
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 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:

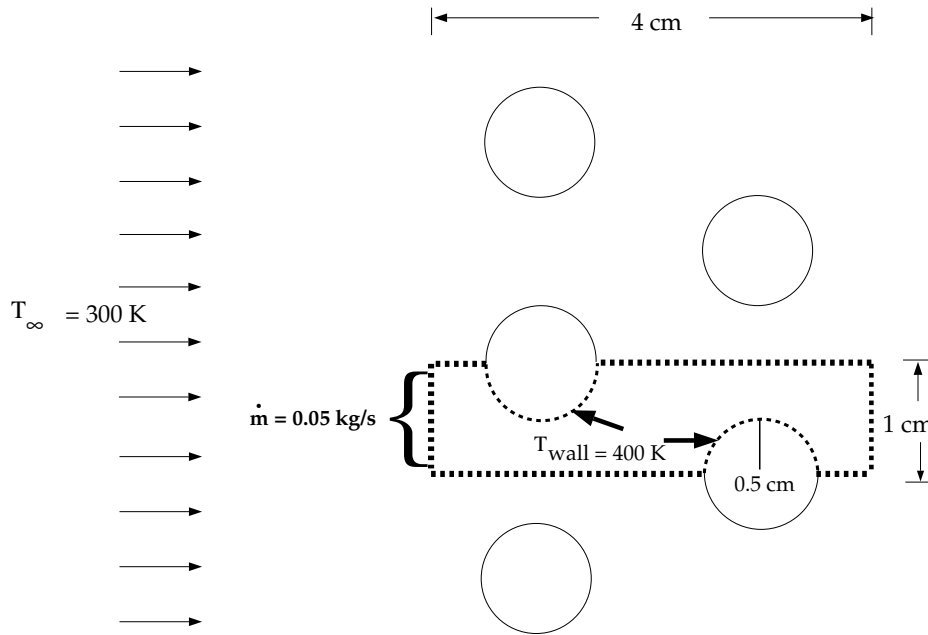
- Create periodic zones.
- Define a specified periodic mass flow rate.
- Model periodic heat transfer with specified temperature boundary conditions.
- Calculate a solution using the pressure-based solver.
- Plot temperature profiles on specified isosurfaces.

Prerequisites

This tutorial assumes that you are familiar with the menu structure in ANSYS 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.



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

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 [User Services Center](#) 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. Use FLUENT Launcher to start the 2D version of ANSYS FLUENT.

For more information about FLUENT Launcher, see Section 1.1.2 in the separate [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.*

Step 1: Mesh

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

File → Read → Mesh...

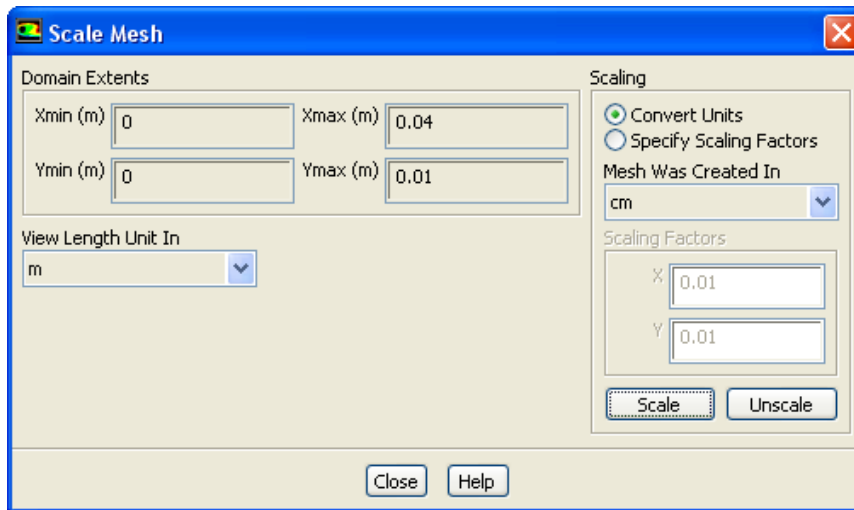
2. Check the mesh.

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 2.2).

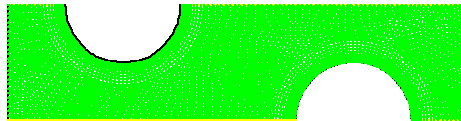


Figure 2.2: Mesh for the Periodic Tube Bank

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 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

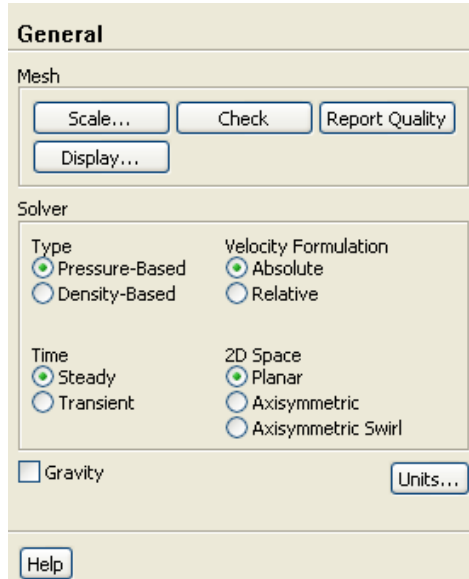
created periodic zones.
```

Step 2: General Settings

◆ General

1. Retain the default settings for the solver.

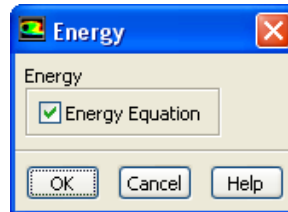
◆ General



Step 3: Models

◆ Models

1. Enable heat transfer.



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

Step 4: Materials

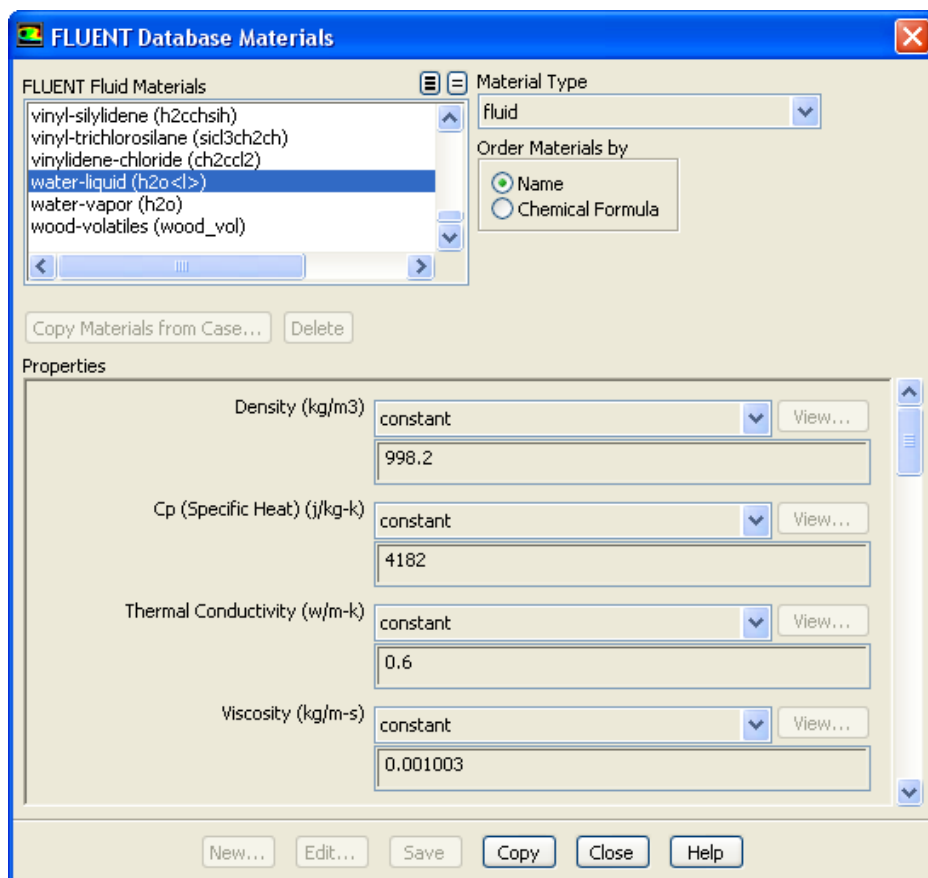
Materials

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.

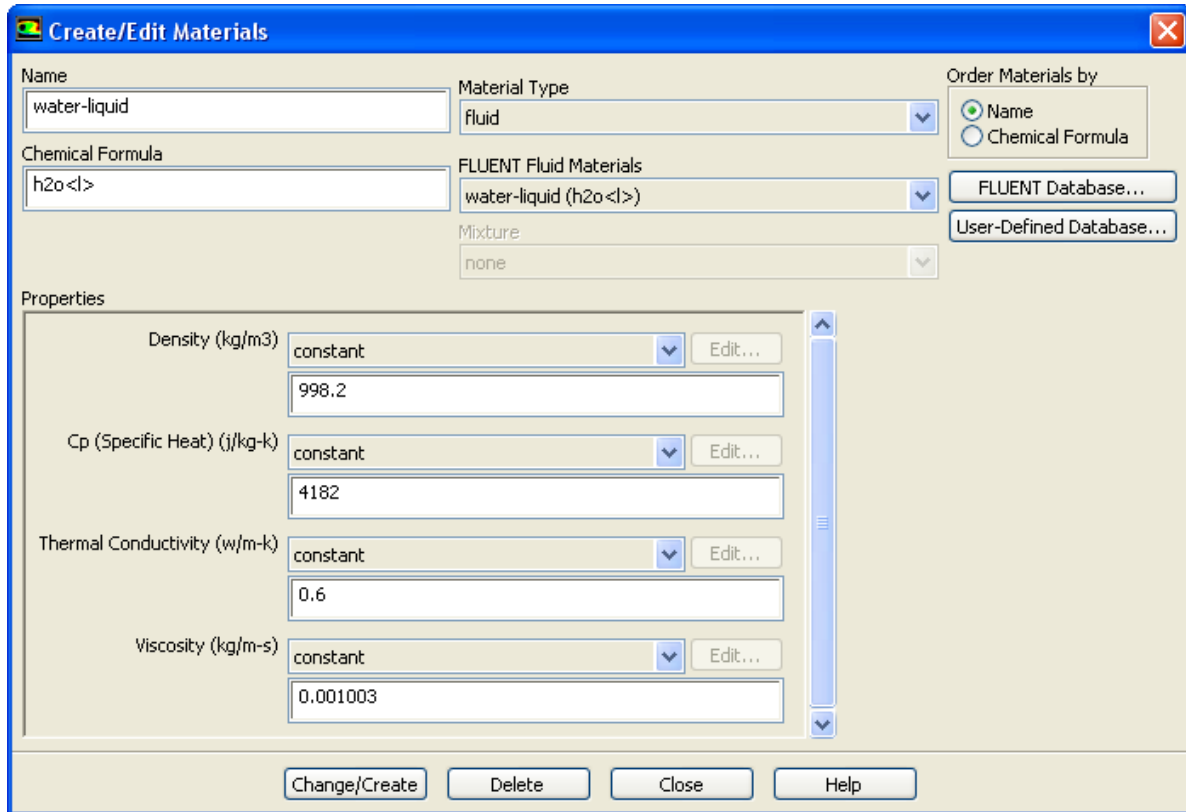
Materials → Fluid → Create/Edit...

- (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.



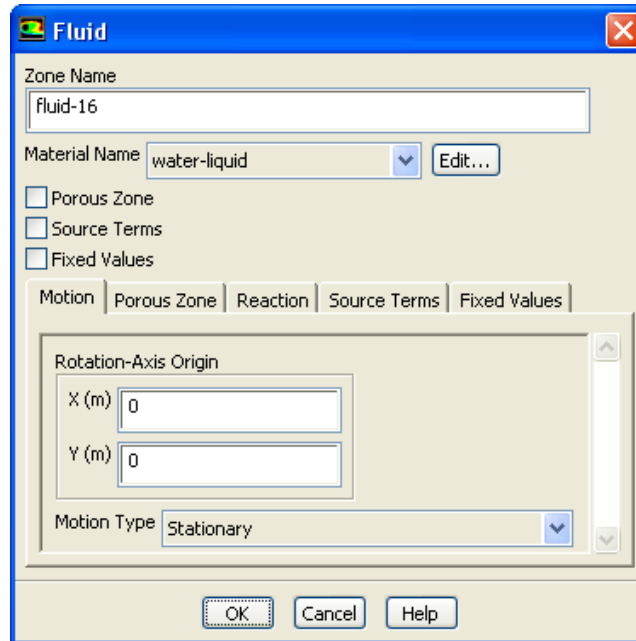
(b) Click Change/Create and close the Create/Edit Materials dialog box.

Step 5: Cell Zone Conditions

◆ Cell Zone Conditions

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

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

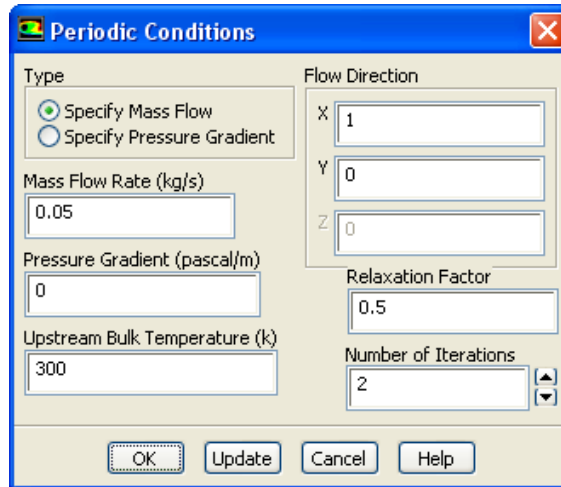


- (a) Select water-liquid from the Material Name drop-down list.
- (b) Click OK to close the Fluid dialog box.

Step 6: Periodic Conditions

◆ Boundary Conditions

1. Define the periodic flow conditions.



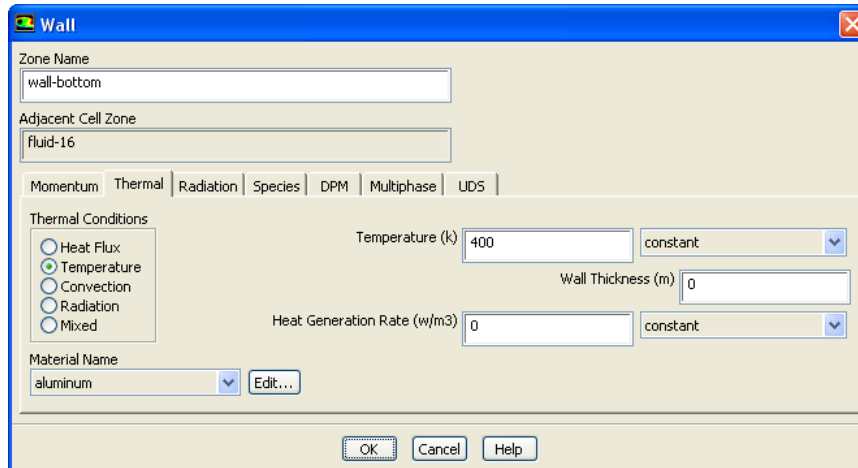
- (a) Select Specify Mass Flow in the Type list.
This will allow you to specify the Mass Flow Rate.
- (b) Enter 0.05 kg/s for Mass Flow Rate.
- (c) Click OK to close the Periodic Conditions dialog box.

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.

Step 8: Solution

1. Set the solution parameters.

↻ **Solution Methods**

The image shows the 'Solution Methods' dialog box in ANSYS. It is divided into several sections:

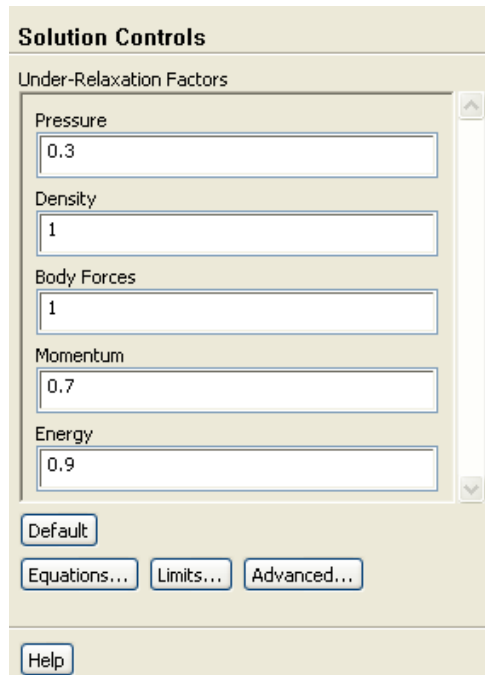
- Pressure-Velocity Coupling:** A dropdown menu for 'Scheme' is set to 'SIMPLE'.
- Spatial Discretization:** A scrollable area containing four dropdown menus:
 - 'Gradient' is set to 'Least Squares Cell Based'.
 - 'Pressure' is set to 'Standard'.
 - 'Momentum' is set to 'Second Order Upwind'.
 - 'Energy' is set to 'Second Order Upwind'.
- Transient Formulation:** A dropdown menu is currently empty. Below it are two unchecked checkboxes: 'Non-Iterative Time Advancement' and 'Frozen Flux Formulation'.

At the bottom of the dialog, there are two buttons: 'Default' and '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.

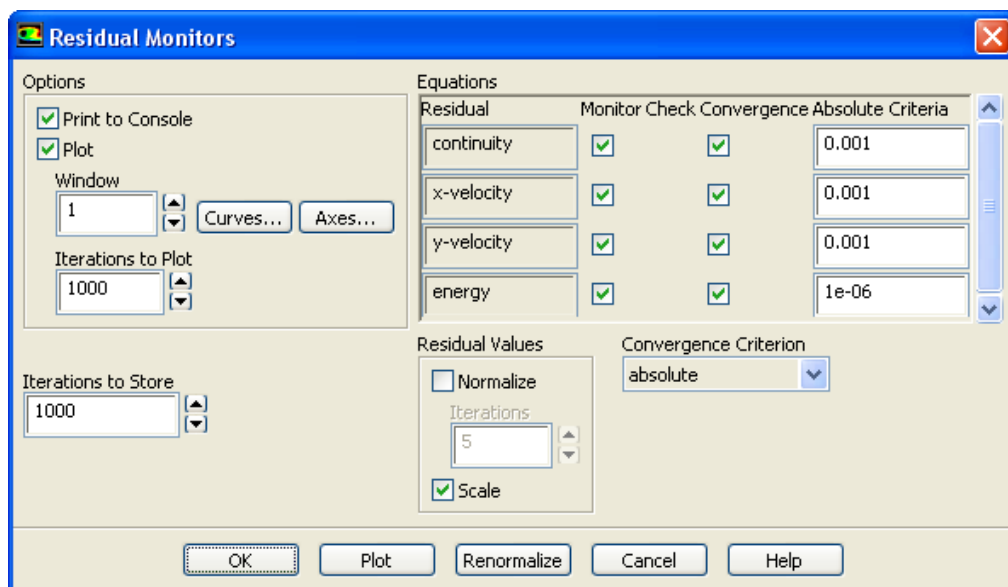
◆ **Solution Controls**



(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

The image shows the 'Solution Initialization' dialog box. It has a title bar 'Solution Initialization'. Below the title bar, there is a 'Compute from' dropdown menu. Underneath is the 'Reference Frame' section with two radio buttons: 'Relative to Cell Zone' (which is selected) and 'Absolute'. The 'Initial Values' section contains five input fields: 'Gauge Pressure (pascal)' with '0', 'X Velocity (m/s)' with '0', 'Y Velocity (m/s)' with '0', and 'Temperature (k)' with '300'. At the bottom of the dialog are several buttons: 'Initialize', 'Reset', 'Patch...', 'Reset DPM Sources', 'Reset Statistics', and 'Help'.

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

(b) Click Initialize.

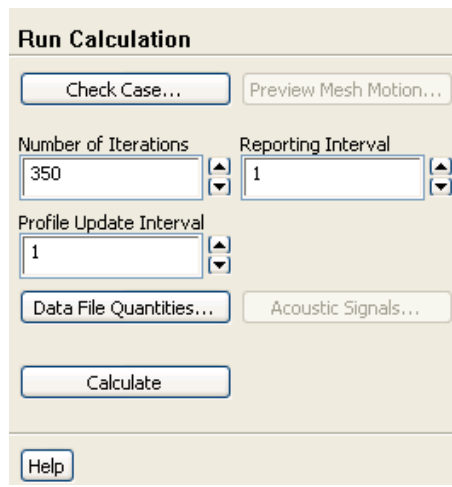
The values shown in the task pane will be used as the initial condition for the solution.

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

File → Write → Case...

- Start the calculation by requesting 350 iterations.

◆ **Run Calculation**



- Enter 350 for Number of Iterations.
- 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.

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

◆ **Solution Controls**

- 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 580 iterations.

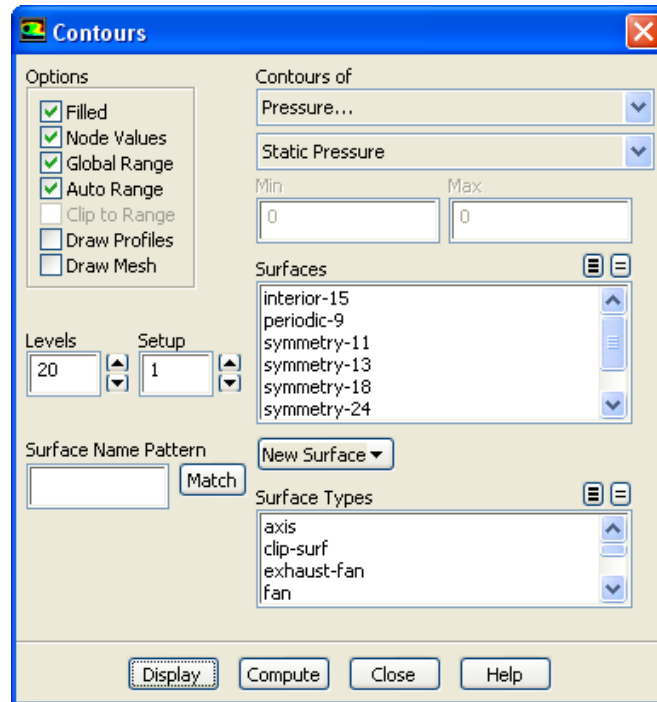
- Save the case and data files (tubebank.cas and tubebank.dat).

File → Write → Case & Data...

Step 9: Postprocessing

◆ Graphics and Animations

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



- (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.

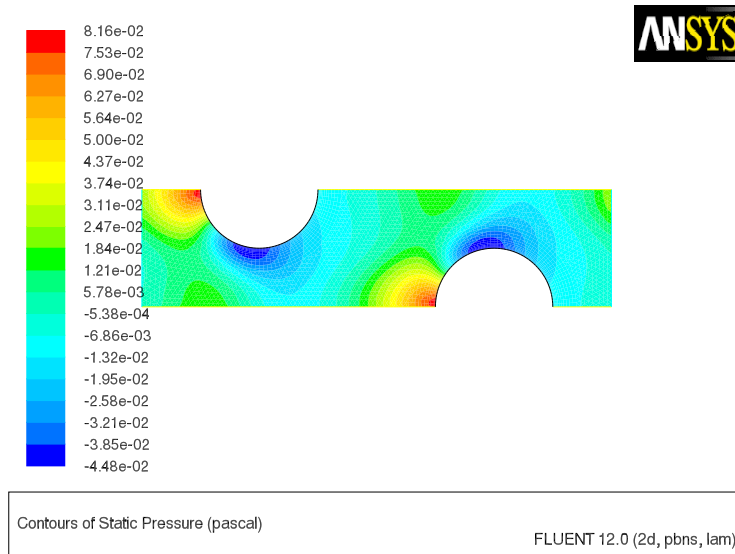
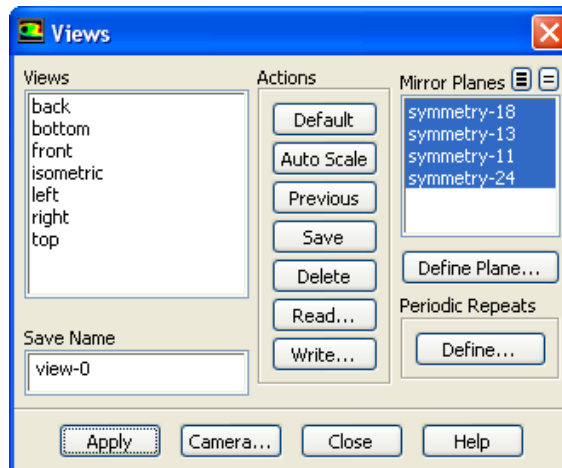


Figure 2.3: Contours of Static Pressure

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



- 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 2.4 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 (Figure 2.4).

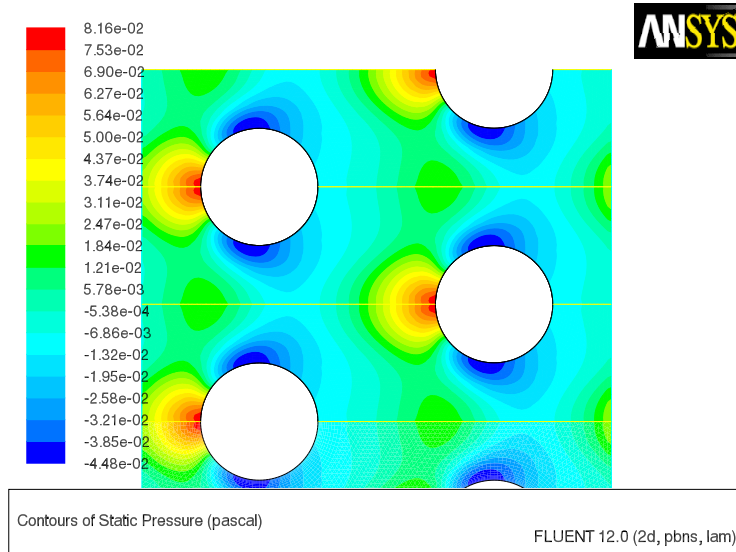
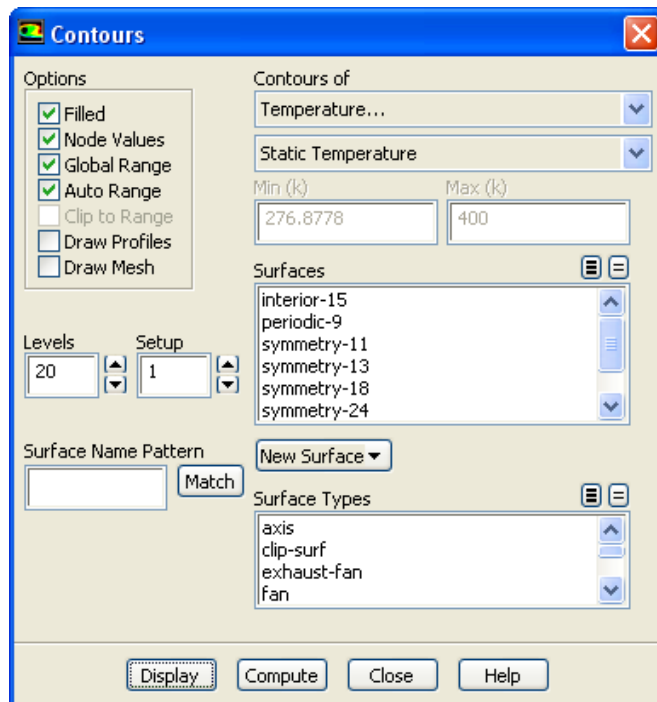


Figure 2.4: Contours of Static Pressure with Symmetry

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.

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



- (a) Select Temperature... and Static Temperature from the Contours of drop-down lists.
- (b) Click Display and close the Contours dialog box.

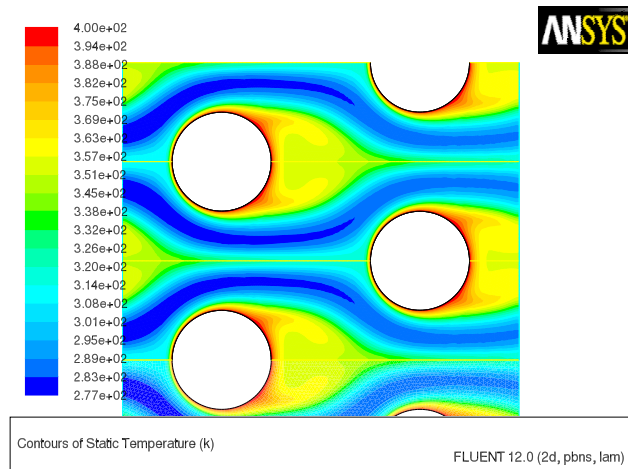
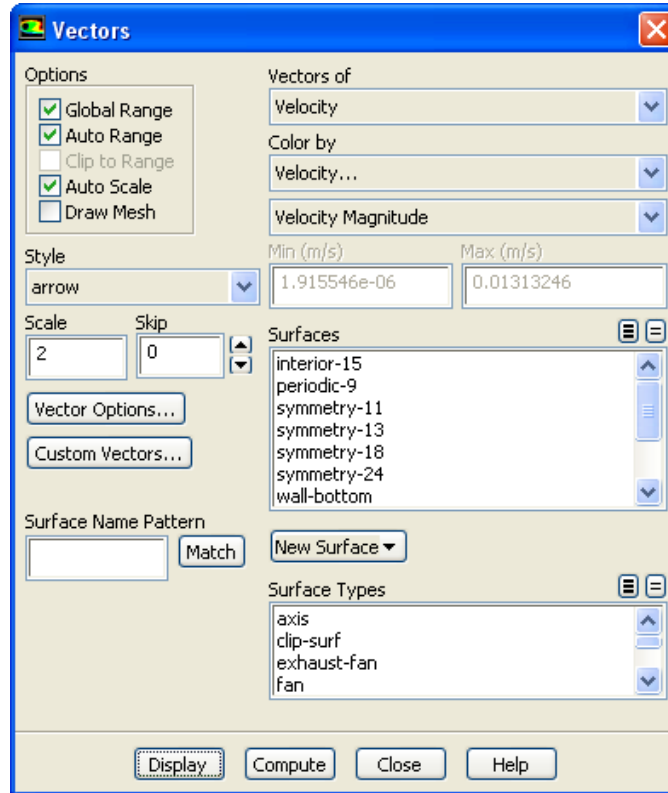


Figure 2.5: Contours of Static Temperature

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.

3. Display the velocity vectors (Figure 2.6).



- (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 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 recirculating flow behind the tube and the boundary layer development along the tube surface.

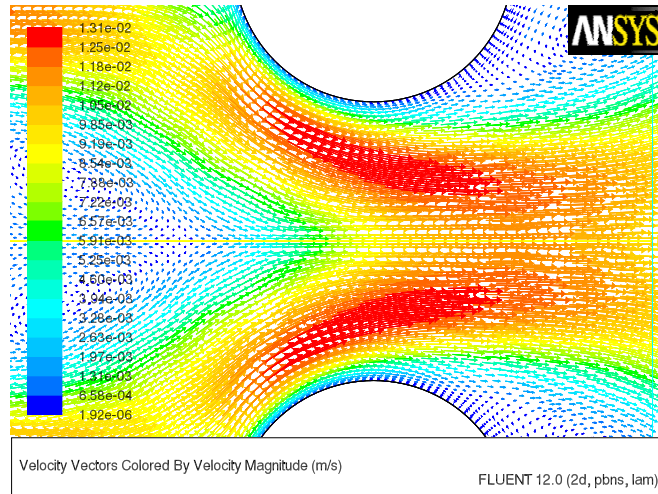
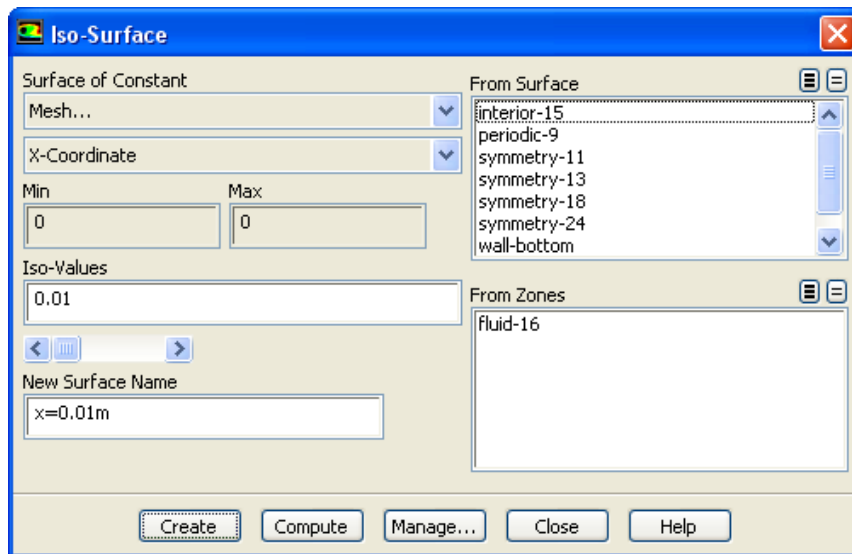


Figure 2.6: Velocity Vectors

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.

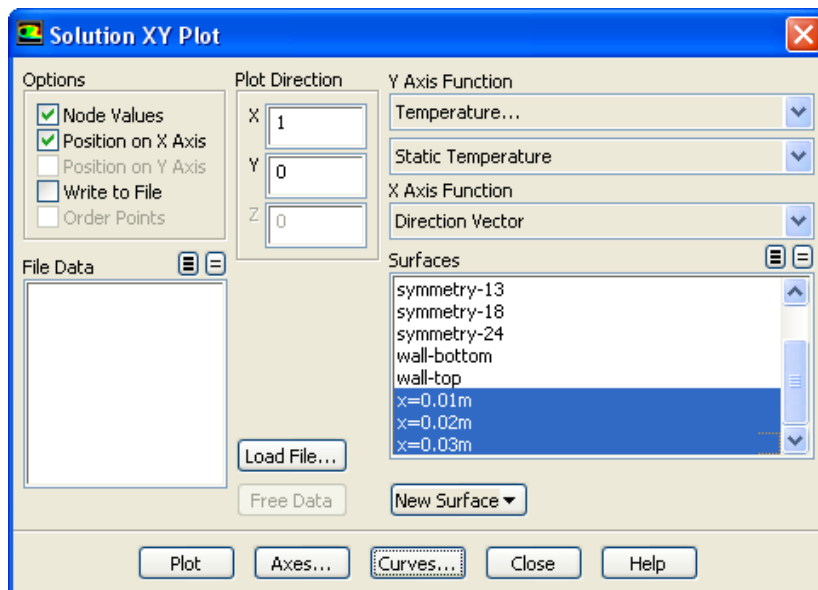
Surface → Iso-Surface...



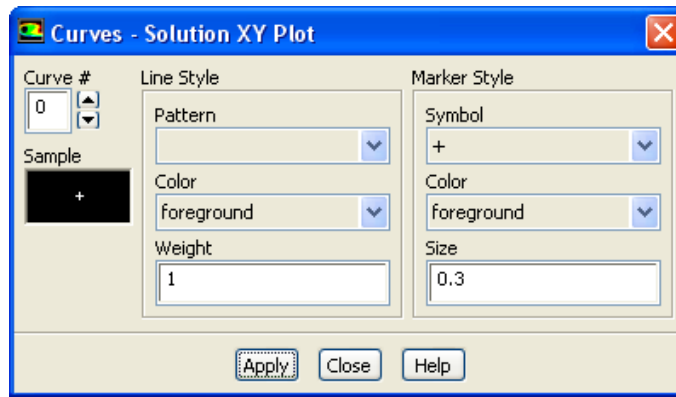
- (a) Select Mesh... and X-Coordinate from the Surface of Constant drop-down lists.
- (b) Enter 0.01 for Iso-Values.
- (c) Enter x=0.01m for New Surface Name.

- (d) Click Create.
5. 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 $x=0.02m$.
6. 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 dialog box.
7. Create an XY plot of static temperature on the three isosurfaces (Figure 2.7).

 **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=0.01m, 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 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.

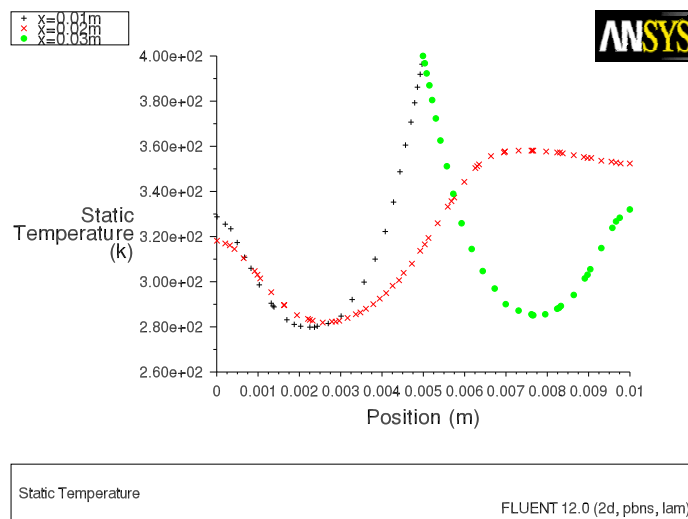


Figure 2.7: Static Temperature at $x=0.01$, 0.02 , and 0.03 m

Summary

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

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 adaptation can also ensure that the solution is independent of the mesh. These steps are demonstrated in [Tutorial 1](#).