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:

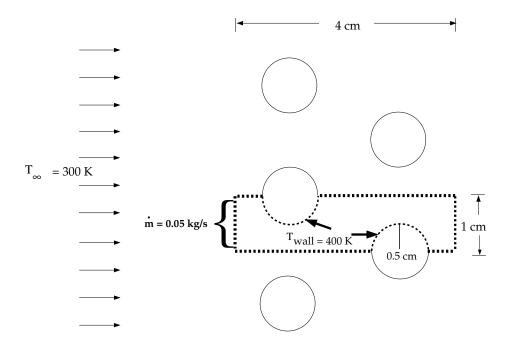
- Create periodic zones.
- Define a specified periodic mass flow rate.
- Model periodic heat transfer with specified temperature boundary conditions.
- Calculate a solution using the pressure-based solver.
- Plot temperature profiles on specified isosurfaces.

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 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 $(T_{\rm wall})$ 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

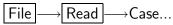
- 1. Download periodic_flow_heat.zip from the Fluent Inc. User Services Center or copy it from the FLUENT documentation CD to your working folder (as described in Tutorial 1).
- 2. Unzip periodic_flow_heat.zip.

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.



2. Check the grid.

 $Grid \longrightarrow Check$

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	\longrightarrow Scale
------	-------------------------

Scale Grid		×	
Scale Factors	Unit Conversion		
× 0.01	Grid Was Created In cm 👻		
Y 0.01	Change Length Units		
Domain Extents			
Xmin (m) 🔋	Xma× (m) 0.04	ſ	
Ymin (m) 👔 Ymax (m) 📴 9.01			
Scale	Unscale Close Help		

- (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).

 $\mathsf{Display} \longrightarrow \mathsf{Grid}...$

Grid Display			×
Options Nodes ✓ Edges Faces Partitions Shrink Factor F 0	Edge Type • All • Feature • Outline eature Angle	Surfaces symmetry-11 symmetry-13 symmetry-18 symmetry-24 wall-12 wall-21 wall-3 wall-9	
Surface Name I	Pattern Match	Surface Types I axis clip-surf exhaust-fan fan Outline Interior	
Display	Colors	Close Help	

- (a) Retain the default settings.
- (b) Click Display and close the Grid Display panel.



Grid		
	FLUENT 6.3 (2d, pbns, lam)	

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 –	\rightarrow	Models	\longrightarrow Solver
----------	---------------	--------	--------------------------

Solver	×
Solver © Pressure Based © Density Based	Formulation C Implicit C Explicit
Space © 2D © Axisymmetric © Axisymmetric Swirl © 3D Velocity Formulation © Absolute	Time Steady Unsteady
C Relative Gradient Option Green-Gauss Cell Bas Green-Gauss Node Ba	
C Least Squares Cell Ba	

2. Activate heat transfer.

 $\fbox{Define} \longrightarrow \rag{Models} \longrightarrow \rag{Energy...}$

Energy 🔀		
Energy		
Energy Equation		
OK Cancel Help		

- (a) Enable the Energy Equation option.
- (b) Click OK to close the Energy panel.

3. Define the periodic flow conditions.

Define \longrightarrow Periodic Conditions...

Periodic Conditions	
Type Specify Mass Flow Specify Pressure Gradient	Flow Direction
Mass Flow Rate (kg/s) 0.05 Pressure Gradient (pascal/m)	Z 0 Relaxation Factor
0	0.5
Upstream Bulk Temperature (k) 300	Number of Iterations
OK Update C	ancel Help

- (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 $\mathsf{Periodic}$ 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.

 $Define \longrightarrow Materials...$

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

Fluent Database Materials	
Fluent Fluid Materials = vinyl-trichlorosilane (sicl3ch2ch) vinylidene-chloride (ch2ccl2) water-liquid (h2o<1>) water-vapor (h2o) wood-volatiles (wood_vol) Copy Materials from Case Delete Properties	Material Type fluid Order Materials By Name Chemical Formula
Density (kg/m3) Cp (j/kg-k)	998.2
Thermal Conductivity (w/m-k) Viscosity (kg/m-s)	0.6
New Edit Sav	Close Help

- 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.

Name	Material Type	Order Materials By
water-liquid	fluid	▼ 🖲 Name
Chemical Formula	Fluent Fluid Materials	Chemical Formula
h2o<1>	water-liquid (h2o <l>)</l>	▼ Fluent Database
	Mixture	User-Defined Database
	none	
Properties	,	
Density (kg/m3)		<u> </u>
	constant Edit	
	998.2	
Cp (j/kg-k)	constant 🗸 Edit	
	4182	
	4182	
Thermal Conductivity (w/m-k)	constant 👻 Edit	
	9.6	
Viscosity (kg/m-s)	constant 👻 Edit	
	0.001003	
		·

(b) Close the Materials panel.

Step 4: Boundary Conditions

Boundary Cond	itions 🛛 🔀	
Zone fluid-16 interior-15 periodic-9 symmetry-11 symmetry-13 symmetry-24 wall-21 wall-3	Type fluid solid	
ID 16		
Set Co	py Close Help	

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

Fluid 🛛
Zone Name fluid-16
Material Name water-liquid Edit
☐ Source Terms ☐ Fixed Values
Motion Porous Zone Reaction Source Terms Fixed Values
Rotation-Axis Origin
Y (m) 0
Motion Type Stationary
OK Cancel Help

- (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).

Wall						X
Zone Name						
wall-bottor	n					
Adjacent Cell	Zone					
fluid-16						
Momentum	Thermal	Radiation	Species DPM	Multi	ohase UDS	
Thermal Con	ditions					
O Heat Flu	x		Temper	ature (k)	400	constant 🔹
 Tempera Convection 			Wall Thickr	iess (m)	0	
C Radiatio		Heat	Generation Rat	e (w/m3)	0	constant 👻
iniacu					,	,
Material Nan aluminum	ne	▼ Edit				
			OK C	ancel	Help	

- (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).

Wall							X
Zone Name							
wall-top							
Adjacent Cell	Zone						
fluid-16							
Momentum	Thermal	Radiation	Species DPM	Multi	phase UDS		
Thermal Cor	ditions						
C Heat Flu	x		Temper	ature (k)	400	constant .	-
 Tempera Convect 			Wall Thickr	iess (m)	0		
C Radiatio	n	Heat	Generation Rate	: (w/m3)	0	constant -	-
						, _	-
Material Nar aluminum	ne	▼ Edit					
			ОК	ancel	Help		

- (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.	
--	--

Solution Controls			
Equations 📃 🖃	Under-Relaxation Factors		
Flow Energy	Density 1		
<u>Energy</u>	Body Forces 1		
	Momentum 0.7		
	Energy 0.9		
Pressure-Velocity Coupling Discretization			
SIMPLE	Pressure Standard		
	Momentum Second Order Upwind 👻		
	Energy Second Order Upwind		
OK Default Cancel Help			

- (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.

 $\fbox{Solve} \longrightarrow \fbox{Monitors} \longrightarrow \r{Residual} \dots$

Iterations	5 1000 +	Wind	
ormalization		Iterations	1000 숮
🗆 Normal	lize 🗹 Scale	Axes	Curves
onvergence Crit	terion		
bsolute	•		
Check Monitor Converç	Absolute jence Criteria	À	
V V	0.001		
V V	0.001	_	
V V	0.001		
V V	1e-06		
	,		
		*	
	bsolute Check Monitor Converg V V V V V V	Check Absolute Monitor Convergence Criteria Image: Conve	check Absolute Check Absolute Monitor Convergence Criteria V V 0.001 V V 0.001

- (a) Enable Plot in the Options group box.
- (b) Click OK to close the Residual Monitors panel.
- 3. Initialize the solution.

Solve \longrightarrow Initialize \longrightarrow Initialize...

Solution Initialization
Compute From Reference Frame Reference Frame Reference Frame Absolute
Initial Values
Gauge Pressure (pascal) g
× Velocity (m/s) g
Y Velocity (m/s) g
Temperature (k) 300
Init Reset Apply Close Help

- (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).

$$\mathsf{File} \longrightarrow \mathsf{Write} \longrightarrow \mathsf{Case}...$$

5. Start the calculation by requesting 350 iterations.

Solve \longrightarrow Iterate...

Iterate 🔀		
Iteration		
Number of Iterations 350		
Reporting Interval 1		
UDF Profile Update Interval 1		
Iterate Apply Close Help		

- (a) Enter 350 for Number of Iterations.
- (b) Click Iterate.
- (c) Close the **lterate** 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.

7. Continue the calculation by requesting another **300** iterations.

Solve \longrightarrow 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).

 $\mathsf{File} \longrightarrow \mathsf{Write} \longrightarrow \mathsf{Case} \And \mathsf{Data...}$

Step 6: Postprocessing

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

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

 $\mathsf{Display} \longrightarrow \mathsf{Contours}...$

Contours	X
Options	Contours of
🗹 Filled	Pressure 👻
✓ Node Values ✓ Global Range	Static Pressure
Auto Range	Min (pascal) Max (pascal)
Clip to Range	-0.04486338 0.0818333
Draw Profiles	Surfaces =
Levels Setup 20 1 4 Surface Name Pattern	interior-15 periodic-9 symmetry-11 symmetry-13 symmetry-18 v
	Surface Types
Match	axis clip-surf
	exhaust-fan fan 💌
Display Co	mpute Close Help

- (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.

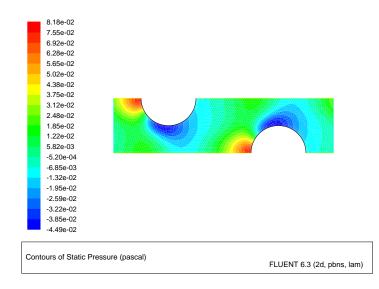


Figure 2.3: Contours of Static Pressure

2. Change the view to mirror the display across the symmetry planes (Figure 2.4). \Box isplay \longrightarrow Views...

Views		×	
Views	Actions	Mirror Planes = =	
back bottom	Default	symmetry-18 symmetry-13	
front	Auto Scale	symmetry-11	
isometric left	Previous	symmetry-24	
right top	Save	Define Plane	
	Delete	Periodic Repeats	
Save Name	Read	Define	
view-0	Write		
Apply Camera Close Help			

- (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).

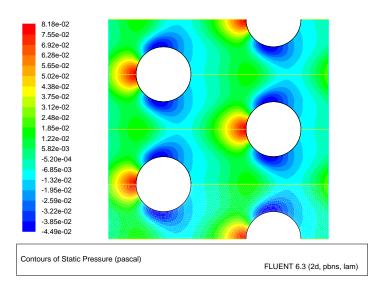


Figure 2.4: Contours of Static Pressure with Symmetry

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).

$Display \longrightarrow$	Contours
---------------------------	----------

Contours	X	
Options	Contours of	
🗹 Filled	Temperature	
✓ Node Values ✓ Global Range	Static Temperature 👻	
Auto Range	Min (k) Max (k)	
Clip to Range	277.2219 400	
Draw Profiles Draw Grid	Surfaces =	
Levels Setup 20 1 2	interior-15 periodic-9 symmetry-11 symmetry-13	
	symmetry to	
Match	Surface Types = = axis clip-surf exhaust-fan fan	
Display Compute Close Help		

- (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.

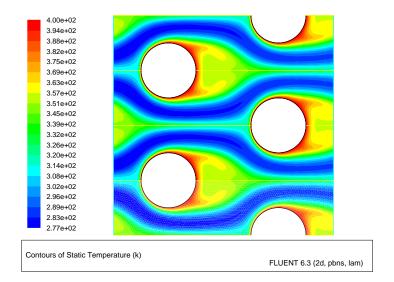


Figure 2.5: Contours of Static Temperature

4. Display the velocity vectors (Figure 2.6).



Vectors					
Options	Vectors of				
🗖 Node Values	Velocity 👻				
🗹 Global Range	Color by				
Auto Range Clip to Range	Velocity 🝷				
Auto Scale Draw Grid	Velocity Magnitude				
	Min (m/s) Max (m/s)				
Style arrow 🝷	1.951977e-06 0.01312803				
Scale 2	Surfaces 📃				
Skip 👩 🔺	interior-15				
Sub A	periodic-9 symmetry-11				
Vector Options	symmetry-13				
Custom Vectors	symmetry-18 symmetry-24				
Surface Name Pattern	Surface Types				
	axis				
Match	clip-surf exhaust-fan				
fan 💉					
Display Compute Close Help					

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

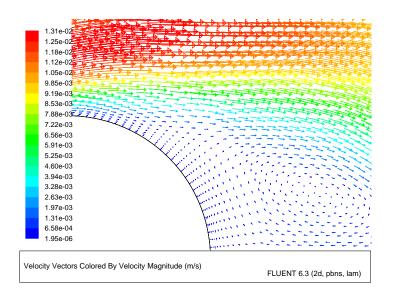


Figure 2.6: Velocity Vectors

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...

lso-Surface					×
Surface of Constant			From Surface		==
Grid		•	interior-15		~
		=	periodic-9		
X-Coordinate		•	symmetry-11 symmetry-13		=
Min Ma	x		symmetry-18		
0 0			symmetry-24		~
, Iso-Values			From Zones		E E
0.01			fluid-16		
New Surface Name					
x=0.01m					
			r.		
Create	Compute Ma	naç	e Close	He	p

- (a) Select Grid... and X-Coordinate from the Surface of Constant drop-down lists.
- (b) Enter 0.01 for lso-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.
- Create an XY plot of static temperature on the three isosurfaces (Figure 2.7).
 Plot → XY Plot...

Solution XY Plot				
Options Plot Direction Y Axis Function				
Node Values	XØ	Temperature	•	
Position on X Axis Position on Y Axis	Y 1	Static Temperature	•	
Write to File	ZO	X Axis Function		
C Order Points		Direction Vector	•	
File Data 📃 📃		Surfaces	II.	
		symmetry-18 symmetry-24 wall-bottom wall-top	~	
		x=0.01m		
	Load File	x=0.02m x=0.03m	~	
	Free Data			
Plot Axes Curves Close Help				

(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 dropdown 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.

Curves - Solution XY Plot					
Curve # Ø • Sample +	Line Style Pattern Color	Marker Style Symbol + Color			
	foreground Weight 1	foreground Size 0.3			
Apply Close Help					

- 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 \times 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.

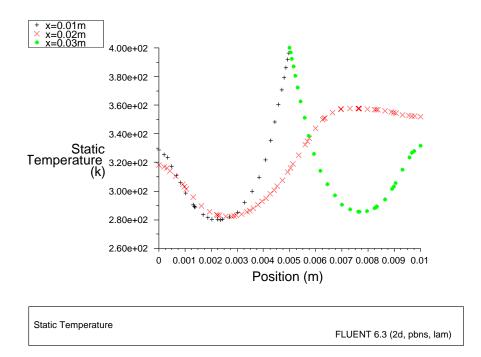


Figure 2.7: Static Temperature at x=0.01, 0.02, and 0.03 m

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.