
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 inflow boundary will be redefined as a periodic zone, and the outflow boundary defined as its shadow.

In this tutorial you will learn how to:

- Create periodic zones
- Define a specified periodic mass flow rate
- Model periodic heat transfer with specified temperature boundary conditions
- Calculate a solution using the segregated 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 solved or read Tutorial 1. Some steps 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, that are staggered in the direction of 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.

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 inflow 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.

Preparation

1. Copy the file `tubebank/tubebank.msh` from the FLUENT documentation CD to your working directory (as described in Tutorial 1).
2. Start the 2D version of FLUENT.

Step 1: Grid

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

`File` → `Read` → Case...

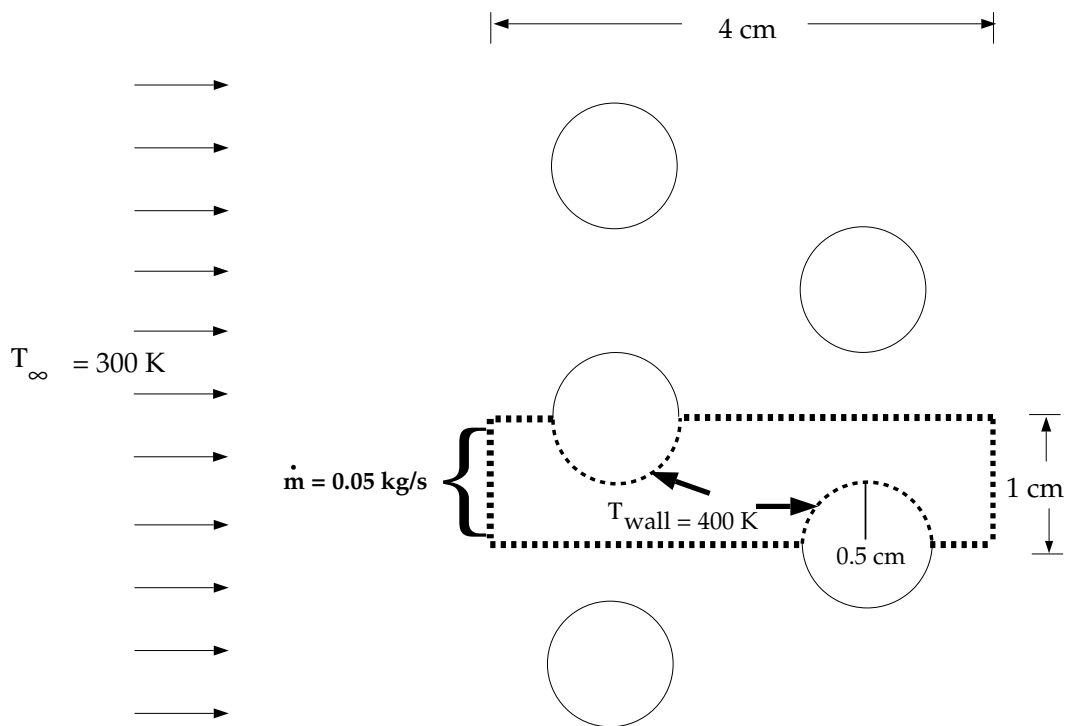
2. Check the grid.

`Grid` → Check

FLUENT *will perform various checks on the mesh and will report the progress in the console window. Pay particular attention to the reported minimum volume. Make sure this is a positive number.*

3. Scale the grid.

`Grid` → Scale...



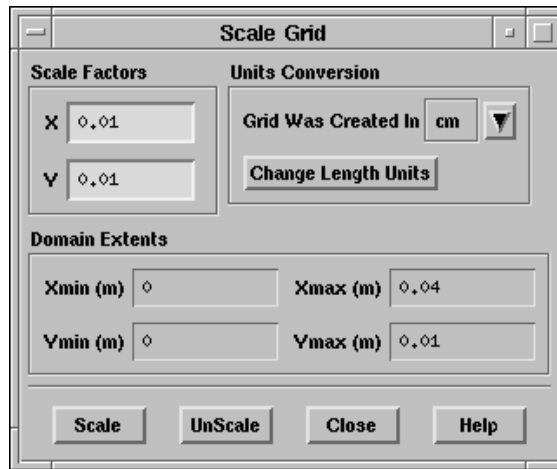
$$\rho = 998.2 \text{ kg/m}^3$$

$$\mu = 0.001003 \text{ kg/m-s}$$

$$c_p = 4182 \text{ J/kg-K}$$

$$k = 0.6 \text{ W/m-K}$$

Figure 2.1: Schematic of the Problem

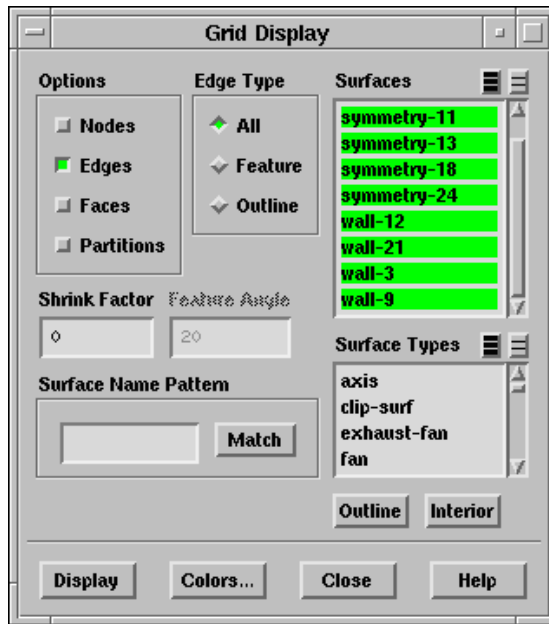


- (a) In the Units Conversion drop-down list, select cm to complete the phrase Grid Was Created In cm (centimeters).
- (b) Click on Scale to scale the grid.

The final Domain Extents should appear as in the panel above.

- 4. Display the mesh (Figure 2.2).

Display → Grid...



In Figure 2.2 you can see that 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. The quadrilateral cells provide better resolution of the viscous gradients near the tube walls. The remainder of the computational domain is conveniently filled with triangular cells.

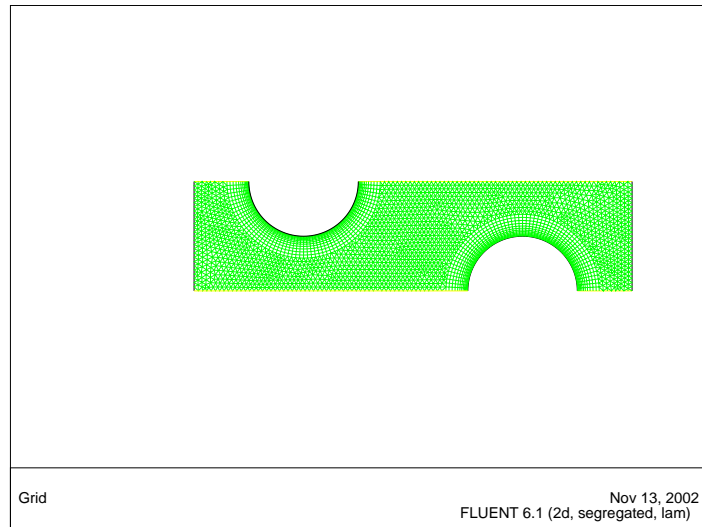


Figure 2.2: Mesh for the Periodic Tube Bank

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

5. Create the periodic zone.

wall-9 and *wall-12*, the inflow and outflow boundaries, respectively, are currently defined as wall zones and need to be redefined as periodic. *wall-9* will be made into a translationally periodic zone, and *wall-12* will be deleted and redefined as *wall-9*'s periodic shadow.

- (a) In the console window, type the commands shown in boxes in the dialog below.

Hint: You may need to enter press the `<Enter>` key to get the `>` prompt.

```
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.
```

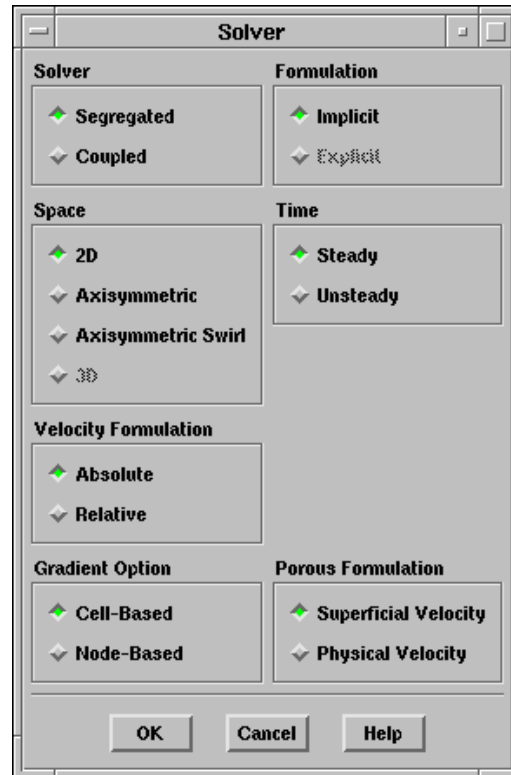
```
zone 12 deleted
```

```
created periodic zones.
```

Step 2: Models

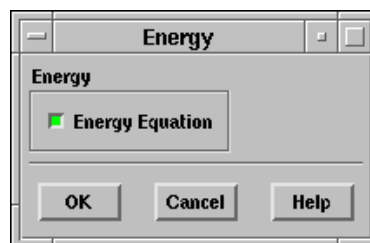
1. Keep the default solver settings.

Define → Models → Solver...



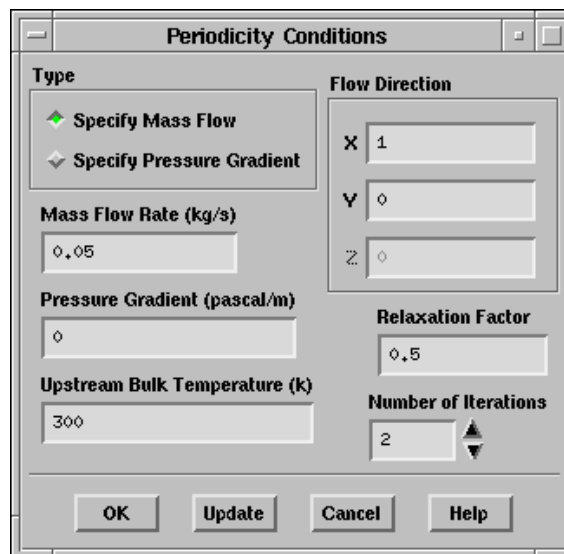
2. Enable heat transfer by activating the energy equation.

Define → Models → Energy...



3. Set the periodic flow conditions.

Define → Periodic Conditions...



- (a) Select Specify Mass Flow under Type.
This will allow you to specify the Mass Flow Rate.
- (b) Enter a Mass Flow Rate of 0.05 kg/s.
- (c) Click OK.

Step 3: Materials

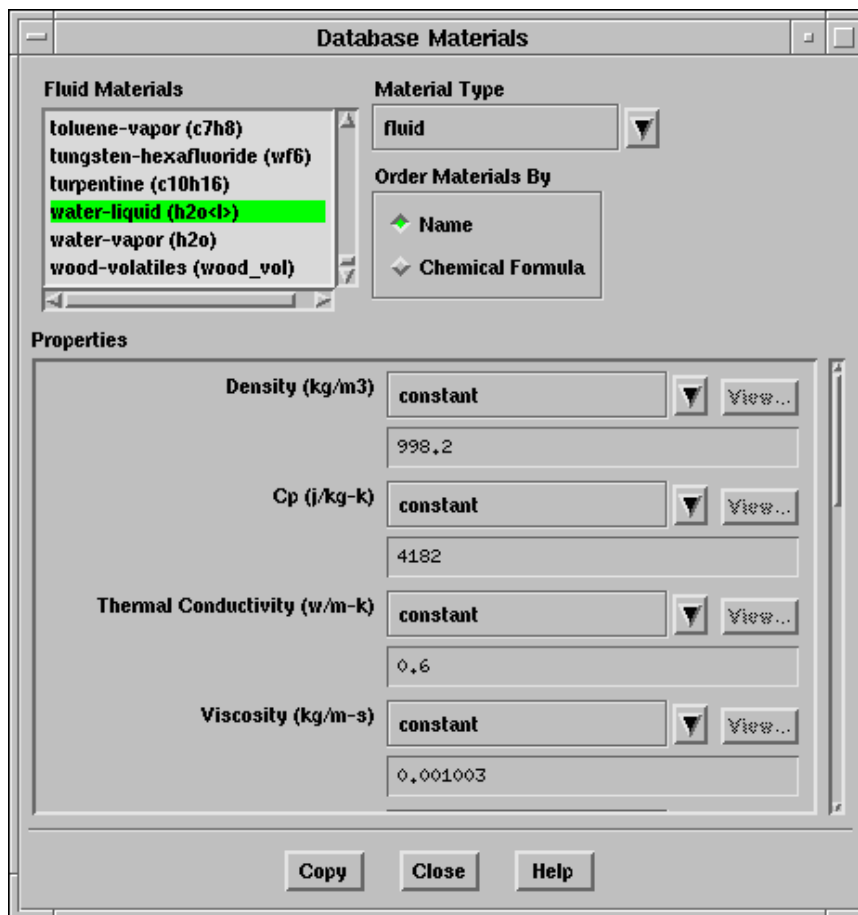
You will need to add liquid water to the list of fluid materials by copying it from the materials database.

1. Copy the properties of liquid water from the database.

Define → Materials...

- (a) Click on the Database... button.

This will open the Database Materials panel.



- (b) Scroll down the Fluid Materials list to the bottom, and select **water-liquid (h2o<l>)**.

This will display the default settings for water-liquid as shown in the panel above.

- (c) Click Copy, and Close the Database Materials panel.

The Materials panel will now display the copied information for water.

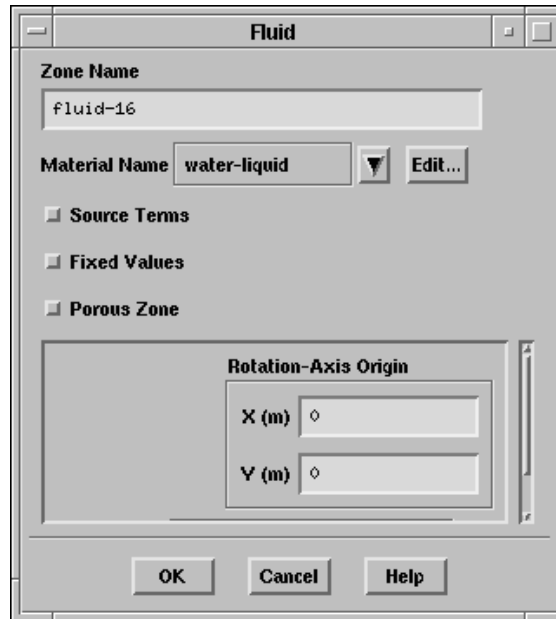
The screenshot shows a window titled "Materials" with the following fields and controls:

- Name:** water-liquid
- Material Type:** fluid
- Order Materials By:** Name (selected), Chemical Formula
- Chemical Formula:** h2o<l>
- Fluid Materials:** water-liquid (h2o<l>)
- Mixture:** (empty)
- Properties:**
 - Density (kg/m³):** constant, value: 998.2
 - Cp (j/kg-k):** constant, value: 4182
 - Thermal Conductivity (w/m-k):** constant, value: 0.6
 - Viscosity (kg/m-s):** constant, value: 0.001003
- Buttons:** Change/Create, Delete, Close, Help

Step 4: Boundary Conditions

Define → Boundary Conditions...

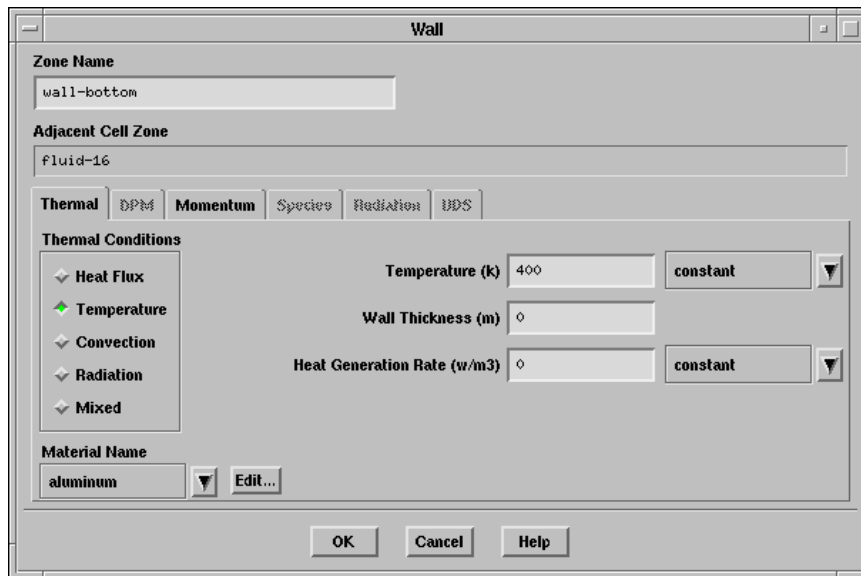
1. Set the conditions for fluid-16.



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

2. Set the boundary conditions for wall-21.

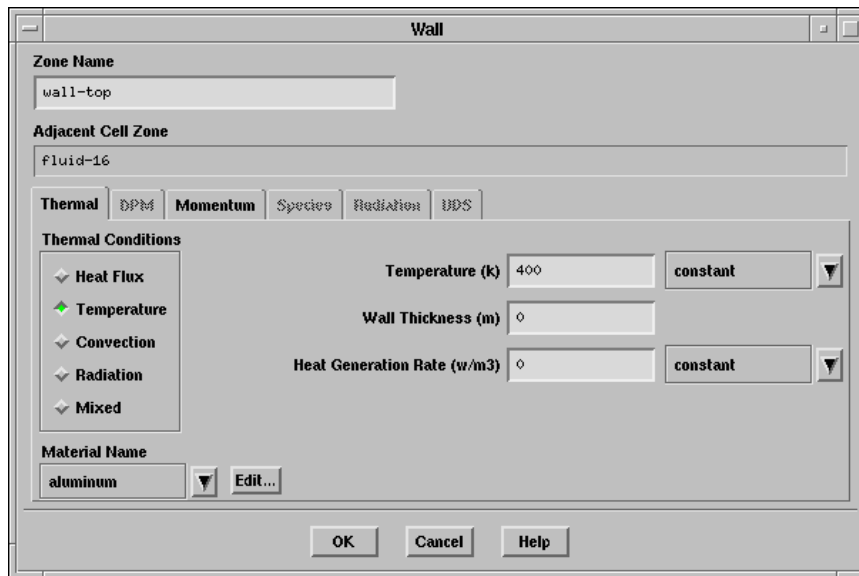
wall-21 is the bottom wall of the left tube in the periodic module shown in Figure 2.1.



- Change the Zone Name from wall-21 to wall-bottom.
- Select Temperature under Thermal Conditions.
- Change the Temperature to 400 K.

3. Set the boundary conditions for wall-3.

wall-3 is the top wall of the right tube in the periodic module shown in Figure 2.1.

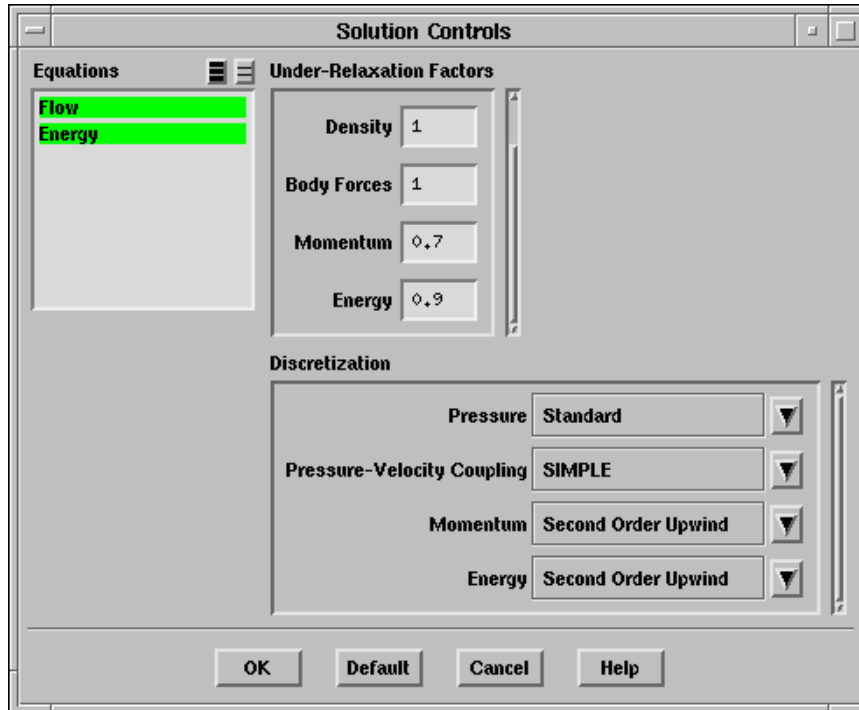


- Change the Zone Name from wall-3 to wall-top.
- Select Temperature under Thermal Conditions.
- Change the Temperature to 400 K.

Step 5: Solution

1. Set the solution parameters.

Solve → Controls → Solution...



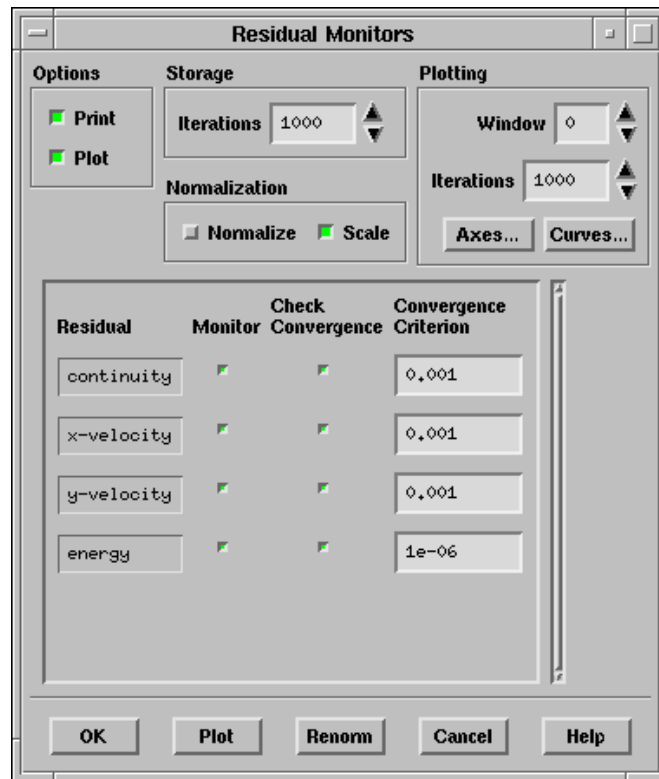
- (a) Change the Under-Relaxation Factor for Energy to 0.9.

Hint: You will need to scroll down the Under-Relaxation Factors list to see Energy.

- (b) Under Discretization, select Second Order Upwind for Momentum and Energy.

2. Enable the plotting of residuals.

Solve → Monitors → Residual...

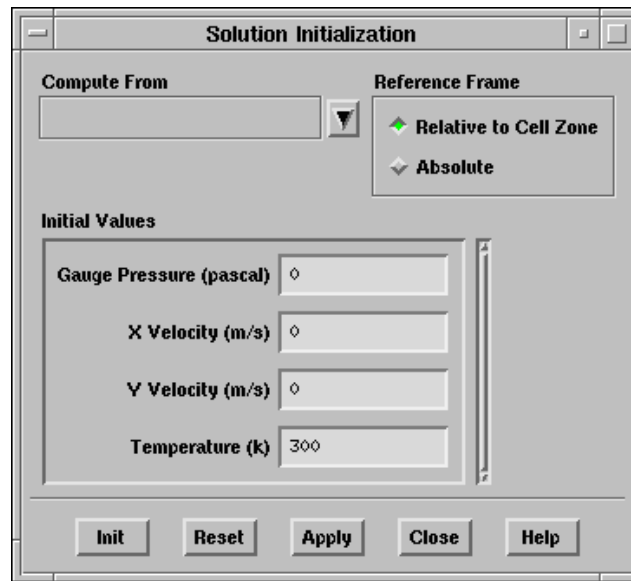


(a) Under Options, select Plot.

(b) Click the OK button.

3. Initialize the solution.

Solve → Initialize → Initialize...



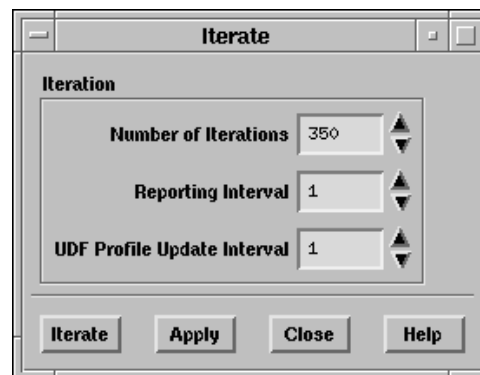
- (a) Under Initial Values, check that the value for Temperature is set to 300 K.
(b) Click Init, and Close the panel.

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

File → Write → Case...

5. Start the calculation by requesting 350 iterations.

Solve → Iterate...



- (a) Set the Number of Iterations to 350.
(b) Click Iterate.

The energy residual curve begins to flatten out after about 350 iterations. In order for the solution to converge, the relaxation factor for energy will have to be further reduced.

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, you will see an initial dip in the plot of the energy residual, resulting from a reduction in the under-relaxation factor. The solution will converge in a total of approximately 580 iterations.

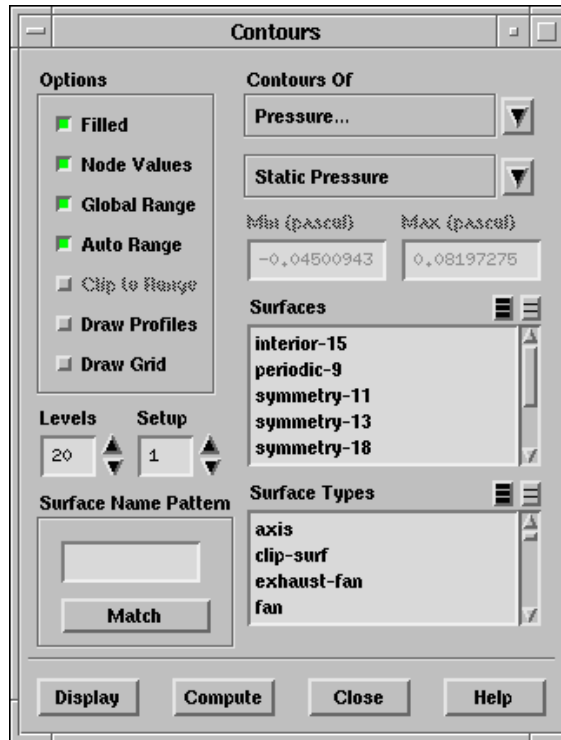
8. Save the case and data files (tubebank.cas and tubebank.dat).

File → **Write** → Case & Data...

Step 6: Postprocessing

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

Display → Contours...



- (a) Select Filled under Options.
- (b) Select Pressure... and Static Pressure in the Contours Of drop-down list.
- (c) Click Display.

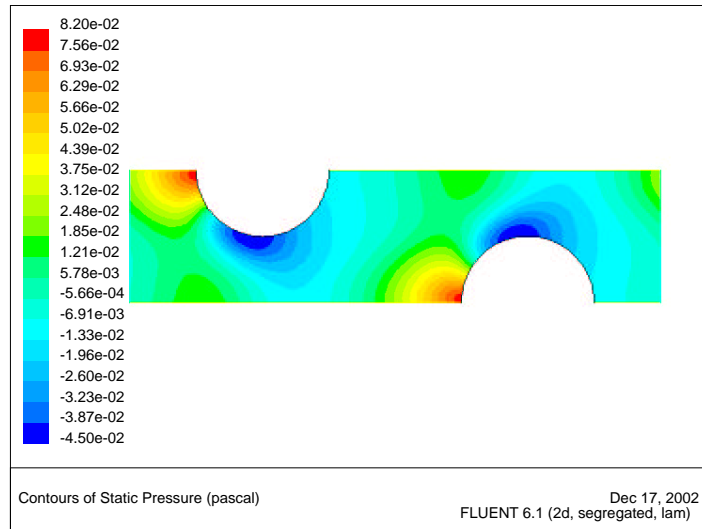
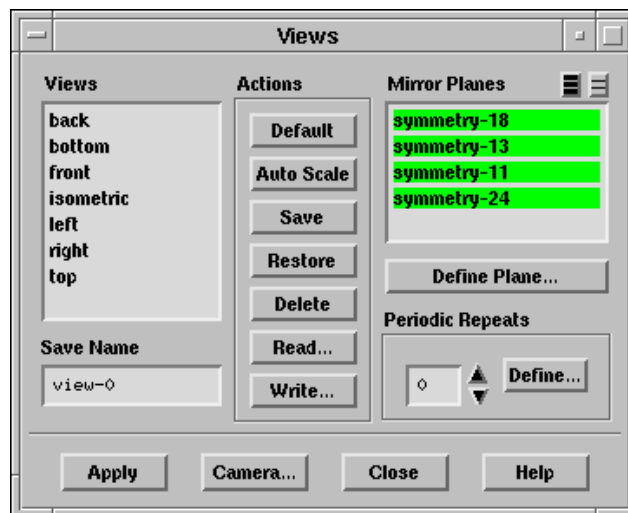


Figure 2.3: Contours of Static Pressure

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

Display → Views...



- (a) Select all of the symmetry zones by clicking the shaded icon to the right of Mirror Planes.

Note: *There are four symmetry zones in the Mirror Planes 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. 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 panel.
- (c) Using the left button of your mouse, translate the view so that it is centered in the window.

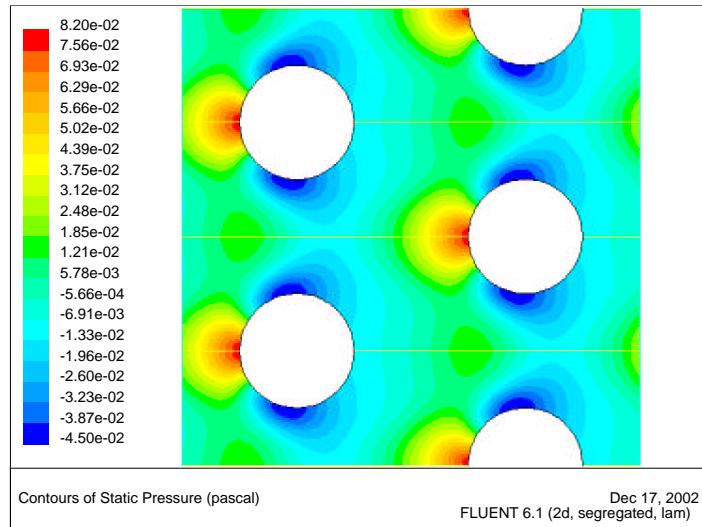
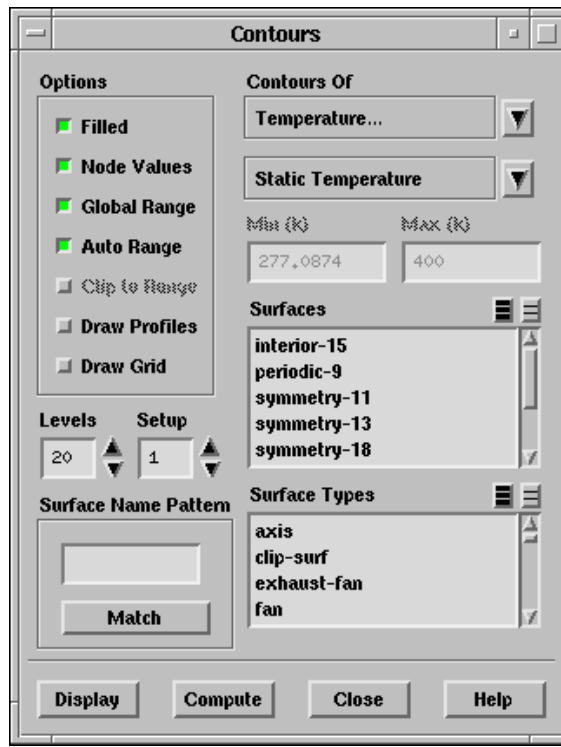


Figure 2.4: Contours of Static Pressure with Symmetry

Note: *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 inflow and outflow boundaries.*

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

Display → Contours...



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

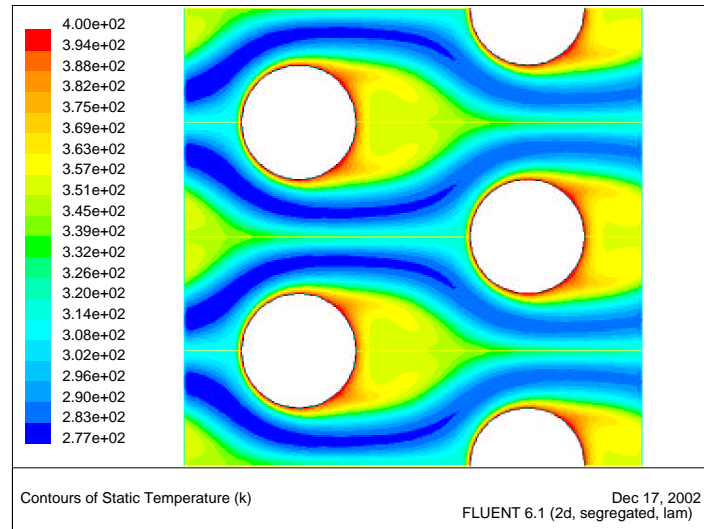
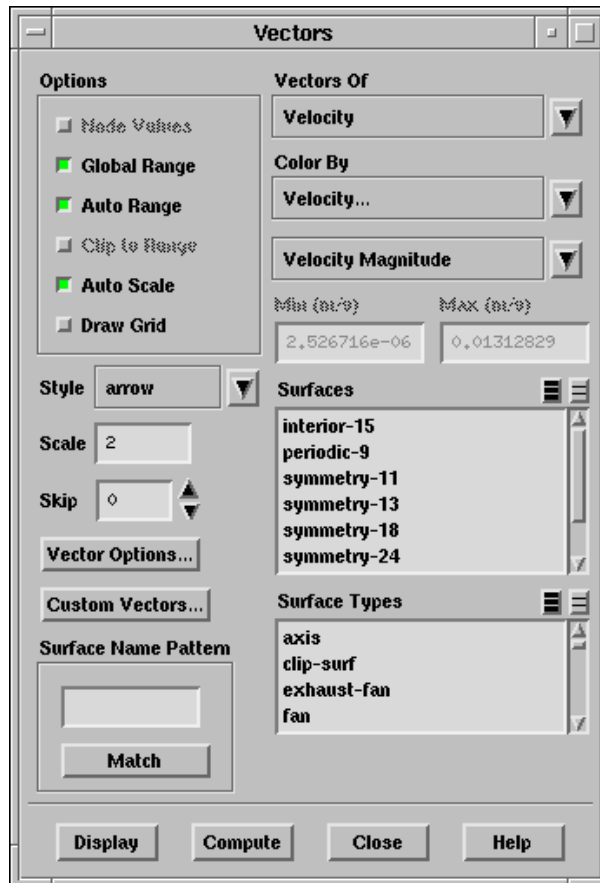


Figure 2.5: Contours of Static Temperature

The contours 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.

4. Display the velocity vectors (Figure 2.6).

Display → Vectors...



- (a) Select Velocity... and Velocity Magnitude in the Color By drop-down list.
- (b) Change the Scale to 2.

This will enlarge the vectors that are displayed, making it easier to view the flow patterns.

- (c) Click Display.
- (d) Zoom in on the upper right portion of the left tube using your middle mouse button, to get the display shown in Figure 2.6.

This zoomed-in view of the velocity vector plot clearly shows the recirculating flow behind the tube and the boundary layer development along the tube surface.

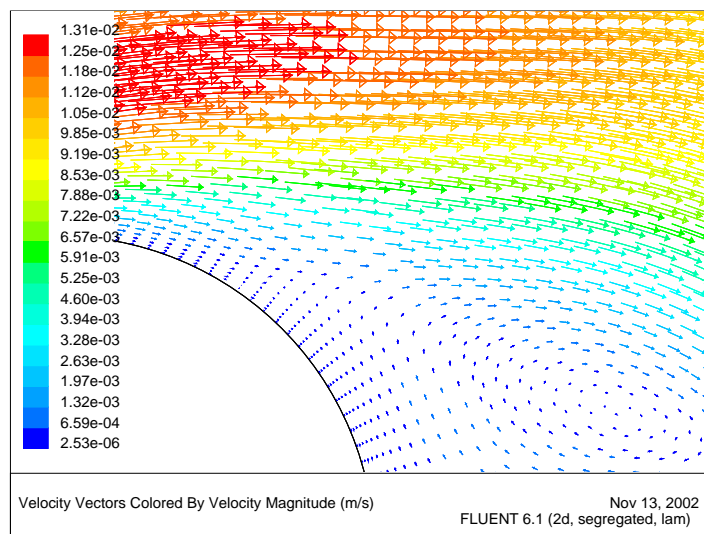


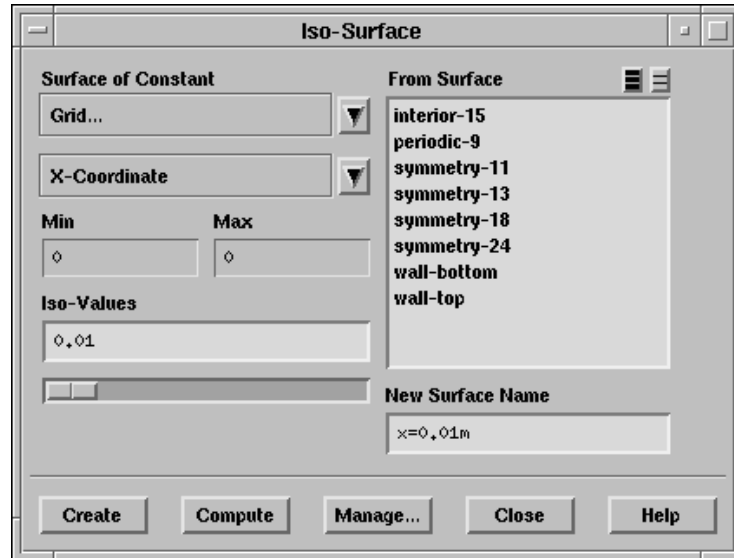
Figure 2.6: Velocity Vectors

5. Plot the temperature profiles at three cross-sections of the tube bank.

- (a) Create an isosurface on the periodic tube bank at $x = 0.01$ m (through the first tube).

You will first need to create a surface of constant x coordinate for each cross-section: $x = 0.01$, 0.02 , and 0.03 m. These isosurfaces correspond to the vertical cross-sections through the first tube, halfway between the two tubes, and through the second tube.

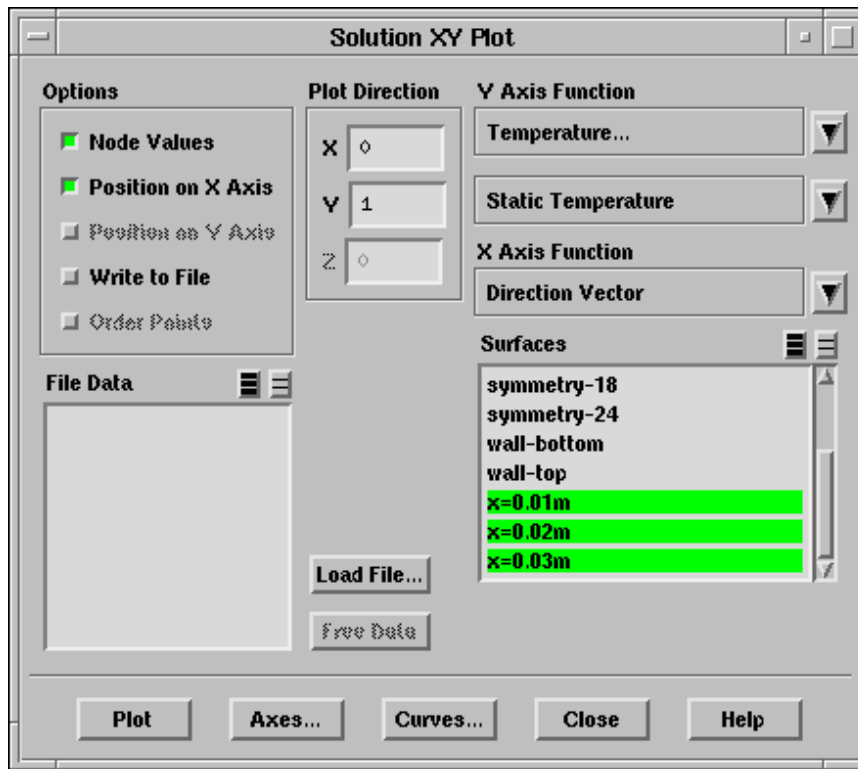
Surface → Iso-Surface...



- i. In the Surface of Constant drop-down lists, select Grid... and X-Coordinate.
 - ii. Enter $x=0.01\text{m}$ under New Surface Name.
 - iii. Enter 0.01 for Iso-Values.
 - iv. Click Create.
- (b) Follow the same procedure to create surfaces at:
- $x = 0.02$ m (halfway between the two tubes)
 - $x = 0.03$ m (through the middle of the second tube)

- (c) Create an XY plot of static temperature on the three isosurfaces.

Plot → XY Plot...

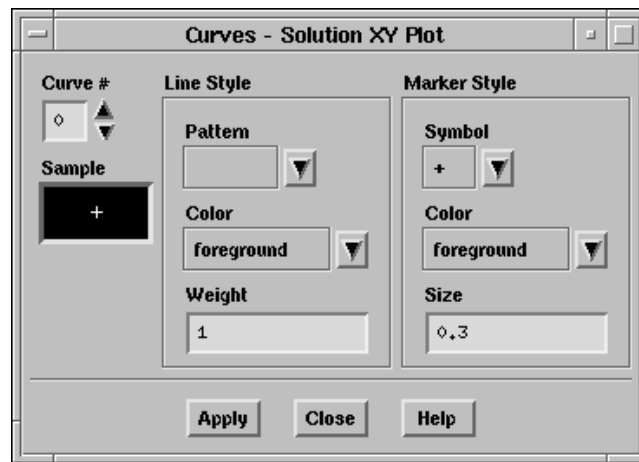


- i. Change the Plot Direction for X to 0, and the Plot Direction for Y to 1.

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

- ii. Select Temperature... and Static Temperature in the Y-Axis Function dropdown lists.
- iii. Scroll down the Surfaces list and select $x=0.01\text{m}$, $x=0.02\text{m}$, and $x=0.03\text{m}$.
- iv. Click Curves... to define different styles for the different plot curves.

This will open the Curves - Solution XY Plot panel.



- v. Select + in the Symbol drop-down list.
- vi. Click Apply.

This assigns the + symbol to the $x = 0.01$ m curve.
- vii. Increase the Curve # to 1 to define the style for the $x = 0.02$ m curve.
- viii. Select x in the Symbol drop-down list.
- ix. Change the Size to 0.5.
- x. Click Apply, and Close the panel.

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

- xi. In the Solution XY Plot panel, click Plot.

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 Periodicity Conditions panel makes it easy to run this type of model over 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.

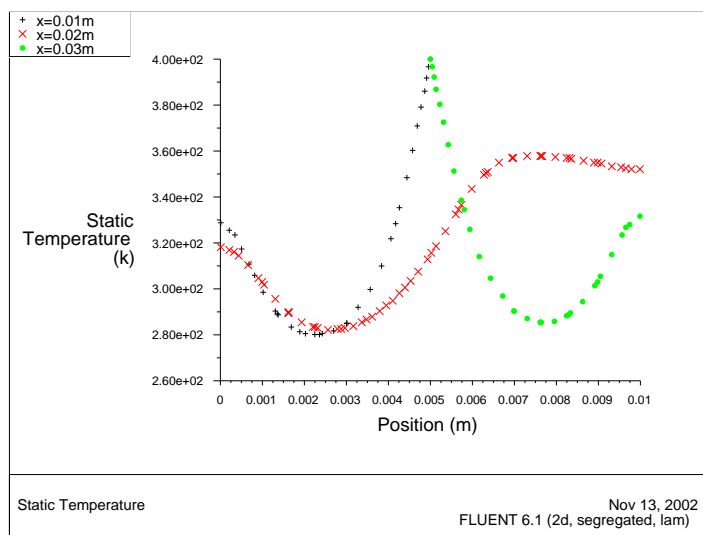


Figure 2.7: Static Temperature at $x=0.01, 0.02,$ and 0.03 m