Studying the Effects of Vortex Generators on Heat Exchangers

William Cusato, William Fischer

Introduction

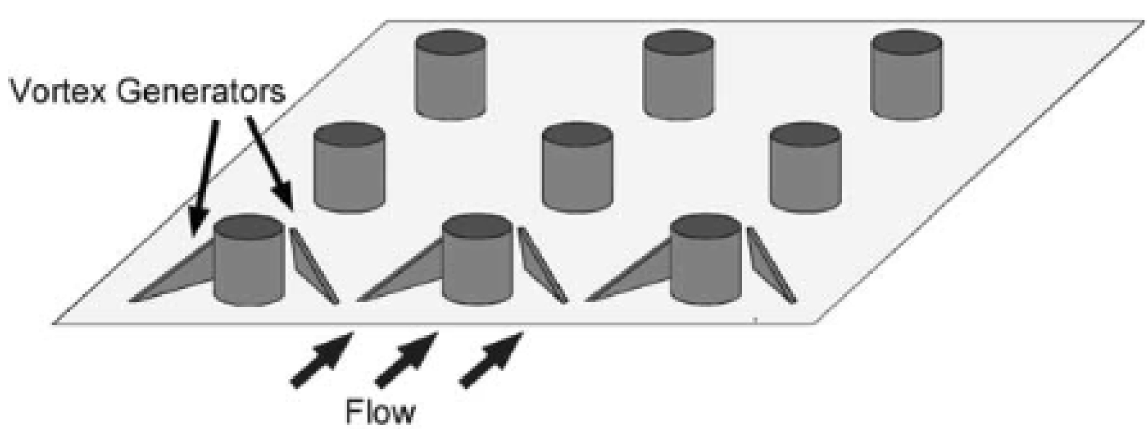
The flow we chose to study is fluid moving through a fin-tube heat exchanger. Heat exchangers are commonly used in many engineering applications, including air conditioning and power generation. Our work investigates the effect of adding vortex generators to a heat exchanger on the pressure loss and heat transfer. We evaluate the pressure loss because including vortex generators increases the effectiveness of heat transfer, but a significant pressure loss could offset this benefit by requiring more energy to operate. We aim to understand the trade-offs by considering these two results.

Methods

In our experiment, we focused on the flow and heat transfer differences between a heat exchanger with and without vortex generators. This is reflected in two separate geometries we created and two separate simulations run for both the flow and the heat-transfer.

Geometry

Torri et al. (2002) discuss two main varieties of heat exchanger geometry: in-line and staggered. The differences in these geometries stem from the positions of tubes that exchange heat between the two systems. While both geometries were discussed in the paper, we decided to focus on the in-line heat exchanger geometry in our COMSOL simulations.



Shown above are rough guidelines for the geometry arrangement, with heat exchanger tubes placed collinear to each other and vortex generators placed at an angle with respect to a line that runs perpendicular to the bottom horizontal axis. The research paper outlines specific geometric constraints their experiment adhered to (Torri et al., 2002). These constraints can be seen below:

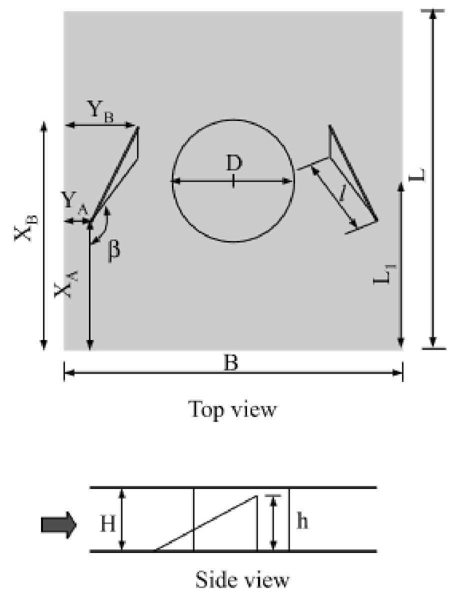


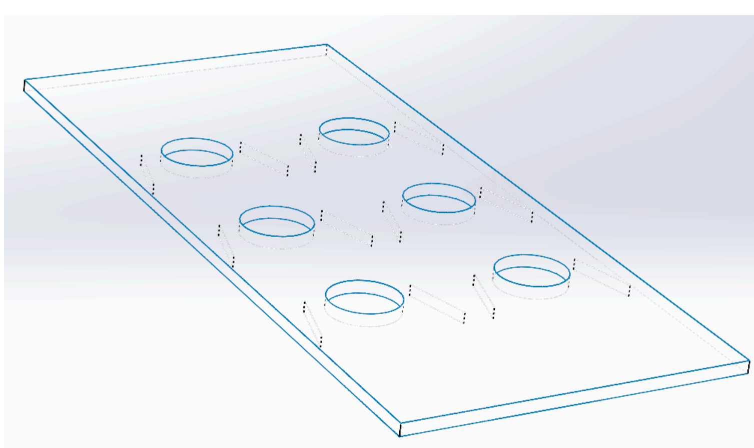
Table 1
Summary of geometrical arrangements of test-cores and vortex generators

Variables	Arrangement proposed by the present study[a,b]	Arrangement proposed by Fiebig et al. [2][a]
B/H	13.39	13.39
L/H	13.39	13.39
D/H	5.36	5.36
L_1/D	1.25	1.25
D/B	0.40	0.40
X_A/H	2.44	9.38
X_B/H	7.61	10.79
Y_A/H	1.19	4.02
Y_B/H	2.57	2.60
h/H	0.9	1.0
l/D	1.0	0.37
β	165°	135°

[a] In-line tube arrangement.
[b] Staggered tube arrangement.

These guidelines were followed absolutely in our model with one exception: the vortex generators were made to be rectangles instead of triangles. We also picked the length of one heat exchanger unit to be L = 75 mm because all the given parameters are non-dimensionalized. From this, all other values could be derived.

Our virtual model consisted of three rows and two columns of vortex generators and heat exchanger units. This layout was chosen to observe the impact of vortex generators on adjacent rows of tubes. There are surfaces enclosing the entire setup on the sides and above. We also extended the inlet and outlet to be displaced from vortex generators to reduce any potential errors in the results from disturbed flow near the boundary conditions. We chose to make our geometry 3D because it better matches the setup of the paper (Torri et al., 2002), and we think the flow created by the vortex generators is very much three-dimensional. Our final geometry setup can be seen below.



Boundary Conditions

To define the flow, we use an inlet velocity to match the Reynolds numbers used in the academic paper (Torri et al., 2002). In particular, they used a wide range of laminar flows, we chose to fixate on flow with a Reynolds number of 300.

In the context of this experiment, Reynolds number is expressed as $Re = \frac{U_{in} \cdot 2H}{\nu}$. Values of $Re = 300$, $H = 5.69 \cdot 10^{-3}\, m$, $\nu = 1.48 \cdot 10^{-5} \frac{m^2}{s}$, yield a velocity of $U_{in} = 0.39516 \frac{m}{s}$.

This velocity is used as the boundary condition for inlet velocity.

The outlet boundary condition is defined to have a static pressure of 0 because it allows easy comparison of flow setups and contributes to more reliable solution convergence. The rest of the boundaries in our flow setup are defined as walls. We considered using open boundaries but decided that containing the flow with walls to create an internal flow is a better representation of how the flow would behave in a small section of a heat exchanger (Torri et al., 2002).

For the heat transfer analysis, we used the same inlet and outlet conditions. The inlet flow was set to a temperature of 293.15 K and surfaces of the tubes to a constant 343.15 K. Conduction from the fluid in the tubes through their walls is not something we were interested in, so setting the tube surface temperature abstracts that from the simulation. Setting the temperature as a boundary condition also meant we could probe the heat flux through the tubes as a measure of the heat transfer effectiveness. We picked the temperatures arbitrarily within the range of room temperature because we know the experiments from the journal were also done around that range (Torri et al., 2002). The specific temperatures do not really matter because we non-dimensionalize the results. We set the remaining boundaries to be thermal insulators as the heat transfer through the tubes is the entire purpose of a heat exchanger, and the other boundaries are mainly there to confine the flow rather than represent real geometry.

The driving mechanism of heat transfer in the heat exchanger is forced convection, so we ran our simulation to only account for that. Neglecting natural convection makes the simulation much easier to run and preserves the purpose of our investigation.

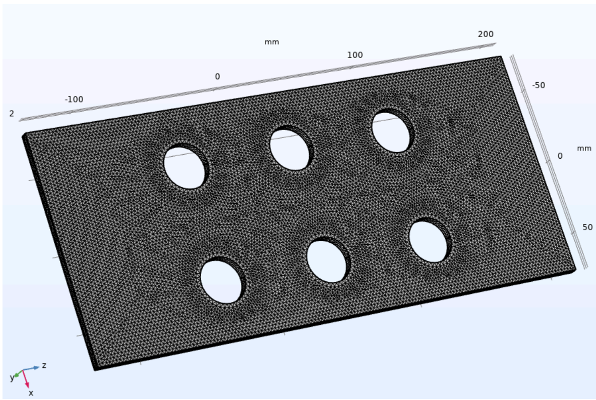
Mesh

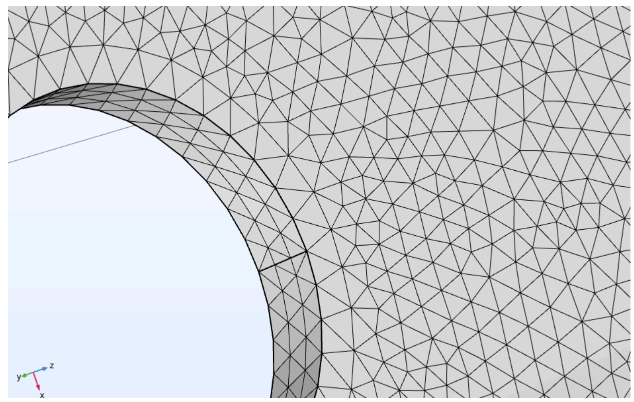
We used coarse mesh, which gave us quick computation times of about 1-2 minutes. Because our entire geometry was modeled using surfaces and not solids, the mesh computation was simpler. The mesh was concentrated near the locations of intersections between surfaces. Because the geometries used in both the flow simulation and the heat transfer simulation were identical, the meshes were as well. This resulted in the mesh that is seen below:

Geometry	Number of Domain Elements

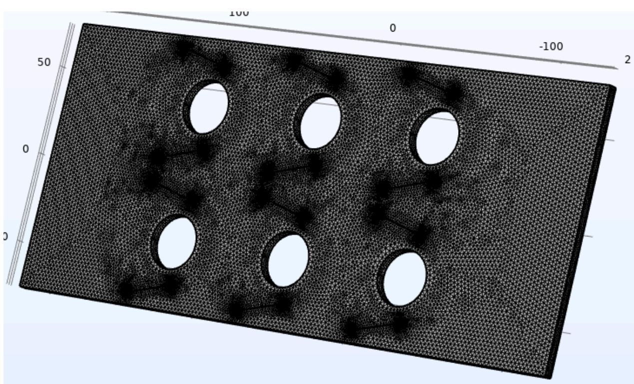
With Fins	585548
Without Fins	321947

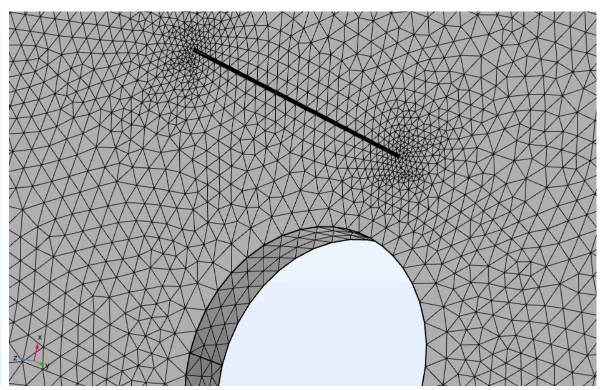
Mesh without Fins





Mesh with Fins





We believe the area around the heat exchanger tube will provide interesting results because the flow will form the boundary layer around the tube due to no-slip. The vortex generators will heighten this interesting effect due to the constricted areas in which the flow travels through (Torri et al., 2002).

Computation

The simulation run is supposed to simulate internal flow with a Reynolds number of 300. A Reynolds number of 300 is well within the bounds for laminar flow, so we used the laminar flow condition. Our computation times are as follows:

Experiment Run - Flow Simulation	Computation Time (s)
Re = 300 With Fins	63
Re = 300 Without Fins	31

Experiment Run - Heat Transfer Simulation	Computation Time (s)
Re = 300 With Fins	275
Re = 300 Without Fins	147

Results and Validation

Results

Experiment Run - Flow Simulation	Pressure Drop (Pa)
Re = 300 With Fins	2.59
Re = 300 Without Fins	1.62

Torri et al. (2002) measures improvement in heat transfer through the J-factor, a dimensionless parameter that quantifies convective heat transfer. It is defined as $j = \frac{Nu}{Re \cdot Pr^{1/3}}$. Therefore, finding Nu, Re and Pr will allow us to quantify heat transfer improvement as a result of adding vortex generators.

In the context of our simulation, Re is kept constant at 300.

The Prandtl number can be defined as $Pr = \frac{\nu}{\alpha}$. Using the 'derived values' function in COMSOL, volume averages for these parameters were found. With these values, Pr can be calculated.

$$Pr_{Fin} = \frac{\nu}{\alpha} = \frac{1.5613 \cdot 10^{-5}}{2.2100 \cdot 10^{-5}} = .70647, \quad Pr_{NoFin} = \frac{\nu}{\alpha} = \frac{1.5703 \cdot 10^{-5}}{2.2233 \cdot 10^{-5}} = .70629$$

In the context of the experiment run, Nu is defined as $Nu = h_m \cdot \frac{2H}{k_{air}}$ (Torri et al., 2002), where H is the height of the heat exchanger (5.69 mm). To find the convective heat transfer coefficient, we used the 'derived values' function once again to find overall heat flux. Using the relationship, $Q'' = h_m (T_s - T_{inf})$, we are able to solve for h_m since we set T_s and T_{inf} as boundary conditions.

$$h_{m,\,Fin} = \frac{Q''}{(T_s - T_{inf})} = \frac{753.37}{50} = 15.0674, \quad h_{m,\,NoFin} = \frac{Q''}{(T_s - T_{inf})} = \frac{607.73}{50} = 12.154$$

To calculate Nu, only the thermal conductivity of air is unknown. Once again, using the 'derived values' function, these values could be determined.

$$Nu_{Fin} = h_m \cdot \frac{2H}{k_{air}} = 15.0674 \cdot \frac{2(5.69 \cdot 10^{-3})}{(.026243)} = 6.539$$

$$Nu_{NoFin} = h_m \cdot \frac{2H}{k_{air}} = 12.154 \cdot \frac{2(5.69 \cdot 10^{-3})}{(.026220)} = 5.261$$

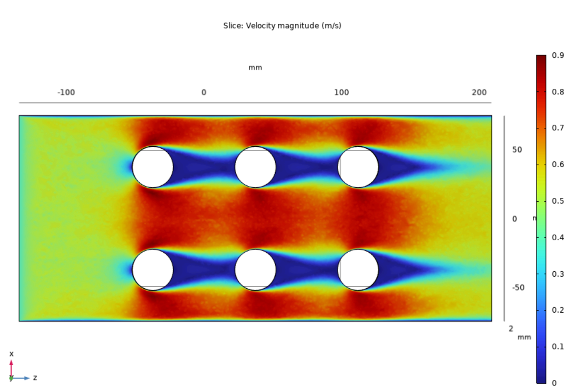
Finally, the j-factor can be calculated,

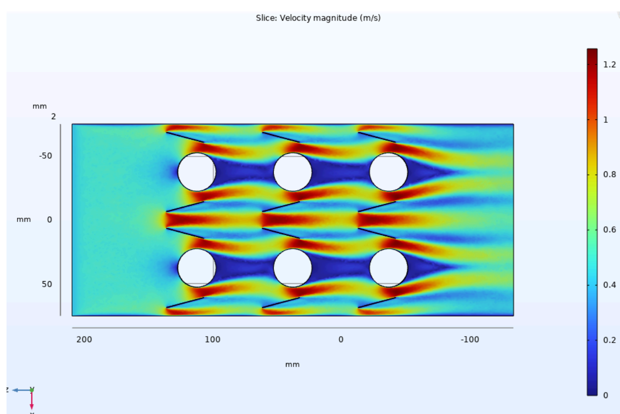
$$j_{Fin} = \frac{Nu}{Re \cdot Pr^{1/3}} = \frac{6.539}{300(.70647)^{1/3}} = .024473, \quad j_{NoFin} = \frac{Nu}{Re \cdot Pr^{1/3}} = \frac{5.261}{300(.70629)^{1/3}} = .019688$$

Experiment Run - Heat Transfer Simulation	J-factor
Re = 300 With Fins	.024473
Re = 300 Without Fins	.019688

The higher J-factor for the geometry with fins means the vortex generators increased heat transfer efficiency.

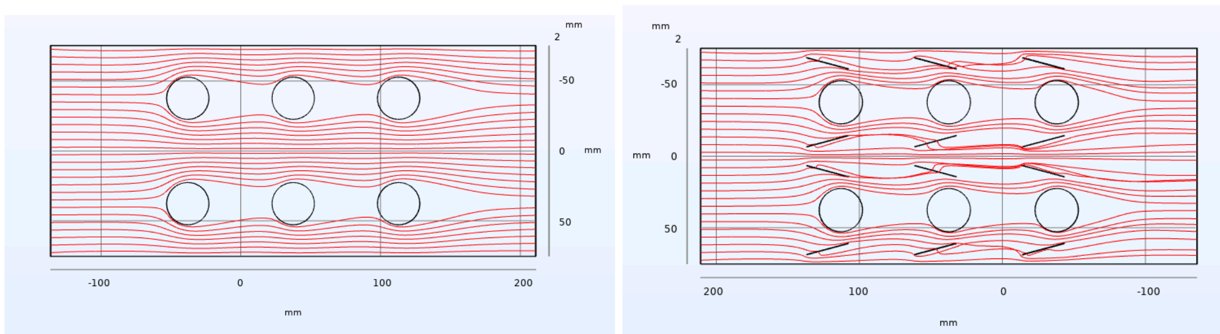
Plots showing velocity magnitude through the xz plane in m/s with both no vortex generators (left) and vortex generators (right)





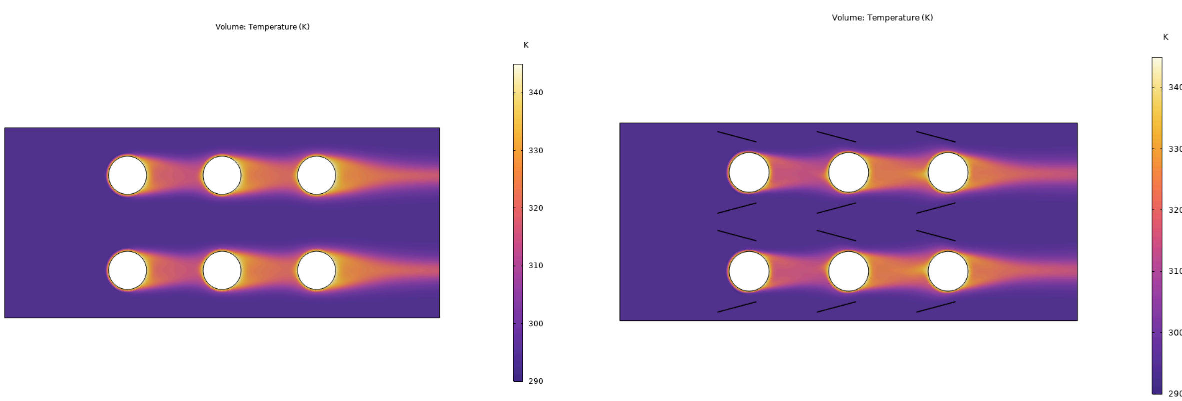
Comparing these plots, it is easy to see the effect that vortex generators have on the flow through the heat exchanger. The area of flow separation with vortex generators is greatly reduced due to the increased y-direction movement of flow. It can also be noted that the flow with vortex generators achieves a much higher maximum velocity.

Plots showing streamline of flow through the xz plane with both no vortex generators (left) and vortex generators (right)

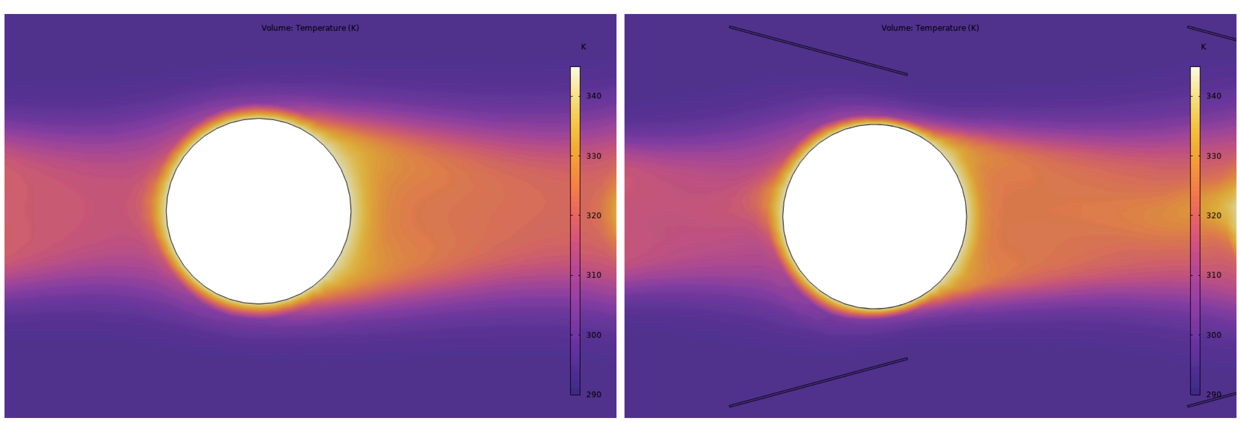


It is evident from this plot that the vortex generators, once again, decrease flow separation, as the area with no streamlines on the right hand side is much smaller than the area with no streamlines on the right hand side. Notably, there do not seem to be visible vortices in the flow.

Plots showing temperature in Kelvin of the flow through the xz plane with both no vortex generators (left) and vortex generators (right).



Zoomed in view of above plots, with no vortex generators (left) and with vortex generators (right)



The most notable difference is between the thermal boundary layers downstream of the tubes. The shape of the boundary layers are pretty similar, but thinner on the model with fins downstream of the tube. This difference makes sense given the results of our flow simulation. The fins increase flow near the downstream side of the tubes, which advects more heat away.

Validation

The paper we are referencing (Torri et al., 2002) reported results as a ratio of friction factors, f/f_{Go} and j-factors, j/j_{Go}. The variable f is the Fanning friction factor, and the subscript Go represents the flow without vortex generators. The fanning friction factor is directly proportional to pressure drop, so we can compare a ratio of our pressure drops,

$$\Delta P/\Delta P_{Go} = 2.59/1.62 = 1.60$$

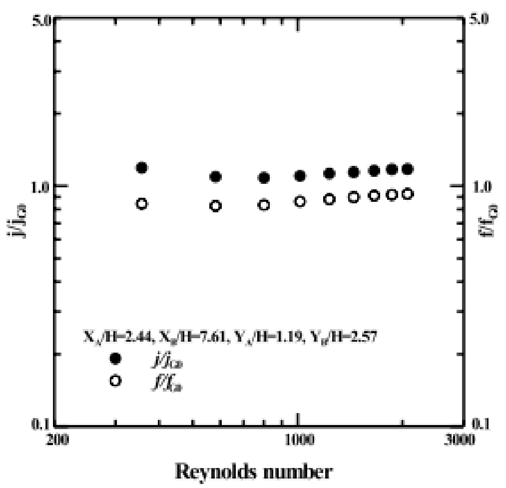


Fig. 5. The comparison of j/j_{Go} and f/f_{Go} with respect to Reynolds number (for the configuration of winglet in in-line tube arrangement illustrated in Fig. 3(a)).

From the plot to the left (Torri et al., 2002), the experimental results from the paper show the f/f_{Go} ratio at Re = 300 is actually about 0.8. This corresponds to a 20% reduction in drag from including the vortex generators. Our result shows the opposite, a 60% increase in drag. Our result intuitively makes sense–all the fins in our model provide extra surface area and redirect the flow, creating more drag. However, the drag reduction seen in the study is different due to the vortices and unique flow characteristics created by their delta-shaped fins. As seen in the streamline plot, there are no noticeable vortices in our simulation (we ran a static study but vortices are dynamic), which we believe to be a contributing factor to the differences between the simulation and the experiment. Although the data does not validate our result, we don't think it invalidates it because of the aforementioned differences.

Referring to the same plot (Torri et al., 2002), the experimental results from the paper show the j/j_{Go} ratio at Re = 300 is roughly 1.2. This matches the data from our simulation quite nicely, as our calculated ratio is $j/j_{Go} = \frac{.024473}{.019688} = 1.2430$. This result makes sense, because the vortex generators cut into the area of flow separation in between each heat exchanger tube. This, in turn, increases heat transfer as the fluid traveling around the heat exchanger tubes is faster and can take away heat from more surface area. The results of our heat transfer analysis show great correlation with the experimental results.

Conclusions and Future Work

Including rectangular vortex generators in our flow simulation resulted in a significant increase in pressure drop and heat transfer compared to flow without the vortex generators. Our setup shows a clear tradeoff between heat transfer and pressure drop since both metrics increased with fins added. Using the fins increases the effectiveness of the heat exchanger, but requires more power to operate because of the pressure drop.

We faced numerous challenges throughout the process of creating this simulation. Starting with issues stemming from the geometry, we initially imported it as an STL which led to messed up boundaries in the mesh. We resolved this issue by importing the SolidWorks file directly. In the process of creating our simulation, we initially had open boundaries along the top and side surfaces of our model. We found this to be problematic as the flow would leave our model without really interacting with the exchanger tubes, and we were left with very little flow at our outlet. We fixed this by making all boundaries that weren't our inlet or outlet walls and found this simulated our flow much better. The heat transfer simulation went surprisingly smooth, and we didn't encounter any issues.

If given more time to work on this study, we would choose to expand the number of tube units that our model included. With more units, we believe the flow simulation will better mirror that of reality. We would have also liked to create simulations based on the exact geometry that the experiment covers. Thus, cross-comparison between our simulation and the experiment could be conducted, and areas of improvement could be more readily observed.

Finally, if more time was given, we would have looked to run transient study, specifically for reasons relating to flow. We were made aware that vortices in flow are a transient property, and the fact that our study was conducted under steady-state conditions is likely the reason we did not see vortices in the flow. We believe the prevalence of vortices would have made our pressure loss data more accurate compared to that of the paper (Torri et al., 2002).

Citations

Torri, K., et al. "Heat Transfer Enhancement Accompanying Pressure-Loss Reduction with Winglet-Type Vortex Generators for Fin-Tube Heat Exchangers." *Harvard Astrophysics Data System*, International Journal of Heat and Mass Transfer 45 (2002) 3795–3801, 5 Sept. 2001, https://ui.adsabs.harvard.edu/abs/2002IJHMT..45.3795T/abstract. Accessed 9 Nov. 2024.