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.

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

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.

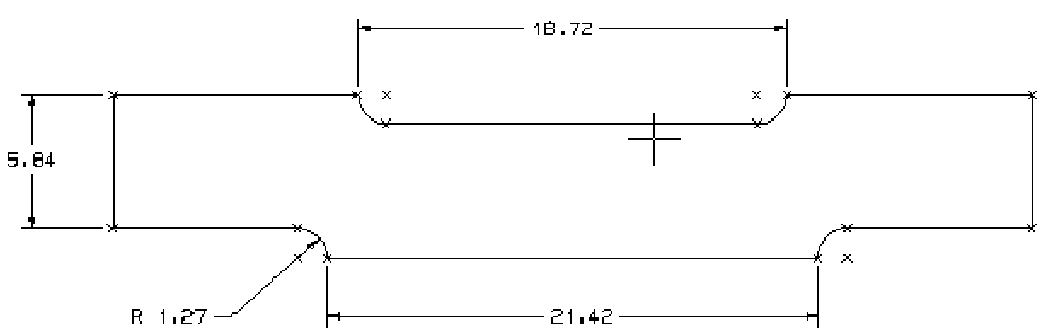


Figure 1: Heat Exchanger Geometry

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-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.

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

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

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:

$$St = \frac{h}{\rho U c_p} \tag{2}$$

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.*

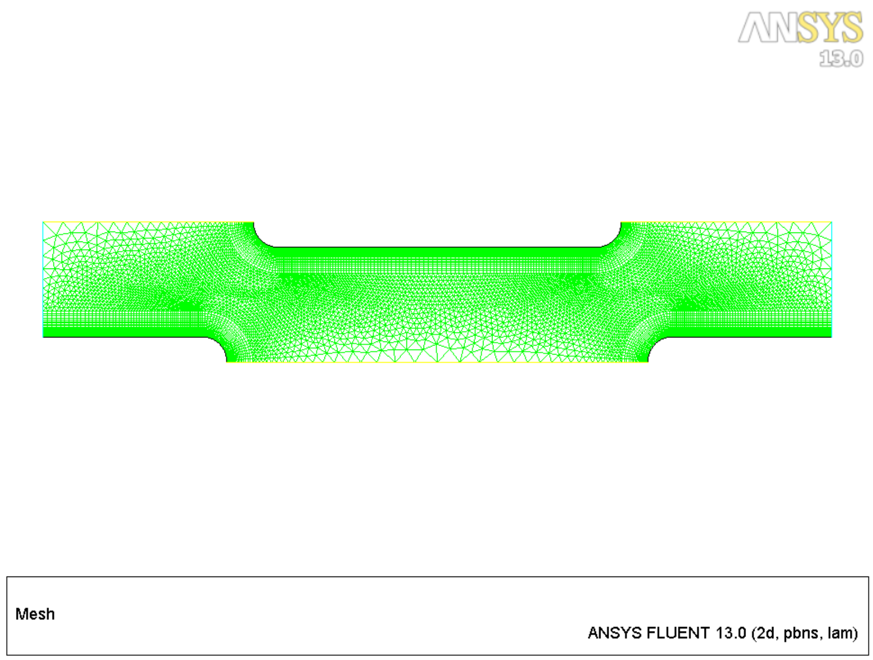


Figure 2: Graphics Display of the Mesh

Step 2: General Settings

If you created the mesh using GAMBIT*, then you have already defined the periodic zones, else, you need to perform this step here in* ANSYS FLUENT*. Change the left and right walls to periodic zones using the following TUI commands.*

Note: *The references to zone ID numbers below may differ from your case, since your zones may be numbered differently.*

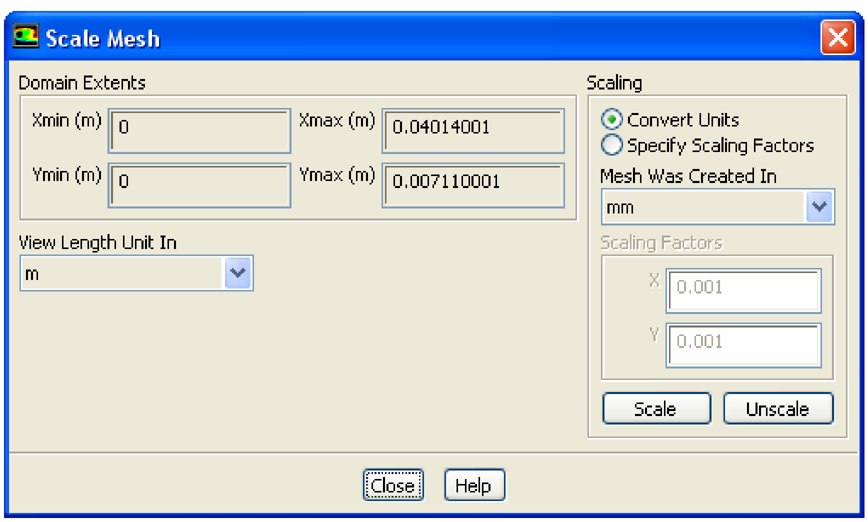
```
/mesh/modify-zones> mp
Periodic zone [()] wall-3
Shadow zone [()] wall-4
Rotational periodic? (if no, translational) [yes] no
Create periodic zones? [yes] yes
  computed translation deltas: 0.401400 0.000000
  all 48 faces matched for zones 3 and 4.
  zones 4 deleted
  created periodic zones.
```

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.

   Models ⟶ Energy ⟶ Edit..

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

   Models ⟶ Viscous ⟶ Edit..

   (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<l>) from the database.

Materials ⟶ Create/Edit...

1. Click FLUENT Database....

   (a) Select ammonia-liquid (nh3<l>) 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.

Cell Zone Conditions ⟶ fluid ⟶ Edit...

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.

   Boundary Conditions ⟶ wall-bottom ⟶ Edit...

   (a) Click on the Thermal tab.

   (b) Select Temperature from the Thermal Conditions group box.

   (c) Set the Temperature to 350.

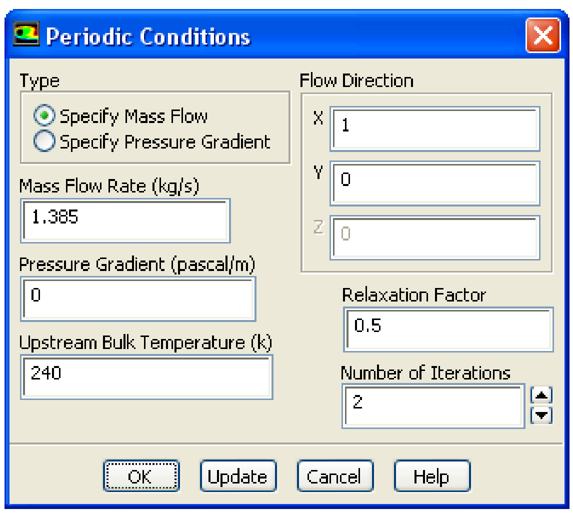
   (d) Click OK to close the Wall dialog box.

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

   Boundary Conditions ⟶ wall-top ⟶ Edit...

3. Specify the periodic flow conditions.

   Boundary Conditions ⟶ Periodic Conditions...



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

   (b) Set Mass Flow Rate to 1.385.

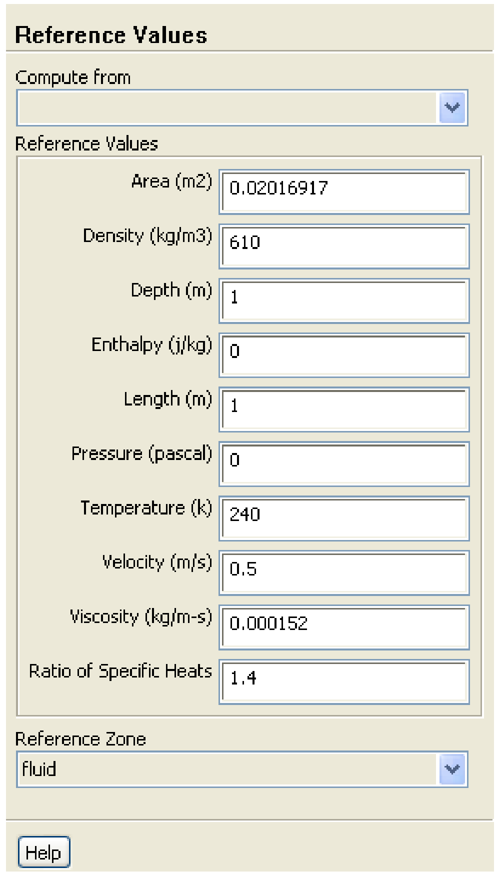
   (c) Set Upstream Bulk Temperature to 240.

   (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



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.

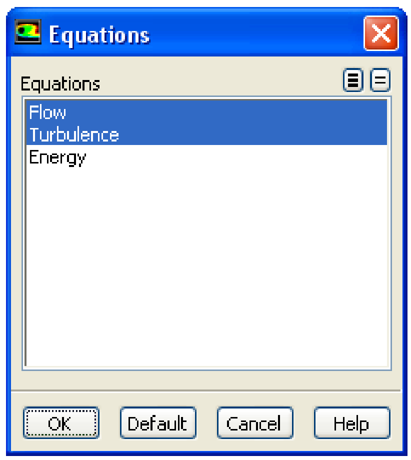
2. Set the Under-Relaxation Factors

   Solution Controls

   (a) Set Energy to `0.95`.

   (b) Deselect Energy in the list of equations.

   Solution Controls ⟶ Equations...



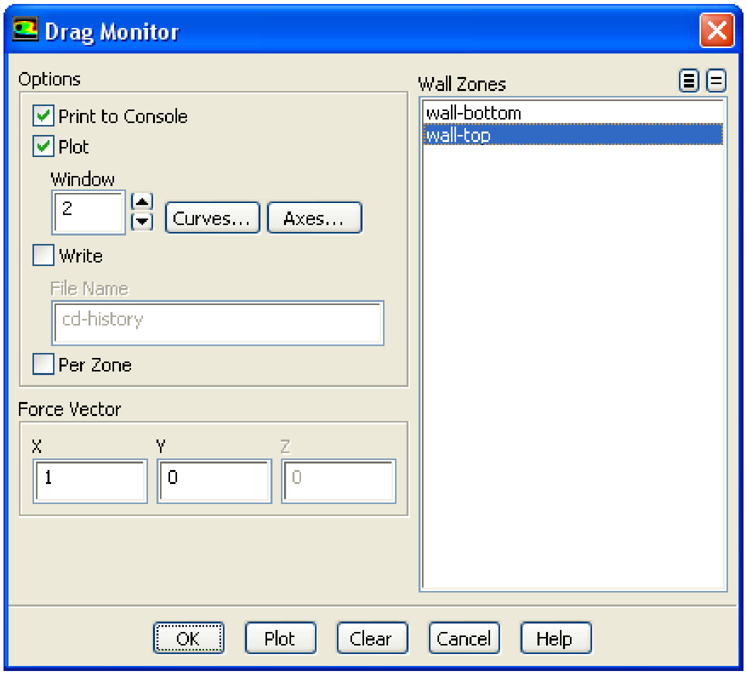
   You will begin the calculation by solving only the flow and turbulence equations. Once this solution has converged you will disable the flow and turbulence equations and solve only the energy equation. Finally, you will solve all equations together.

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...



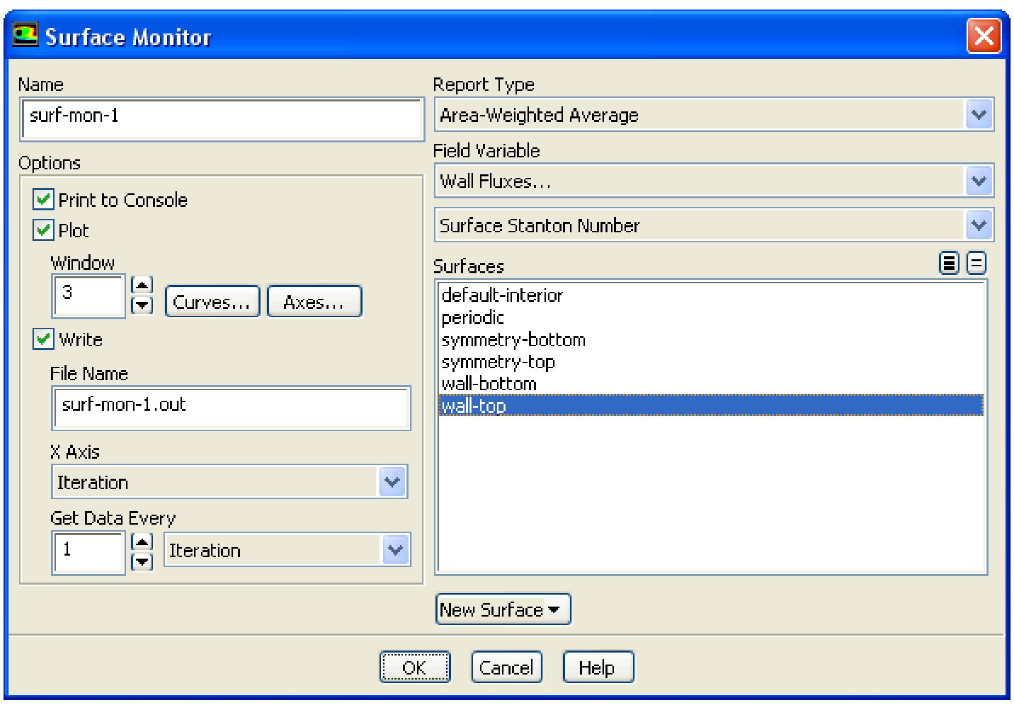
(a) Enable Print to Console and Plot.

(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.

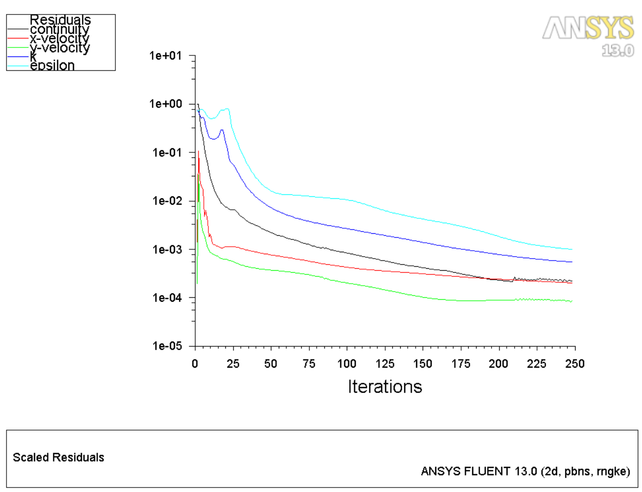


Figure 3: Scaled Residuals

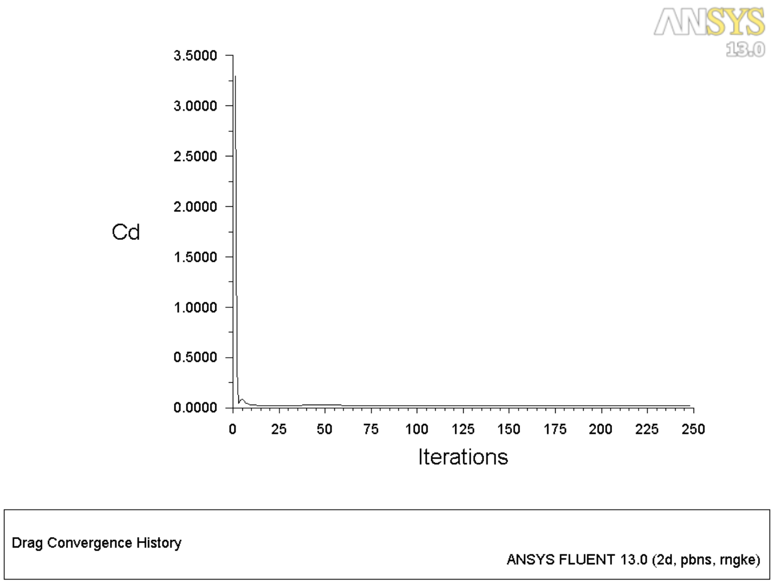


Figure 4: Drag Convergence History

9. Check the value of Yplus and adapt if necessary.

   Adapt ⟶Yplus/Ystar...

   You will use mesh adaption to refine the mesh if the Yplus *value is improper:*

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

   (b) If you use the two-layer zonal model, Yplus at the adjacent cell should be in the order of Yplus = 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...

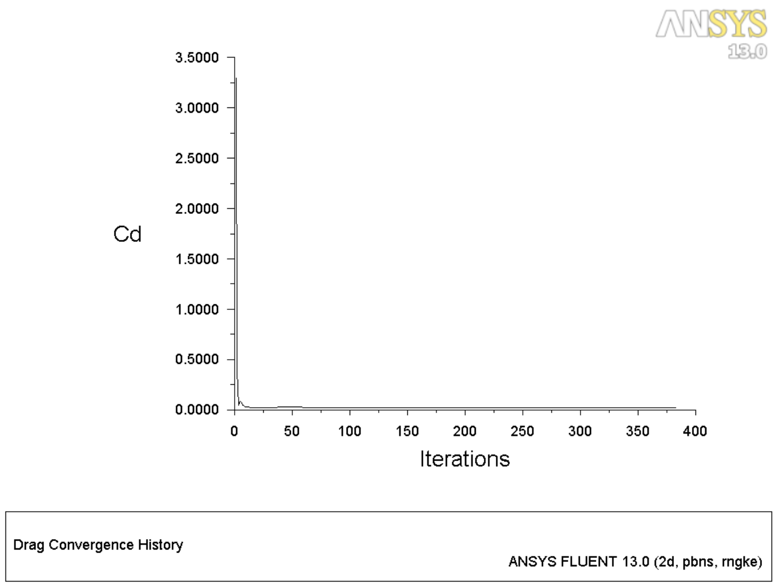


Figure 5: Drag Convergence History After Enabling Energy

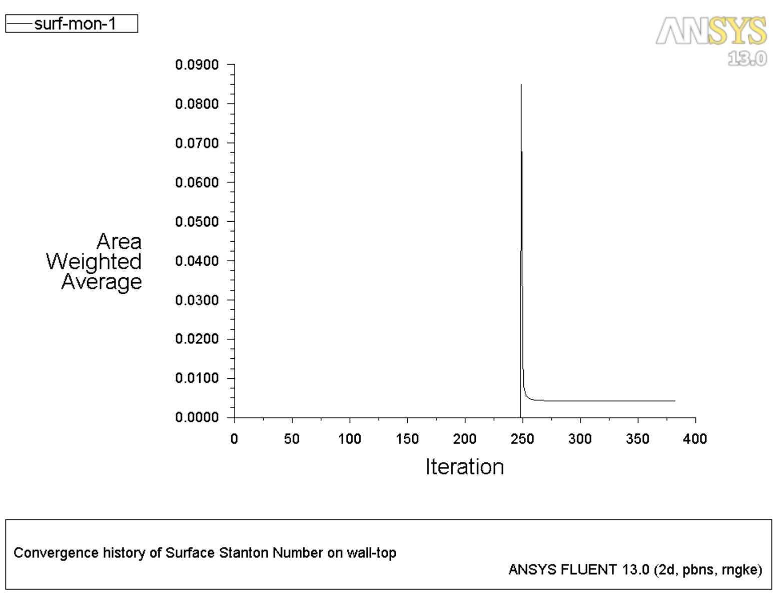


Figure 6: Convergence History of Surface Stanton Number on wall-top

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...

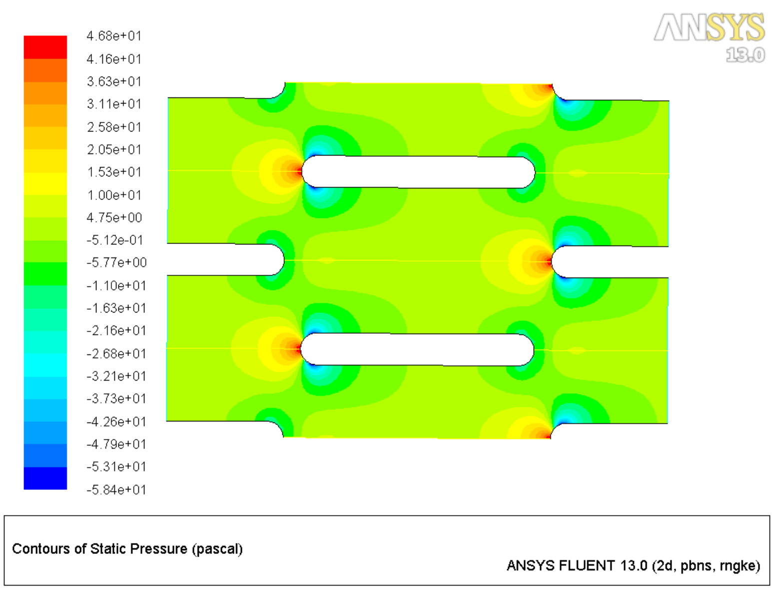


Figure 7: Contours of Static Pressure

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.

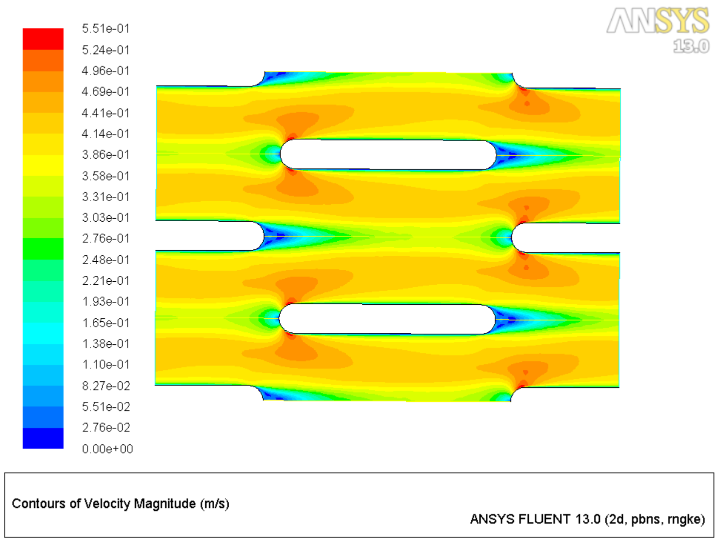


Figure 8: Contours of Velocity Magnitude

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).

Graphics and Animations ⟶ Vectors ⟶ Set Up...

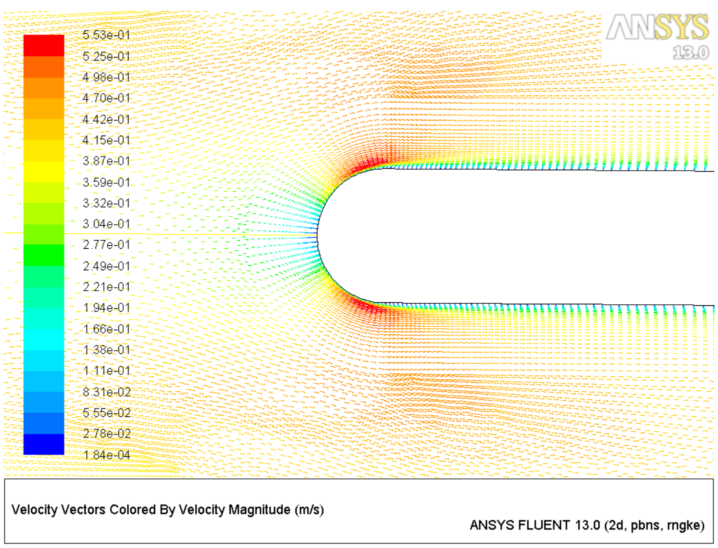


Figure 9: Velocity Vectors at the Front Showing the Stagnation Point Region

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

   Graphics and Animations ⟶ Contours ⟶ Set Up...

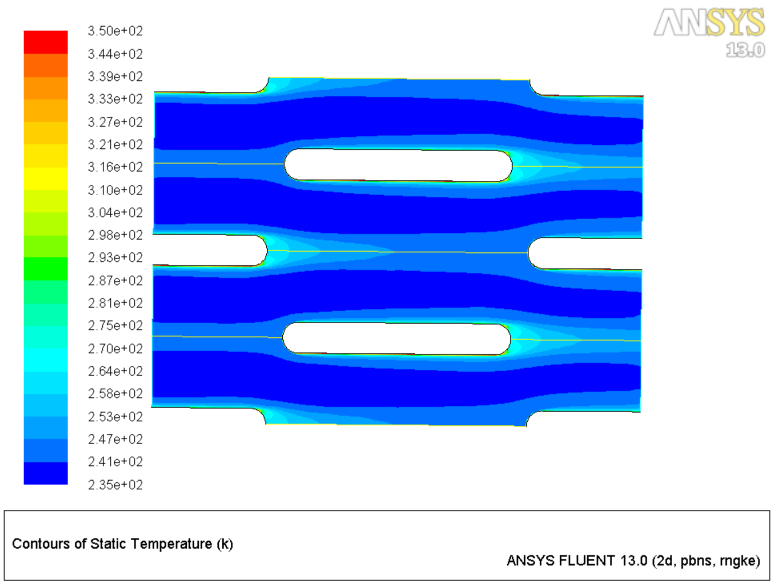


Figure 10: Contours of Temperature

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.