Advanced Turbulence Tutorial 2. Turbulent Flow in a Compact Heat Exchanger

Purpose

The purpose of this tutorial is to evaluate the pressure drop and heat transfer characteristics for liquid ammonia flowing through the specified interrupted fin heat exchanger at a given mass flux. Taking into account all of the factors that must be considered in turbulence modeling, as well as your time and hardware constraints, estimate the average drag coefficient and average Stanton number. At the end of the session, we will compare the computed values with the experimental data.

Prerequisites

This tutorial assumes that you have successfully completed the introductory training course for the FLUENT solver and the GAMBIT preprocessor.

Problem Description

The compact heat exchanger for this problem utilizes interrupted and staggered fins that enhance the heat transfer relative to a continuous fin heat exchanger by promoting turbulent mixing in the wake region behind each fin and taking advantage of the relatively high heat transfer coefficients as the boundary layers continuously re-form on the interrupted fins. This particular heat exchanger surface is known as Surface #11.32-0.737-SR.

A sketch of the geometry, with dimensions in mm, is shown in Figure 2.1. The geometry contains symmetry boundary conditions at the top and bottom planes.

Assume that pressurized liquid ammonia at 240 K is to be heated in the heat exchanger and that the fin walls are at a constant temperature of 350 K. The mass flux of liquid ammonia through the heat exchanger is 303.14 kg/s-m^2 and the hydraulic diameter is 3.51 mm. Considering the viscosity of liquid ammonia, 0.000152 kg/m-s, the Reynolds number based on hydraulic diameter is 7000 and falls in the weakly turbulent region (i.e., low Reynolds number turbulent regime).

Periodic boundaries will be used at the beginning and ending planes of the domain and a periodic mass flow rate of 1.385 kg/s will be used.
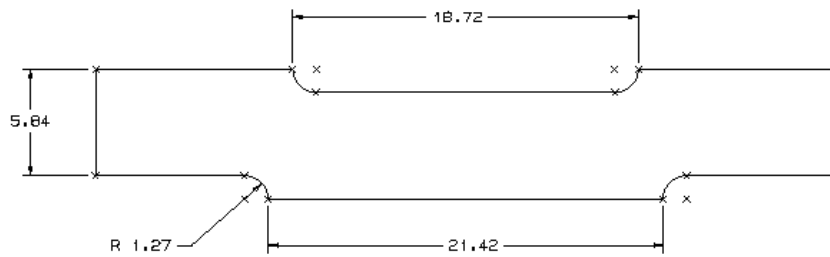
Figure 2.1: The Geometry

The performance of this heat exchanger will be expressed in terms of a drag coefficient c_D and a Stanton number St. c_D is based on the drag force D, the fin surface area A, the density of the fluid ρ, and the mean velocity based on the minimum cross-sectional flow area U:

$$c_D = \frac{D}{\frac{1}{2}A\rho U^2} \tag{2.1}$$

where c_D represents both the viscous and pressure drag components.

The Stanton number depends upon the average heat transfer coefficient and the thermal capacitance of the flow:

$$St = \frac{h}{\rho U c_p} \tag{2.2}$$

Preparation

1. Copy the IGES file containing the geometry for this problem, `htx.igs`, to your working directory.

2. Develop a strategy for modeling this turbulent flow. The strategy will include decisions on which turbulence model to use, what type of near-wall treatment to use, what type of grid to use (quad, tri, or hybrid), and how to distribute the nodes.

   Hint: *Be sure to estimate the friction velocity u_τ so that you will be able to specify the proper node spacing in the direction normal to the fin walls.*

Setup and Solution

Step 1: Grid

1. Read the mesh file htx.msh.

2. If you used another preprocessor to generate your mesh, create the periodic zones.

   If you created your mesh using GAMBIT*, then you have already defined the periodic zones. If you used another preprocessor, however, you may need to perform this step here in* FLUENT*. Before you turn on the models, you will need to change the left and right walls to periodic zones. Note that references to zone ID numbers below may differ from your case, since your zones may be numbered differently.*

   You will use the following TUI command for the purpose

   grid ⟶ modify-zones ⟶make-periodic

```
/grid/modify-zones> mp
Periodic zone [()] wall-3
Shadow zone [()] wall-4
Rotational periodic? (if no, translational) [yes] no
Create periodic zones? [yes] yes

  computed translation deltas: 0.401400 0.000000
  all 48 faces matched for zones 3 and 4.

  zones 4 deleted

  created periodic zones.
```

3. Check, and scale the grid from microns to meters.
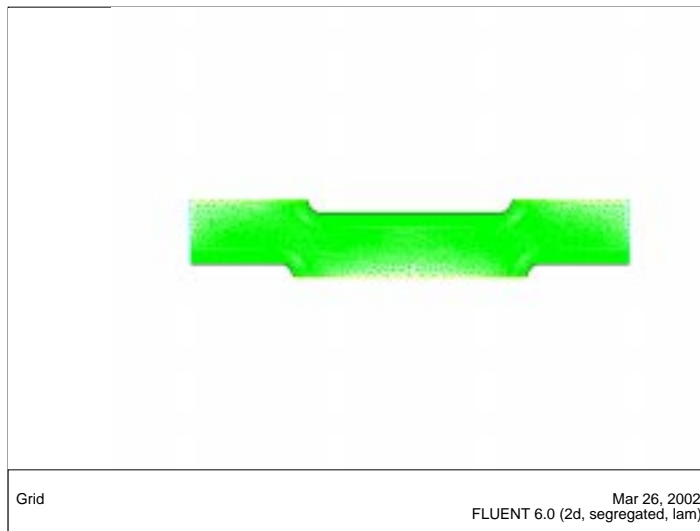
4. Display the grid.

Figure 2.2: Graphics Display of the Grid

Step 2: Models

1. Retain the default solver settings.

2. Define k-epsilon (2 eqn) turbulence model.

   (a) Select RNG for k-epsilon Model and Enhanced Wall Treatment for Near-Wall Treatment.

   (b) Retain the default values for the other parameters.

3. Turn on the Energy Equation.

Step 3: Materials

1. Copy ammonia-liquid (nh3<l>) from the database.

Step 4: Operating Conditions

1. Retain the default operating conditions.

Step 5: Boundary and Periodic Conditions

1. Set the boundary conditions for the fluid zone.

   (a) Select ammonia-liquid for Material Name and retain the default values for the other parameters.

2. Set the boundary conditions for the wall zones.

   (a) Set the Temperature for wall-bottom and wall-top zone to 350.

3. Specify the periodic flow conditions.

   (a) Select Specify Mass Flow for Type and set the Mass Flow Rate, and Upstream Bulk Temperature to 1.385 kg/sec, and 240 K respectively.

Step 6: Solution

1. Initialize the flow field.

   (a) Set the X Velocity to 0.5 m/s, the Temperature to 240 K, and the Turbulence Dissipation Rate to 1e5 m^2/s^3.

2. Enable the plotting of residuals during the calculation.

3. Enable the monitoring of the surface Stanton number on the longest wall (wall-top).

   (a) Specify a report of Surface Stanton Number for wall-top.
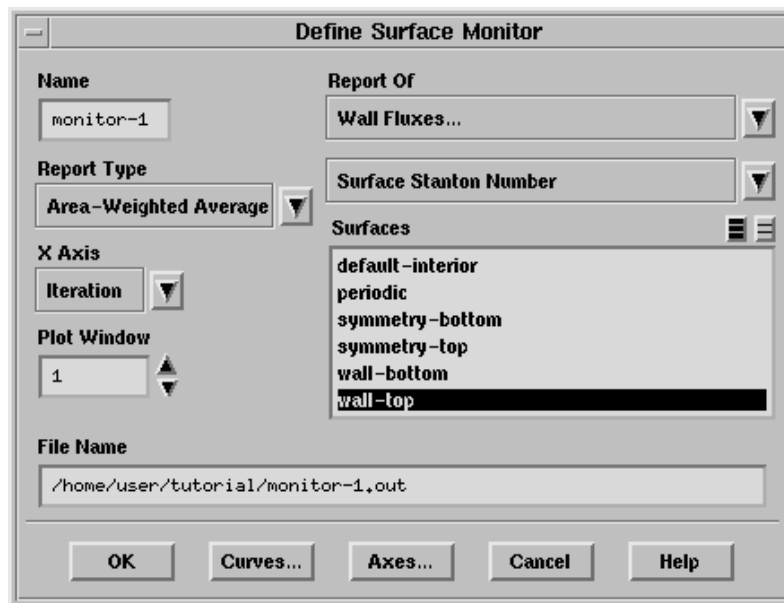
4. Enable the monitoring of the drag coefficient on the longest wall (wall-top).

   (a) Specify a report of the Drag Coefficient for wall-drop in the X direction.

5. Set the appropriate reference values for use in reporting the drag coefficient and surface Stanton number.
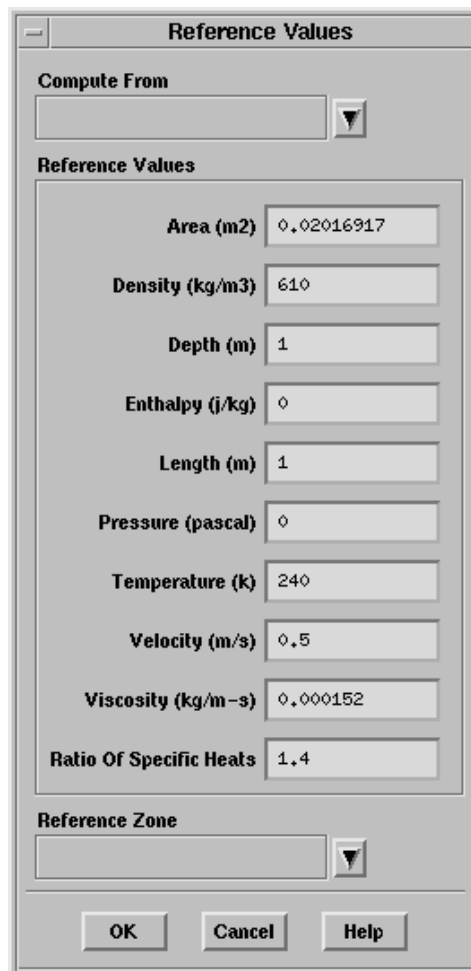
Note: *The reference* Area *should be equal to the area of* wall-top, *which can be calculated using the* Surface Integrals *panel.*

6. Set the solution parameters.

    (a) Set the under-relaxation factor for Energy to 0.95.

    (b) Set the Discretization for Momentum, Turbulence Kinetic Energy, Turbulence Dissipation Rate and Energy to Second Order Upwind.

    (c) Deselect Energy in the list of equations.

    You will begin the calculation by solving only the flow and turbulence equations. Once this solution has converged, you will turn off the flow and turbulence equations and solve only the energy equation. Finally, you will solve all equations together.

7. Calculate the flow and turbulence solution until convergence.
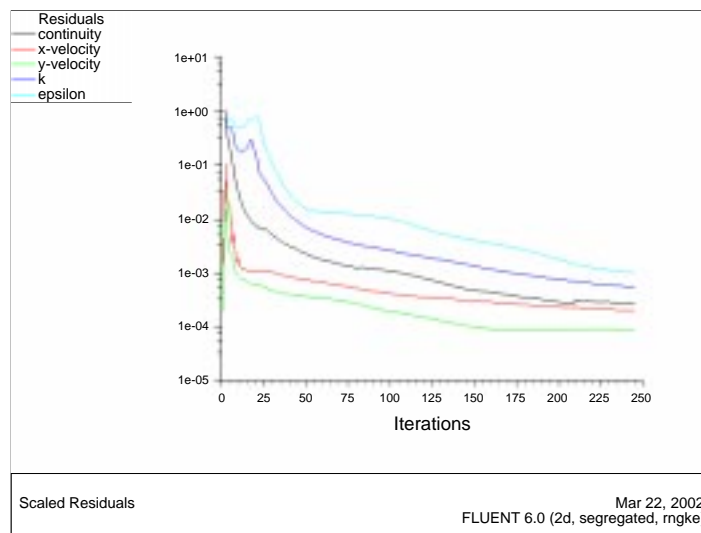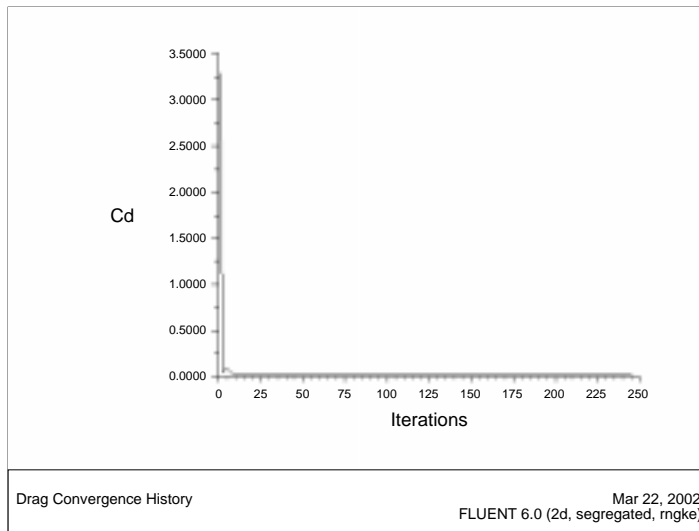

Figure 2.3: Scaled Residuals

Figure 2.4: Drag Convergence History

8. Check the value of y^+ and adapt if necessary.

    Adapt ⟶Y+/Y*...

    You will use grid adaption to refine the mesh if the y^+ value is improper:

    (a) If you use wall functions, the y^+ value should be between 50 and 500.

    (b) If you use the two-layer zonal model, y^+ at the adjacent cell should be in the order of $y^+ = 1$.

9. Turn off solution of the flow and turbulence equations, and turn on solution of only the energy equation.

10. Calculate the energy solution until convergence and then save the case and data files.
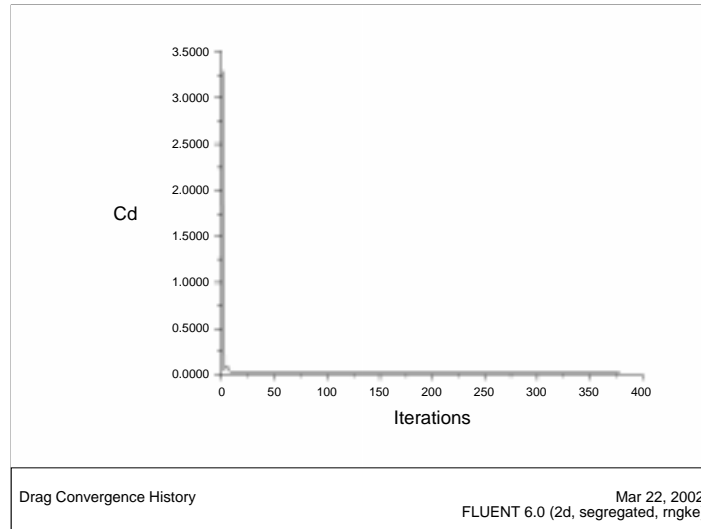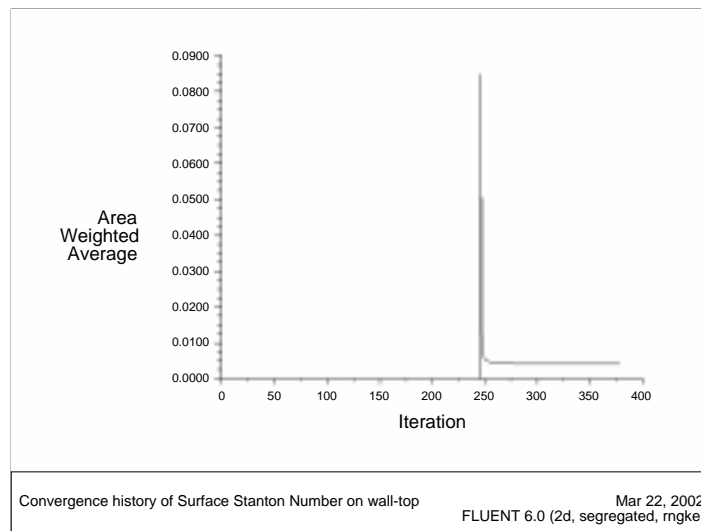
Figure 2.5: Drag Convergence History

Figure 2.6: Convergence History of Surface Stanton Number on wall-top

11. Turn on solution of all equations, solve until convergence, and save the final case and data files.

Step 7: Postprocessing

1. Enable reflection of the display across the symmetry planes.

   (a) Select all planes under Mirror Planes, click Apply, and close the panel.

   The display will be updated, as shown in Figure 2.7. The pressure contours show a region of high pressure near the stagnation point at the front of the fin, and regions of relatively low static pressure where the flow accelerates around the curved front portion of the fin.

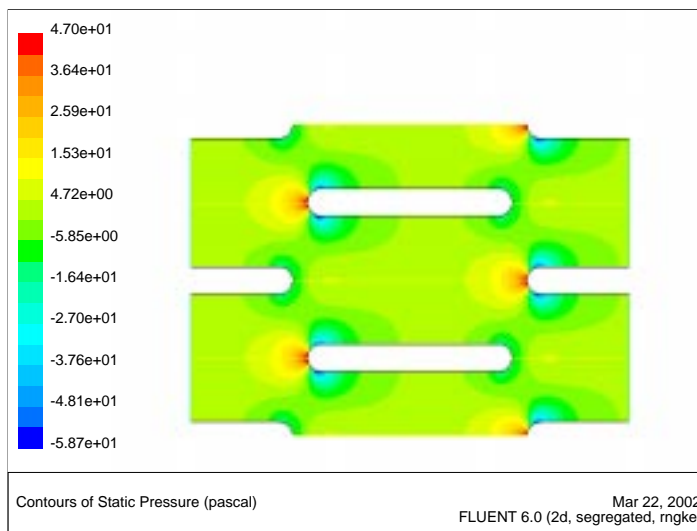2. Display filled contours of static pressure.

Figure 2.7: Contours of Static Pressure

3. Display filled contours of velocity magnitude (Figure 2.8).

   The velocity contours show the stagnation and flow acceleration regions at the front of the fin, the growth of a thin boundary layer along the length of the fin, and a wake region immediately downstream of the fin. The wake region promotes turbulent mixing and enhances overall heat transfer. Note that for this flow rate, the wake region settles out before the next fin is encountered, and ensures that a new boundary layer with a relatively high heat transfer coefficient will grow on subsequent fins.
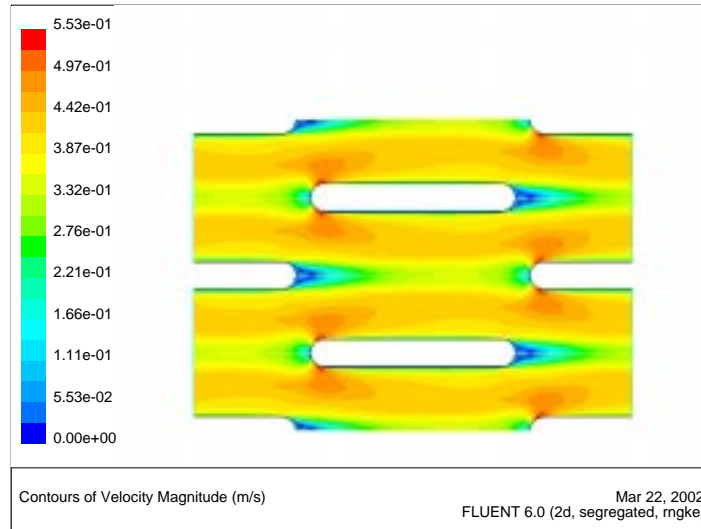
Figure 2.8: Contours of Velocity Magnitude
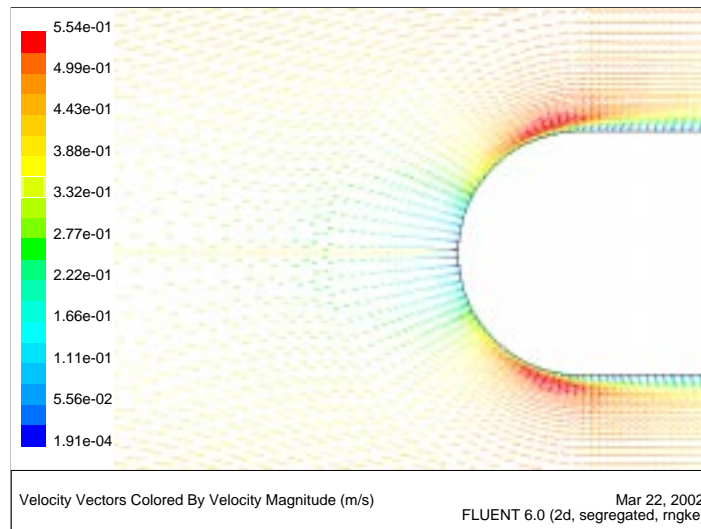
4. Display velocity vectors.

Figure 2.9: Velocity Vectors at the Front Showing the Stagnation Point Region
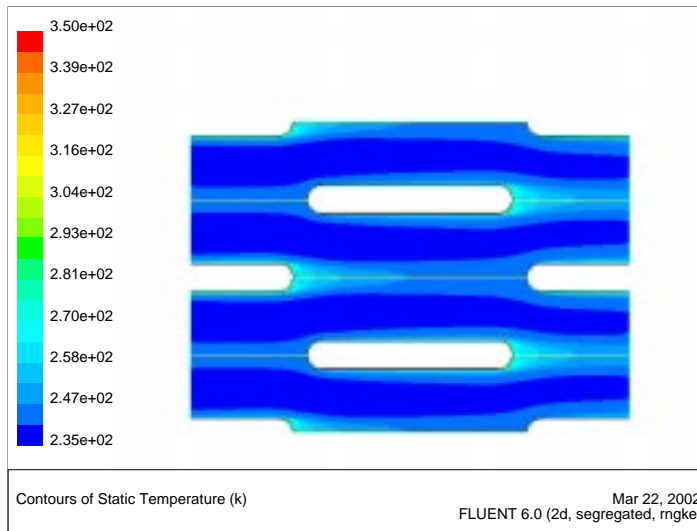
5. Display filled contours of temperature.

Figure 2.10: Contours of Temperature

Summary

For liquid ammonia at $Re = 7000$ over heat exchanger surface #11.32-0.737-SR, the experimental data in terms of pressure drop performance is $c_D = 0.019$ and in terms of heat transfer performance is $St = 3.87 \times 10^{-3}$.

Now that you have successfully completed the tutorial, you can review this work by checking the following:

1. Which type of grid did you use?

2. How many cells were required?

3. How accurate was the calculated drag coefficient?

4. How accurate was the calculated Stanton number?

5. What would you have done differently?