Brian Menard

June 10th, 2021

Parallel and Counter Flow Heat Exchanger Analysis

1. Introduction

The following term project examines simulating a parallel flow cylindrical heat exchanger composed of two regions of water. Heat exchangers are essential components in almost every thermal system, and parallel heat exchangers are one of the simplest variations of heat exchangers. A numerical model was created in FLUENT along with an analytical model in Matlab. These models were simulated using several different mesh sizes and mass flow rates, and their results compared in the following report.

2. Purpose

The purpose of the following analysis is to compare the theoretical heat exchanger model with the numerical simulation results from FLUENT. The fluid mass flow rates were set equal to each other to create a uniform heat flux boundary condition on the boundary between the fluid. The fluid velocities were defined such that their Reynolds numbers were below 2300 and the flow remained laminar.

3. Problem Description

The geometry of the heat exchanger consists of an outer tube with a 0.056 m ID and a copper dividing tube with a 0.025 m ID and thickness of 0.003 m. The tubes are located concentric to each other with the hot fluid flowing in the outer region and the cold fluid flowing in the inner region. This configuration results in both fluid regions having the same hydraulic diameter. The Reynolds number and Prandtl numbers were calculated to find the hydrodynamic and thermal entry lengths, after which the length of the tube was defined to be slightly longer to examine the heat transfer in the fully developed regions. The length of the heat exchanger is 3 m long.

3.1 Fluid Properties

The hot fluid has a uniform inlet temperature of 343 K, and the cold fluid has an inlet temperature of 303 K. Both fluids have a uniform inlet mass flow rate of 0.01 kg/s, or 0.0204 m/s for the cold fluid and 0.0059 for the hot fluid. The default water properties in FLUENT were used with a density of 998.2 kg/m^3, specific heat of 4182 J/kg-K, viscosity of 0.001003 kg/m-s, and thermal conductivity of 0.6 W/m-K. These properties were assumed to be constant throughout the simulation for both fluids.

Table 1. Fluid Properties

	Mass Flow Rate [kg/s]	Inlet Velocity [m/s]	Reynold's Number	Prandtl Number	Hydrodynamic Entry Length [m]	Thermal Entry Length [m]
Hot Fluid	0.01	0.0059	146	6.99	0.182	0.644
Cold Fluid	0.01	0.0204	509	6.99	1.257	4.437*

*The tube length of 3 m was sized when there was an error in this calculation, so the cold fluid should not reach thermal fully developed flow.

3.2 Flow Physics
The hot fluid Reynolds number was calculated to be 146 while the cold fluid Reynolds number was calculated to be 508. The hot and could fluid's Prandtl numbers were found to be 6.99. Table XX summarizes these values and their associated entry lengths prior to fully developed flow.

3.3 Boundary Conditions
The heat exchanger is modeled in 2D with an axisymmetric boundary condition along the center line of the tube. A perfectly insulated boundary condition was placed on the outer surface of the heat exchanger, along with the sides of the copper tube perpendicular to the flow at the ends of the tube. A uniform mass flow rate was placed on the inlet to the tube and a pressure outlet of 101.3 kPa on the opposing ends of the tube. The thermal back flow condition of each region was set at the analytical outlet temperatures of 332.4 K and 313.5 K for the hot and cold regions, respectively.

4. Grid Convergence

The numerical solution was performed over three different mesh sizes to ensure that the results were mesh-size independent. Figures 1.A-1.C show the three different mesh sizes used in the study, while Table 2 summarizes the results of the various mesh sizes in comparison to the analytical solution.

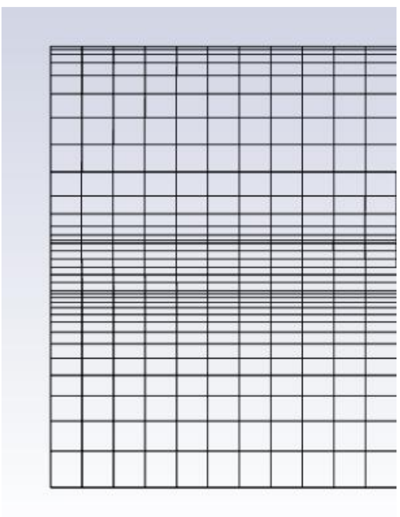

Figure 1.A Coarse Mesh

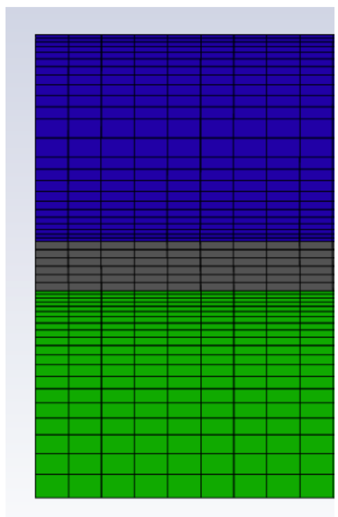

Figure 1.B Medium Mesh

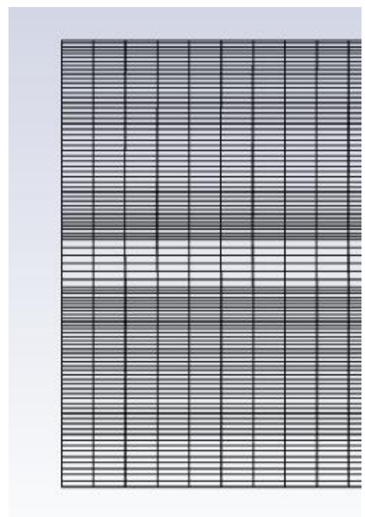

Figure 1.C Fine Mesh

Table 2. Comparison of various mesh sizes with analytical results for a mass flow of 0.01 kg/s

	Coarse Mesh	Medium Mesh	Fine Mesh	Analytical
Number of Cells	51000	75000	141000	-
Total Heat Transfer [W]	470.5	482.7	483.2	434.0
Hot Fluid Exit Temp. [K]	331.6	331.4	331.4	332.4
Cold Fluid Exit Temp. [K]	317.3	317.1	317.1	313.5

5. Case setup

Three different cases were set up in which the mass flow rate was varied between 0.005, 0.01 and 0.015 kg/s to examine the effect of fluid speed on the rate and amount of heat transfer. Table 3. Summarizes the findings from the three different cases using the fine mesh pictured in Figure 1.C

Table 3. Comparison of various mass flow rates with analytical results using the fine mesh

	Numerical Results			Analytical		
Mass Flow Rate [kg/s]	0.05	0.010	0.015	0.05	0.010	0.015
Total Heat Transfer [W]	327.2	483.2	581.6	325	434.0	491
Hot Fluid Exit Temp. [K]	327.3	331.4	333.6	327.5	332.4	335.2
Cold Fluid Exit Temp. [K]	320.1	317.1	315.2	318.5	313.5	310.8

6. Calculation

The SIMPLE algorithm was used from pressure velocity coupling, and second order unwinding was used for both momentum, pressure and energy. The viscous -laminar solver was used with energy turned on while other advanced simulation techniques were turned off. Gravity was not enabled for this simulation as natural convection effects were deemed to be negligible. The solution has some difficulty converging to residuals less than 10e-6, so under relaxation was used for both momentum (0.2) and pressure (0.3). This improved convergence some, however it still took ~6500 iterations to fully converge

7. Results

The results from the 0.01 kg/s mass flow rate case have been post processed and included below. The validity of the results along with other influencing variables are discussed in Section 7.1 below.

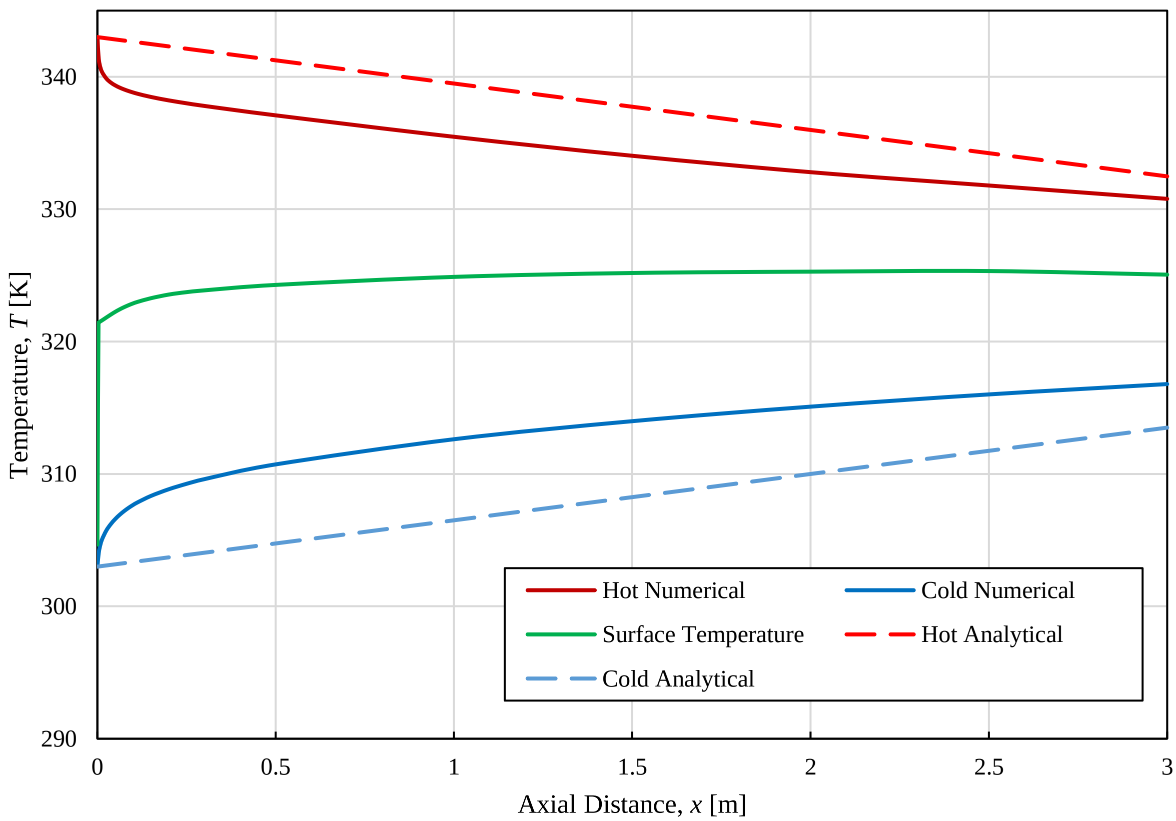

Figure 2. Temperature distributions along the length of the heat exchanger

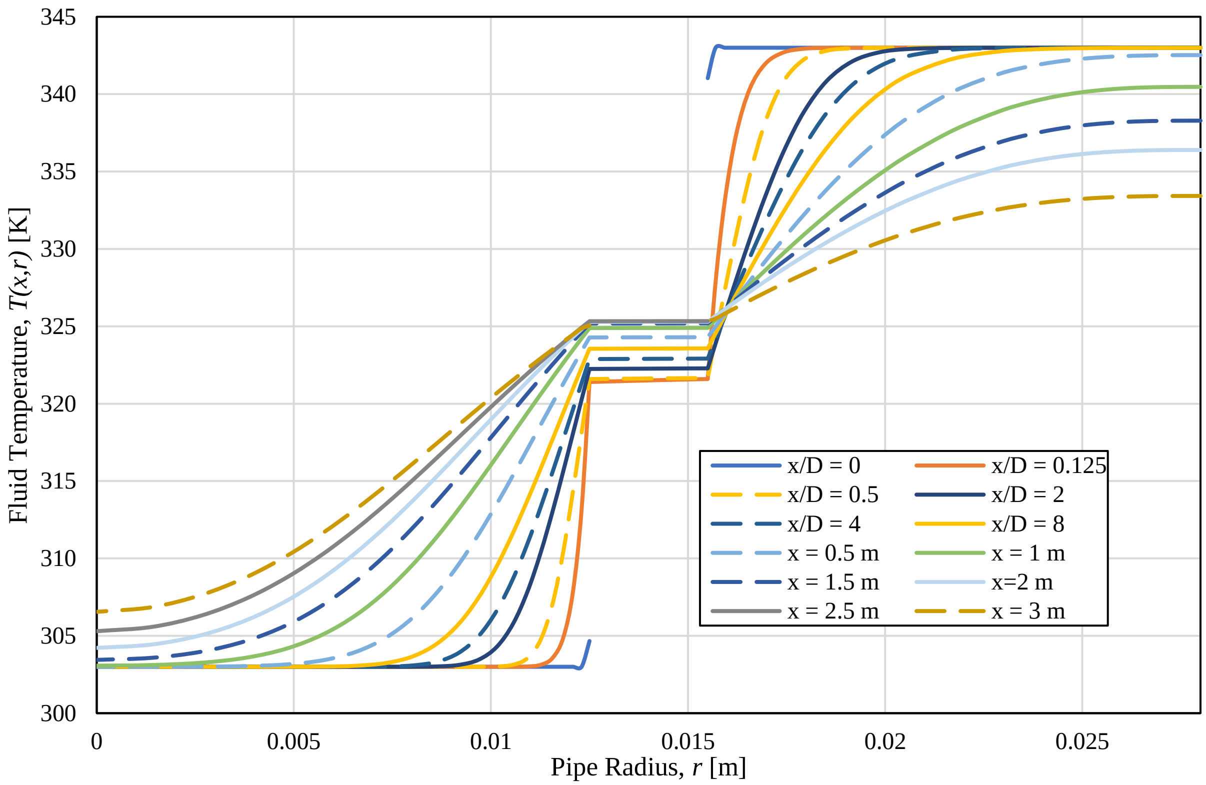

Figure 3. Temperature profiles across the radius of the heat exchanger at several different axial locations

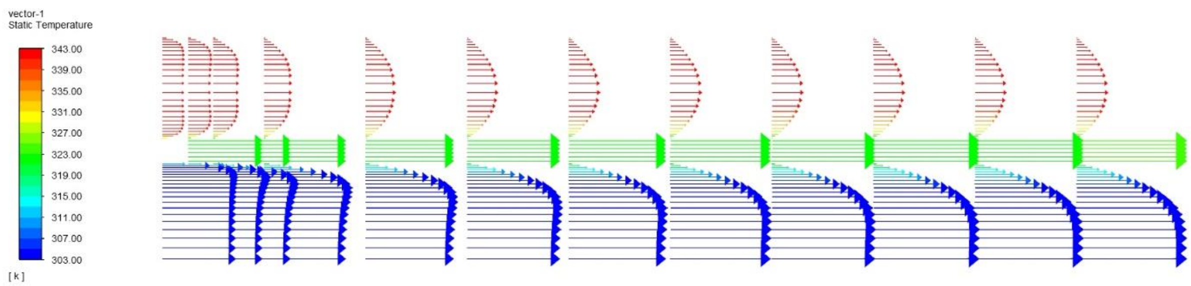

Figure 4. Velocity vectors in the entry region of the heat exchanger (green arrows have zero velocity)

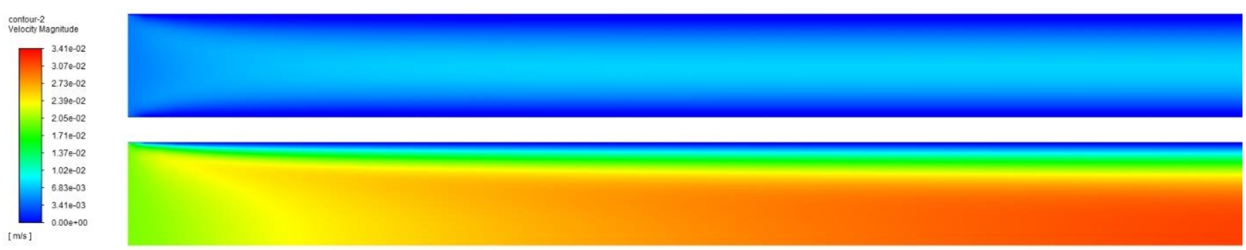

Figure 5. Velocity distribution contour plot at the start of the pipe

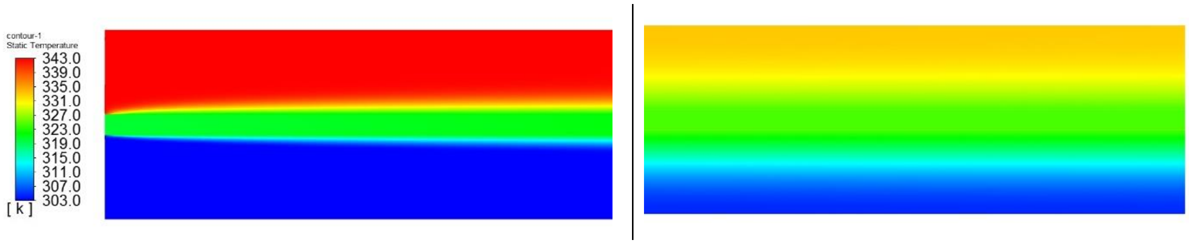

Figure 6. Contour plots of temperature distribution at the start and end of the pipe

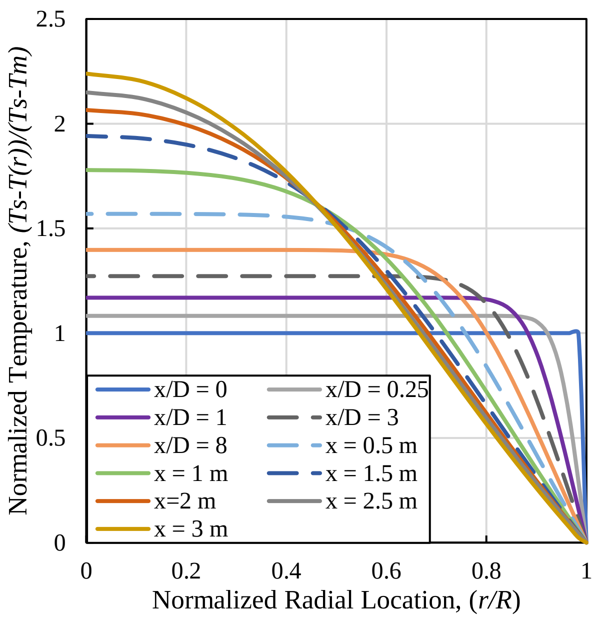

Figure 7.A Cold fluid entry region temperature profiles

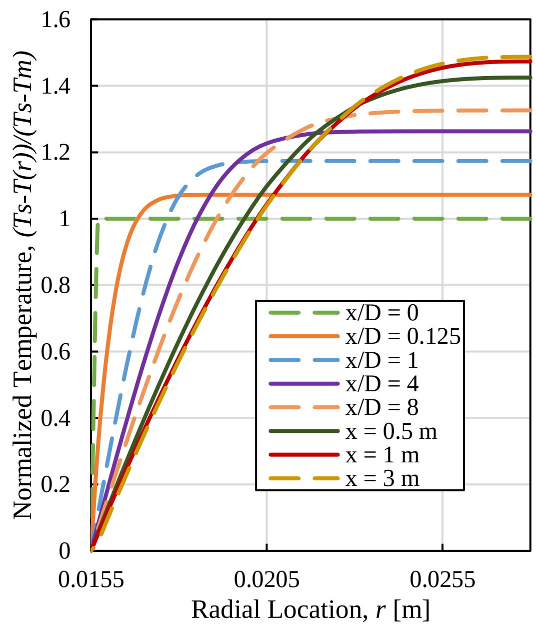

Figure 7.B Hot fluid entry region temperature profiles

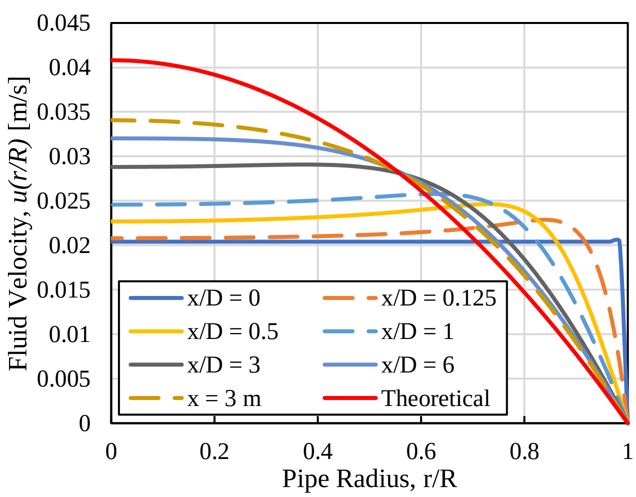

Figure 8.A Center tube (cold) entry region velocity profiles

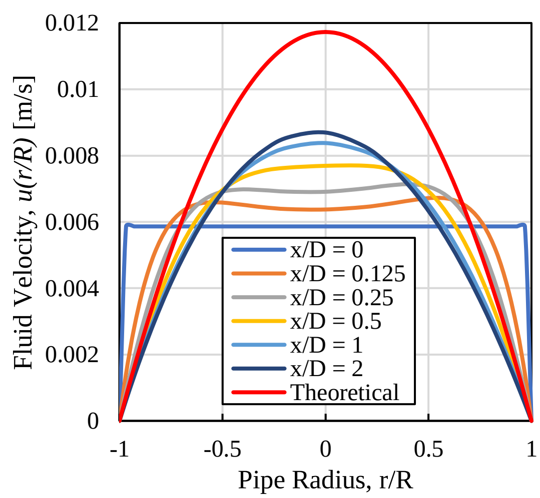

Figure 8.B Outer tube (hot) entry region velocity profiles

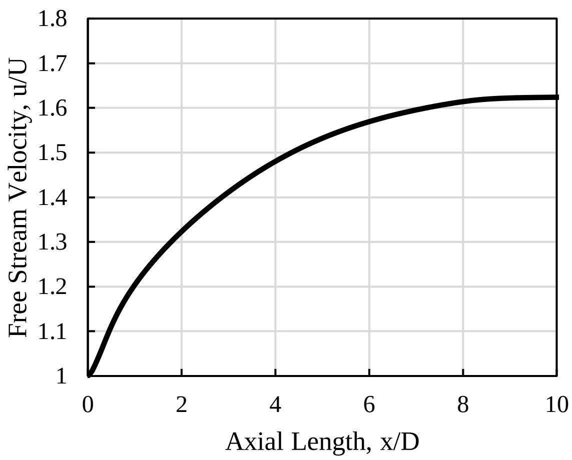

Figure 9.A Center tube (cold) region center line velocity vs. axial length

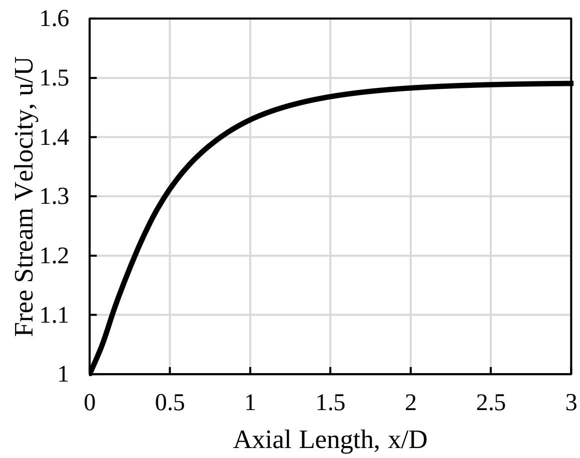

Figure 9.B Outer tube (hot) region center line velocity vs. axial length

7.1 Results Discussion

Simulating this simple heat exchanger proved much more difficult than anticipated, and due to this the presented results represent a best-effort to obtain accurate solutions. While there may exist several errors in the simulations, the results presented in Tables 2 and 3 indicate decent agreement between the analytical and numerical solutions after considering the discussion below. Figure 2 also presents a graphical comparison of the two models, and while the temperature along the length of the tube is different, the output temperatures are relatively close. The Matlab model has been attached to the end of this report

In terms of the analytical model, several key assumptions were made which do not entirely hold up in the numerical model. The most critical assumption made was that the flow was hydrodynamically and thermally fully developed at the inlet of both fluid regions. This was not the case in the numerical simulation, where the fully developed flow did not appear until farther down the pipe as shown in Figures 7 and 8. This difference in modeling approach accounts for most of the discrepancies between the two models, as the entry region of the numerical model shows higher heat fluxes than its fully developed region. This results in more heat being transferred between the fluids and in some cases less temperature difference at the outlet of the heat exchanger. Simulating the analytical model starting the fully developed region of the numerical model with modified input temperatures resulted in good agreeance between the models.

There are several issues with the FLUENT model, primarily illustrated in Figures 8 and 9. As discussed in Table 1, the hydrodynamic entry length for the hot and cold fluids were 0.182 m and 1.257 m respectively. However as illustrated in Figure 9.A, the hot tube reaches fully developed flow in just 0.075 m and the cold tube reaches fully developed flow in about 0.25 m. Furthermore, after normalizing the velocity and radii in Figures 8.A and 8.B, the fluids do not come even close to reaching the theorical profile with a maximum velocity of 2. I am not sure where this issue is coming from, though it could have come from convergence issues or incorrect simulation schemes.

8. Conclusion

Several different simulation conditions and mesh sizes were modeled both analytically through MATLAB and numerically through Fluent. While there exist some discrepancies between the analytical and numerical solutions, the models are in general agreement after accounting for differing assumptions.