Tutorial 2. Modeling Periodic Flow and Heat Transfer

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 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.

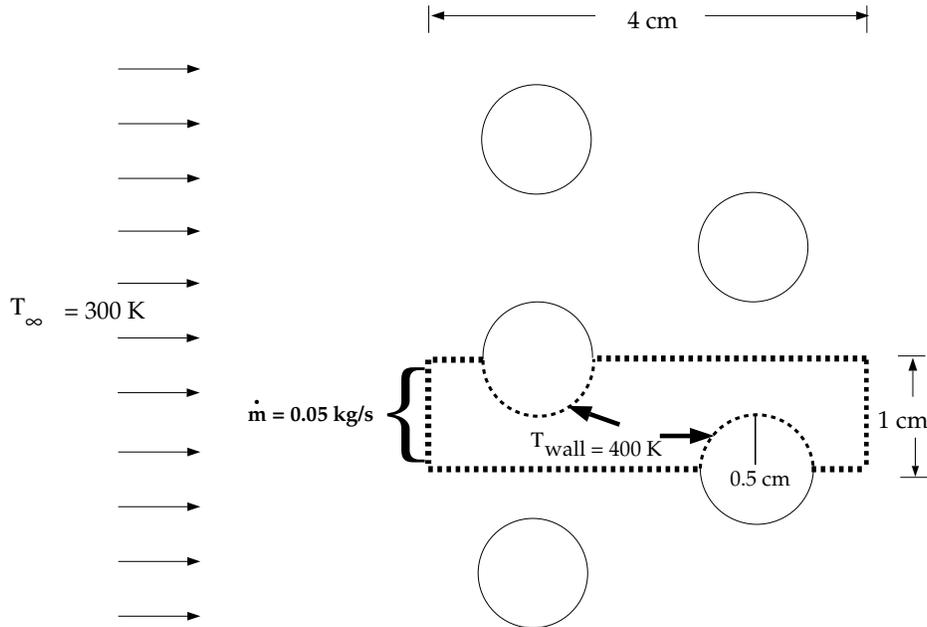This tutorial demonstrates how to do the following:

- 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.

Prerequisites

This tutorial assumes that you are familiar with the menu structure in FLUENT and that you have completed Tutorial 1. Some steps in the setup and solution procedure will not be shown explicitly.

Problem Description

This problem considers a 2D section of a tube bank. A schematic of the problem is shown in Figure 2.1. 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.

$$\rho = 998.2 \text{ kg/m}^3$$
$$\mu = 0.001003 \text{ kg/m-s}$$
$$c_p = 4182 \text{ J/kg-K}$$
$$k = 0.6 \text{ W/m-K}$$

Figure 2.1: 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 2.1. 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 2.1.

Setup and Solution

Preparation

1. Download `periodic_flow_heat.zip` from the Fluent Inc. User Services Center or copy it from the FLUENT documentation CD to your working folder (as described in Tutorial 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. Start the 2D (2d) version of FLUENT.

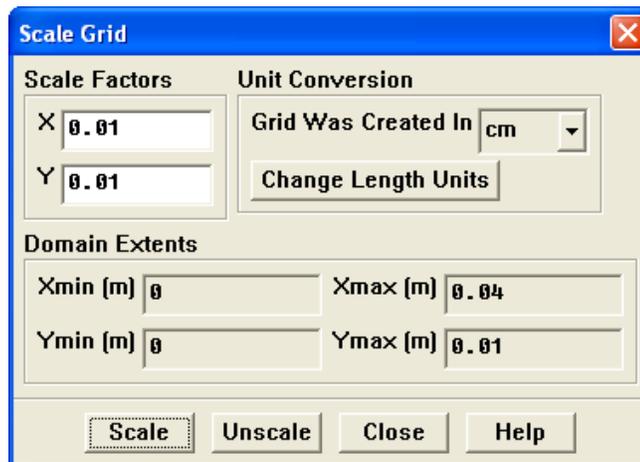### Step 1: Grid

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

   File ⟶ Read ⟶Case...

2. Check the grid.

   Grid ⟶Check

   FLUENT *will perform various checks on the mesh and report the progress in the console. Make sure that the minimum volume reported is a positive number.*
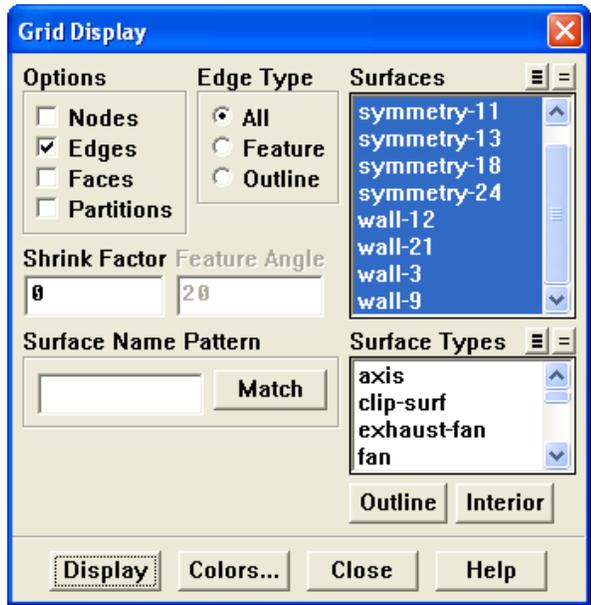
3. Scale the grid.

   Grid ⟶Scale...

   (a) Select cm (centimeters) from the Grid Was Created In drop-down list in the Unit Conversion group box.

   (b) Click Scale to scale the grid.

   (c) Close the Scale Grid panel.

4. Display the mesh (Figure 2.2).

Display ⟶Grid...

(a) Retain the default settings.

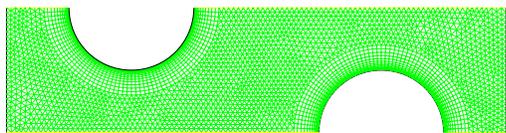(b) Click Display and close the Grid Display panel.

Figure 2.2: 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 2.2). 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 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.*

5. 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 responses outlined in boxes as shown:

```
> grid/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.
```
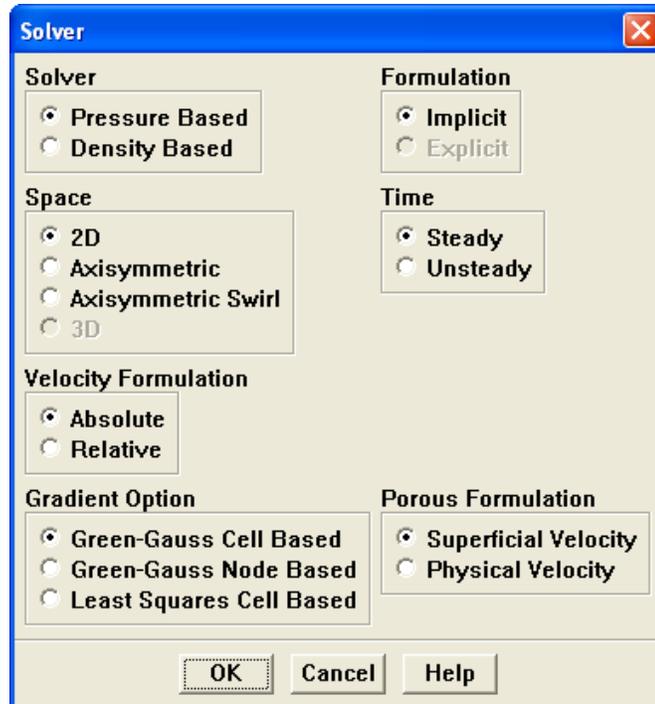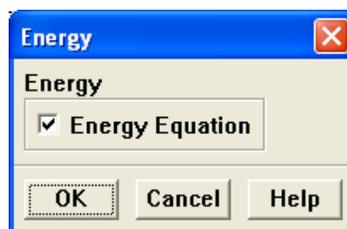
Step 2: Models

1. Retain the default settings for the solver.

   Define $\longrightarrow$ Models $\longrightarrow$Solver...

2. Activate heat transfer.

   Define $\longrightarrow$ Models $\longrightarrow$Energy...

   (a) Enable the Energy Equation option.

   (b) Click OK to close the Energy panel.

3. Define the periodic flow conditions.

$\boxed{\text{Define}} \longrightarrow \text{Periodic Conditions...}$

(a) Select Specify Mass Flow from the Type list.

This will allow you to specify the Mass Flow Rate.

(b) Enter 0.05 kg/s for Mass Flow Rate.
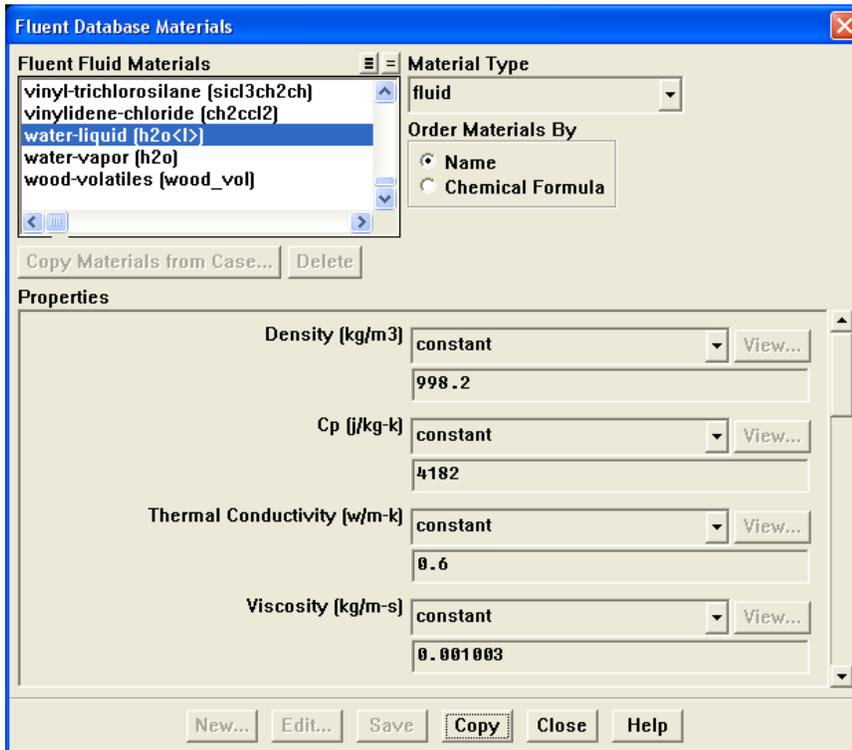
(c) Click OK to close the Periodic Conditions panel.

Step 3: Materials

The default properties for water defined in 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 FLUENT materials database.
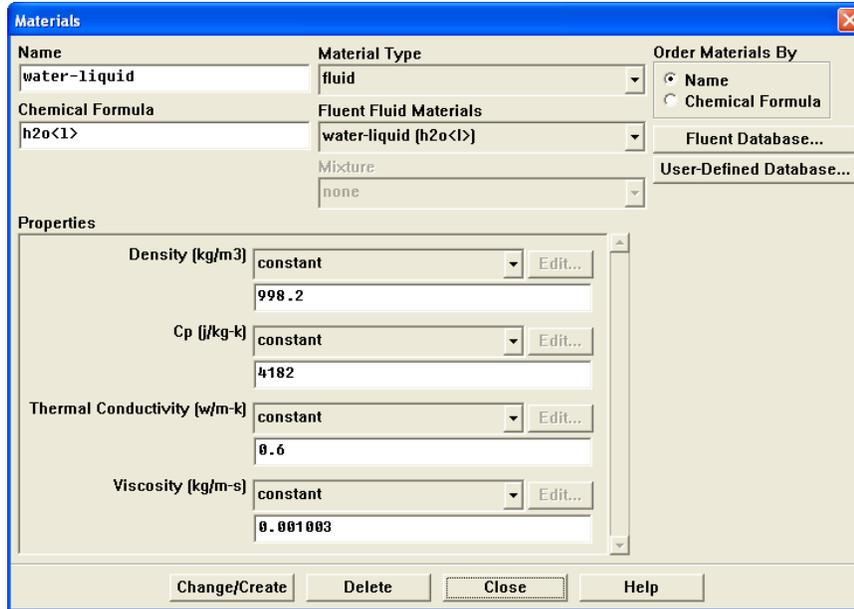
   Define ⟶Materials...

   (a) Click the Fluent Database... button to open the Fluent Database Materials panel.

   i. Select water-liquid (h2o<l>) from 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 panel.*

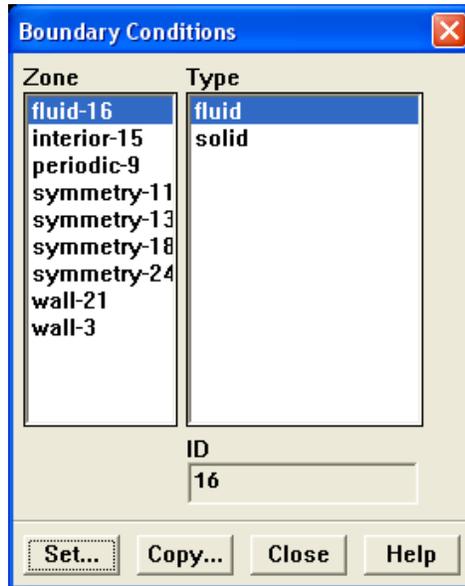   ii. Click Copy and close the Fluent Database Materials panel.

   The Materials *panel will now display the copied properties for* water-liquid.
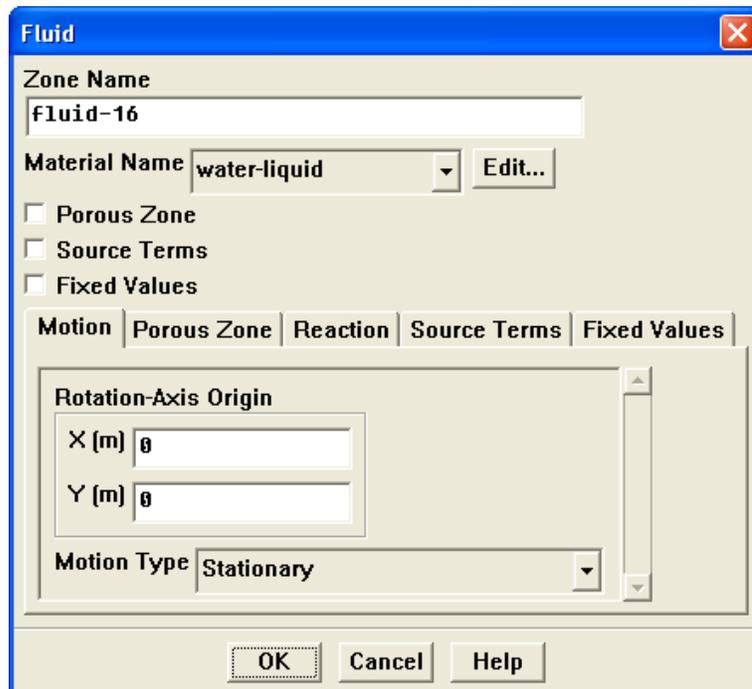
(b) Close the Materials panel.

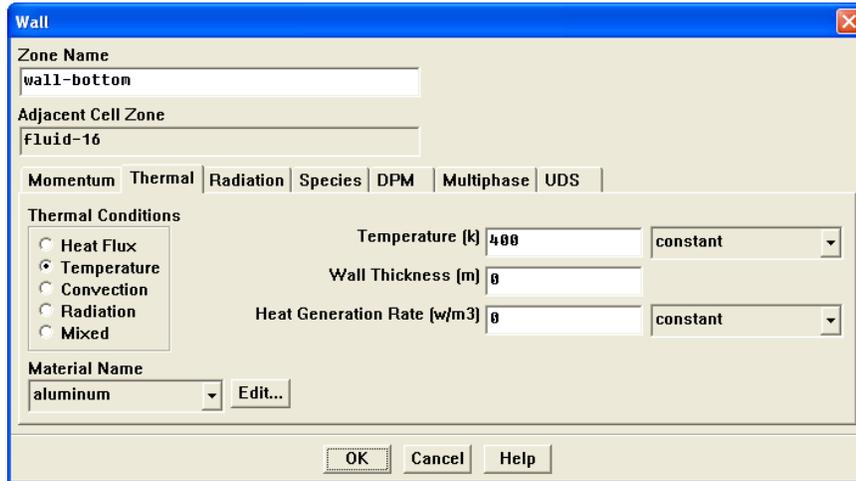## Step 4: Boundary Conditions

Define ⟶Boundary Conditions...

1. Set the boundary conditions for the continuum fluid zone (fluid-16).

(a) Select water-liquid from the Material Name drop-down list.

(b) Click OK to close the Fluid panel.

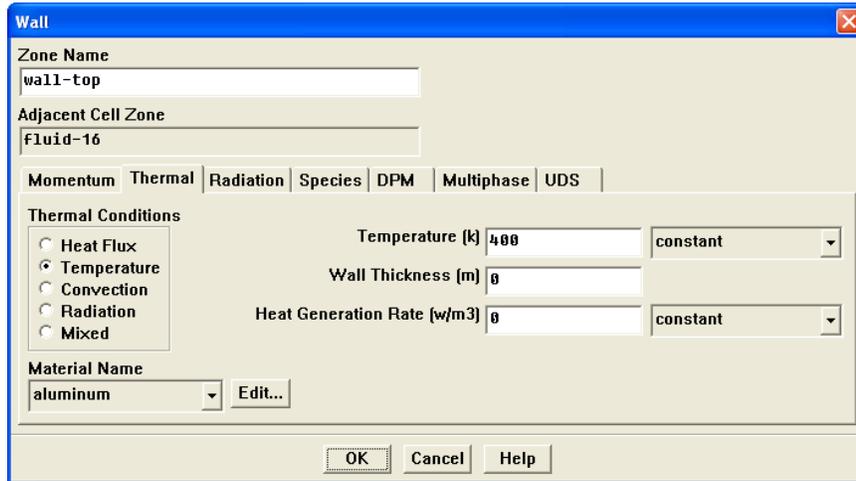2. Set the boundary conditions for the bottom wall of the left tube (wall-21).

(a) Enter `wall-bottom` for Zone Name.

(b) Click the Thermal tab.

    i. Select Temperature from 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 panel.

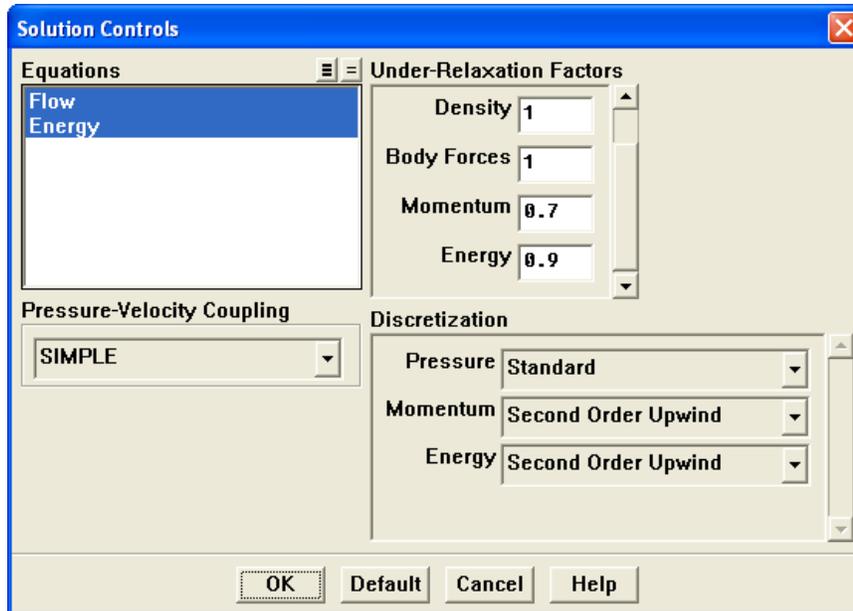3. Set the boundary conditions for the top wall of the right tube (wall-3).

(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 panel.

4. Close the Boundary Conditions panel.

Step 5: Solution

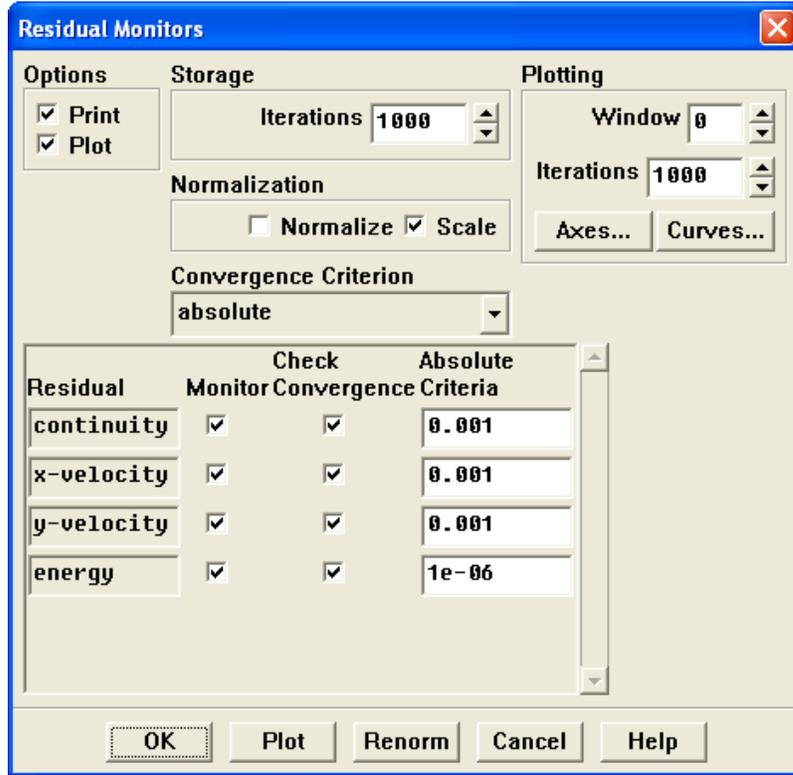1. Set the parameters that control the solution.

   Solve $\longrightarrow$ Controls $\longrightarrow$ Solution...

   (a) Enter `0.9` for Energy in the Under-Relaxation Factors group box.

   Scroll down to find the Energy *number-entry box.*

   (b) Select Second Order Upwind from the Momentum and Energy drop-down lists in the Discretization group box.

   (c) Click OK to close the Solution Controls panel.

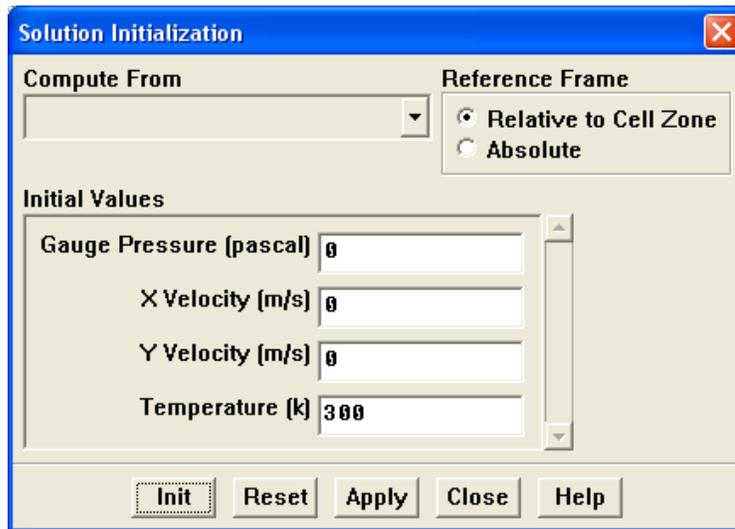2. Enable the plotting of residuals during the calculation.

| Solve | ⟶ | Monitors | ⟶Residual...

(a) Enable Plot in the Options group box.

(b) Click OK to close the Residual Monitors panel.

3. Initialize the solution.

| Solve | ⟶ | Initialize | ⟶Initialize...

    (a) Retain the default setting of 300 K for Temperature in the Initial Values group box.

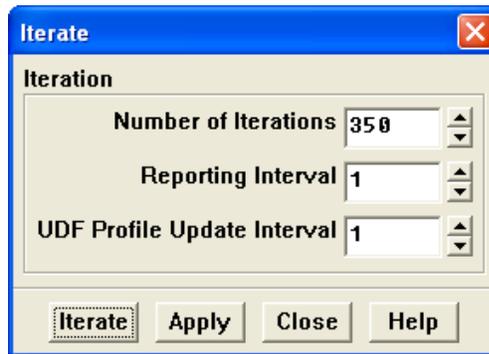   (b) Click Init and close the Solution Initialization panel.

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

4. Save the case file (`tubebank.cas`).

    File ⟶ Write ⟶Case...

5. Start the calculation by requesting 350 iterations.

    Solve ⟶Iterate...

    (a) Enter 350 for Number of Iterations.

    (b) Click Iterate.

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

       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.

6. Change the Under-Relaxation Factor for Energy to 0.6.

    Solve ⟶ Controls ⟶Solution...

7. Continue the calculation by requesting another 300 iterations.

    Solve ⟶Iterate...

    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 580 iterations.

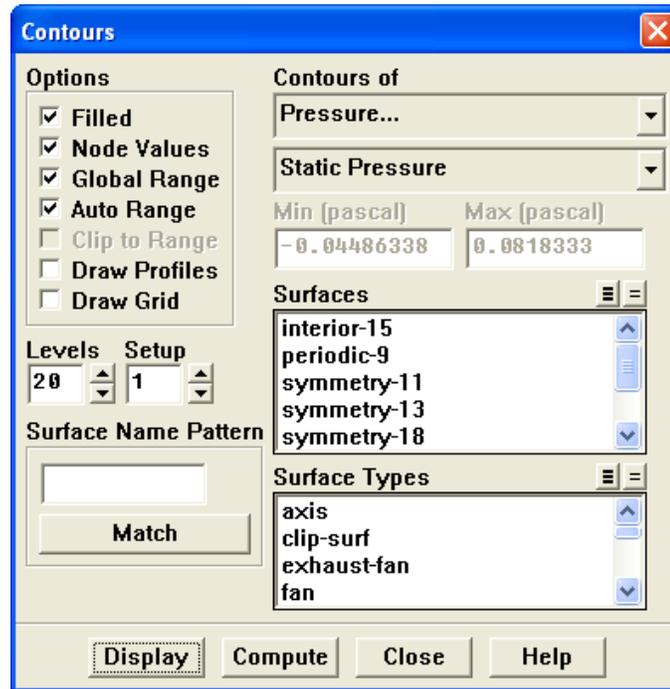8. Save the case and data files (`tubebank.cas` and `tubebank.dat`).

    File ⟶ Write ⟶Case & Data...

Step 6: Postprocessing

Postprocess the results and create plots and graphs of the solution.

1. Display filled contours of static pressure (Figure 2.3).

    Display ⟶Contours...

(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.
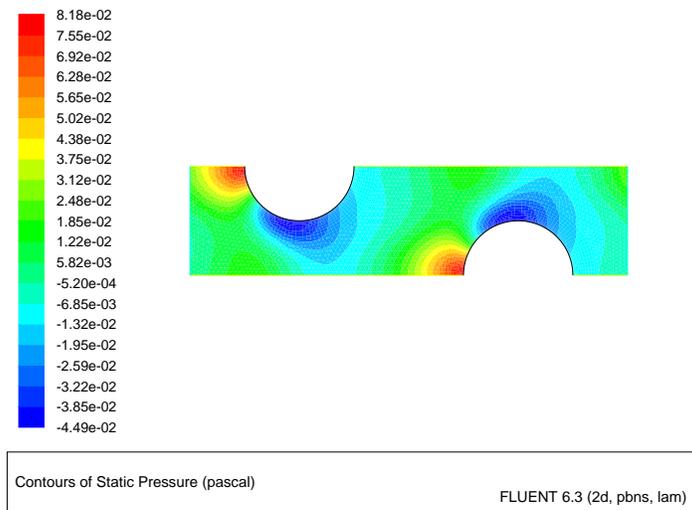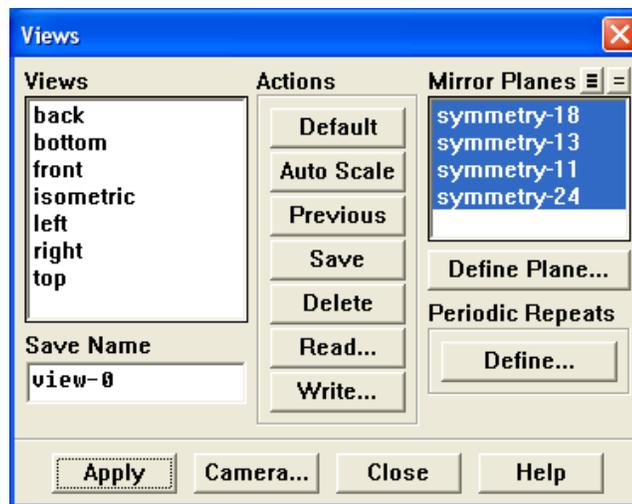
(c) Click Display and close the Contours panel.

Contours of Static Pressure (pascal)

FLUENT 6.3 (2d, pbns, lam)

Figure 2.3: Contours of Static Pressure

2. Change the view to mirror the display across the symmetry planes (Figure 2.4).

Display ⟶Views...

(a) 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 2.4 by selecting just one of the symmetry zones on the top symmetry plane, and one on the bottom.

(b) Click Apply and close the Views panel.

(c) Translate the display of symmetry contours so that it is centered in the graphics window by using the left mouse button (Figure 2.4).
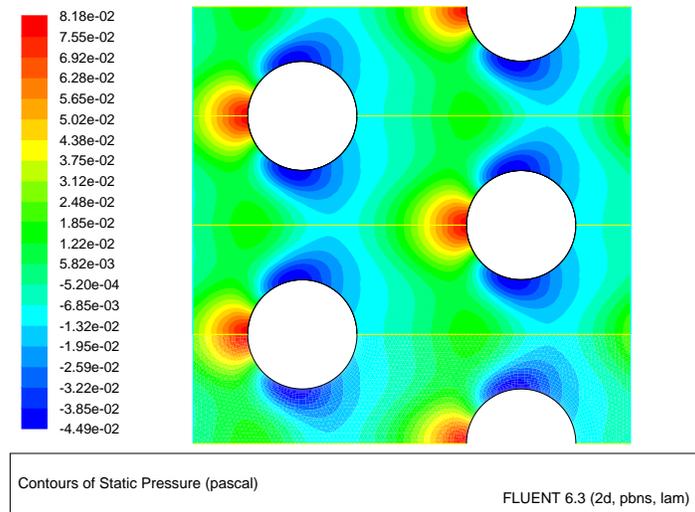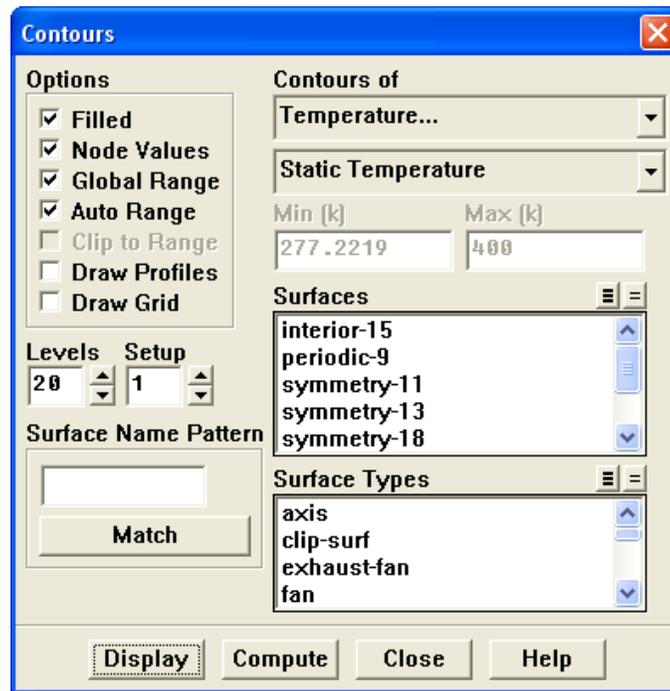

Figure 2.4: Contours of Static Pressure with Symmetry

The pressure contours displayed in Figure 2.4 do not include the linear pressure gradient computed by the solver. Thus, the contours are periodic at the inlet and outflow boundaries.

3. Display filled contours of static temperature (Figure 2.5).

$\boxed{\text{Display}} \longrightarrow \text{Contours...}$

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

(b) Click Display and close the Contours panel.

The contours in Figure 2.5 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.
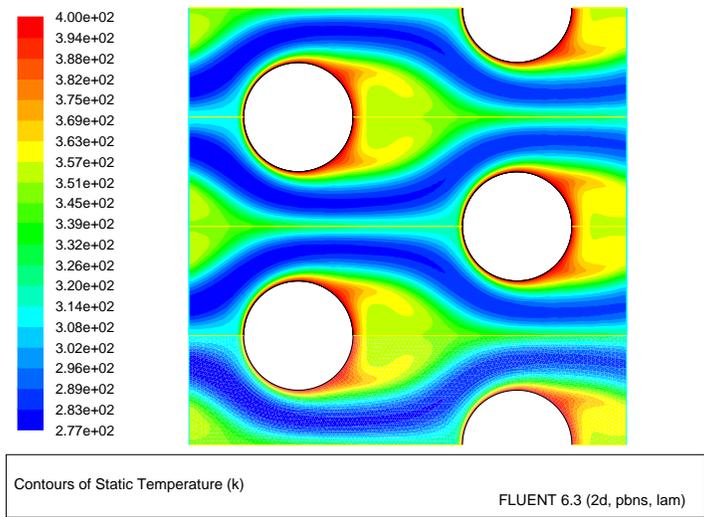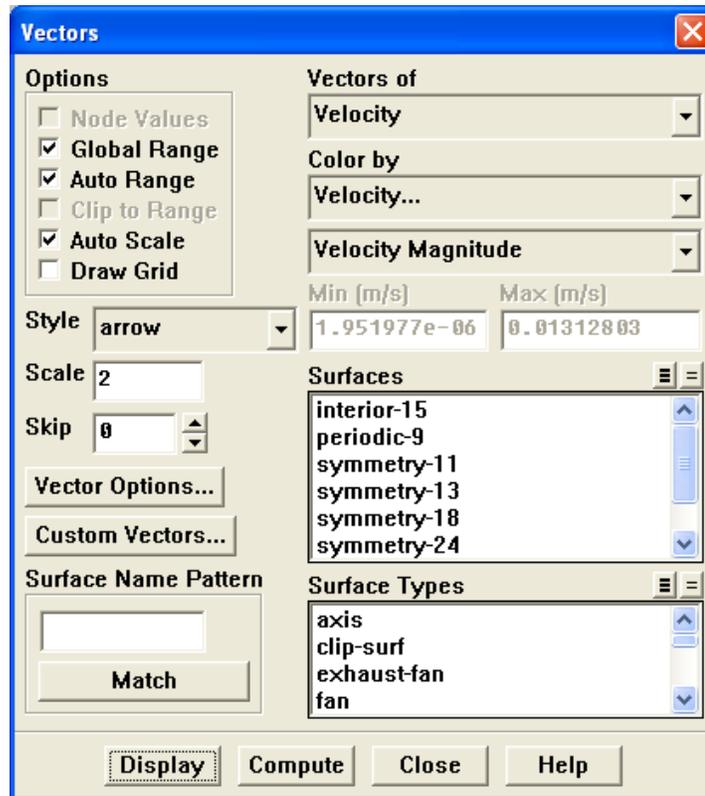
Figure 2.5: Contours of Static Temperature

4. Display the velocity vectors (Figure 2.6).

$\boxed{\text{Display}} \longrightarrow \text{Vectors...}$

(a) Enter 2 for the 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 panel.

(e) Zoom in on the upper right portion of one of the left tubes to get the display shown in (Figure 2.6), by using the middle mouse button in the graphics window.

The magnified view of the velocity vector plot in Figure 2.6 clearly shows the re-circulating flow behind the tube and the boundary layer development along the tube surface.

1.31e-02
1.25e-02
1.18e-02
1.12e-02
1.05e-02
9.85e-03
9.19e-03
8.53e-03
7.88e-03
7.22e-03
6.56e-03
5.91e-03
5.25e-03
4.60e-03
3.94e-03
3.28e-03
2.63e-03
1.97e-03
1.31e-03
6.58e-04
1.95e-06

Velocity Vectors Colored By Velocity Magnitude (m/s)
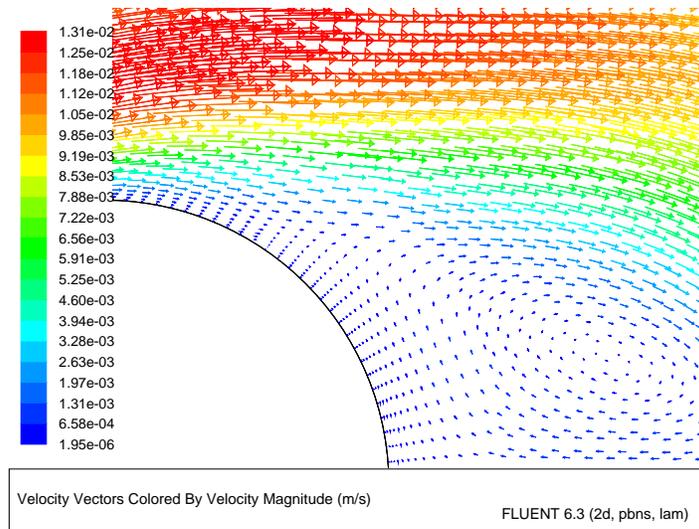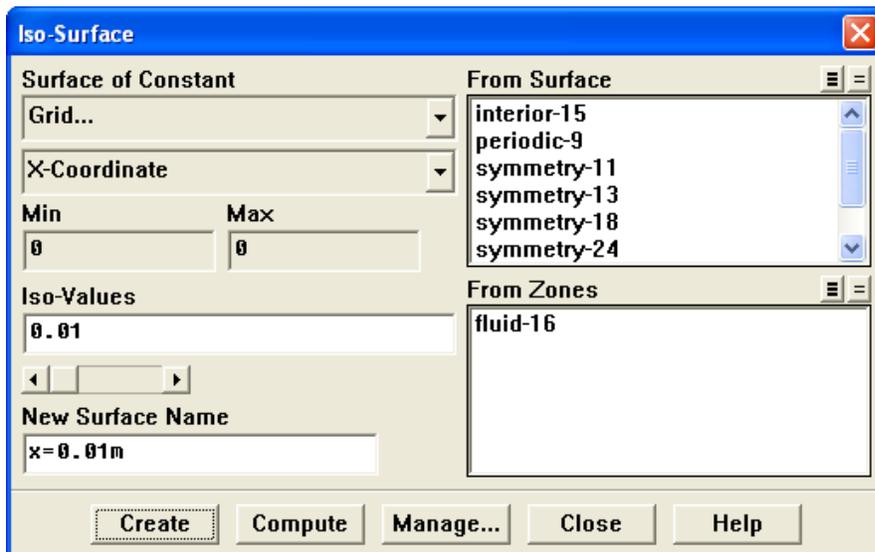
FLUENT 6.3 (2d, pbns, lam)

Figure 2.6: Velocity Vectors

5. 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.
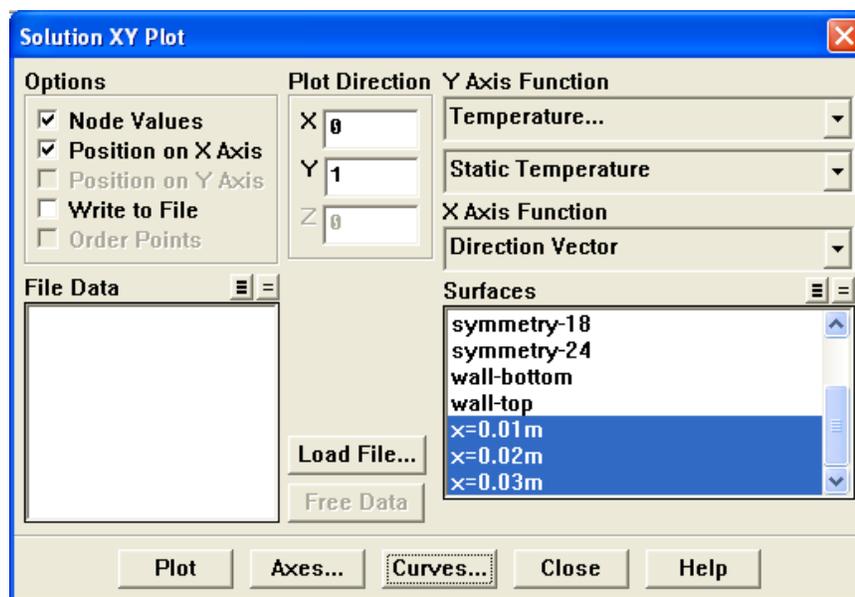
Surface $\longrightarrow$ Iso-Surface...

(a) Select Grid... and X-Coordinate from the Surface of Constant drop-down lists.

(b) Enter 0.01 for Iso-Values.

    (c) Enter `x=0.01m` for New Surface Name.

    (d) Click Create.

6. In a similar manner, create an isosurface on the periodic tube bank at $x = 0.02$ m (halfway between the two columns of tubes) named `x=0.02m`.

7. In a similar manner, create an isosurface on the periodic tube bank at $x = 0.03$ m (through the middle of the second column of tubes) named `x=0.03m`, and close the Iso-Surface panel.

8. Create an XY plot of static temperature on the three isosurfaces (Figure 2.7).

    Plot ⎯→XY Plot...

    (a) Enter 0 for X and 1 for Y in the Plot Direction group box, as shown in the previous panel.

       With a Plot Direction *vector of (0,1),* 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=0.01m, x=0.02m, and x=0.03m in the Surfaces selection list.

       Scroll down to find the x=0.01m, x=0.02m, *and* x=0.03m *surfaces.*

(d) Click Curves... to open the Curves - Solution XY Plot panel.

This panel 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 panel.

Since you did not change the curve style for the $x = 0.03$ m curve, the default symbol will be used.
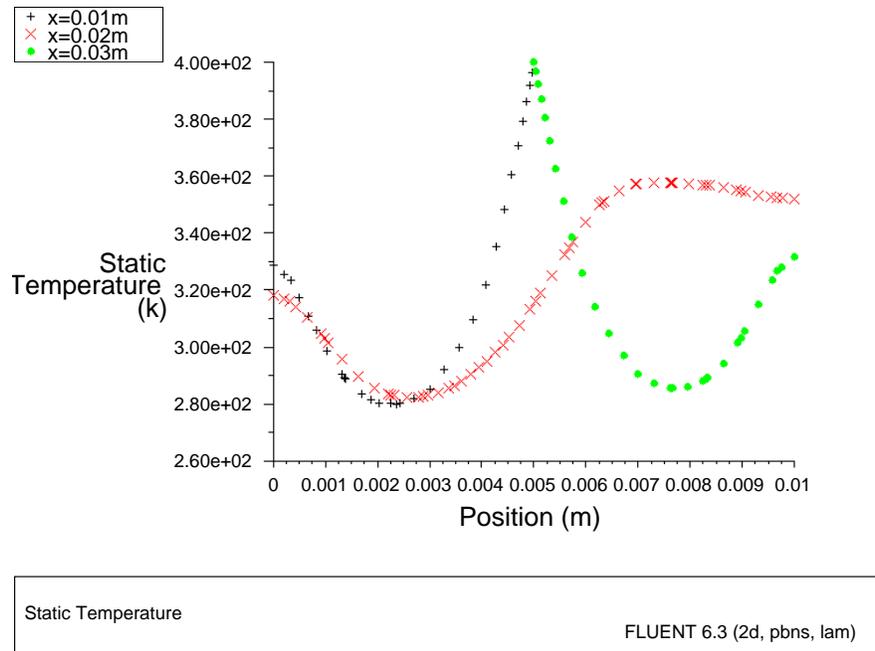
(e) Click Plot and close the Solution XY Plot panel.

Figure 2.7: Static Temperature at x=0.01, 0.02, and 0.03 m

Summary

In this tutorial, periodic flow and heat transfer in a staggered tube bank were modeled in 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 panel 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.

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 grid. Grid adaption can also ensure that the solution is independent of the grid. These steps are demonstrated in Tutorial 1.