
Chapter 4: Modeling Periodic Flow and Heat Transfer

This tutorial is divided into the following sections:

- 4.1. Introduction
- 4.2. Prerequisites
- 4.3. Problem Description
- 4.4. Setup and Solution
- 4.5. Summary
- 4.6. Further Improvements

4.1. Introduction

Many industrial applications, such as steam generation in a boiler or air cooling in the coil of an air conditioner, can be modeled as two-dimensional periodic heat flow. This tutorial illustrates how to set up and solve a periodic heat transfer problem, given a pregenerated mesh.

The system that is modeled is a bank of tubes containing a flowing fluid at one temperature that is immersed in a second fluid in cross flow at a different temperature. Both fluids are water, and the flow is classified as laminar and steady, with a Reynolds number of approximately 100. The mass flow rate of the cross flow is known and the model is used to predict the flow and temperature fields that result from convective heat transfer.

Due to symmetry of the tube bank and the periodicity of the flow inherent in the tube bank geometry, only a portion of the geometry will be modeled in ANSYS FLUENT, with symmetry applied to the outer boundaries. The resulting mesh consists of a periodic module with symmetry. In the tutorial, the inlet boundary will be redefined as a periodic zone, and the outflow boundary defined as its shadow.

This tutorial demonstrates how to do the following:

- Create periodic zones.
- Define a specified periodic mass flow rate.
- Model periodic heat transfer with specified temperature boundary conditions.
- Calculate a solution using the pressure-based solver.
- Plot temperature profiles on specified isosurfaces.

4.2. Prerequisites

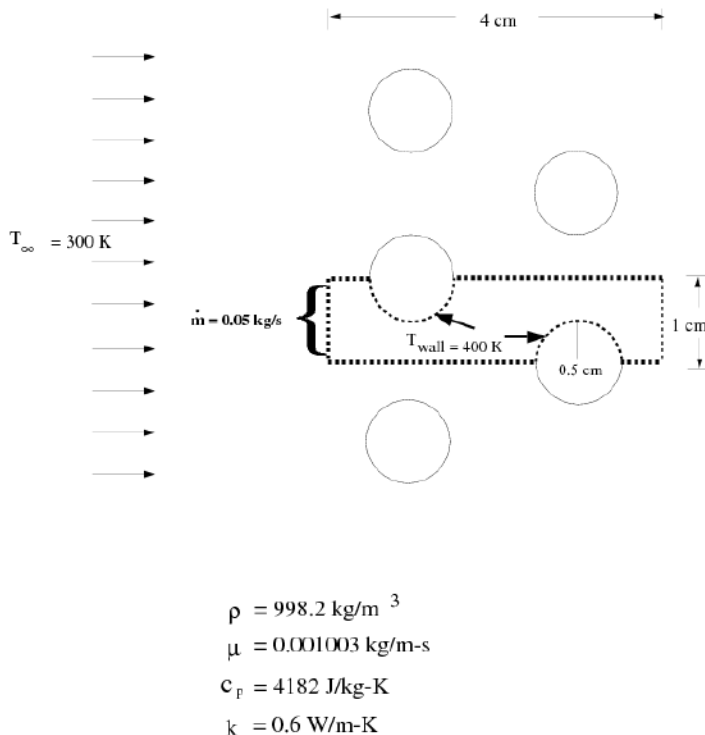
This tutorial assumes that you are familiar with the menu structure in ANSYS FLUENT and that you have completed *Introduction to Using ANSYS FLUENT: Fluid Flow and Heat Transfer in a Mixing Elbow* (p. 111). Some steps in the setup and solution procedure will not be shown explicitly.

4.3. Problem Description

This problem considers a 2D section of a tube bank. A schematic of the problem is shown in *Figure 4.1* (p. 178). The bank consists of uniformly spaced tubes with a diameter of 1 cm, which are staggered across the cross-

fluid flow. Their centers are separated by a distance of 2 cm in the x direction, and 1 cm in the y direction. The bank has a depth of 1 m.

Figure 4.1 Schematic of the Problem



Because of the symmetry of the tube bank geometry, only a portion of the domain needs to be modeled. The computational domain is shown in outline in [Figure 4.1 \(p. 178\)](#). A mass flow rate of 0.05 kg/s is applied to the inlet boundary of the periodic module. The temperature of the tube wall (T_{wall}) is 400 K and the bulk temperature of the cross flow water (T_{∞}) is 300 K. The properties of water that are used in the model are shown in [Figure 4.1 \(p. 178\)](#).

4.4. Setup and Solution

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

- 4.4.1. Preparation
- 4.4.2. Step 1: Mesh
- 4.4.3. Step 2: General Settings
- 4.4.4. Step 3: Models
- 4.4.5. Step 4: Materials
- 4.4.6. Step 5: Cell Zone Conditions
- 4.4.7. Step 6: Periodic Conditions
- 4.4.8. Step 7: Boundary Conditions
- 4.4.9. Step 8: Solution
- 4.4.10. Step 9: Postprocessing

4.4.1. Preparation

1. Download `periodic_flow_heat.zip` from the ANSYS [Customer Portal](#) or the [User Services Center](#) to your working folder (as described in [Preparation \(p. 4\)](#) of *Introduction to Using ANSYS FLUENT in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow* (p. 1)).

2. Unzip `periodic_flow_heat.zip`.

The file `tubebank.msh` can be found in the `periodic_flow_heat` folder created after unzipping the file.

3. Use FLUENT Launcher to start the **2D** version of ANSYS FLUENT.

For more information about FLUENT Launcher, see [Starting ANSYS FLUENT Using FLUENT Launcher in the User's Guide](#).

Note

The **Display Options** are enabled by default. Therefore, once you read in the mesh, it will be displayed in the embedded graphics windows.

4.4.2. Step 1: Mesh

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

File → **Read** → **Mesh...**

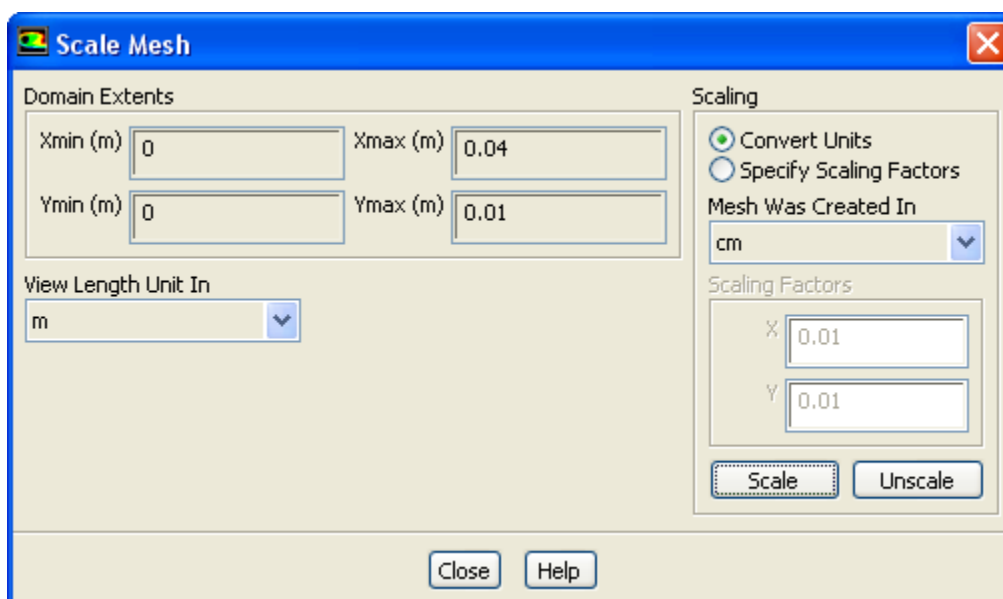
2. Check the mesh.

 **General** → **Check**

ANSYS FLUENT will perform various checks on the mesh and report the progress in the ANSYS FLUENT console window. Ensure that the minimum volume reported is a positive number.

3. Scale the mesh.

 **General** → **Scale...**



- a. Select **cm** (centimeters) from the **Mesh Was Created In** drop-down list in the **Scaling** group box.
- b. Click **Scale** to scale the mesh.

- c. Close the **Scale Mesh** dialog box.
4. Check the mesh.

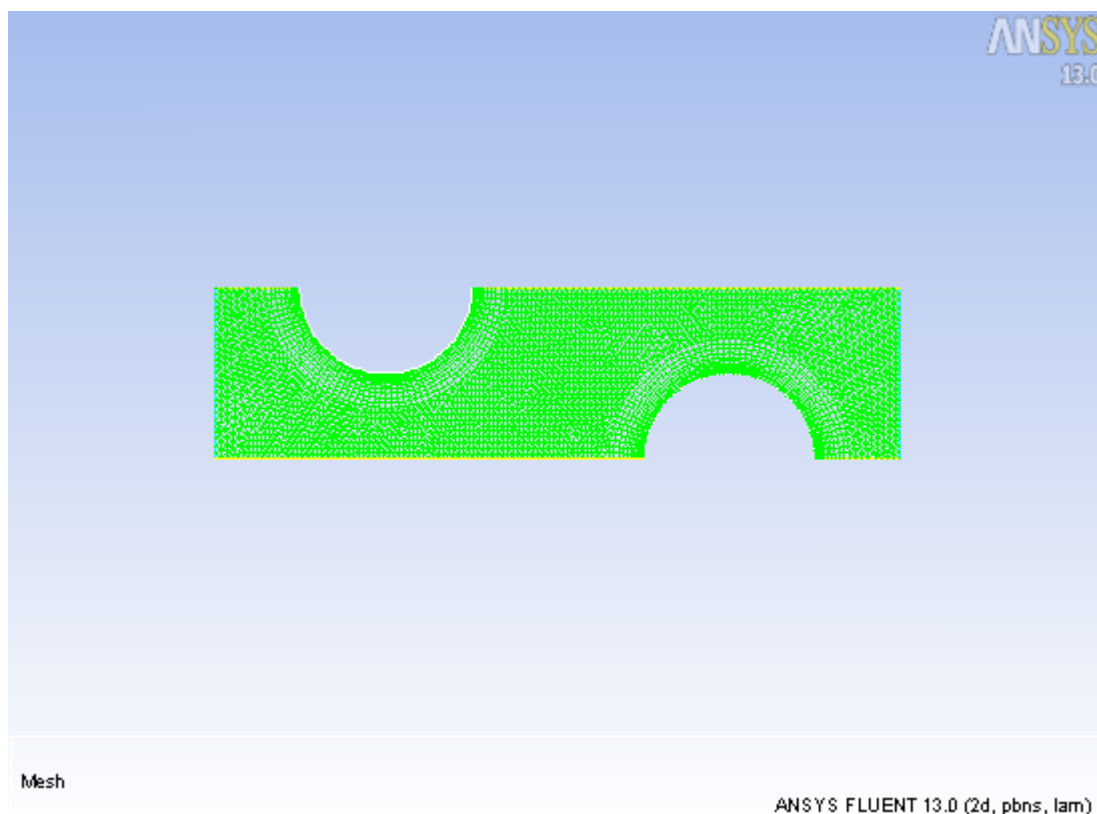
↔ **General** → **Check**

Note

It is a good idea to check the mesh after you manipulate it (i.e., 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.

5. Examine the mesh ([Figure 4.2 \(p. 180\)](#)).

Figure 4.2 Mesh for the Periodic Tube Bank



Quadrilateral cells are used in the regions surrounding the tube walls and triangular cells are used for the rest of the domain, resulting in a hybrid mesh (see [Figure 4.2 \(p. 180\)](#)). The quadrilateral cells provide better resolution of the viscous gradients near the tube walls. The remainder of the computational domain is filled with triangular cells for the sake of convenience.

Extra

You can use the right mouse button to probe for mesh information in the graphics window. If you click the right mouse button on any node in the mesh, information will be displayed in the ANSYS 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.

6. Create the periodic zone.

*The inlet (**wall-9**) and outflow (**wall-12**) boundaries currently defined as wall zones need to be redefined as periodic using the text user interface. The **wall-9** boundary will be redefined as a translationally periodic zone and **wall-12** as a periodic shadow of **wall-9**.*

- a. Press < Enter > in the console to get the command prompt (>).
- b. Enter the text command and input the responses outlined in boxes as shown:

```
mesh/modify-zones/make-periodic
```

```
Periodic zone [()] 9
Shadow zone [()] 12
Rotational periodic? (if no, translational) [yes] no
Create periodic zones? [yes] yes
Auto detect translation vector? [yes] yes
```

```
computed translation deltas: 0.040000 0.000000
all 26 faces matched for zones 9 and 12.
```

```
zone 12 deleted
```

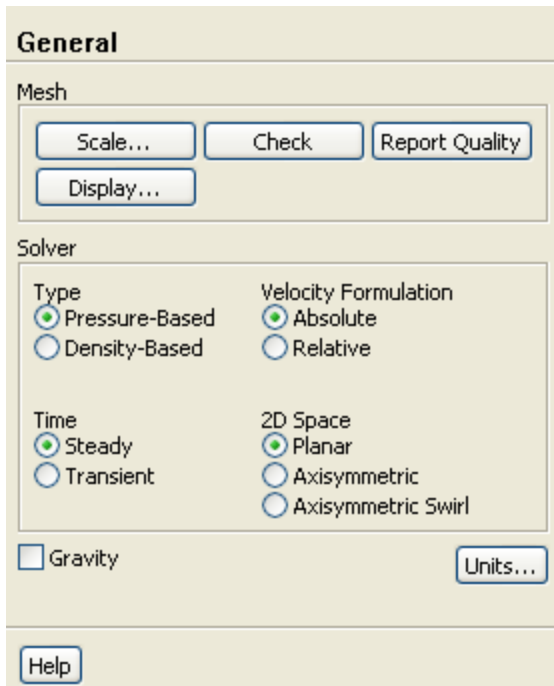
```
created periodic zones.
```

4.4.3. Step 2: General Settings

General

1. Retain the default settings for the solver.

General

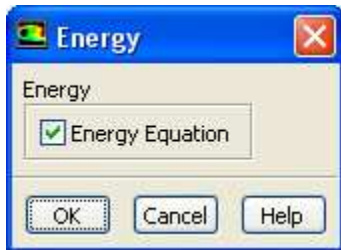


4.4.4. Step 3: Models

Models

1. Enable heat transfer.

Models → **Energy** → **Edit...**



- a. Enable **Energy Equation**.
- b. Click **OK** to close the **Energy** dialog box.

4.4.5. Step 4: Materials

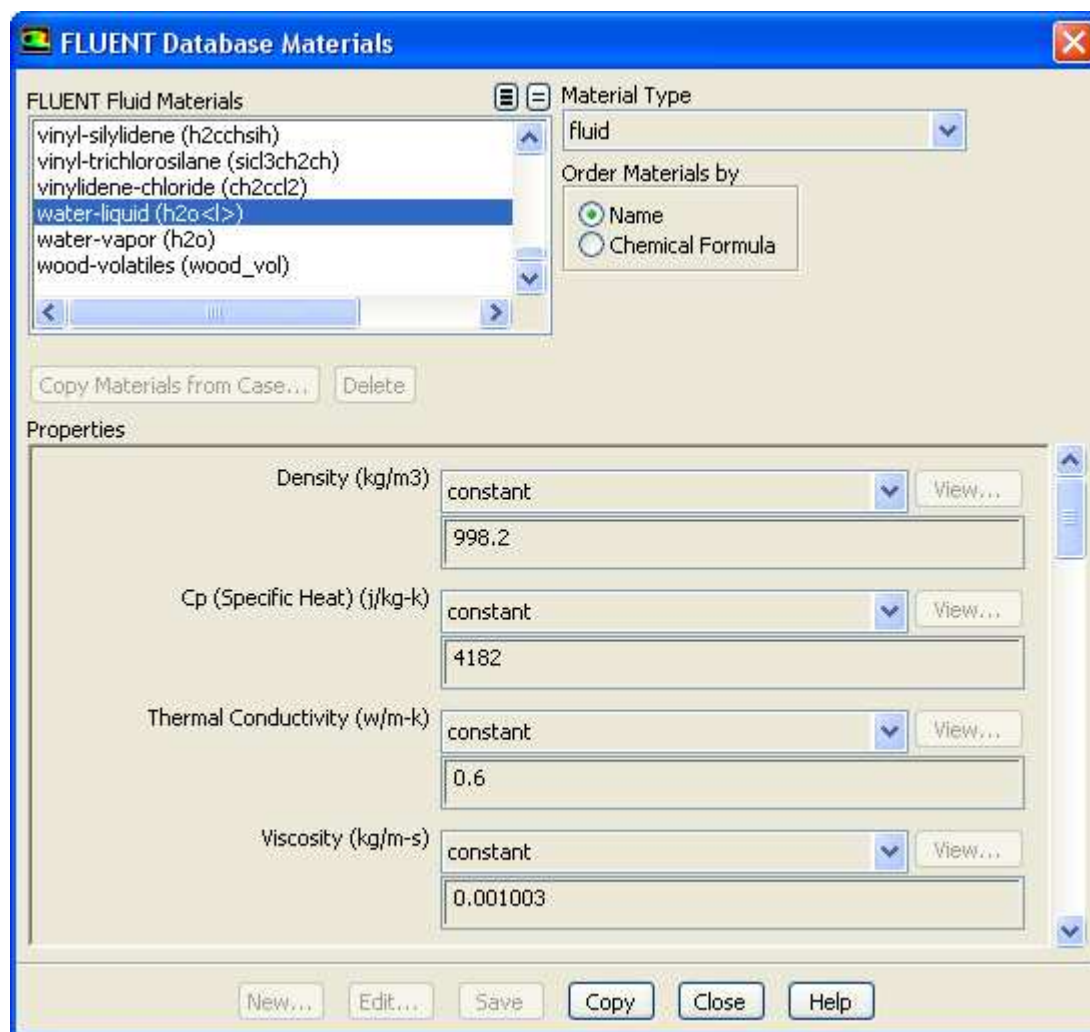
Materials

The default properties for water defined in ANSYS FLUENT are suitable for this problem. In this step, you will make sure that this material is available for selecting in future steps.

1. Add water to the list of fluid materials by copying it from the ANSYS FLUENT materials database.

Materials → **Fluid** → **Create/Edit...**

- a. Click **FLUENT Database...** in the **Create/Edit Materials** dialog box to open the **FLUENT Database Materials** dialog box.

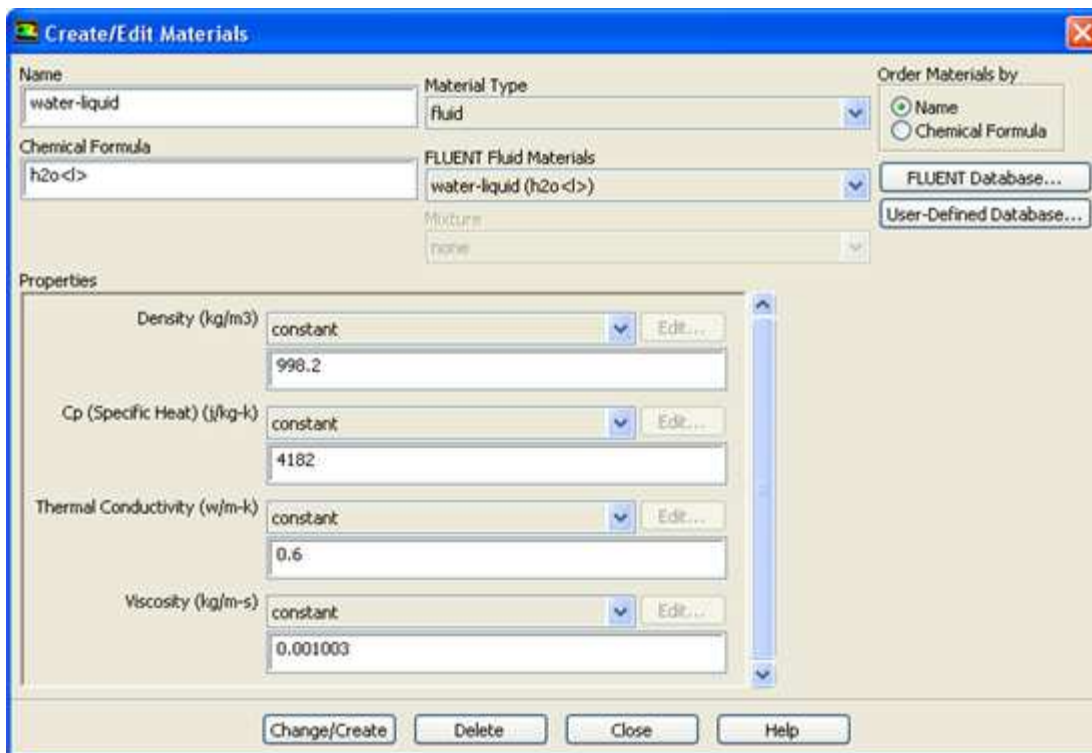


- i. Select **water-liquid (h2o l)** in the **FLUENT Fluid Materials** selection list.

*Scroll down the list to find **water-liquid (h2o l)**. Selecting this item will display the default properties in the dialog box.*

- ii. Click **Copy** and close the **FLUENT Database Materials** dialog box.

*The **Create/Edit Materials** dialog box will now display the copied properties for **water-liquid**.*



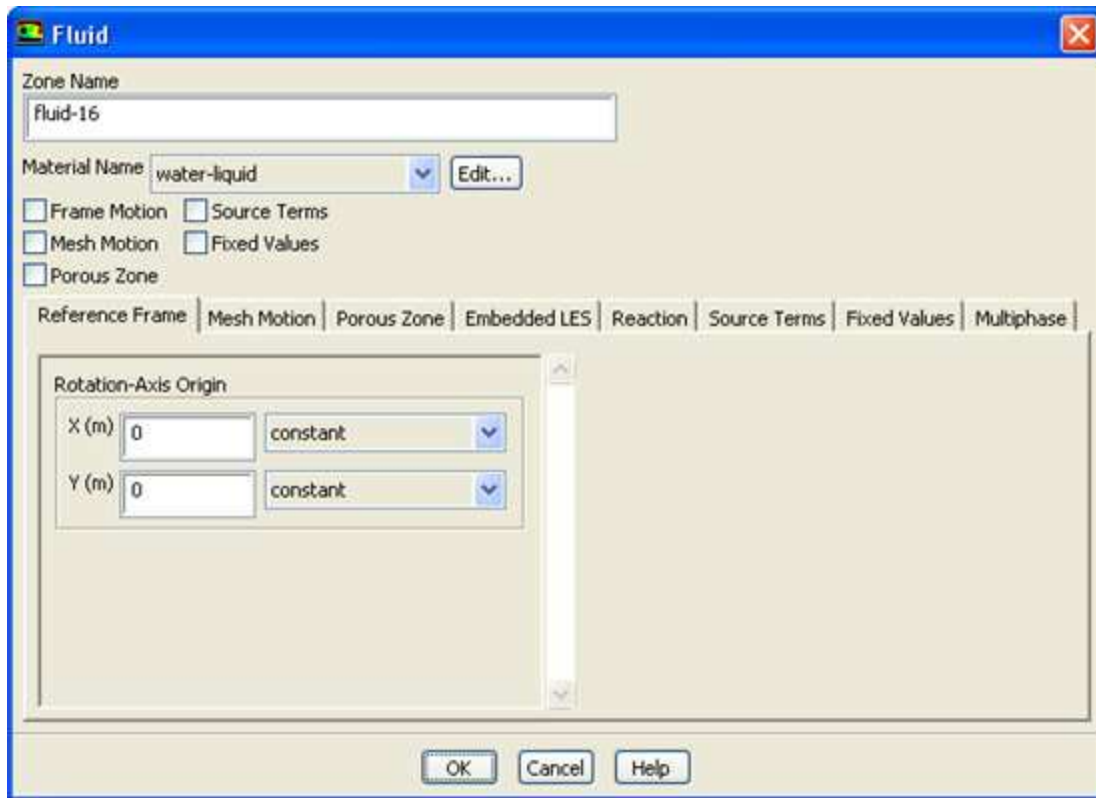
- b. Click **Change/Create** and close the **Create/Edit Materials** dialog box.

4.4.6. Step 5: Cell Zone Conditions

Cell Zone Conditions

- Set the cell zone conditions for the continuum fluid zone (**fluid-16**).

Cell Zone Conditions → fluid-16 → Edit...



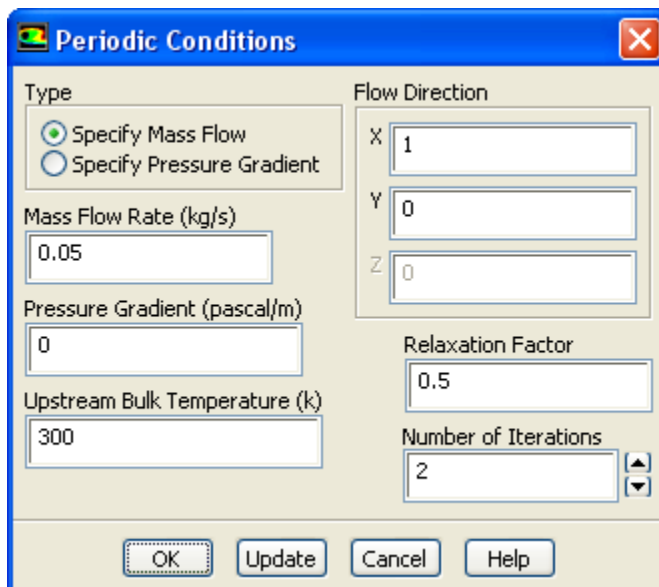
- Select **water-liquid** from the **Material Name** drop-down list.
- Click **OK** to close the **Fluid** dialog box.

4.4.7. Step 6: Periodic Conditions

◆ Boundary Conditions

- Define the periodic flow conditions.

◆ **Boundary Conditions** → **periodic-9** → **Periodic Conditions...**



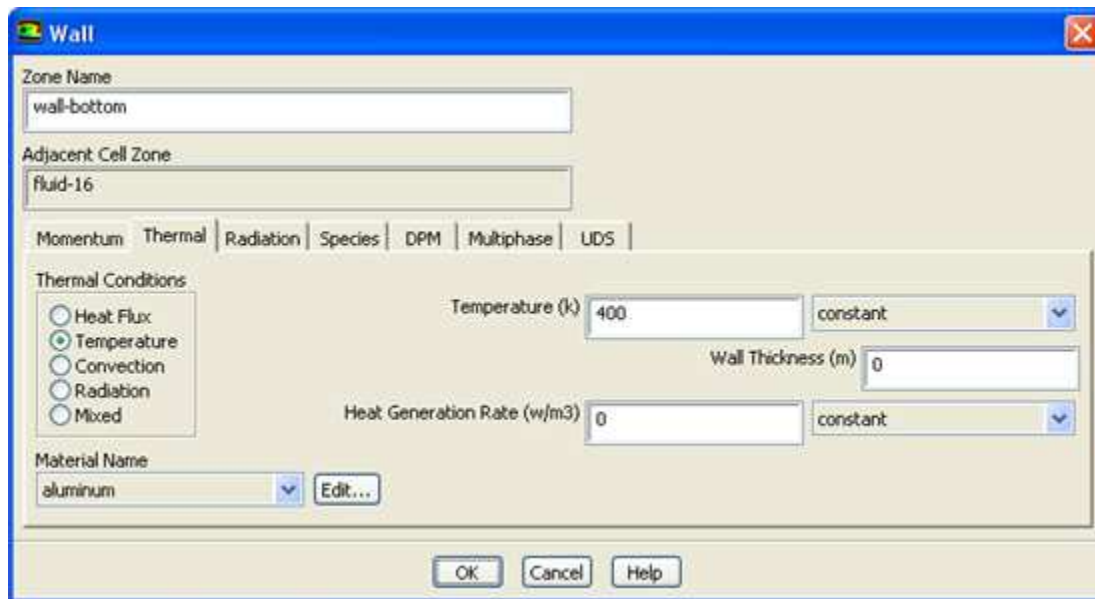
- a. Select **Specify Mass Flow** in the **Type** list.
*This will allow you to specify the **Mass Flow Rate**.*
- b. Enter 0.05 kg/s for **Mass Flow Rate**.
- c. Click **OK** to close the **Periodic Conditions** dialog box.

4.4.8. Step 7: Boundary Conditions

❖ Boundary Conditions

1. Set the boundary conditions for the bottom wall of the left tube (**wall-21**).

❖ **Boundary Conditions** →  **wall-21** → **Edit...**



- a. Enter wall-bottom for **Zone Name**.
- b. Click the **Thermal** tab.
 - i. Select **Temperature** in the **Thermal Conditions** list.
 - ii. Enter 400 K for **Temperature**.

These settings will specify a constant wall temperature of 400 K.

- c. Click **OK** to close the **Wall** dialog box.
2. Set the boundary conditions for the top wall of the right tube (**wall-3**).

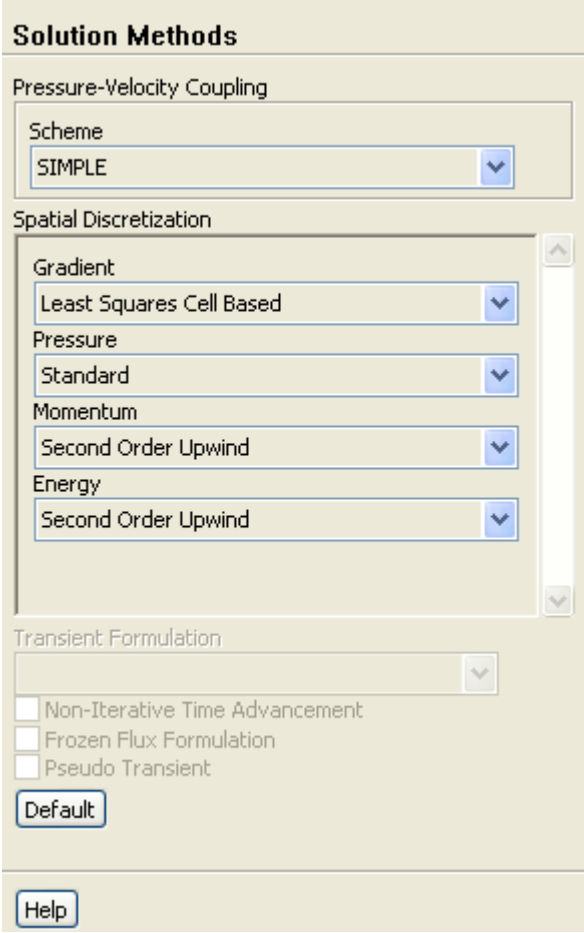
❖ **Boundary Conditions** →  **wall-3** → **Edit...**

- a. Enter wall-top for **Zone Name**.
- b. Click the **Thermal** tab.
 - i. Select **Temperature** from the **Thermal Conditions** list.
 - ii. Enter 400 K for **Temperature**.
- c. Click **OK** to close the **Wall** dialog box.

4.4.9. Step 8: Solution

1. Set the solution parameters.

Solution Methods



Solution Methods

Pressure-Velocity Coupling

Scheme
SIMPLE

Spatial Discretization

Gradient
Least Squares Cell Based

Pressure
Standard

Momentum
Second Order Upwind

Energy
Second Order Upwind

Transient Formulation

☐ Non-Iterative Time Advancement

☐ Frozen Flux Formulation

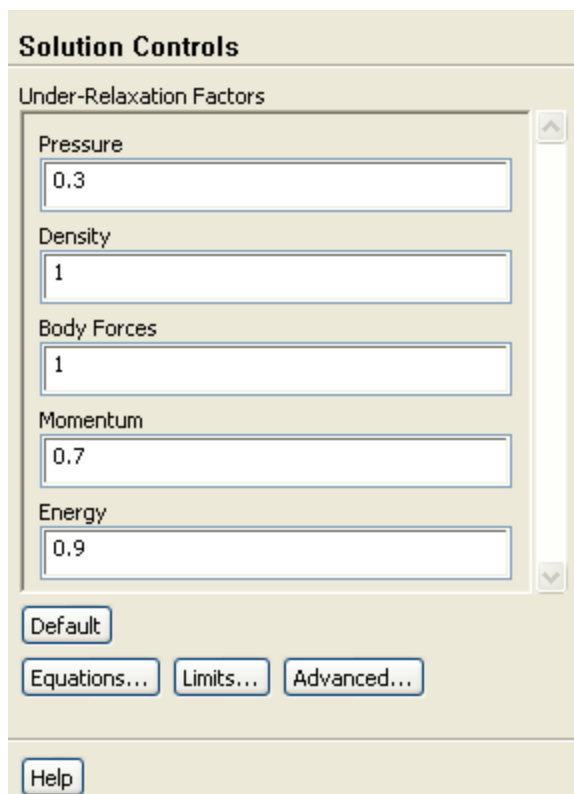
☐ Pseudo Transient

Default

Help

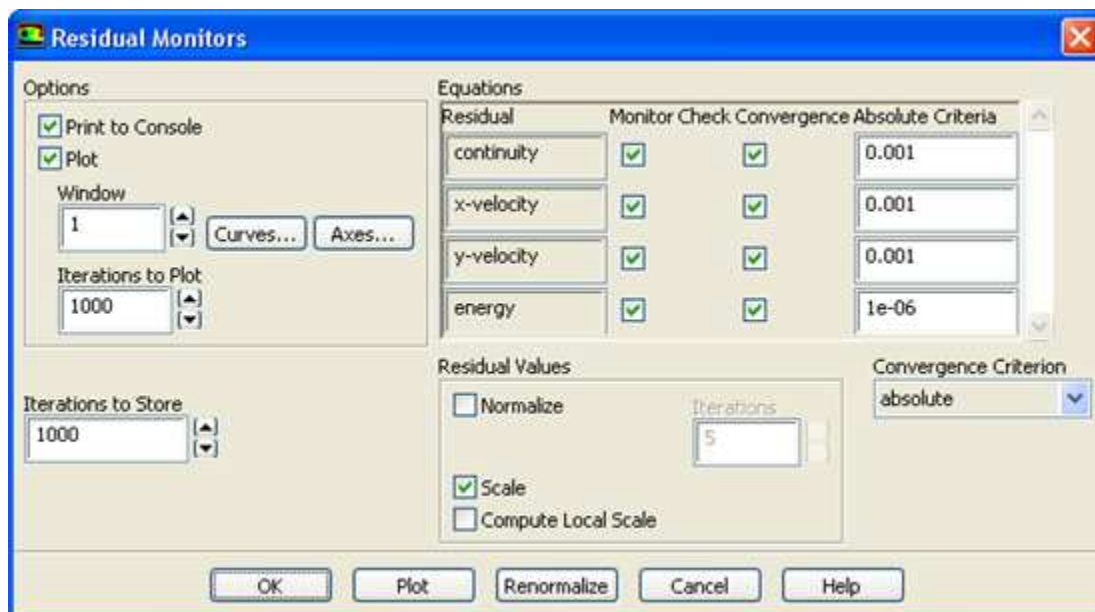
- a. Retain the default setting of **Least Squares Cell Based** for the **Gradient** in the **Spatial Discretization** group box.
 - b. Select **Second Order Upwind** from the **Momentum** and **Energy** drop-down lists.
2. Set the solution controls.

Solution Controls



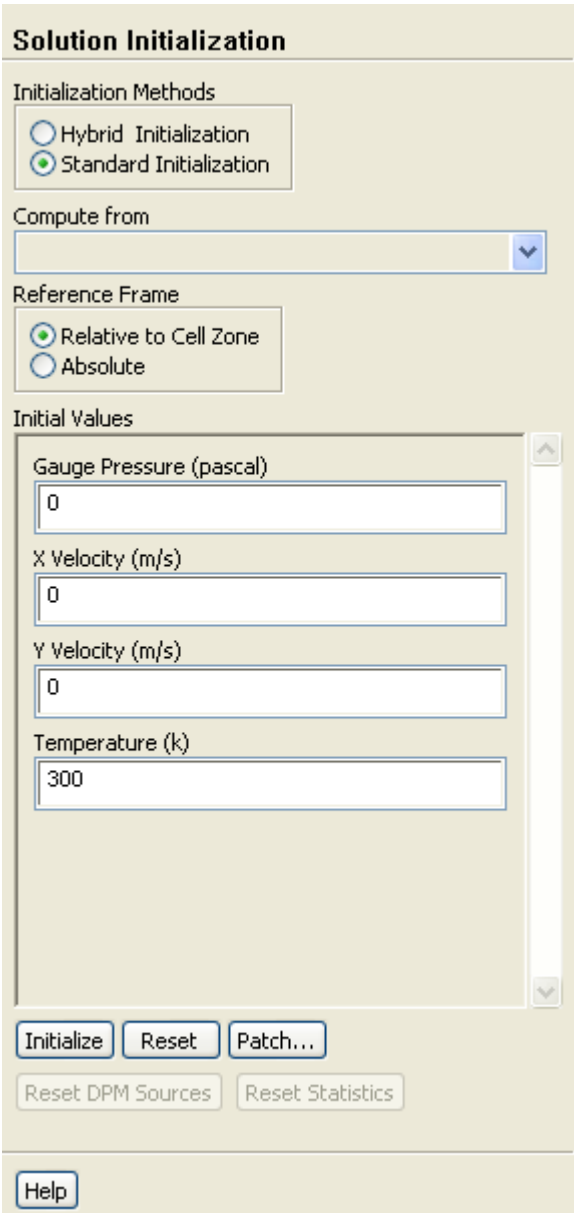
- a. Enter 0.9 for **Energy** in the **Under-Relaxation Factors** group box.
3. Enable the plotting of residuals during the calculation.

❖ **Monitors** → **Residuals** → **Edit...**



- a. Make sure **Plot** is enabled in the **Options** group box.
- b. Click **OK** to close the **Residual Monitors** dialog box.
4. Initialize the solution.

Solution Initialization



Solution Initialization

Initialization Methods

☐ Hybrid Initialization
☒ Standard Initialization

Compute from

Reference Frame

☒ Relative to Cell Zone
☐ Absolute

Initial Values

Gauge Pressure (pascal)

0

X Velocity (m/s)

0

Y Velocity (m/s)

0

Temperature (K)

300

Initialize Reset Patch...

Reset DPM Sources Reset Statistics

Help

- a. Retain the default setting of 300 K for **Temperature** in the **Initial Values** group box.
- b. Click **Initialize**.

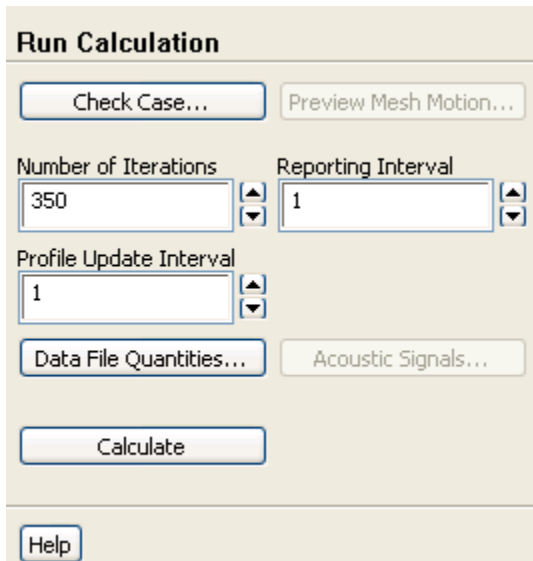
The values shown in the task pane will be used as the initial condition for the solution.

5. Save the case file (tubebank.cas.gz).

File → **Write** → **Case...**

6. Start the calculation by requesting 350 iterations.

Run Calculation



- a. Enter 350 for **Number of Iterations**.
- b. Click **Calculate**.

The energy residual curve that is displayed in the graphics window will begin to flatten out as it approaches 350 iterations. For the solution to converge to the recommended residual value of 10^{-6} , you need to reduce the under-relaxation factor for energy.

7. Change the **Under-Relaxation Factor** for **Energy** to 0.6.

◆ Solution Controls

8. Continue the calculation by requesting another 300 iterations.

◆ Run Calculation

After restarting the calculation, the plot of the energy residual will display an initial dip as a result of the reduction of the under-relaxation factor. The solution will converge in a total of approximately 610 iterations.

9. Save the case and data files (tubebank.cas.gz and tubebank.dat.gz).

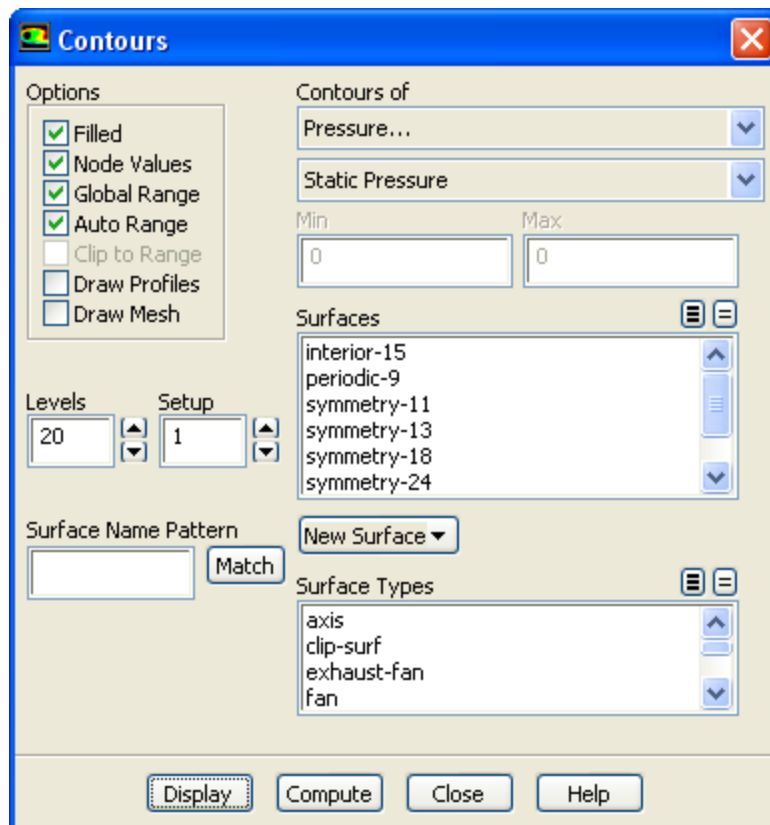
File → Write → Case & Data...

4.4.10. Step 9: Postprocessing

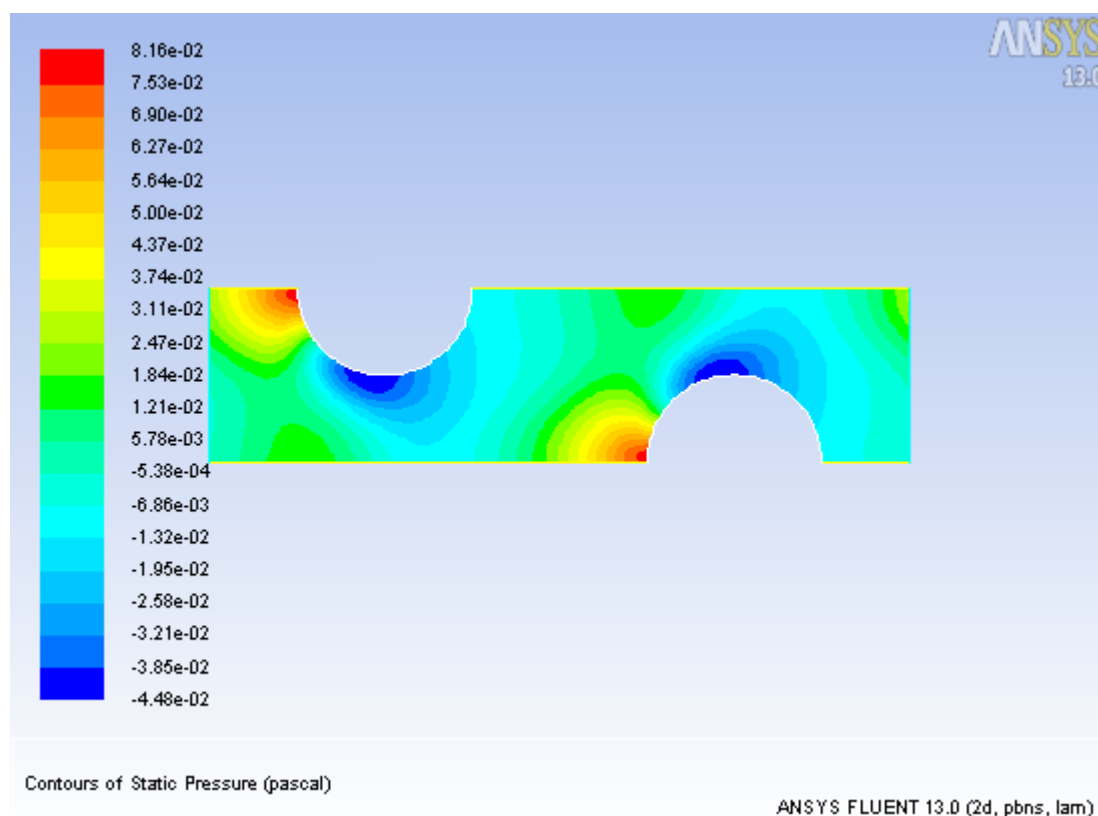
◆ Graphics and Animations

1. Display filled contours of static pressure (*Figure 4.3* (p. 192)).

◆ Graphics and Animations →  **Contours** → **Set Up...**

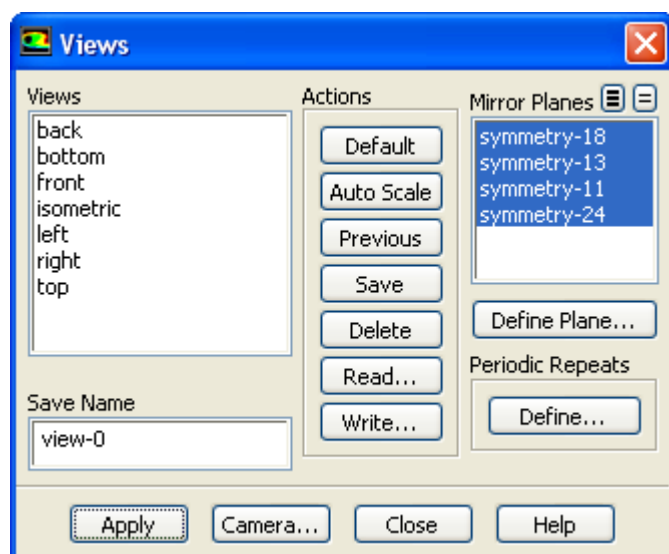


- a. Enable **Filled** in the **Options** group box.
- b. Retain the default selection of **Pressure...** and **Static Pressure** from the **Contours of** drop-down lists.
- c. Click **Display**.

Figure 4.3 Contours of Static Pressure

- d. Change the view to mirror the display across the symmetry planes (*Figure 4.4* (p. 193)).

◆ **Graphics and Animations** → **Views...**



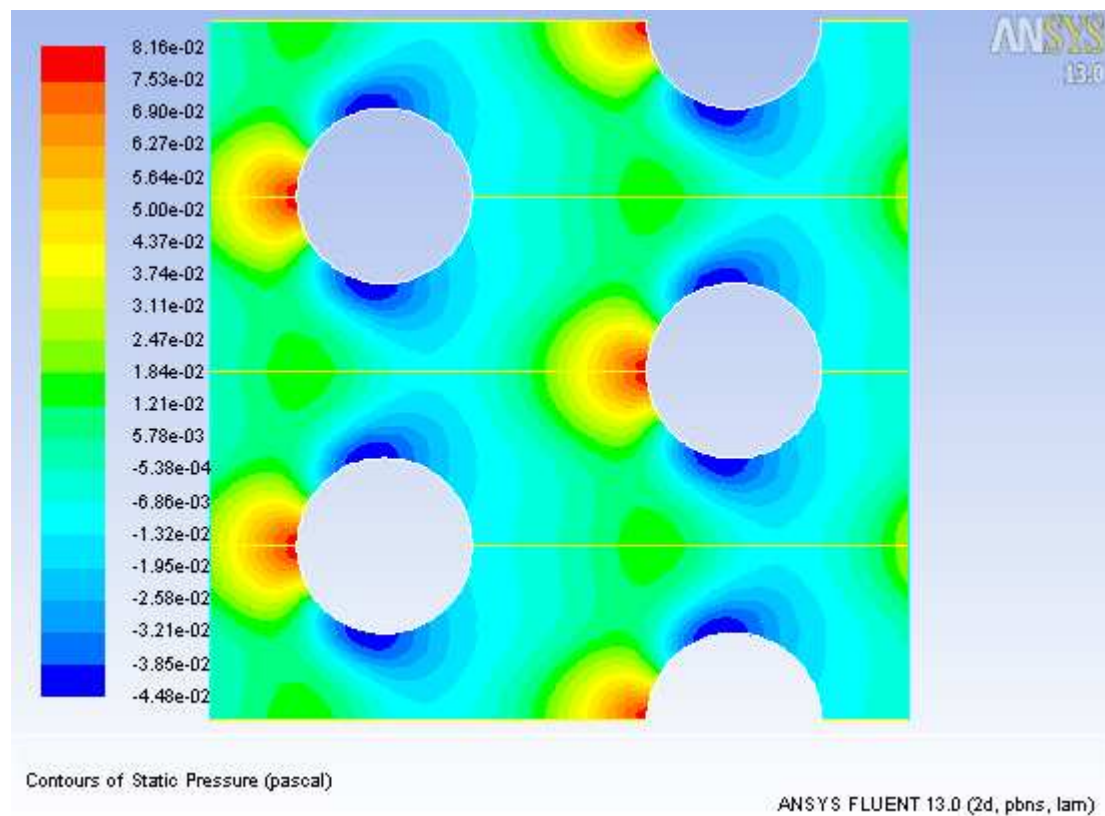
- i. Select all of the symmetry zones (**symmetry-18**, **symmetry-13**, **symmetry-11**, and **symmetry-24**) in the **Mirror Planes** selection list by clicking on the shaded icon in the upper right corner.

Note

There are four symmetry zones in the **Mirror Planes** selection list because the top and bottom symmetry planes in the domain are each comprised of two symmetry zones, one on each side of the tube centered on the plane. It is also possible to generate the same display shown in [Figure 4.4 \(p. 193\)](#) by selecting just one of the symmetry zones on the top symmetry plane, and one on the bottom.

- ii. Click **Apply** and close the **Views** dialog box.
- iii. Translate the display of symmetry contours so that it is centered in the graphics window by using the left mouse button ([Figure 4.4 \(p. 193\)](#)).

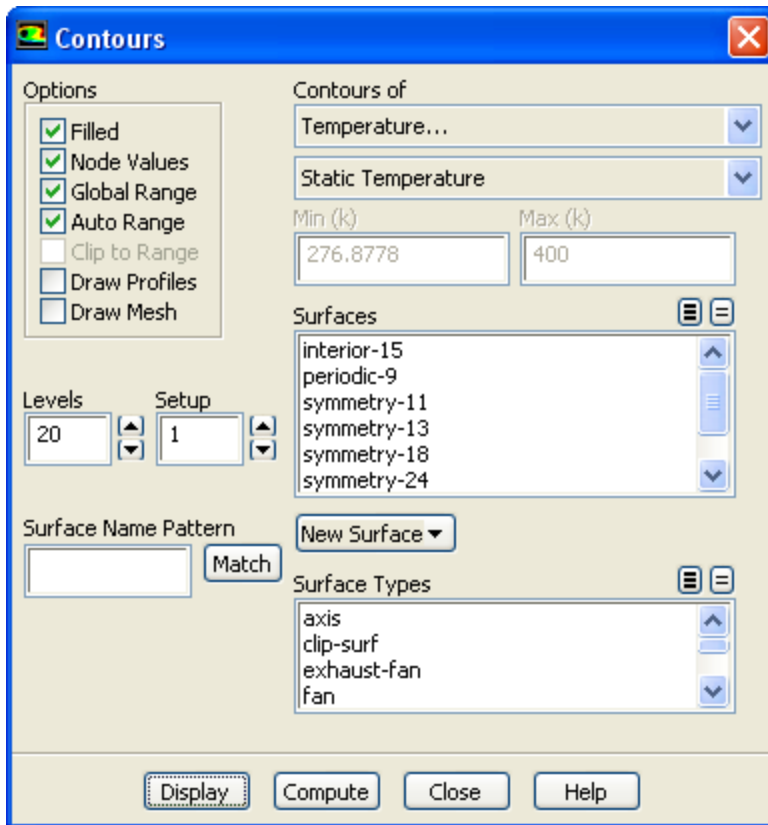
Figure 4.4 Contours of Static Pressure with Symmetry



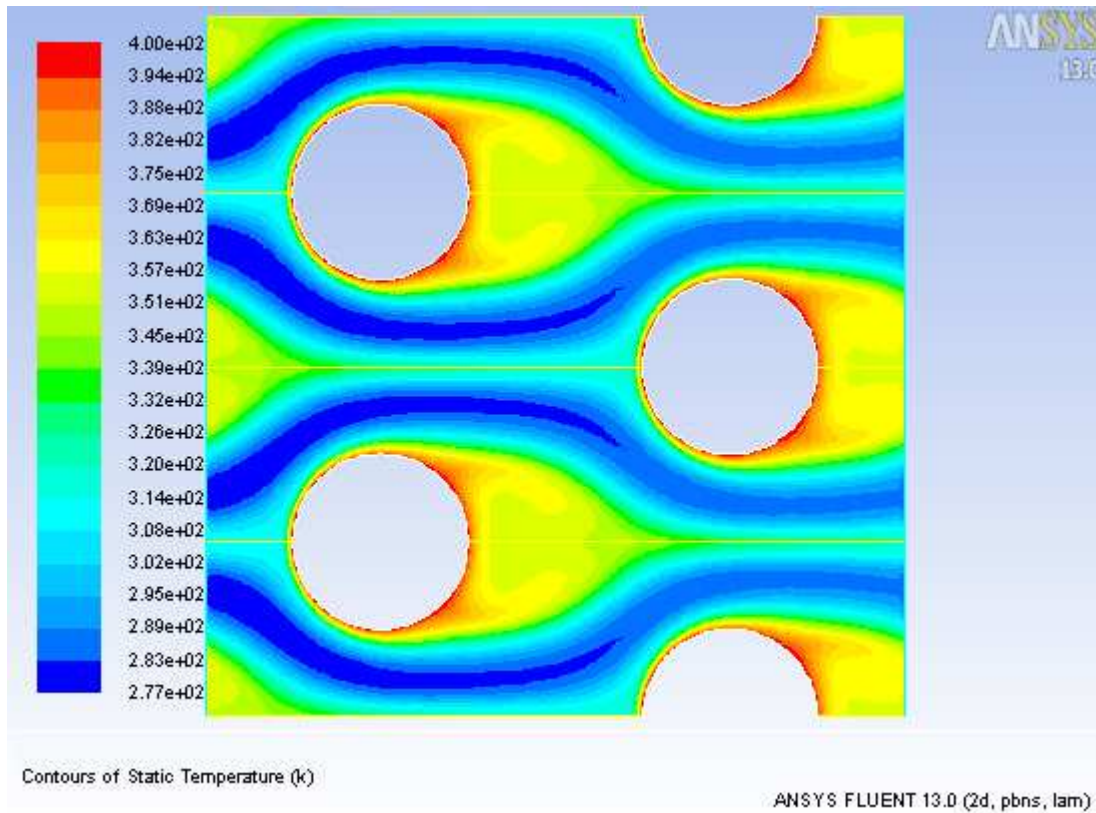
The pressure contours displayed in [Figure 4.4 \(p. 193\)](#) do not include the linear pressure gradient computed by the solver. Thus, the contours are periodic at the inlet and outflow boundaries.

2. Display filled contours of static temperature ([Figure 4.5 \(p. 195\)](#)).

Graphics and Animations → **Contours** → **Set Up...**



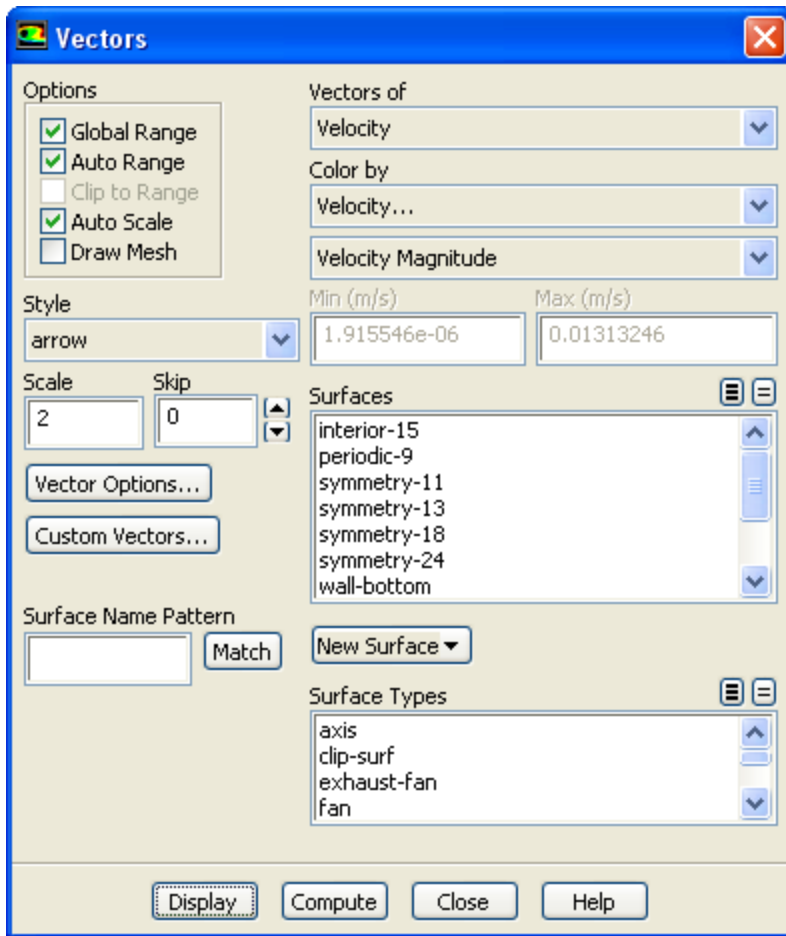
- Select **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.
- Click **Display** and close the **Contours** dialog box.

Figure 4.5 Contours of Static Temperature

The contours in [Figure 4.5 \(p. 195\)](#) reveal the temperature increase in the fluid due to heat transfer from the tubes. The hotter fluid is confined to the near-wall and wake regions, while a narrow stream of cooler fluid is convected through the tube bank.

3. Display the velocity vectors ([Figure 4.6 \(p. 197\)](#)).

Graphics and Animations → **Vectors** → **Set Up...**

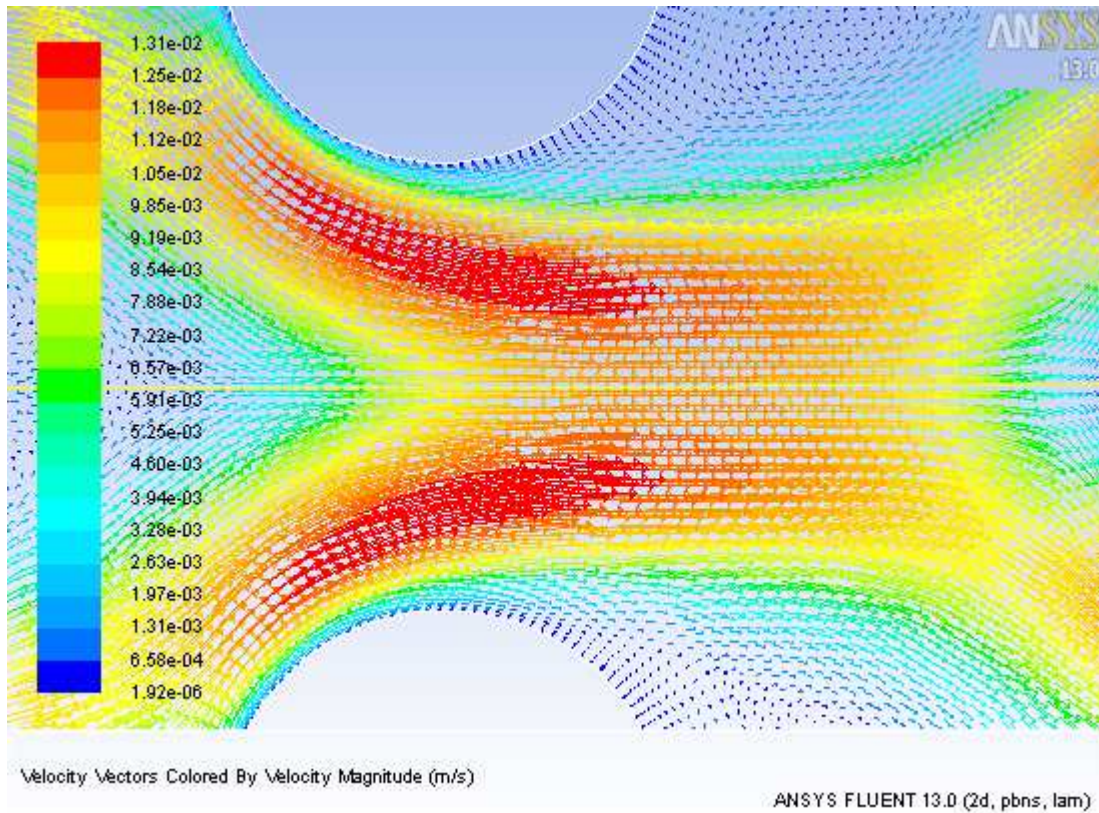


- a. Enter 2 for **Scale**.

This will increase the size of the displayed vectors, making it easier to view the flow patterns.

- b. Retain the default selection of **Velocity** from the **Vectors of** drop-down list.
- c. Retain the default selection of **Velocity...** and **Velocity Magnitude** from the **Color by** drop-down lists.
- d. Click **Display** and close the **Vectors** dialog box.
- e. Zoom in on the upper right portion of one of the left tubes to get the display shown in (Figure 4.6 (p. 197)), by using the middle mouse button in the graphics window.

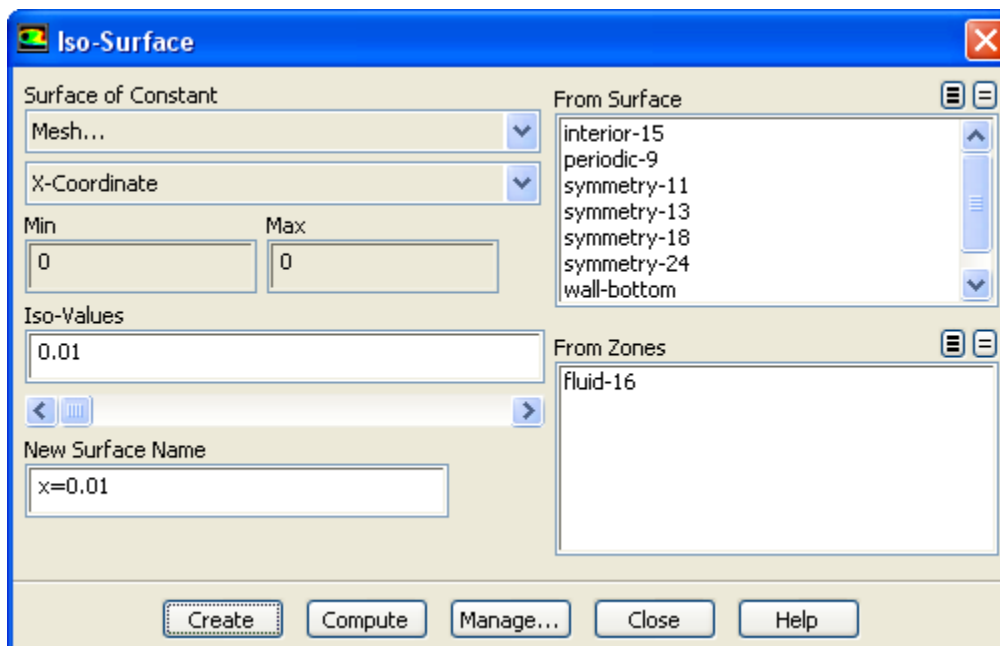
The magnified view of the velocity vector plot in Figure 4.6 (p. 197) clearly shows the recirculating flow behind the tube and the boundary layer development along the tube surface.

Figure 4.6 Velocity Vectors

4. Create an isosurface on the periodic tube bank at $x = 0.01$ m (through the first column of tubes).

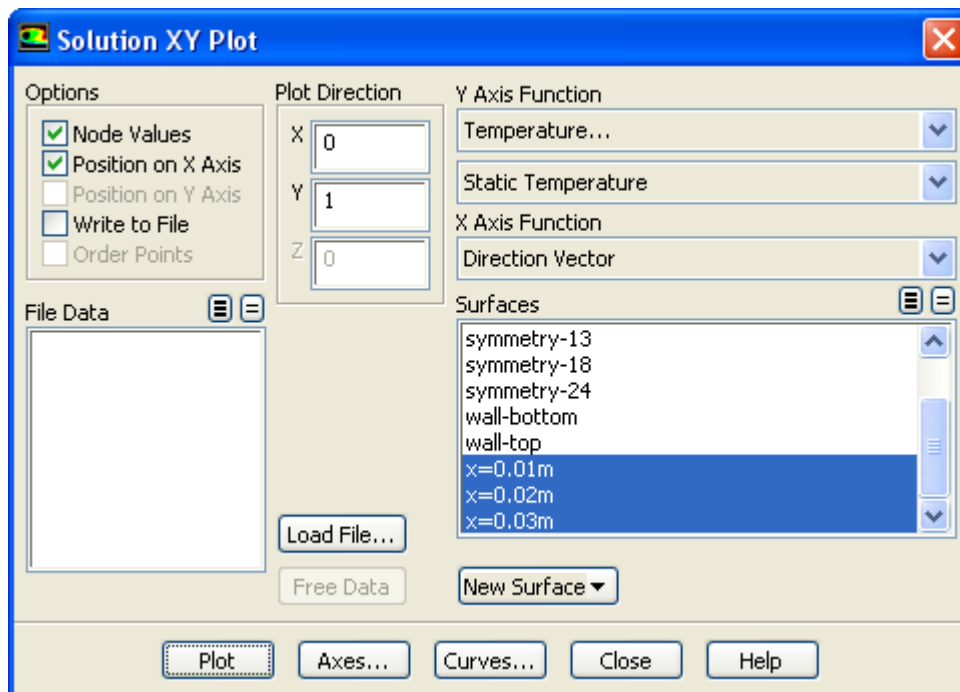
This isosurface and the ones created in the steps that follow will be used for the plotting of temperature profiles.

Surface → **Iso-Surface...**



- a. Select **Mesh...** and **X-Coordinate** from the **Surface of Constant** drop-down lists.
 - b. Enter 0.01 for **Iso-Values**.
 - c. Enter $x=0.01\text{m}$ for **New Surface Name**.
 - d. Click **Create**.
5. In a similar manner, create an isosurface on the periodic tube bank at $x = 0.02\text{ m}$ (halfway between the two columns of tubes) named $x=0.02\text{m}$.
 6. In a similar manner, create an isosurface on the periodic tube bank at $x = 0.03\text{ m}$ (through the middle of the second column of tubes) named $x=0.03\text{m}$, and close the **Iso-Surface** dialog box.
 7. Create an XY plot of static temperature on the three isosurfaces (*Figure 4.7* (p. 200)).

 **Plots** →  **XY Plot** → **Set Up...**



- a. Enter 0 for **X** and 1 for **Y** in the **Plot Direction** group box.

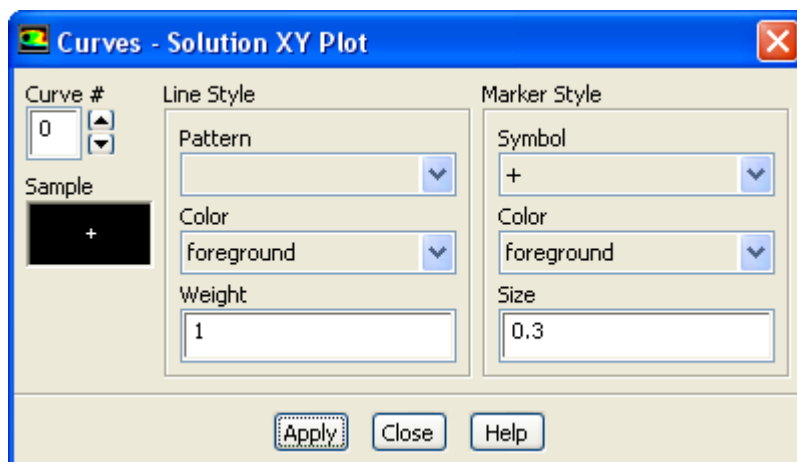
*With a **Plot Direction** vector of $(0, 1)$, ANSYS FLUENT will plot the selected variable as a function of y . Since you are plotting the temperature profile on cross sections of constant x , the temperature varies with the y direction.*

- b. Select **Temperature...** and **Static Temperature** from the **Y-Axis Function** drop-down lists.
- c. Select **$x=0.01\text{m}$** , **$x=0.02\text{m}$** , and **$x=0.03\text{m}$** in the **Surfaces** selection list.

*Scroll down to find the **$x=0.01\text{m}$** , **$x=0.02\text{m}$** , and **$x=0.03\text{m}$** surfaces.*

- d. Click the **Curves...** button to open the **Curves - Solution XY Plot** dialog box.

This dialog box is used to define plot styles for the different plot curves.



- i. Select **+** from the **Symbol** drop-down list.

*Scroll up to find the **+** item.*

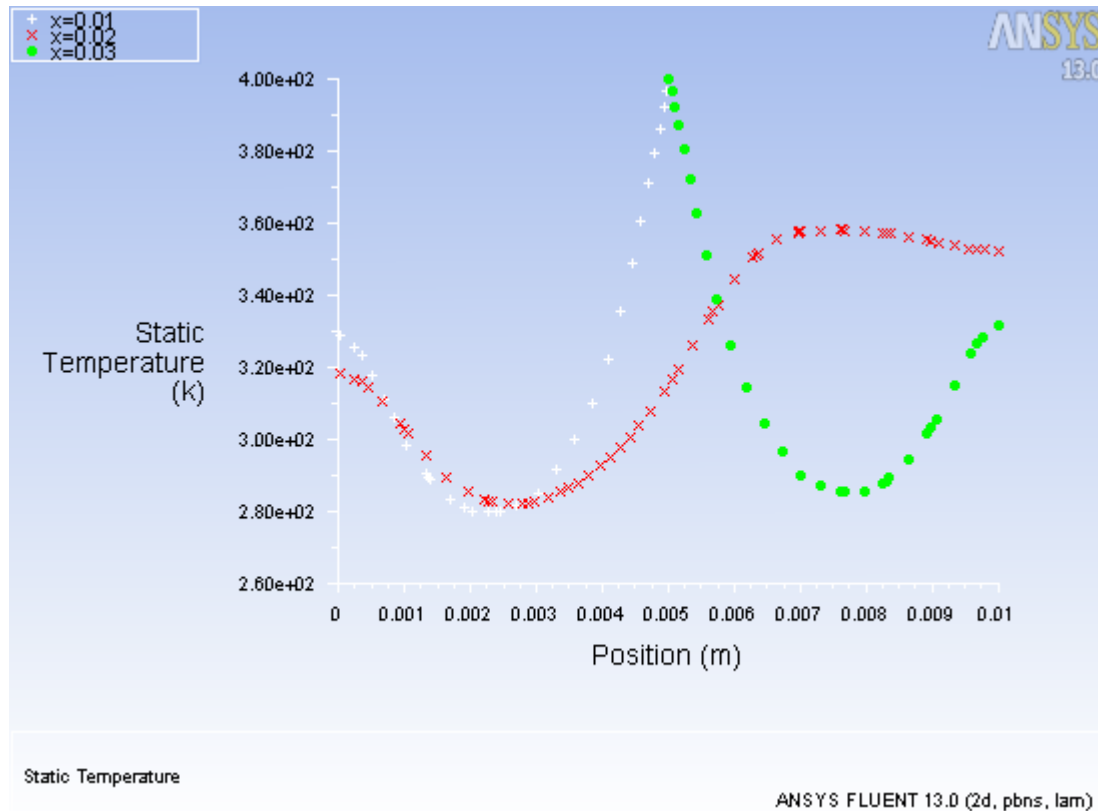
- ii. Click **Apply** to assign the **+** symbol to the $x = 0.01$ m curve.
- iii. Set the **Curve #** to 1 to define the style for the $x = 0.02$ m curve.
- iv. Select **x** from the **Symbol** drop-down list.

*Scroll up to find the **x** item.*

- v. Enter 0.5 for **Size**.
- vi. Click **Apply** and close the **Curves - Solution XY Plot** dialog box.

Since you did not change the curve style for the $x = 0.03$ m curve, the default symbol will be used.

- e. Click **Plot** and close the **Solution XY Plot** dialog box.

Figure 4.7 Static Temperature at $x=0.01$, 0.02 , and 0.03 m

4.5. Summary

In this tutorial, periodic flow and heat transfer in a staggered tube bank were modeled in ANSYS FLUENT. The model was set up assuming a known mass flow through the tube bank and constant wall temperatures. Due to the periodic nature of the flow and symmetry of the geometry, only a small piece of the full geometry was modeled. In addition, the tube bank configuration lent itself to the use of a hybrid mesh with quadrilateral cells around the tubes and triangles elsewhere.

The **Periodic Conditions** dialog box makes it easy to run this type of model with a variety of operating conditions. For example, different flow rates (and hence different Reynolds numbers) can be studied, or a different inlet bulk temperature can be imposed. The resulting solution can then be examined to extract the pressure drop per tube row and overall Nusselt number for a range of Reynolds numbers.

4.6. Further Improvements

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in *Introduction to Using ANSYS FLUENT: Fluid Flow and Heat Transfer in a Mixing Elbow* (p. 111).