SHELL AND TUBE HEAT EXCHANGER

MENG 3211 LABORATORY REPORT

031L-Th2-Group 7

REPORT SUBMITTED BY:

Usman Akram

Jeremy Baker

Johnathan Ferguson

Daniel Sigala

We certify that the narrative, diagrams, figures, tables, calculations and analysis in this report are our own work.

DATE EXPERIMENT PERFORMED:	March 30, 2022
DATE REPORT DUE:	April 22, 2022
DATE REPORT SUBMITTED:	April 22, 2022

MECHANICAL ENGINEERING DEPARTMENT
HOUSTON ENGINEERING CENTER
COLLEGE OF ENGINEERING
THE UNIVERSITY OF TEXAS AT TYLER

ABSTRACT

In this experiment, a shell and tube heat exchanger is modeled and simulated to find the heat transfer rate of the heat exchanger and its efficiency. A heat exchanger was designed using solidworks and a fluid simulation was done to see how the heat exchanger performed under ideal conditions. The shell and tube design had an efficiency of .37 with an overall heat transfer of Q_{actual} =2.32 kW and theoretical heat transfer of $Q_{theoretical}$= 6.27 kW. Some of the parameters that were modified was the mass flow rate of the fluids and surface area of the pipes to achieve maximum heat transfer rate.

TABLE OF CONTENTS

INTRODUCTION

The purpose of a heat exchanger is to transfer heat from one fluid to another. Heating and cooling of fluids are required in many processes such as boilers, refrigeration and many more. In any type of heat exchange, there is a fluid that requires a change in energy by heating or cooling. One fluid passes through the inside of the pipe while another passes over the outside of that same pipe. This contact allows for heat transfer to occur from fluid to fluid through the pipe material. Ideally this pipe material is picked for its high thermal conductivity to allow for the maximum transfer in energy to occur. With this configuration, the fluids do not mix and energy transfer occurs through convection and conduction with radiation being negligible.

The goal in this experiment was to find the optimal configuration for a shell and tube heat exchanger . A shell and tube heat exchanger is made up of several components that work in conjunction to allow for the maximum heat transfer of the fluids in use. A shell and tube heat exchange consists of a shell enclosed by a stationary head at each end as shown in Figure 1. The shell houses the fluid and the pipes as well as the baffles. A bulkhead separates the fluid from the head and the shell. When designing a heat exchanger one needs to take into consideration if it is going to be a single pass or multiple passes through the shell as shown in Figure 2. If a design with multiple passes is made, the stationary head is divided into different sections to separate the fluids returning from mixing on the next pass through the shell.

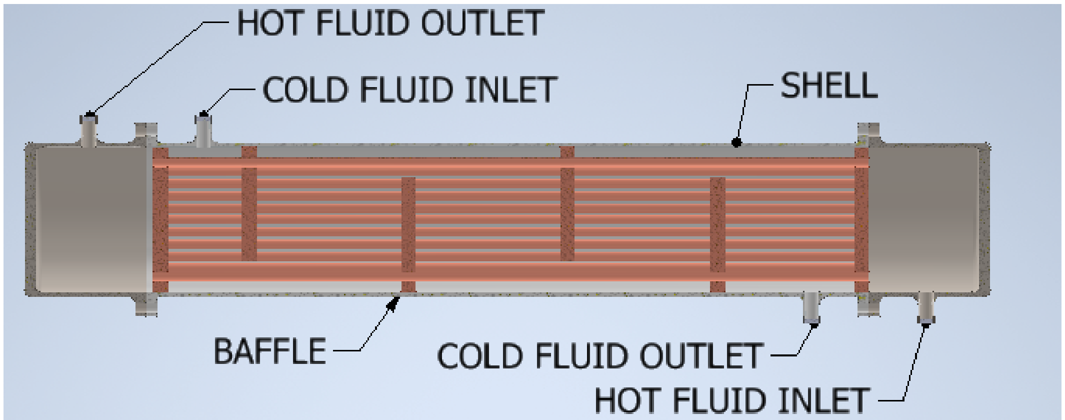

Figure 1. Single Pass Shell and Tube Heat Exchanger.

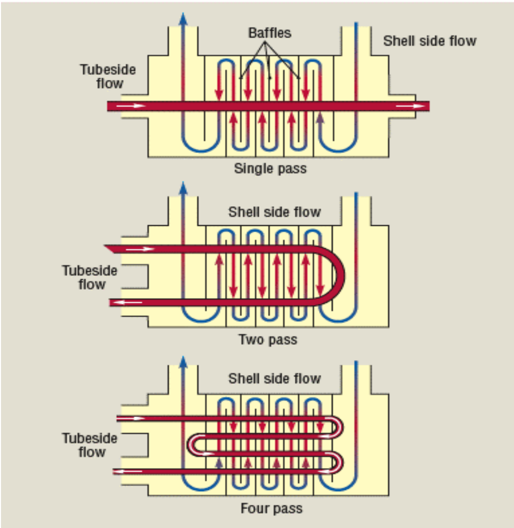

Figure 2. Shell and Tube Heat Exchanger with Different Pass Configurations.

Baffles in the shell help the inlet fluid in the shell make multiple passes across the pipes. These extra passes maximize the amount of heat transferred from the fluids and make a more efficient heat exchanger. The material of the pipes also has to be taken into consideration. Ideally a material with a high thermal conductivity coefficient is preferred. This would allow for the most optimal heat transfer to take place.

The logarithmic mean temperature difference of a heat exchanger is useful in examining the heat driving force in a heat exchanger. The larger the value the more, the more heat is transferred. To calculate the LMTD we use Equation 1.

$$T_{lm} = \frac{(T_{hi} - T_{ci}) - (T_{ho} - T_{co})}{ln(\frac{(T_{hi} - T_{ci})}{(T_{ho} - T_{co}})} \quad (1)$$

where T_{hi} is the hot fluid inlet, T_{ci} is the cold fluid inlet, T_{ho} is the hot fluid outlet, and T_{co} is the cold fluid outlet. This equation is used in counter flow systems and we are assuming there is no phase change if the fluid is happening in the heat exchanger.

To calculate the heat transfer rate in the heat exchanger Equation 2 is used.

$$\dot{Q} = UAT_{lm} \quad (2)$$

where $\dot{Q}$ is the heat transfer rate in W, U is the overall heat transfer coefficient in $\frac{W}{m^2 \cdot K}$, A is the heat flow area in m^2, and T_{lm} is the log mean temperature difference in K.

To solve for effectiveness of the heat exchanger we use equation (3)

$$\epsilon = \frac{\dot{Q}}{\dot{Q}_{max}} \quad (3)$$

To solve for $\dot{Q}_{max}$ we use Equation 4.

$$\dot{Q}_{max} = \dot{m}c_p(T_{hi} - T_{ci}) \quad (4)$$

where $\dot{Q}_{max}$ is the heat transfer, $\dot{m}$ is the mass flow rate, c_p is the specific heat capacity of water at constant pressure.

To solve for $\dot{Q}$ we use Equation 5.

$$\dot{Q} = \dot{m}c_p(T_{hi} - T_{ho}) \text{ or } \dot{Q} = \dot{m}c_p(T_{ci} - T_{co}) \quad (5)$$

where Q is the actual transfer rate.

METHODOLOGY

Equipment

Solidworks 2019 Education Edition was utilized in designing the 3D model of the heat exchanger. The CFD (Computational Fluid Dynamics) calculation was performed using the One UT Tyler servers to handle the fluid simulation and rendering.

Experimental Apparatus

When designing the shell and tube heat exchanger we started by listing the components and designing them one by one. The first thing was to get an overall estimate of how large we wanted the heat exchange to be. Since it was a class project, we started with a size of 1 meter in length for the purpose of practicality. After figuring out the initial length was decided upon, we went about making the components of the heat exchanger.

The tubes were the first part we went about designing. We selected the tube length to be 1 meter long. The tube design had a diameter of 12.5 mm and an outer diameter of 13.5 mm as shown in Figure 3.

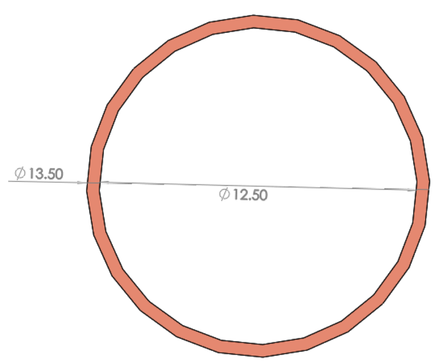

Figure 3. The Tube's Inner and Outer Diameter.

After settling on a tube design we started on the baffle. We wanted as much surface area as possible so we designed a baffle with a geometry to allow as many pipes as possible. The baffles at the ends of the heat exchanger were designed to be 203.20 mm and 20 mm wide as shown in Figure 4.

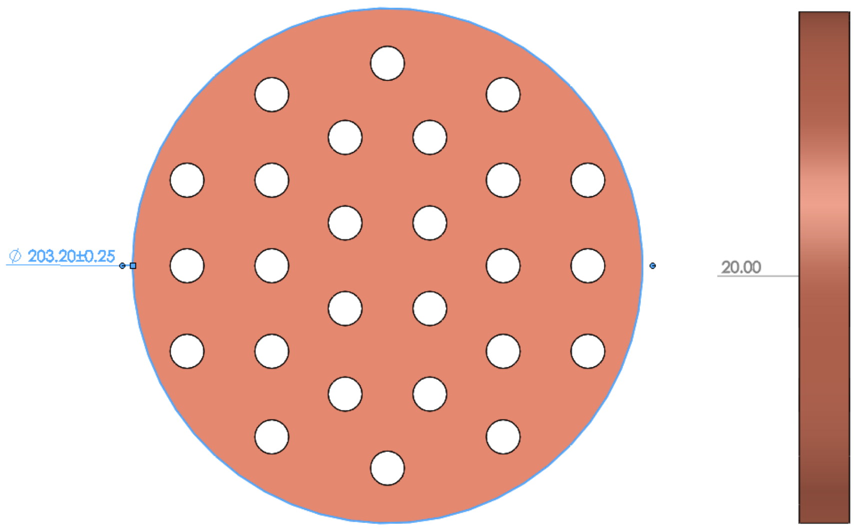

Figure 4. The Baffle Design.

To get as many tubes to fit, a pattern was chosen for the tube layout. The baffle was designed to accommodate 26 tubes. This would allow enough space between the tubed for water to flow properly and allow for sufficient heat transfer.

The inner baffles were of the same design as the baffles at the end only with a section but out to allow for the water to flow. The baffles are positioned in a pattern to allow for the water in the shell to make multiple passes on the tubes. The inner baffle design is with the tube holes spaced out evenly at 33.76 mm as shown in Figure 5.

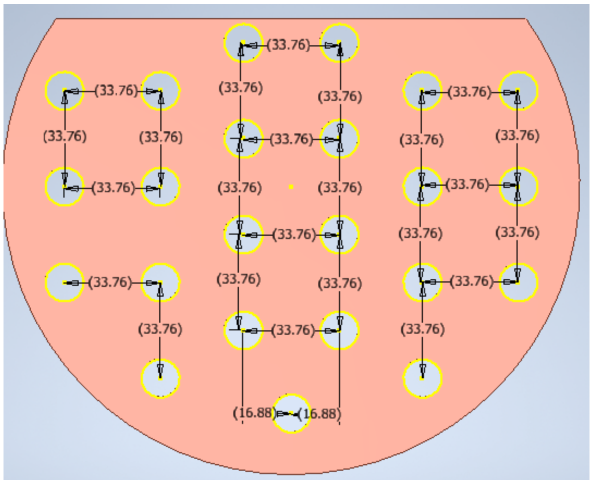

Figure 5. The Inner Baffle with Tube Hole Dimensions.

Once the baffles and te tubes were created, an assembly of the inner tubes and baffles was made. Using the mate function in Solidworks, we combined all the pieces together. The final assembly of the tubes is shown in Figure 6.

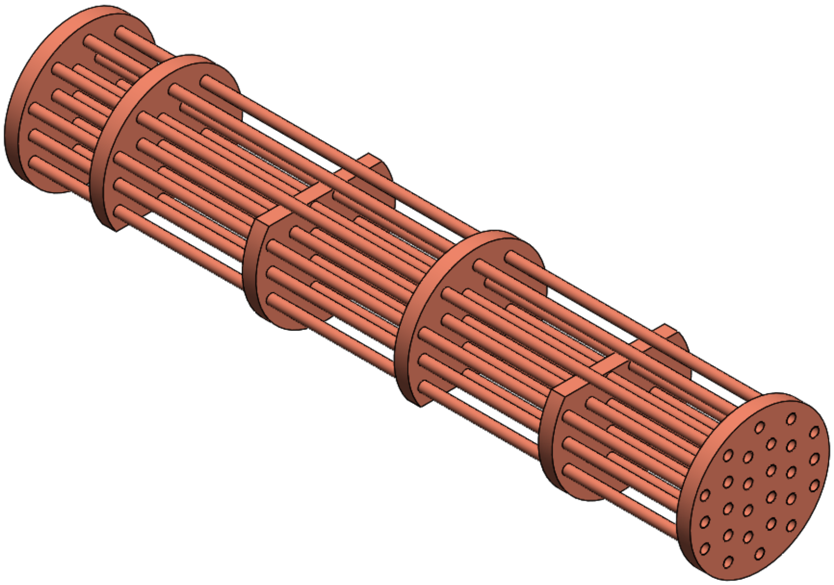

Figure 6. The Tube Assembly with Baffles.

After the tube assembly was completed, the shell was designed to fit the tube assembly. The shell was designed with a length of 1 meter and the same diameter as the baffle to create a water tight fit. The inlets were created with a diameter of 20 mm and were placed at opposite sides of the shell as shown in Figure 7.

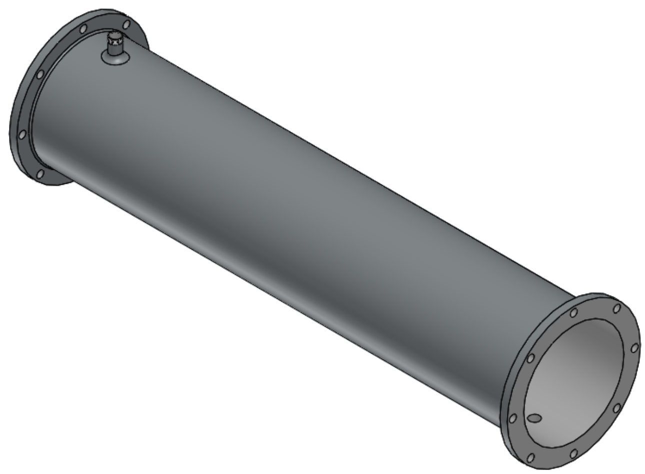

Figure 7. The Shell of the Heat Exchanger.

The end caps were designed to allow fluid enough room to flow and not cause restriction or a vortex. The diameter inside is 203.20 mm and the depth from the flange to the outside is 170 mm. The diameter of the inlet is 20 mm. The design of the endcap is shown in figure 8.

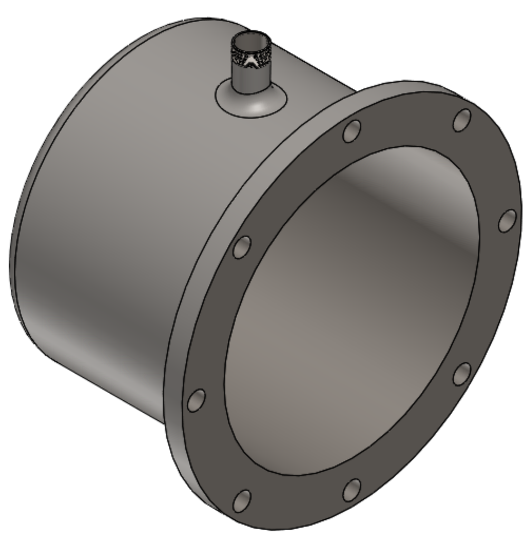

Figure 8. The Endcap of the Heat Exchanger.

Experimental Procedure

After designing all the pieces, the assembly of the full heat exchanger was made as shown in Figure 9. Using the program's fluid simulation, the parameters were set to 50 degrees Celsius for the inlet of the tube and 20 degree Celsius for the inlet of the shell. The simulation was run and the data was collected. The parameters for the simulation were the shell and end caps were adiabatic, the fluid ws water, there was transfer through convection and conduction but no radiation. The system flow rate was set to 3 liters per minute.

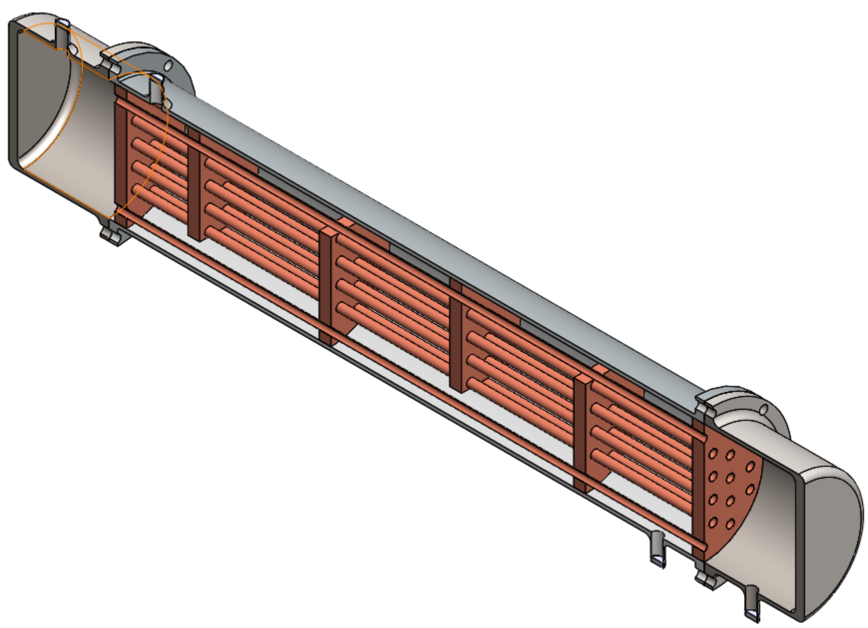

Figure 9. A Cutout View of the Heat Exchanger Assembly.

RESULTS

From the simulation, we can see the temperature distribution of the heat exchanger in Figure 10. The temperature of the fluid drops as it travels through the tube and to the exit. The cold inlet is located at the exit of the tubes, the temperature is lowest by the inlet of the cold water. This counter-flow allows for greater heat transfer since the cold fluid makes its way across the inlet of the hot fluid and absorbs more heat.

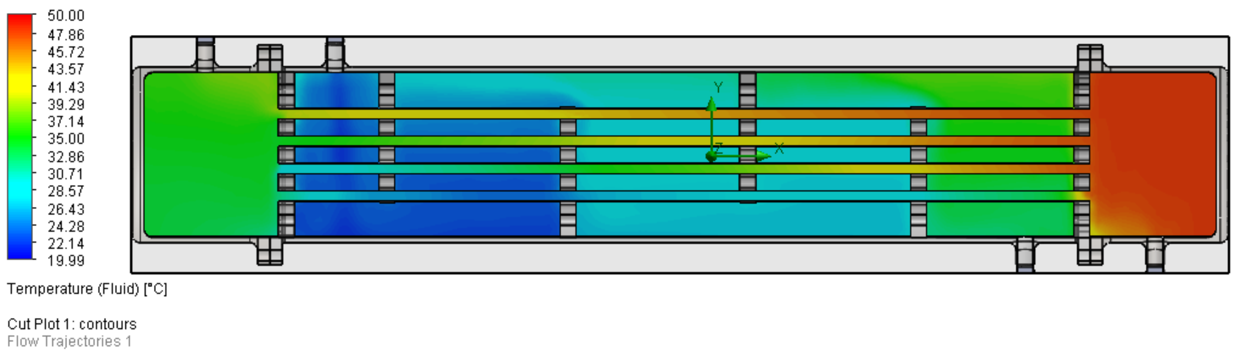

Figure 10. Temperature Distribution of the Fluids Across the Heat Exchanger.

The LMTD was calculated using the program's software. The results of the graph are shown in Figure 11. As the fluid started flowing and heat transfer reached its maximum, the LMTD settled at 18.96℃. The simulation parameters were set to have an adiabatic wall and no roughness in the pipes.

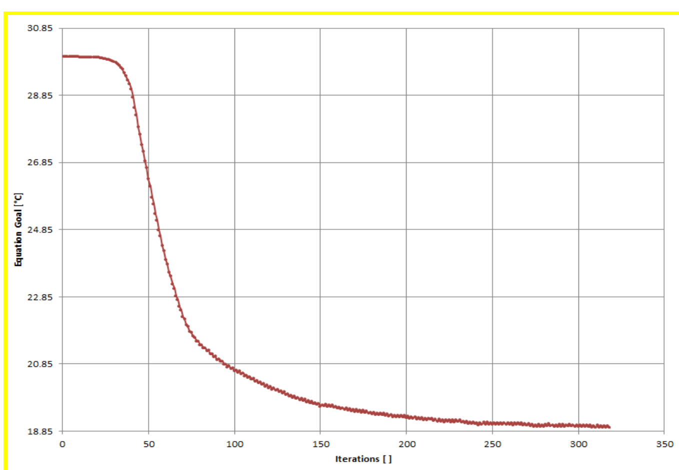

Figure 11. The Calculated LMTD at the Peak Heat Transfer.

To calculate the overall heat transfer rate we use Equation 2. This gives us a value of 2.32 kW. To calculate the Q_{max} , equation 4 is used. This gives us a value of 6.27 kW. With Q and Q_{max} divided the efficiency is about 0.37.

The hot side outlet had a temperature of 38.93℃. The cold side outlet had a temperature of 31.12℃.

DISCUSSION

The results in the simulation provide extensive insight on how the temperature is distributed due to the heat exchanger. With the provided simulation report we understand that the heat transfer that occurs is well within our expectations of the fluids reaching near 35°C. Having an initial cold fluid temperature input of 20°C and an initial hot fluid temperature of 50°C the simulated experiment proved the effects of heat transfer in a shell and tube heat exchanger by revealing that the final cold fluid temperature was 31.12°C and the final hot fluid temperature was 38.93°C. The heat exchanger can be considered successful in terms of providing a mixture of the fluids as shown in Figure 12 below.

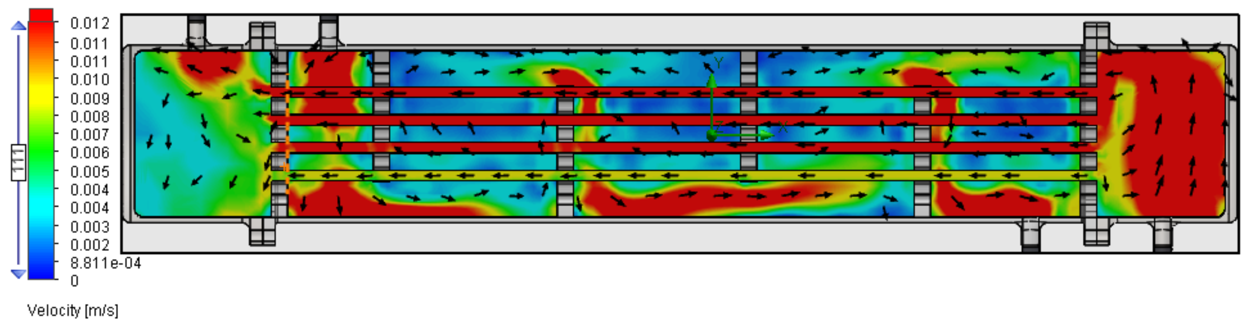

Figure 12. Temperature and Vector Model of Flow and Distribution.

With simulated experimental values the calculations for our Q_{actual} came to be 2.32 kW and this was realistic in terms of how the heat transfer took place during the experiment compared to the calculated $Q_{theoretical}$ which was 6.27 kW. The theoretical value is found under the pretense that there would be a complete transfer of hot and cold fluids taking place. Taking both the actual and theoretical heat transfer we find that the effectiveness of our simulation was roughly .37. This is potentially due to the amount of tubes being used, how the pressure conditions needed to be varied between the inlet and outlet, and also the diameter of the tubes as well for the simulation to take place. Through the simulation it was however confirmed that the temperature of the fluid indeed drops as it travels through the exit tube, establishing the effects of heat transfer taking place in the heat exchanger. As the effectiveness we hoped would be higher, in theory it does give a more practical response to how it would work. And although there are no uncertainties in a simulation, comparing theoretical and actual values will almost always have variance in results.

When analyzing a shell and tube heat exchanger, the types of flow will make a significant difference in how the temperatures are going to differ. Within the shell and tube specifically there are many flows intersecting in the various regions along the tube and baffle areas. Counter-flow and cross-flow for example and between these two the counter-flow allows for the most heat transfer to occur due to the cold fluid flowing across the hot fluid inlet. In Figure 13, we can see that the temperatures are gradually reaching a temperature near each other and this in turn proves that the mixture is taking place over time and the two fluids would verily with enough time and process fully reach an equilibrium temperature.

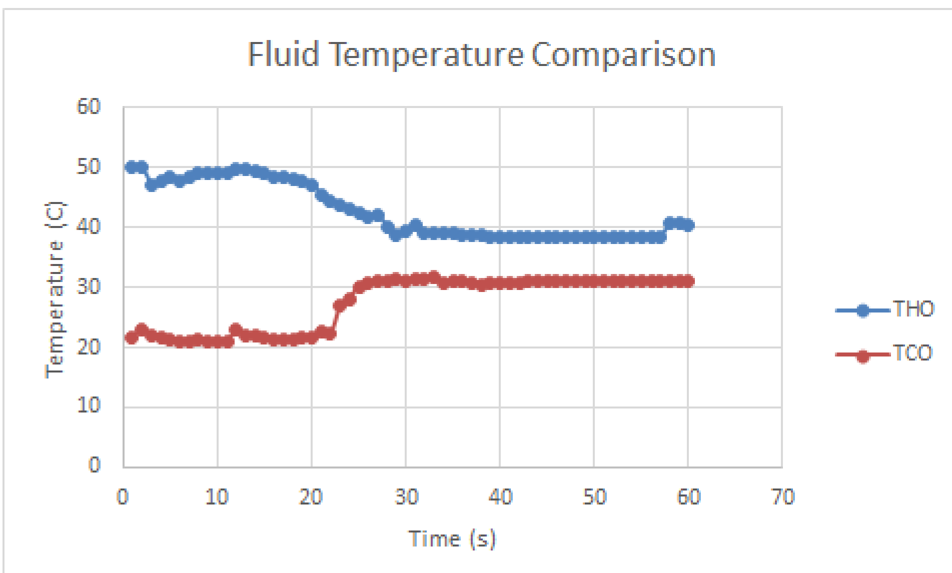

Figure 13. Temperature Change Overtime.

CONCLUSION

Solidworks was used for the experiment instead of Matlab. Solidworks is a better tool when running a program visually. Since Solidworks has 3D visuals to see the flows going in and out of the shell and tube heat exchanger. Solidworks has numerical programming like Matlab does.

Shell and tube heat transfer has an overall equal temperature distribution. The initial conditions of cold inlet of 20℃ and hot inlet of 50℃ both had outlet temperatures of 31.12℃ and 38.93℃. The overall actual heat transfer of the shell and tube heat exchanger was 2.32 kW. The theoretical heat transfer was at 6.27 kW. With those heat transfer values the efficiency was at 37%. The shell and tube heat exchanger had no air inlet and outlet temperature. Since there was no air inlet and outlet temperature to account for, there would be no uncertainty.

REFERENCES

[1] Biswas, Mohammad Abu Rafe. Introduction to Thermal Fluid Laboratory. Houston: THE OPEN EDUCATIONAL RESOURCES FUND, January 19, 2022.

[2] Longwin. (2021, June 11). *LW-9540 Heat Exchanger Experiment*. LW-9540 Heat Exchanger Experiment-Long Win Science and Technology Corporation. Retrieved April 6, 2022, from http://www.longwin.com/english/edu/heat-exchanger.html

APPENDIX A

Name	Task	Hours Spent/Completion
Usman Akram	• Methodology • Conclusion • Reference • Formatting	3.0 hours/completed
Jeremy Baker	• Abstract • Uncertainty • Formatting	6.0 hours/completed
Johnathan Ferguson	• Results • Discussion • Formatting	3.0 hours/completed
Daniel Sigala	• Introduction • Discussion • Formatting • Methodology	12.0 hours/completed