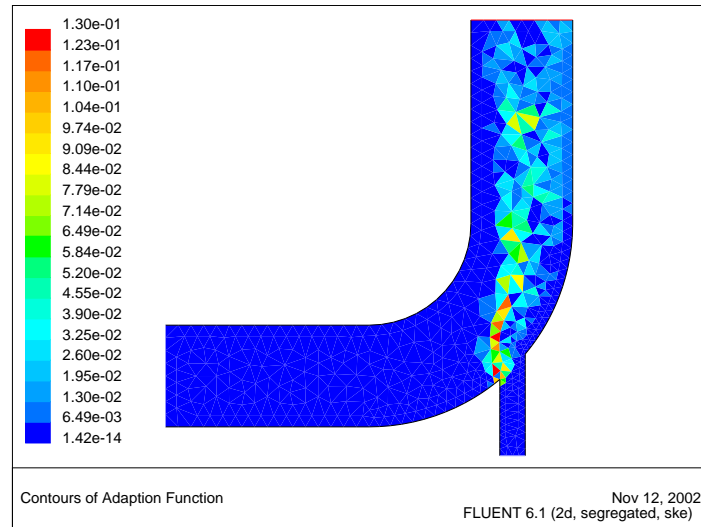


Figure 1.14: Contours of Adaption Function: Temperature Gradient

**Note:** *The quantity Adaption Function defaults to the gradient of the variable whose Max and Min were most recently computed in the Contours panel. In this example, the static temperature is the most recent variable to have its Max and Min computed, since this occurs when the Display button is pushed. Note that for other applications, gradients of another variable might be more appropriate for performing the adaption.*

3. Plot temperature gradients over a limited range in order to mark cells for adaption (Figure 1.15).

- (a) Under **Options**, deselect **Auto Range** so that you can change the minimum temperature gradient value to be plotted.

*The Min temperature gradient is 0 K/m, as shown in the Contours panel.*

- (b) Enter a new **Min** value of 0.02.
- (c) Click **Display**.

*The colored cells in the figure are in the “high gradient” range, so they will be the ones targeted for adaption.*

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

**Adapt** → Gradient...

- (a) Select **Temperature...** and **Static Temperature** in the **Gradients Of** drop-down lists.
- (b) Deselect **Coarsen** under **Options**, so that only a refinement of the grid will be performed.

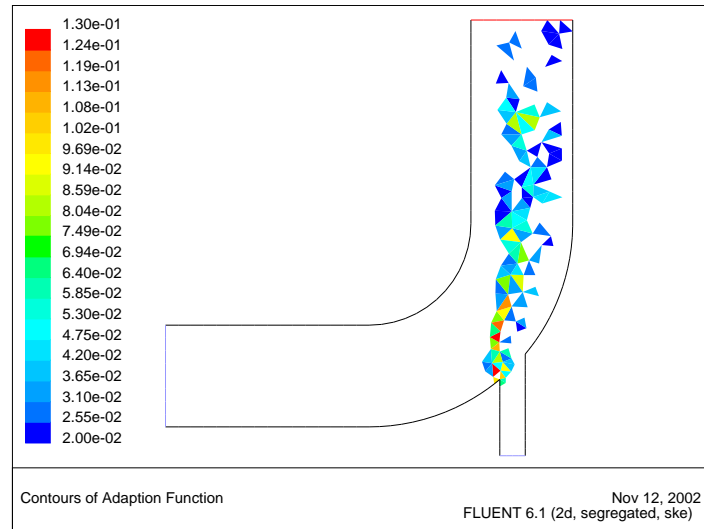
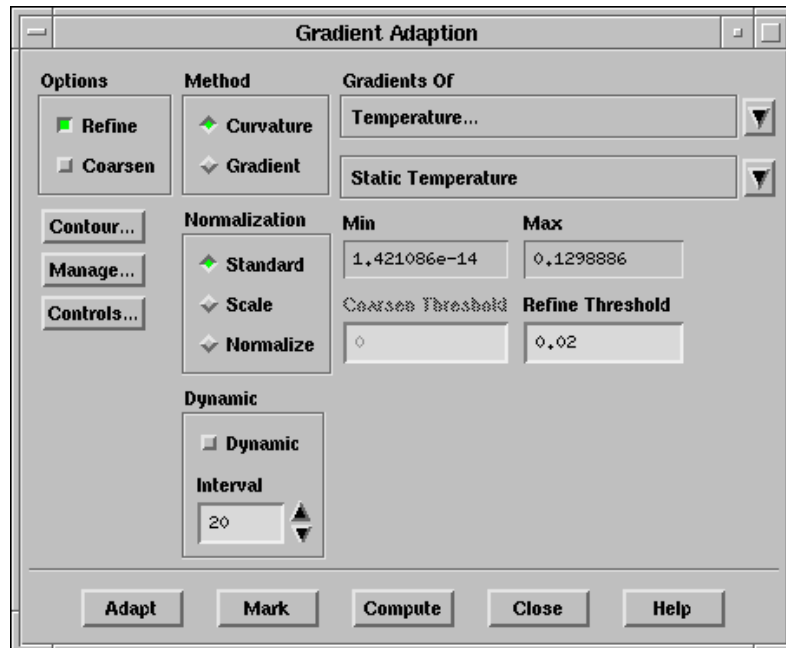


Figure 1.15: Contours of Temperature Gradient Over a Limited Range

(c) Click Compute.

FLUENT will update the Min and Max values.

(d) Enter the value of 0.02 for the Refine Threshold.



- (e) Click Mark.

FLUENT will report the number of cells marked for adaption in the console window.

- (f) Click Manage... to display the marked cells.

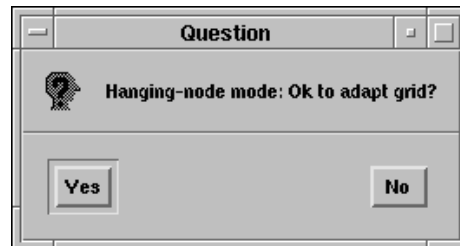
This will open the Manage Adaption Registers panel.



- (g) Click Display.

FLUENT will display the cells marked for adaption (Figure 1.16).

- (h) Click Adapt. Click Yes when you are asked for confirmation.



**Note:** There are two different ways to adapt. You can click on Adapt in the Manage Adaption Registers panel as was just done, or Close this panel and do the adaption in 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.

- (i) Close the Manage Adaption Registers and Gradient Adaption panels.

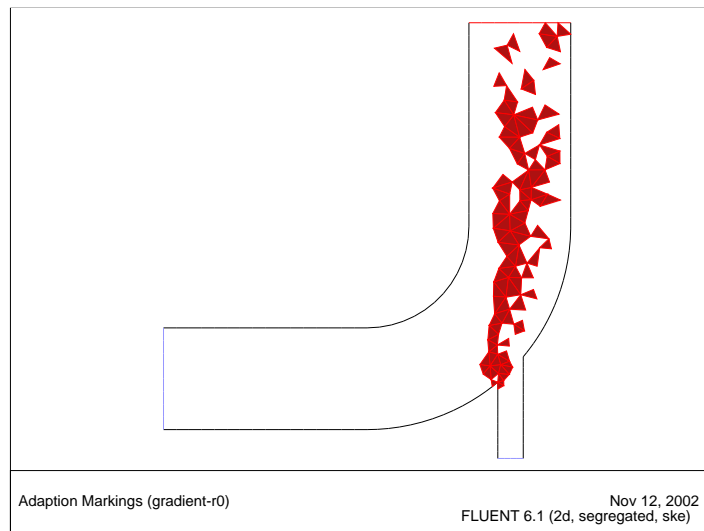


Figure 1.16: Cells Marked for Adaption

5. Display the adapted grid (Figure 1.17).

**Display** → Grid...

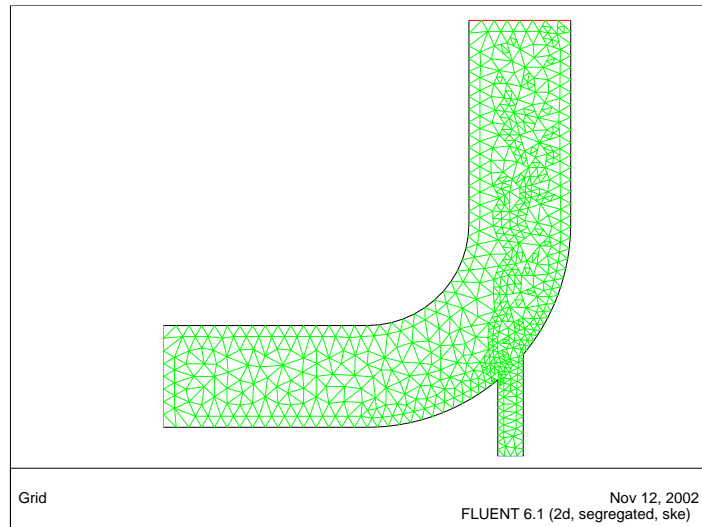
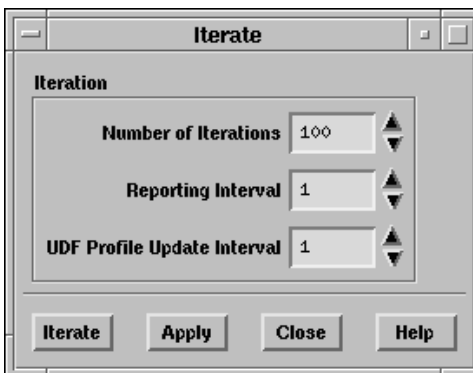


Figure 1.17: The Adapted Grid

6. Request an additional 100 iterations.

**Solve** → Iterate...



*The solution converges after approximately 40 additional iterations.*

7. Write the final case and data files (elbow3.cas and elbow3.dat) using the prefix elbow3.

**File** → **Write** → Case & Data...

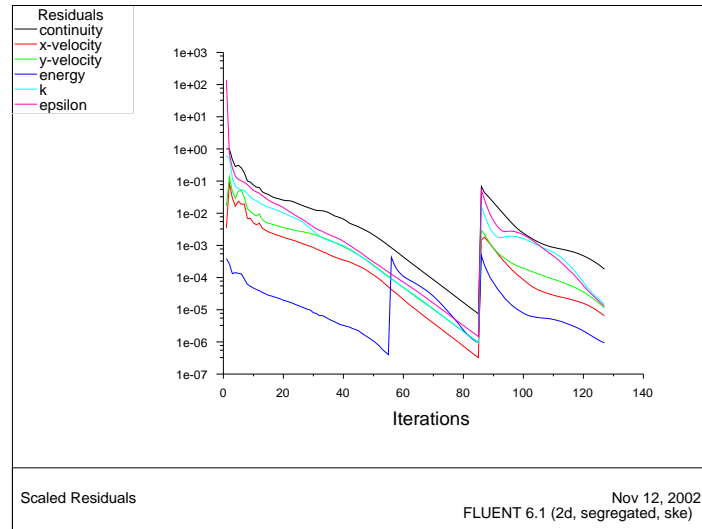


Figure 1.18: The Complete Residual History

8. Examine the filled temperature distribution (using node values) on the revised grid (Figure 1.19).

→ Contours...

**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 solution of the equations on and off during this procedure.

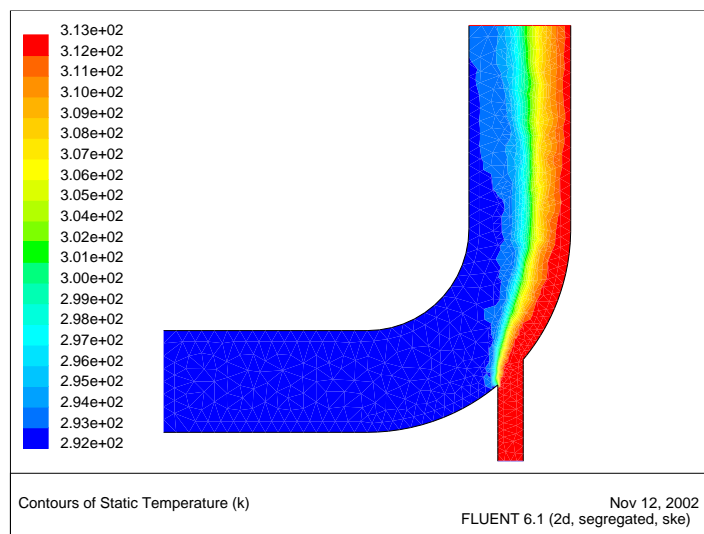


Figure 1.19: Filled Contours of Temperature Using the Adapted Grid