
Tutorial: Turbulent Flow in a Compact Heat Exchanger

Introduction

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. Estimate the average drag coefficient and average Stanton number considering all factors for turbulence modeling, as well as your time and hardware constraints. At the end of the session, you can compare the computed values with the experimental data.

Prerequisites

This tutorial is written with the assumption that you have completed Tutorial 1 from ANSYS FLUENT 13.0 Tutorial Guide, and that you are familiar with the ANSYS FLUENT navigation pane and menu structure. Some steps in the setup and solution procedure will not be shown explicitly.

Problem Description

A sketch of the geometry, with dimensions in mm, is shown in Figure 1. The geometry contains symmetry boundary conditions at the top and bottom planes.

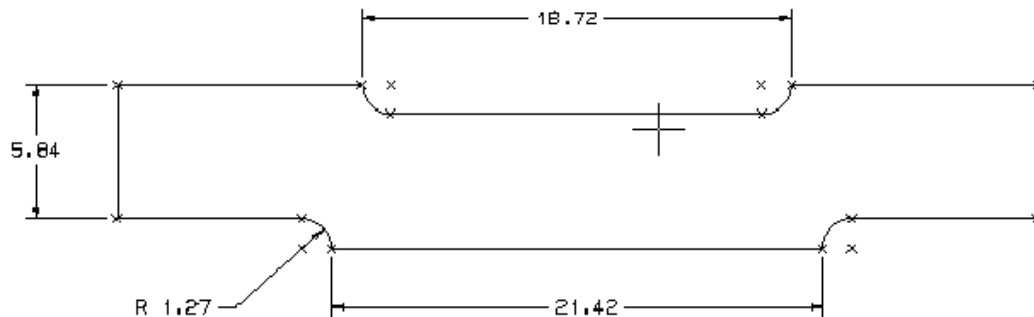


Figure 1: Heat Exchanger Geometry

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.

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-m² 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.

The performance of this heat exchanger will be expressed in terms of a drag coefficient, c_D and a Stanton number, St .

$$c_D = \frac{D}{\frac{1}{2}A\rho U^2} \quad (1)$$

where,

c_D = drag coefficient (represents both the viscous and pressure drag components)

D = the drag force

A = the fin surface area

ρ = the density of the fluid

U = the mean velocity based on the minimum cross-sectional flow area

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

$$St = \frac{h}{\rho U c_p} \quad (2)$$

Preparation

1. Copy the mesh file (htx.mesh.gz), to the working folder.
2. Use FLUENT Launcher to start the 2D version of ANSYS FLUENT.

For more information about FLUENT Launcher see Section 1.1.2 Starting ANSYS FLUENT Using FLUENT Launcher in the ANSYS FLUENT 13.0 User's Guide.

Note: *The Display Options are enabled by default. Therefore, after you read in the mesh, it will be displayed in the embedded graphics window.*

3. 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 mesh to use (quad, tri, or hybrid), and how to distribute the nodes.

Note: *Ensure to estimate the friction velocity, u_t 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: Mesh

1. Read the mesh file, htx.msh.

File → **Read** → Mesh...

As the mesh file is read, ANSYS FLUENT reports the progress in the console.

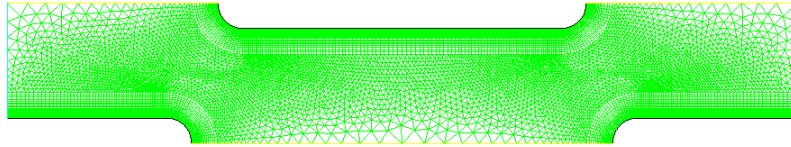


Figure 2: Graphics Display of the Mesh

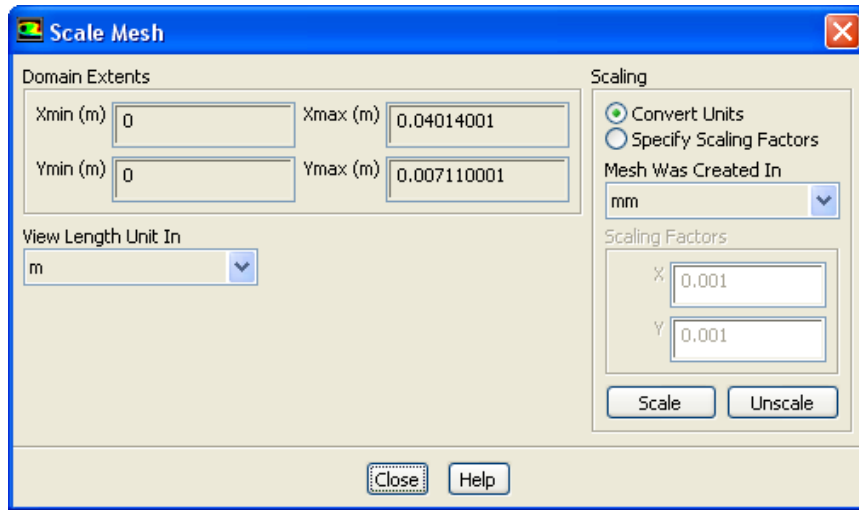
Step 2: General Settings

1. Check the mesh.

 **General** → **Check**

2. Scale the mesh from microns to meters.

 **General** → **Scale...**



- (a) Select mm from the Mesh Was Created In drop-down list.
 - (b) Click Scale twice.
 - (c) Close the Scale Mesh dialog box.
3. Retain the default solver settings.

Step 3: Models

1. Enable the Energy Equation.



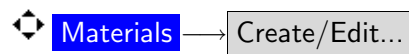
2. Define k-epsilon (2 eqn) turbulence model.



- (a) Select the k-epsilon model.
- (b) Select RNG from k-epsilon Model group box.
- (c) Select Enhanced Wall Treatment from Near-Wall Treatment group box.
- (d) Retain the default values for the other parameters and click OK.

Step 4: Materials

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

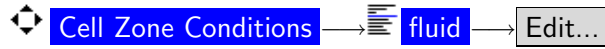


1. Click FLUENT Database....
 - (a) Select ammonia-liquid (nh3<|>) from the list of FLUENT Fluid Materials.
 - (b) Click Copy.
 - (c) Click Close to close the FLUENT Database Materials dialog box.

2. Close the Create/Edit Materials dialog box.

Step 5: Cell Zone Conditions

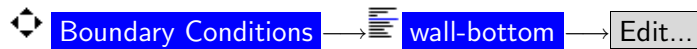
Set the cell zone conditions for the fluid zone.



1. Select ammonia-liquid for Material Name and retain the default values for the other parameters.
2. Click OK to close the Fluid dialog box.

Step 6: Boundary Conditions

1. Set the boundary conditions for the wall zones.

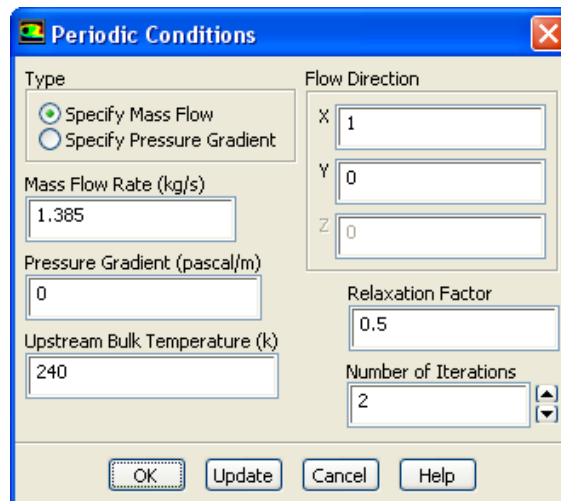
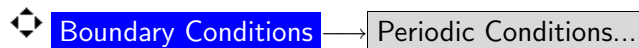


- (a) Click on the Thermal tab.
- (b) Select Temperature from the Thermal Conditions group box.
- (c) Set the Temperature to 350.
- (d) Click OK to close the Wall dialog box.

2. Similarly set the Temperature to 350 for wall-top.



3. Specify the periodic flow conditions.



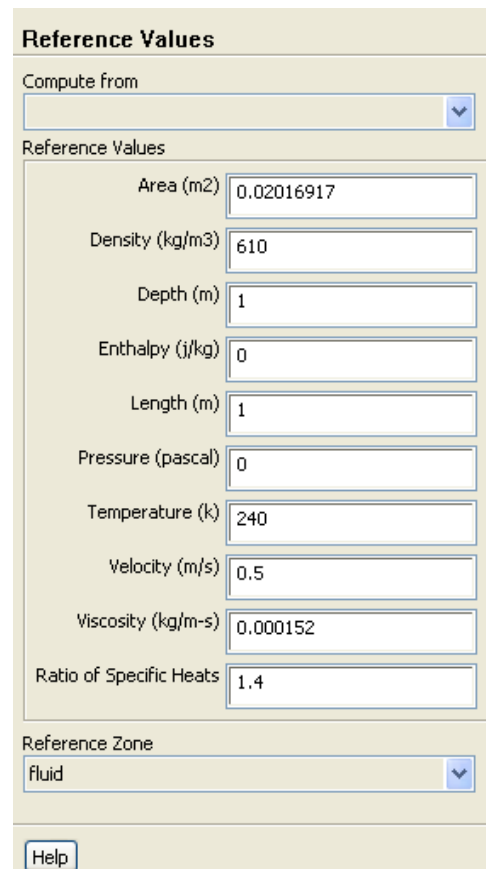
- (a) Select Specify Mass Flow in Type group box.

- (b) Set Mass Flow Rate to 1.385.
- (c) Set Upstream Bulk Temperature to 240.
- (d) Click OK to close the Periodic Conditions dialog box.

Step 7: Reference Values

Set the appropriate reference values for use in reporting the drag coefficient and surface Stanton number.

Reference Values



The image shows a software dialog box titled "Reference Values". It contains several input fields for physical properties and a drop-down menu for the reference zone. The values entered in the fields are: Area (m2) = 0.02016917, Density (kg/m3) = 610, Depth (m) = 1, Enthalpy (j/kg) = 0, Length (m) = 1, Pressure (pascal) = 0, Temperature (k) = 240, Velocity (m/s) = 0.5, Viscosity (kg/m-s) = 0.000152, and Ratio of Specific Heats = 1.4. The Reference Zone is set to "fluid". A "Help" button is located at the bottom left of the dialog box.

Property	Value
Area (m2)	0.02016917
Density (kg/m3)	610
Depth (m)	1
Enthalpy (j/kg)	0
Length (m)	1
Pressure (pascal)	0
Temperature (k)	240
Velocity (m/s)	0.5
Viscosity (kg/m-s)	0.000152
Ratio of Specific Heats	1.4

Reference Zone: fluid

Help

1. Enter 0.02016917 for Area.
2. Enter 610 for Density.
3. Enter 240 for Temperature.
4. Enter 0.5 for Velocity.
5. Enter 0.000152 for Viscosity.
6. Select fluid from the Reference Zone drop-down list.

Note: The reference Area should be equal to the area of wall-top, which can be calculated using the Surface Integrals panel.

Step 8: Solution

1. Set the solution parameters.

◆ **Solution Methods**

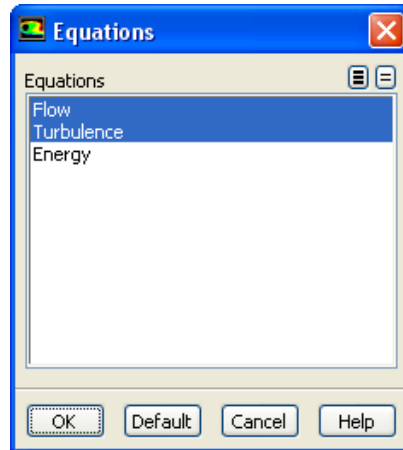
- (a) Select Second Order Upwind from the drop-down list of Momentum, Turbulence Kinetic Energy, Turbulence Dissipation Rate and Energy.

2. Set the Under-Relaxation Factors

◆ **Solution Controls**

- (a) Set Energy to 0.95.
- (b) Deselect Energy in the list of equations.

◆ **Solution Controls** → **Equations...**



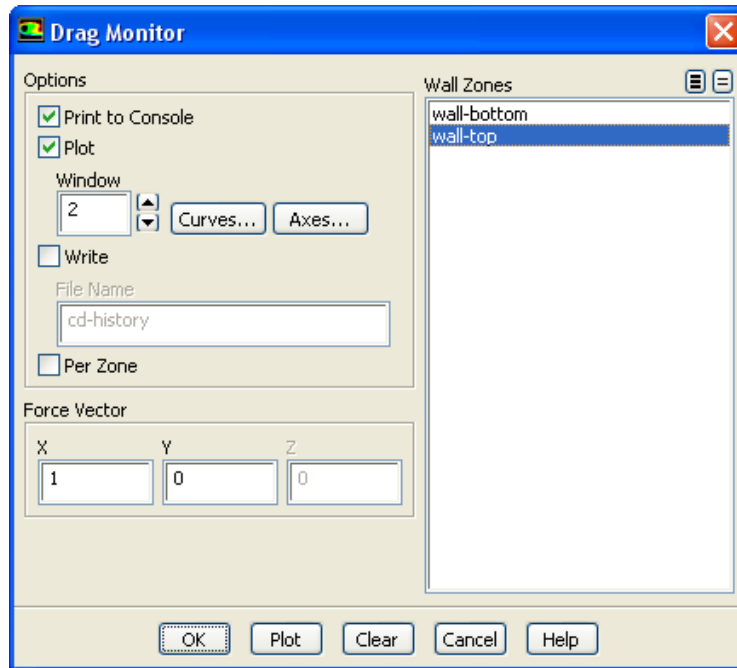
You will begin the calculation by solving only the flow and turbulence equations. Once this solution has converged you will disable the flow and turbulence equations and solve only the energy equation. Finally, you will solve all equations together.

3. Enable the plotting of residuals during the calculation.

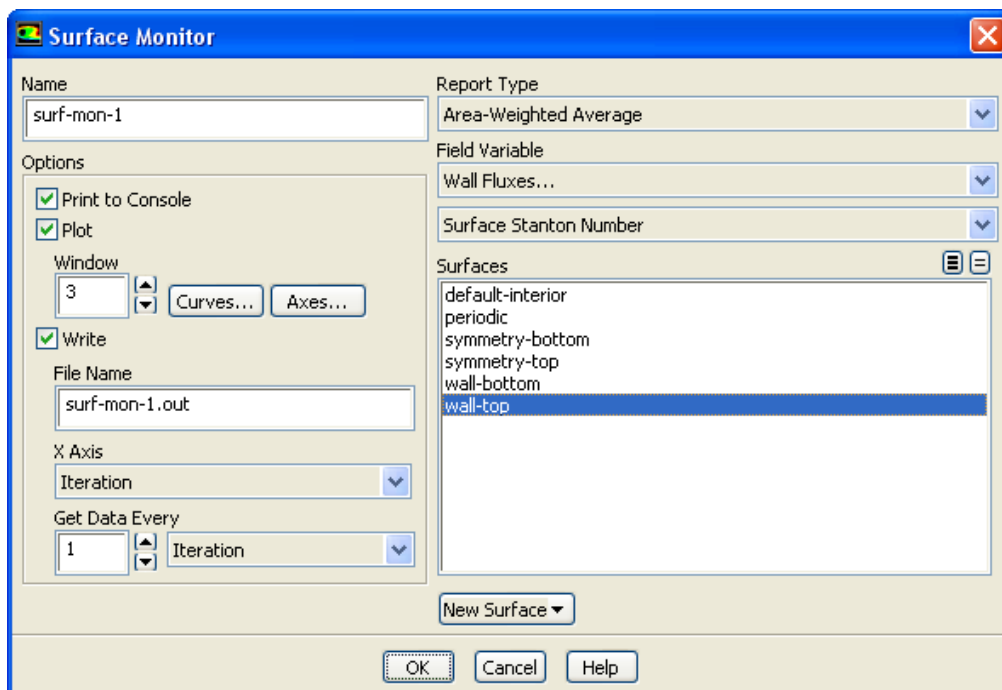
◆ **Monitors** → **Residuals** → **Edit...**

4. Enable the monitoring of the drag coefficient on the longest wall (wall-top).

◆ **Monitors** → **Drag** → **Edit...**



- (a) Enable Print to Console and Plot.
 - (b) Select wall-top from the Wall Zones list.
 - (c) Click OK to close the Drag Monitor dialog box.
5. Enable the monitoring of the surface Stanton number on the longest wall (wall-top).
- ◆ **Monitors** (Surface Monitors) → **Create...**



- (a) Enable Plot and Write.
- (b) Select Area-Weighted Average from the Report Type drop-down list.
- (c) Select Wall Fluxes... and Surface Stanton Number from the Field Variable drop-down lists.
- (d) Select wall-top from the list of Surfaces.
- (e) Click OK to close the Surface Monitor dialog box.

6. Initialize the flow field.

◆ **Solution Initialization**

- (a) Enter 0.5 for X Velocity.
- (b) Enter $1e5$ for Turbulence Dissipation Rate.
- (c) Enter 240 for Temperature.
- (d) Click Initialize.

7. Save the case and data files(htx-setup.cas.gz and htx-setup.dat.gz).

File → **Write** → Case & Data...

8. Calculate the flow and turbulence solution until convergence.

◆ **Run Calculation**

- (a) Enter 260 for Number of Iterations.
- (b) Click Calculate.

Solution will converge in approximately 250 iterations.

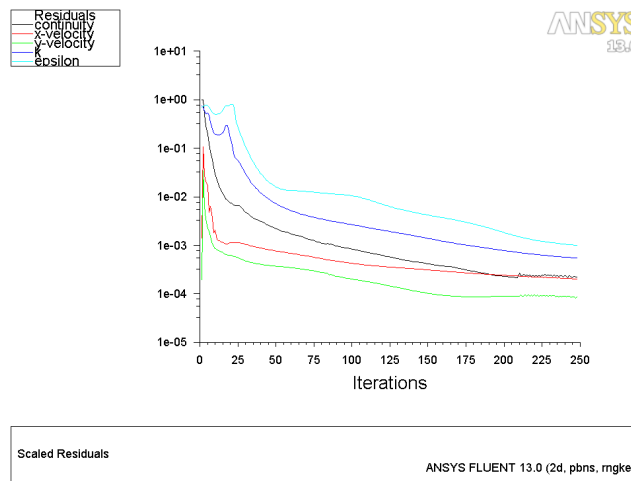


Figure 3: Scaled Residuals

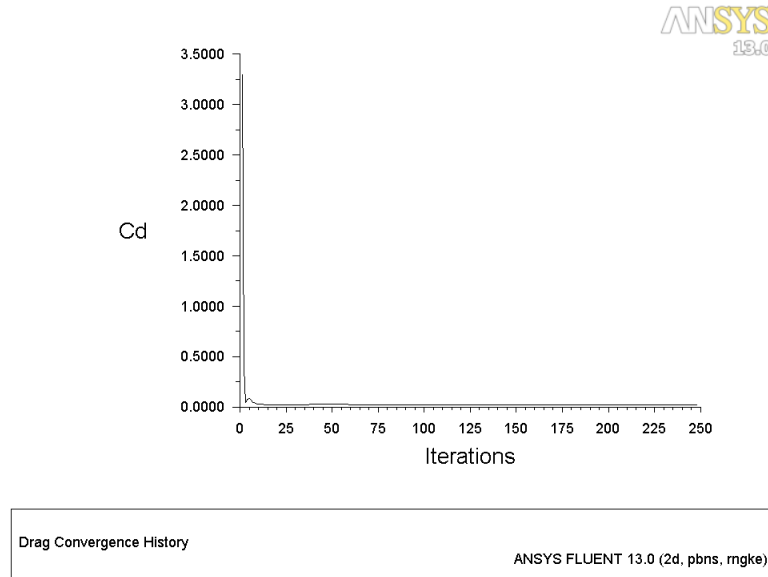


Figure 4: Drag Convergence History

9. Check the value of Y_{plus} and adapt if necessary.

Adapt → $Y_{plus}/Y_{star}...$

You will use mesh adaption to refine the mesh if the Y_{plus} value is improper:

- (a) If you use wall functions, the Y_{plus} value should be between 50 and 500.
- (b) If you use the two-layer zonal model, Y_{plus} at the adjacent cell should be in the order of $Y_{plus} = 1$.

10. Disable the flow and turbulence equations, and enable the energy equation.

 **Solution Controls** → **Equations...**

11. Calculate the energy solution.

 **Run Calculation** → **Calculate**

Solution converges in approximately 150 iterations.

12. Save the case and data files(htx-energy.cas.gz and htx-energy.dat.gz).

File → **Write** → Case & Data...

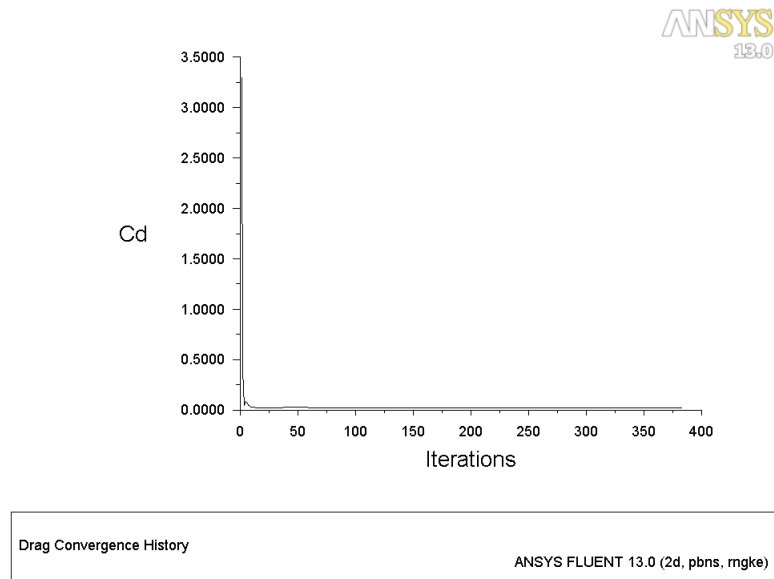


Figure 5: Drag Convergence History After Enabling Energy

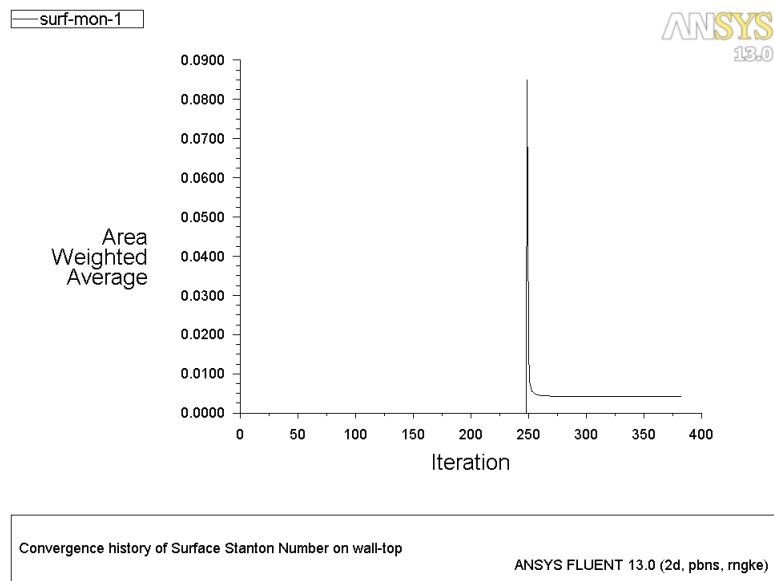


Figure 6: Convergence History of Surface Stanton Number on wall-top

13. Enable solution of all equations, solve until convergence.

Solution converges in approximately 2 iterations.

14. Save the case and data files(htx-final.cas.gz and htx-final.dat.gz).

File → **Write** → Case & Data...

Step 9: Postprocessing

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

Graphics and Animations → **Views...**

(a) Select the planes, symmetry-bottom and symmetry-top under Mirror Planes.

(b) Click Apply and close the Views dialog box.

2. Display filled contours of static pressure (Figure 7).

Graphics and Animations → **Contours** → **Set Up...**

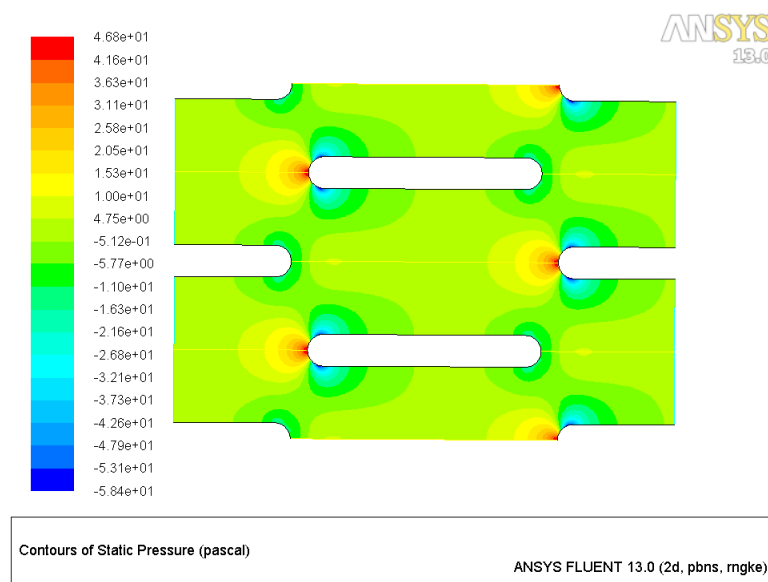


Figure 7: Contours of Static Pressure

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.

3. Display filled contours of velocity magnitude (Figure 8).

Graphics and Animations → **Contours** → **Set Up...**

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.

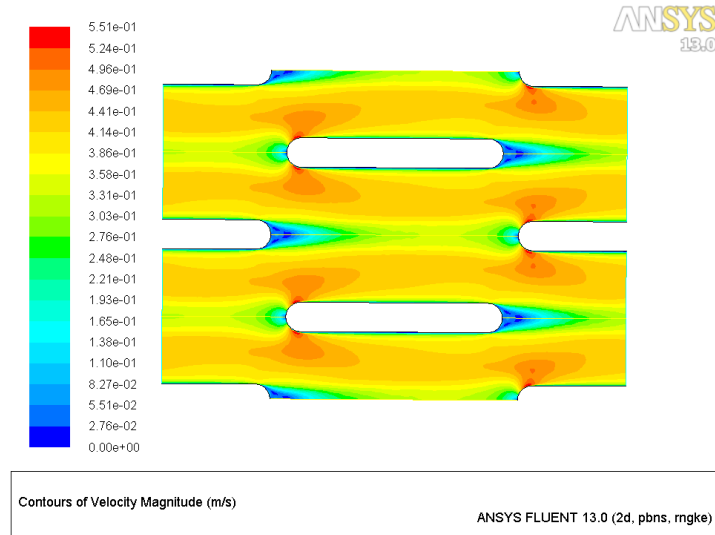


Figure 8: Contours of Velocity Magnitude

Note: 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.

4. Display velocity vectors (Figure 9).

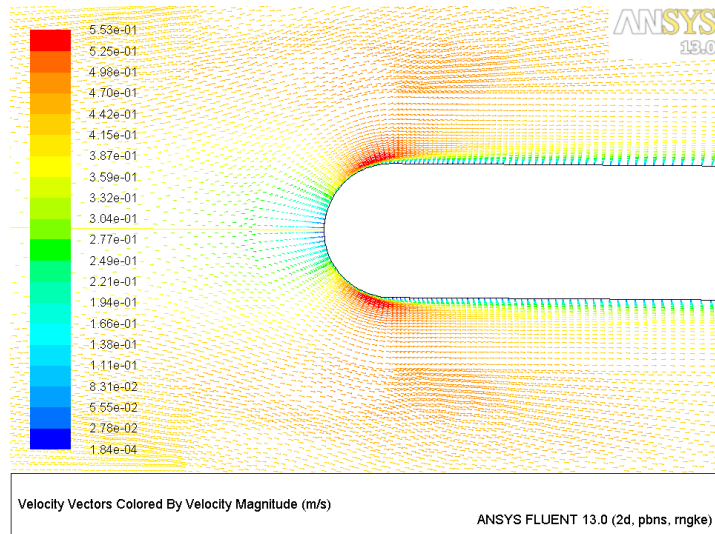
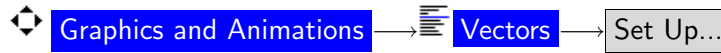


Figure 9: Velocity Vectors at the Front Showing the Stagnation Point Region

5. Display filled contours of temperature (Figure 10).

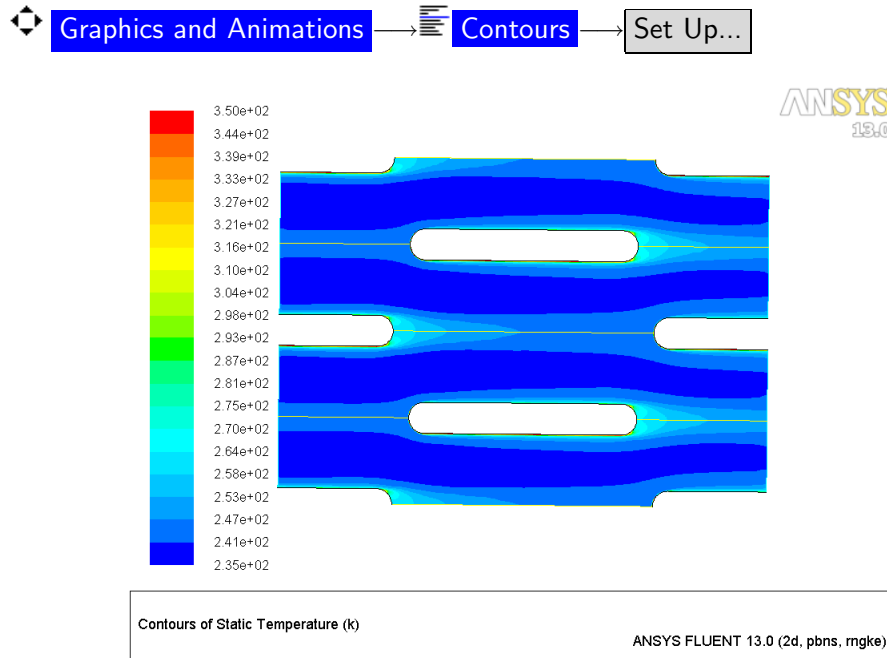


Figure 10: Contours of Temperature

Summary

In this tutorial liquid ammonia flowing through an interrupted fin heat exchanger is simulated using ANSYS FLUENT and the pressure drop and heat transfer characteristics are evaluated. 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:

- Type of mesh used
- Required cells
- Accuracy of calculated drag coefficient?
- Accuracy of calculated Stanton number?

You can also review the tutorial again and think about what you could have done differently.