# Evaluation of a Plate Heat Exchanger for Use in the Unit Operations Laboratory at Worcester Polytechnic Institute

An Major Qualifying Project Report completed in partial fulfillment of the requirements of the Bachelor of Science degree in Chemical Engineering at Worcester Polytechnic Institute

Submitted to Professor W.M. Clark

May 1st 2015

By Julie Laliberte and Blanca Zelaya-Rincon

## Abstract

This project sought to evaluate the use of a new plate heat exchanger to replace the current heat transfer experiment in a unit operations laboratory course. Experimental data collected on a new exchanger showed that it would be a viable addition to the course. A 3D computer model of the exchanger was built and simulation results showed the same trends as the experiment. The simulations will allow students to better visualize flow and heat transfer within the exchanger.

## Acknowledgments

We would like to thank the following individuals for their roles in the success of this project:

Professor Clark- For his continued help and support throughout the year, valuable insight and feedback, and dedication to and interest in our project

Nicholas Medeiros- For his persistent help with troubleshooting issues in COMSOL, and his support and encouragement

Tom Partington- For his assistance with ordering parts for the exchangers, and for cutting open an exchanger so that we could see the true plate configuration

Calvin Clark- For lending us his time to provide advice on modeling the exchanger geometry in Solidworks

**Table of Contents**

## 1.0 Introduction and Background

Heat exchangers are an important type of process equipment, widely used in the chemical, petrochemical, and power industry. As such, chemical engineering students study the operation and analysis of heat exchange equipment in the unit operations laboratory course. Key concepts to be investigated include overall and individual heat transfer coefficients and how these coefficients depend on operating conditions, especially fluid velocity inside the exchanger.[1] Another key concept that every student should understand is how and why a countercurrent flow pattern inside the exchanger is superior to a co-current flow pattern.[2]

The unit operations course at Worcester Polytechnic Institute has benefited for many years from the use of a double pipe heat exchanger with steam in the outer pipe and water in the inner pipe.[3] Students study the effect of water flow rate and steam pressure on the heat transfer coefficients describing the resistance to heat transfer in the fluids on either side of the inner pipe. Students also study the effect of countercurrent versus co-current flow. The flow direction study often leads to confusion because there is a difference between the observed and expected behavior for this exchanger. Theory indicates that there should be no difference between the flow patterns for this special case with steam condensing at constant temperature in the outer pipe, but experiments usually reveal an unexpected and difficult to explain difference between the two flow patterns. The cost of steam to operate this heat exchanger has also become an issue in the past few years and it is unclear how long the physical plant will continue to supply steam for this purpose.

**Figure 1.  Double Pipe Heat Exchanger Currently Used in the Unit Operations Laboratory.**

There are a wide variety of heat exchangers in use today including double pipe, shell and tube, and plate heat exchangers.  Of these, plate heat exchangers are increasing in popularity because they are very compact and provide a large surface area for heat exchange in a small volume.[4, 5] This project aims to investigate the operation and analysis of a newly installed plate heat exchanger shown in Figure 2.  This exchanger utilizes hot and cold water from the utility lines in the building and will not require special preparation to supply steam.  It will also allow study of the expected difference between countercurrent and co-current flow.

**Figure 2.  Newly Installed Plate Heat Exchanger with Associated Hot and Cold Water Lines and Flow Meters.**

Unlike other equipment typical in this setting, such as a transparent reacting vessel, a packed tower, or the double pipe heat exchanger, the fluid flow pattern inside the plate heat exchanger is not easy to visualize.  Moreover, course evaluations often indicate that the heat exchanger lab is difficult because convective heat transfer does not come intuitively to some students.  One goal of this project is therefore to develop a 3D computer model of the plate heat exchanger and compare results of simulations from the model to the physical experiment results. It is anticipated that the simulations will allow students to gain confidence in their understanding of the equations governing the fluid flow and heat transfer within the exchanger as well as help them visualize the complex flow pattern inside the exchanger.  COMSOL Multiphysics[6] is a commercial computer software package that provides graphical representations of solutions to

differential equations in defined geometries and is well suited to model the exchanger.

## 2.0 Methodology

### 2.1 Laboratory Experiment

Prior to the collection of any experimental data for this project, a 10-plate heat exchanger purchased from Duda Energy LLC was purchased for and installed in the unit operations laboratory. The exchanger, offering 0.12 m$^2$ of heat transfer area, is a brazed model with copper-welded 304 stainless steel plates. With compact dimensions of 7.5'' x 2.9'', the exchanger is rated to handle flow rates of up to 17.6 gallons per minute at a maximum pressure of 145 psi. Each plate is imprinted with a raised chevron pattern, shown in Figure 3.
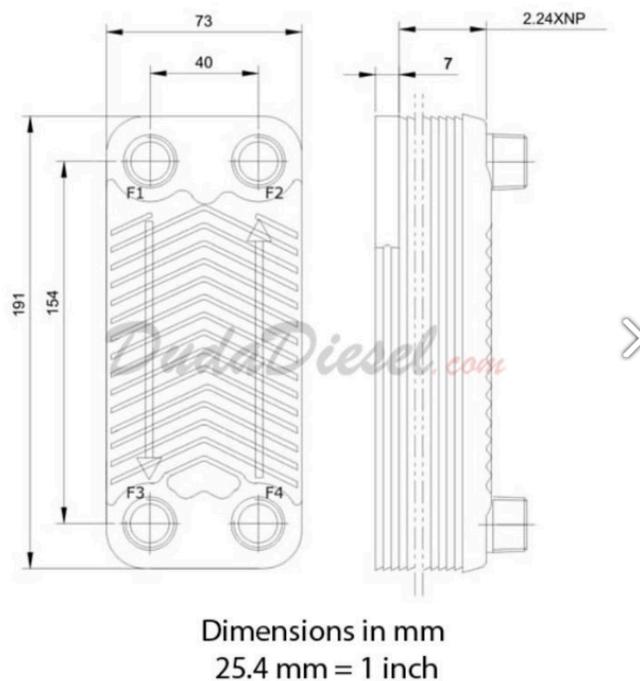


Dimensions in mm
25.4 mm = 1 inch

**Figure 3. Dimensions of the Exchanger and an Individual Plate. Retrieved from Duda Energy LLC http://www.dudadiesel.com/choose_item.php?id=HX1210**

This exchanger was hooked up to a hot and a cold inlet stream branched off of the building's existing plumbing lines. Outlet hoses streams then each flowed through rotameters, and terminated in a nearby floor drain. Immediately threaded into each of the exchanger connections were fittings equipped with thermocouples, four in total. Thermocouples were attached to an electronic readout device, which allowed

temperatures to be viewed on at a time. Furthermore, a differential pressure gauge was hooked in to the cold water inlet and outlet streams to measure pressure drop across the exchanger.

Experimental runs were performed on the exchanger and data recorded on four different dates through the winter of 2014-2015. Prior these trials being carried out, each rotameter was calibrated allowing water to flow for a measured period of time and then weighing that water, enabling the calculation of a volumetric flow rate using the density of water at its recorded outlet temperature. Several flow rates for were measured and plotted for each rotameter to construct a calibration curve that could be fit to a linear equation, permitting the interpolation of unmeasured mass and volumetric flows. During each session, the effect of changing one or more of the following variables was examined; hot water flow rate, cold water flow rate, hot water inlet temperature, and flow configuration. Data recorded for each trial included inlet water temperatures, outlet water temperatures, flow readings, and the time of the run. Cold inlet and outlet pressure were examined during select runs. Additionally, inlet water temperature change with time was recorded from the time the flow was turned on, in order to establish if the building utilities could continuously provide water in the necessary temperature range for this experiment over the course of full laboratory session.

## 2.2 Model Building and COMSOL Simulation

In order to simulate the flow and heat transfer through a model of the exchanger, COMSOL Multiphysics® was chosen as a software program. The unit operations course already utilizes COMSOL for pre-lab assignments, and the chemical engineering department has access to a 'classkit' license to allow multiple students to use to software at once. COMSOL is a finite element analysis program, solving sets of partial differential equations chosen from the physical phenomena selected or input manually by the user, to calculate quantities such as temperature or pressure given a set of initial boundary conditions. COMSOL is able to solve coupled physics

problems, in which variables used to solve for one type of physical occurrence are governed by the solution to another and vice versa.

Use of COMSOL requires that a geometry for a model to apply physics to be constructed. While COMSOL provides tools to create simple geometries within its own interface, it is nowhere close to the capabilities of a dedicated 3D CAD program, and only allows for the drawing of rudimentary shapes and volumes. Due to the complexity and asymmetry of a plate heat exchanger, it was necessary to draw up the model using a separate software program. Solidworks 2014 was chosen for its relative ease of operation by inexperienced users, and its availability to WPI students.

Because of the complicated shape of channel in the exchanger, the metal plates themselves were drawn in Solidworks, instead of a volume in the shape of the fluid flowing through. While the plates were the opposite geometry of where physics would actually occur and need to be studied indicated in COMSOL, these surfaces could be used to slice a block the size of the exchanger into separate fluid channel domains by use of the Boolean operation, 'partition', as shown in Figure 4.
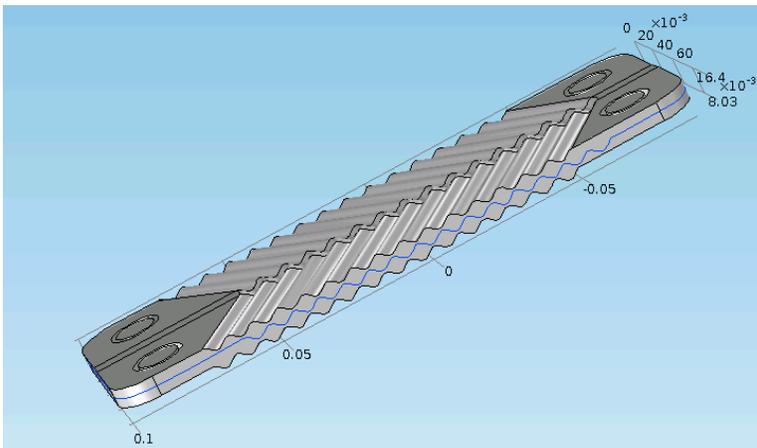


**Figure 4. A Surface Boundary (shown in blue) Used to Partition Two Fluid Domains**

These surfaces represented the finitely thick metal in the physical exchanger, while remaining a 2-dimensional boundary in COMSOL. This allowed for COMSOL to interpret the fluids as touching, and therefore able to exchanger heat between each other. However, in reality, these plates have a thickness and provide resistance to heat transfer. This was accounted for by defining the plate boundaries as a 'thermally resistive layer' with a material of steel added using the built in thermal conductivity of 44 W/m·K and the thickness of the plates indicated by the manufacturer of .118''. Furthermore, this boundary was defined as being an interior wall with a no-slip condition, to prevent the mixing of fluids in adjacent domains and to accompany the laminar flow physics applied to the fluids.

In the same manner as the fluid channels, the inlet and outlet pipe spaces were separated from the channels using the partition function. While the exchanger in the unit operations laboratory is a brazed exchanger, the plate separation had to be simplified to a gasketed model due to limitations of importing geometry from Solidworks that included complex features needed to draw the brazed plates. To represent the gaskets, the ringed domains created around the pipe channels were removed at every other plate from each fluid side, in order to prevent flow into the other fluid's domain. Figure 5 shows the configuration of gaskets, with all but two fluid domains hidden.
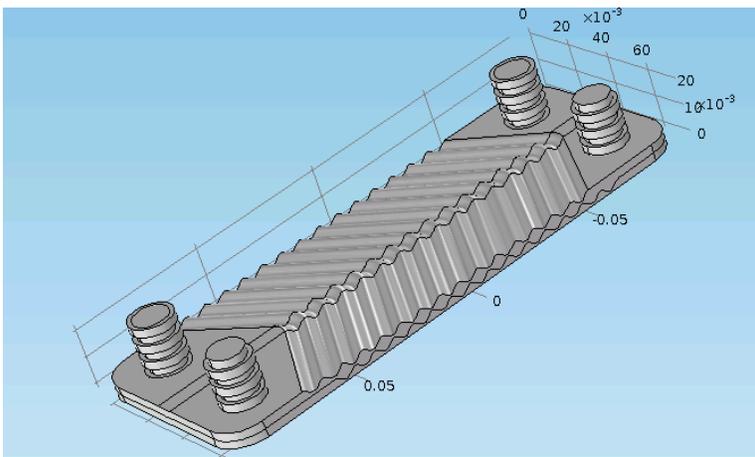


**Figure 5. Gaskets Created Around Piping to Separate the Two Fluids**

The dimensions used for the plates in Solidworks were taken from the manufacturer's information on the exchanger, in order to model it as closely as possible to the physical exchanger in the laboratory that it would be compared to. To mimic the corrugation of the plates, a sine wave was implemented as the side of each plate, with straight segments on the ends. The availability of a second exchanger, identical to the one used for experiments but with the sides cut away, provided insight into how to model the plate set-up. While the exchanger is advertised as having 10 plates, it was determined by looking at the side of the plates that this number counter top and bottom plates which equated to only eight fluid channels being present, as shown in Figure 6. For this reason, only seven heat transfer boundaries were used to split the fluids in the model to create the correct number of domains, therefore allowing for the approximately correct amount of fluid to flow through each when experimental data was applied to the simulation.
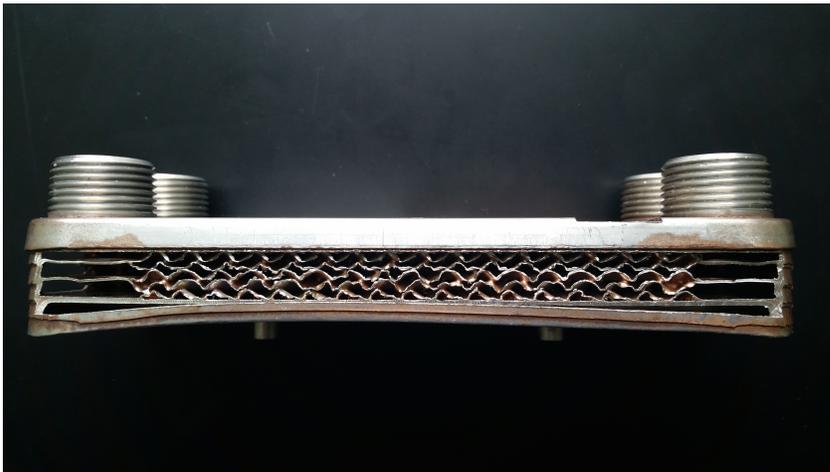


**Figure 6. A Side View of the Brazed Plates of a Cut Exchanger**

Upon the completion of the model geometry, physics were added to the applicable boundaries and domains. Instead of using a flow condition coupled with convective heat transfer, COMSOL offers a single physics module to account for both as 'non-isothermal flow'. This was used in all fluid domains, with each fluid being defined as separate materials of water (1) and water (2), for the hot and cold sides respectively. Laminar flow was used as an approximation, due to turbulent models

being too computationally taxing, and flow was defined as being incompressible. Boundary conditions of pressure and mass flow were added, with atmospheric pressure at the outlets, and flows based on experimental trials at the inlets. Temperatures corresponding to each experimental run were added to the inlets as well. Furthermore, in reality, the exchanger is exposed to ambient conditions and free convective cooling by air at the outer walls. However, for the purpose of this study, it was assumed that all energy is conserved within the fluid and no heat loss occurs. By default, this was accounted for by outside walls being defined as thermally insulated. A complete report on the set-up of one example simulation is shown in Appendix A.

## 3.0 Findings

### 3.1 Experimental Data

After collecting data from the exchanger, it was evaluated based on the fundamental equations governing heat transfer. The amount of heat transfer (Q) was found using the mass flows (m), heat capacity of water at an average temperature between the inlet and outlet (Cp), and the change in temperature for each side ($\Delta T$), where $Q = m \cdot Cp \cdot \Delta T$. This equation was applied to both the hot and cold sides of the exchanger. By completing and energy balance around the exchanger, and essentially comparing the heat taken on by the cold stream to the heat given off by the hot stream, it is evident that some heat is lost to the environment. This amount was found to vary from 1-6 Watts, which was typically one third to one quarter of the total amount of heat transfer, a significant loss.

The heat transfer coefficient (U) was then found for each trial using the amount of energy transferred (Q), the area available for heat transfer (A), and the logarithmic mean temperature difference between the hot fluid and the cold fluid at each side of the exchanger using the equation $Q = U \cdot A \cdot \Delta T_{LM}$. This heat transfer coefficient was the dependent variable that would be compared between runs to determine trends.

An important finding from each laboratory session was that the temperature of the hot water would drop for a period of time, and then either level off or fluctuate. In the first three sessions, the hot water inlet temperature varied enough that the dependence of heat transfer and heat transfer coefficient could be determined. While heat transfer coefficient does not vary much if at all from temperature (only due to the slight change in the viscosity of water), it is a valuable exercise for the unit operations students completing this experiment to be able to find that out on their own, and to see if they can do so correctly. Figure 7 shows the hot water inlet temperature change with time on two different days.
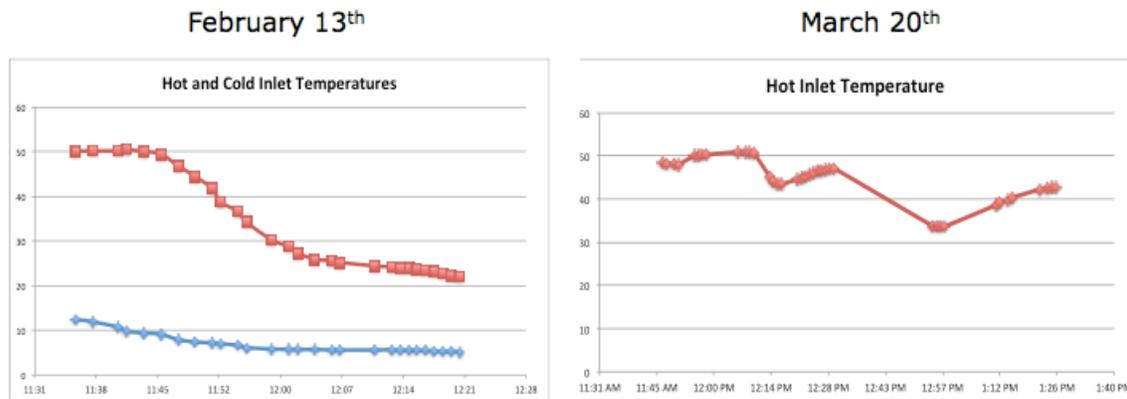


**Figure 7. Change in Temperature of Available Hot Water with Time**

Because the cold temperature always stays fairly steady after an initial drop off period, it was important that students be able to manipulate the hot inlet temperature instead of being at the mercy of the building utilities on any particular day. To make this possible, a second heat exchanger was added to the hot stream before if enters the main exchanger. The hot water hose was equipped with quick connects so that this exchanger could be skipped if not necessary. To cool the hot stream, the cold water supply was split with a tee before it enters the first exchanger, with valves added to control either flow. The charts in Figure 8 show the impact of changing temperature. Each differently colored line represent a set of trials at different cold water flow rates. Each color is a different log mean temperature difference. It is evident that having a higher log mean temperature

difference means more heat will be transferred, but as predicted, it is verified that temperature does not affect overall heat transfer coefficient, as it is not temperature dependent.
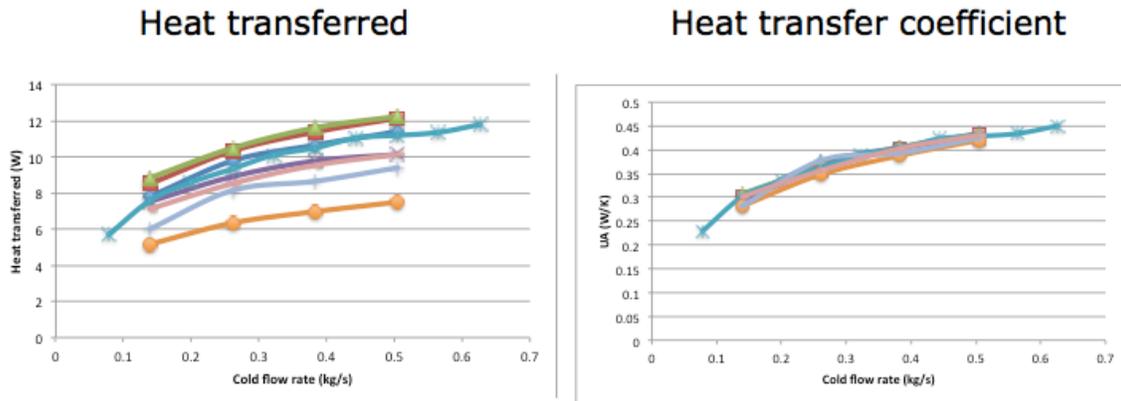


**Figure 8. The Effect of Temperature on Heat Transfer and Heat Transfer Coefficient**

## 3.2 COMSOL Simulations

Aside from the comparison to experimental data, discussed later in this report, the major important finding with the COMSOL simulations was the need to reduce the model to a '1-plate' 2 fluid domain version. While the simulation on the full model did converge, the mesh could not be refined enough to assure the solutions were mesh independent. When the mesh was refined further, COMSOL would give an error message and a solution would not be computed. Figure 9 shows the difference in velocity plots for an 'extra coarse' mesh used on the full model, compared to a 'coarse' mesh on the reduced model. With the finer mesh, it is clear how the velocity increases between the pinched corrugates of the plates.
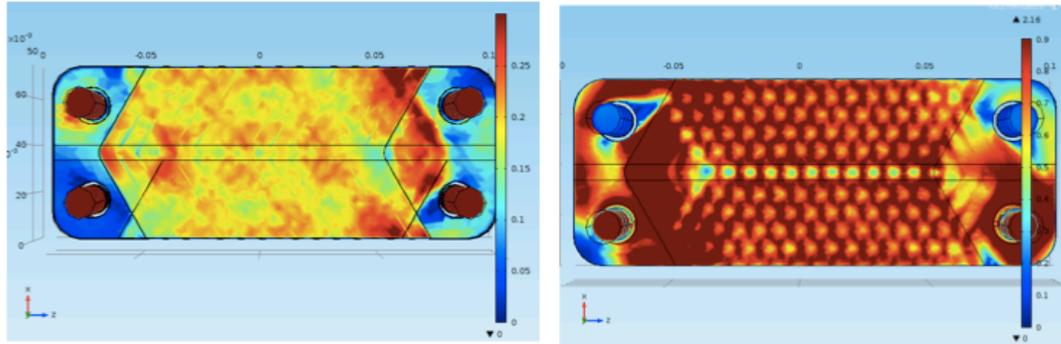
15

**Figure 9. An Extra Coarse Mesh on The Left, Compared to a Coarse Mesh on the Right**

The heat transfer coefficients found from each model were then compared to verify that moving to the reduced model was a valid choice. To accommodate for the smaller amount of total cross sectional area for flow due to the reduced number of channels, flows were decreased by a factor of four for input into COMSOL on the 2-domain model. The area available for heat transfer was taken to be one seventh of that of the larger model, as one plate was used instead of seven. The resulting heat transfer curves are shown in Figure 10. The heat transfer coefficients for each model were very similar and followed the same trendline, making the reduced model a valid option for simulation that allows for the use of a finer mesh and has a faster solution time.

## 3.3 Comparison of Experiment and Simulation

The most significant result to be compared between the simulation and experiment was the overall heat transfer coefficient, with dependence on flow shown in Figure 10.
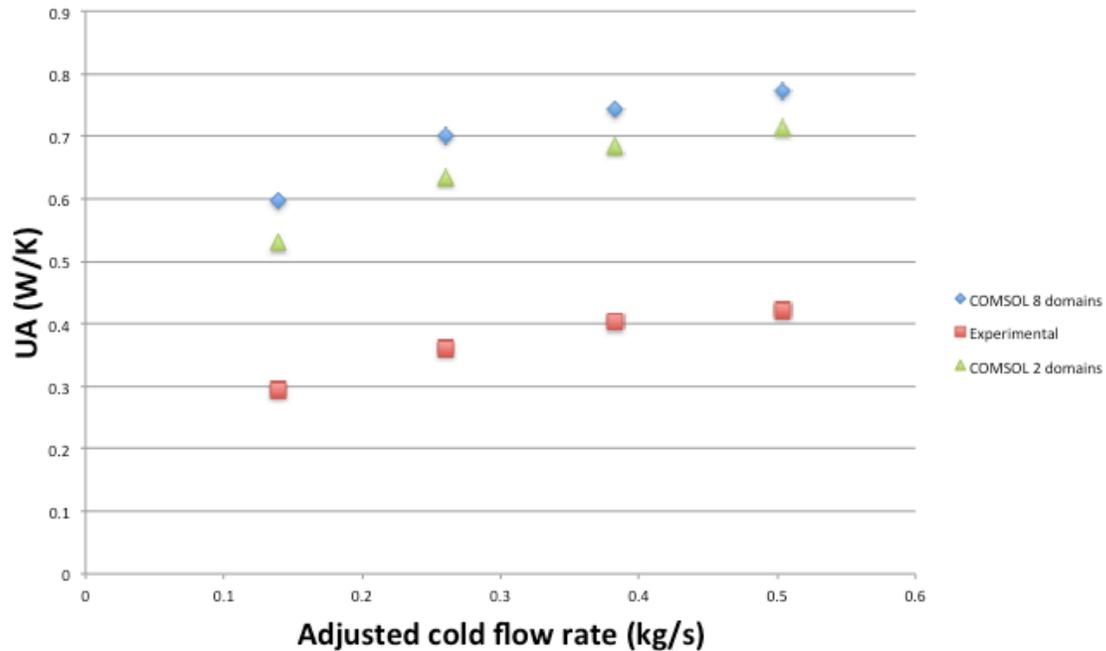


**Figure 10. Heat Transfer Coefficent change with Cold Flow Rate for Two COMSOL Models and the Equivalent Experiment**

While the experiment showed a lower heat transfer coefficient by nearly a factor of two, it followed nearly the same curve as both simulations. The difference may be due to not accounting for heat loss in the simulation, the difference in geometry from the real exchanger, or simply the limitations of simulation due to other assumptions made by the software.

Other results examined such as pressure drop flowed the same pattern of both the simulation and experiment showing the same trends, but at different magnitudes. Pressure drop was much lower in the simulation, even when looking at the full model, which included all channels. Figure 11 shows the pressure drop trends for each case.
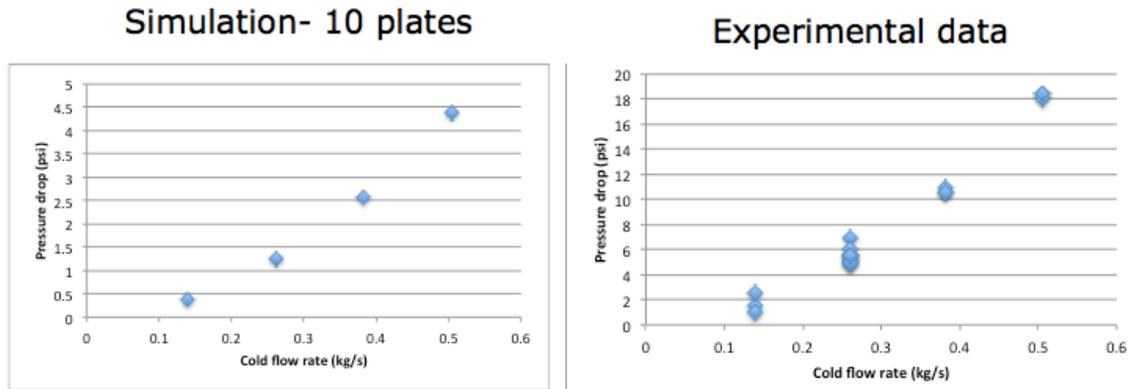
**Figure 11. Pressure Drop Compared Between a Simulation and the Experiment**

The final result compared between each method was the effect of flow configuration on heat transfer. Heat exchangers with a countercurrent flow pattern should have a high heat transfer coefficient than those with concurrent flow, due to a higher total driving force of temperature difference across the exchanger. Having students in the unit operation lab evaluate the impact of flow configuration has been a problem in the past due to the double-pipe heat exchanger not showing the correct trends for this factor. However, the new plate exchanger consistently demonstrated a higher heat transfer coefficient in a countercurrent configuration, in agreement with the simulation, as shown in Figure 12. While the difference between configurations was fairly small, students who make careful measurements should effectively always get the correct result.
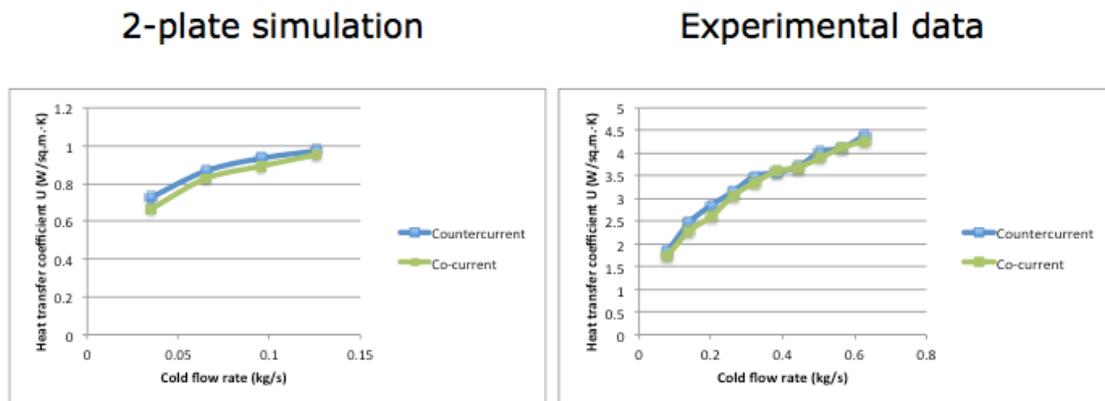


**Figure 12. Effect of Flow Configuration on Heat Transfer Coefficient**

## 4.0 Conclusions and Recommendations

### 4.1 Conclusions

Overall, due to the consistent matching of trends between the simulation and the corresponding experimental data, the model will be able to serve as a tool for unit operations students. Results for heat transfer coefficients, pressure drop, and flow configurations were all in agreement by both methods. It was found that a reduced 2-plate model can serve as an adequate substitute for a larger model when flow rates are reduced accordingly. This smaller model will allow for a finer mesh to be used and quicker computation times. The experiment works as expected and is well suited to the a four hour laboratory period of a unit operations class. With the addition of a second heat exchanger, students are now able to keep the hot water supply at a fairly constant temperature of their choosing. Students will be able to see the impact of flow rate and flow configuration on heat transfer coefficient, and also that temperature does not have an impact on the coefficient.

### 4.2 Recommendations

Several steps could be undertaken to assure that further work with this model:

1) A model with more accurate geometry could be worked towards, to assure this is not causing too much discrepancy between experimental results and the simulation. A brazed model could be developed instead of a gasketed model.

2) Heat loss to ambient conditions should be accounted for the match the reality of convective cooling by air. COMSOL has options to add this condition, but they were not functioning properly when their use was attempted during this project.

3) The chemical engineering department needs to secure access to a more powerful server if unit operations students are to be able to run this simulation during their pre-lab assignments for the experiment. While the 2-plat model simulations take only a few minutes on a coarse mesh, they require up to 90 minutes to compute with the finer mesh that allows students to see the true velocity profile.

4) To facilitate use of the simulation in the unit operations lab, the model could be adapted to the application builder, a new feature in COMSOL 5.0. This tool allows for a simulation to be done in a simpler interface, where only a handful of parameters can be easily changed, and will compute using less server power.

**References**

1. McCabe, W. L., Smith, J. C., and Harriott, P., Unit Operations of Chemical Engineering, Seventh Ed., McGraw Hill, New York, (2005), pp. 335-340.


2. McCabe, W. L., Smith, J. C., and Harriott, P., Unit Operations of Chemical Engineering, Seventh Ed., McGraw Hill, New York, (2005), pp. 326-329.

3. Clark, W. M., "Experiment No. 6: Heat Transfer from Condensing Steam to Water in a Double Pipe Heat Exchanger", ChE 4401, Unit Operations Laboratory I, Experiment Description, (2013).

4. http://www.alfalaval.com/products-and-solutions/plate-heat-exchangers/Pages/plate-heat-exchangers.aspx

5.  McCabe, W. L., Smith, J. C., and Harriott, P., Unit Operations of Chemical Engineering, Seventh Ed., McGraw Hill, New York, (2005), pp. 455-459.

6.  COMSOL Multiphysics

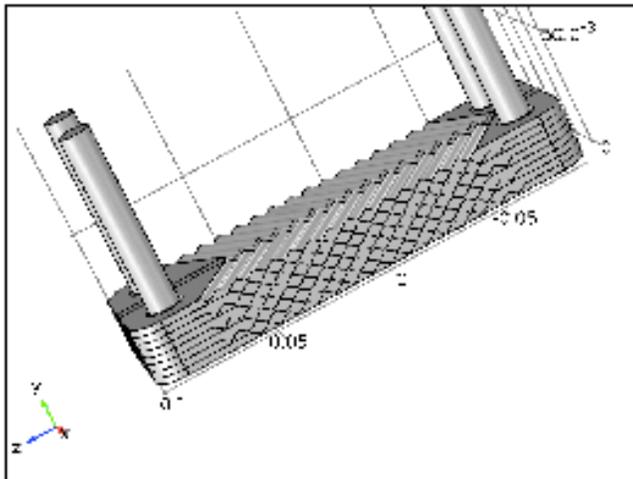- **Appendix A: Example of a COMSOL Run Report**

  - ## Component 1 (comp1)

  - ### Definitions

    - #### Coordinate Systems

*Boundary System 1*

| Coordinate system type | Boundary system |
|---|---|
| Identifier | sys1 |

  - ### Geometry 1



*Geometry 1*

**Units**

| Length unit | m |
|---|---|
| Angular unit | deg |

- **Materials**
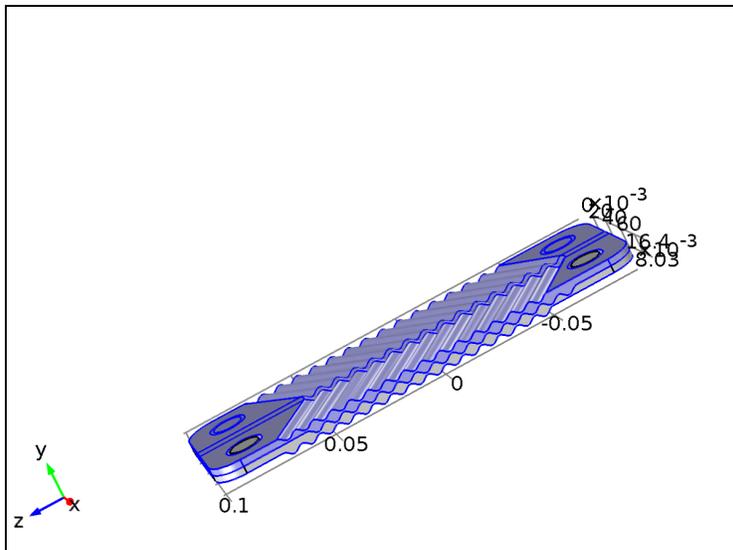
- **Steel AISI 4340**
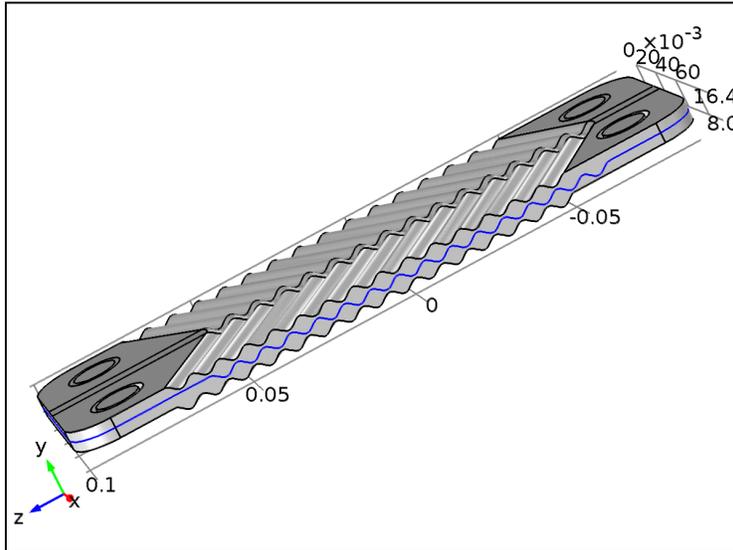


*Steel AISI 4340*

**Selection**

| Geometric entity level | Boundary |
|---|---|
| Selection | Boundaries 2, 6, 8–9, 12, 14–15, 18, 20–21, 30, 33, 42, 44–45, 47–49, 170–175, 178, 182–190, 194, 198–201 |

- **Water, liquid**



*Water, liquid*

**Selection**
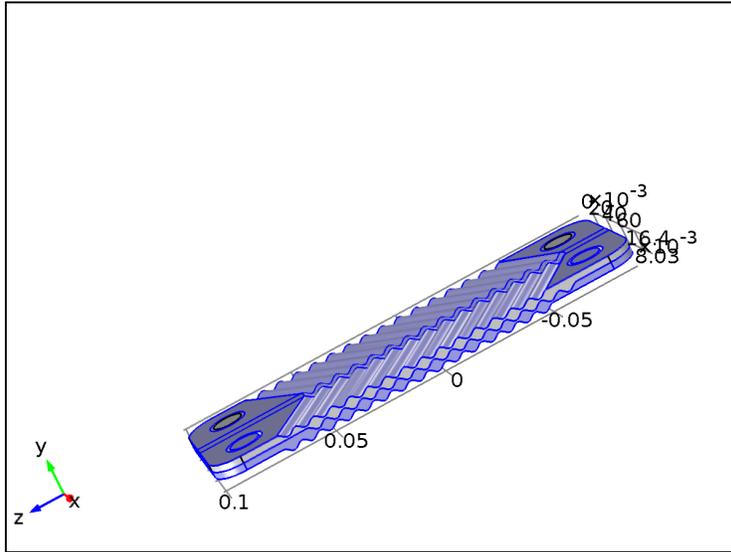
| Geometric entity level | Domain |
| --- | --- |
| Selection | Domains 2, 4, 6, 8–34 |

- **Water, liquid (2)**



*Water, liquid (2)*

**Selection**

| Geometric entity level | Domain |
| --- | --- |
| Selection | Domains 1, 3, 5, 7, 35–60 |

- **Non-Isothermal Flow (nitf)**
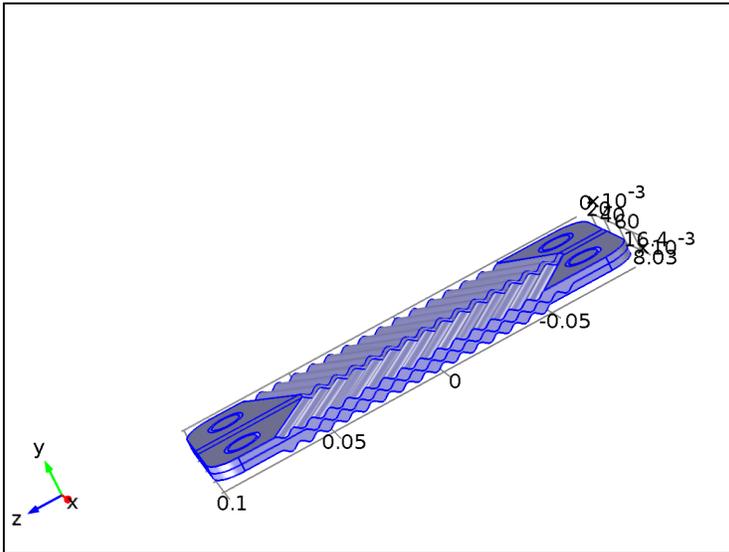


*Non-Isothermal Flow*

**Features**

| |
|---|
| Fluid 1 |
| Thermal Insulation 1 |
| Wall 1 |
| Initial Values 1 |
| Fluid 2 |
| Inlet 1 |
| Outlet 1 |
| Inlet 2 |
| Outlet 2 |
| Temperature 1 |
| Temperature 2 |
| Thin Thermally Resistive Layer 1 |
| Interior Wall 1 |

- **Mesh 1**
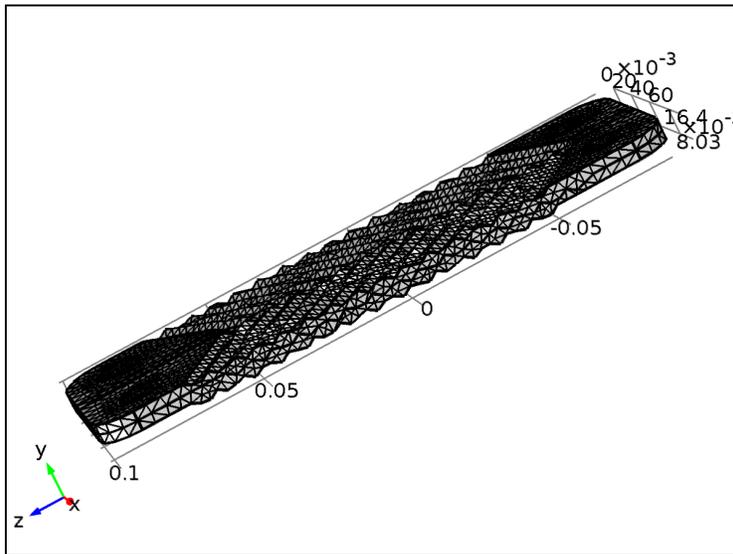


*Mesh 1*

- **Study 1**

- **Stationary**

**Study settings**

| Property | Value |
|---|---|
| Include geometric nonlinearity | Off |

**Mesh selection**

| Geometry | Mesh |
|---|---|
| Geometry 1 (geom1) | mesh1 |

**Physics selection**

| Physics | Discretization |
|---|---|
| Non-Isothermal Flow (nitf) | physics |

## Appendix B: Rotameter Calibration Curves



**Cold Water Calibration**

$y = 0.0965x + 0.2813$
$R^2 = 0.99834$

Flow (gal/min) vs. Rotameter Reading



**Hot Water Calibration**

$y = 1.2599x - 0.1186$
$R^2 = 0.99016$

Flow (gal/min) vs. Rotameter Reading
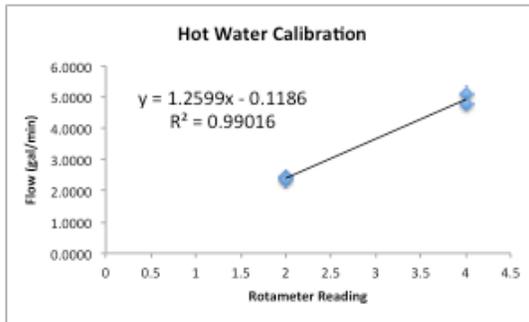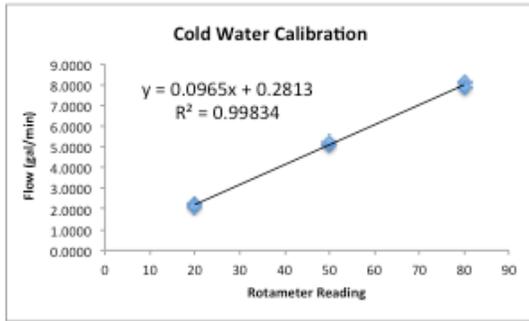
## Appendix C: Raw Data From a Sample of Experimental Trials

run #, Cold inlet temp (°C), Cold outlet temp (°C), Hot inlet temp (°C),  Hot outlet temp (°C), Cold inlet pressure (psi), Cold outlet pressure (psi), Cold flow reading, Hot flow reading

| run # | Cold inlet temp | Cold outlet temp | Hot inlet temp | Hot outlet temp | Cold flow reading | Hot flow reading |
|---|---|---|---|---|---|---|
| 1 | 7.65 | 21 | 48.4 | 32.6 | 20 | 2 |
| 2 | 7.35 | 16.3 | 48.3 | 28.5 | 40 | 2 |
| 3 | 6.75 | 13.4 | 48.1 | 26.1 | 60 | 2 |
| 4 | 6.35 | 11.75 | 48 | 24.6 | 80 | 2 |
| 5 | 6.05 | 20.6 | 50 | 32.9 | 20 | 2 |
| 6 | 5.95 | 15.4 | 50.2 | 28.75 | 40 | 2 |
| 7 | 5.9 | 13 | 50.25 | 26.7 | 60 | 2 |
| 8 | 5.75 | 11.5 | 50.3 | 25.25 | 80 | 2 |
| 9 | 5.75 | 20.9 | 50.9 | 33.15 | 20 | 2 |
| 10 | 5.65 | 15.25 | 51 | 28.65 | 40 | 2 |
| 11 | 5.5 | 12.75 | 50.9 | 26.35 | 60 | 2 |
| 12 | 5.4 | 11.2 | 50.65 | 24.85 | 80 | 2 |
| 13 | 5.5 | 18.4 | 45.2 | 29.9 | 20 | 2 |
| 14 | 5.45 | 13.6 | 44 | 25.2 | 40 | 2 |
| 15 | 5.45 | 11.55 | 43.5 | 23.3 | 60 | 2 |
| 16 | 5.35 | 10.15 | 43.4 | 22.1 | 80 | 2 |
| 17 | 5.5 | 22.8 | 44.65 | 33.5 | 10 | 2 |
| 18 | 5.6 | 18.6 | 44.95 | 29.6 | 20 | 2 |
| 19 | 5.5 | 15.75 | 45.3 | 27.6 | 30 | 2 |
| 20 | 5.5 | 14.05 | 45.7 | 26.2 | 40 | 2 |
| 21 | 5.4 | 12.9 | 46.1 | 25.3 | 50 | 2 |
| 22 | 5.35 | 11.9 | 46.4 | 24.5 | 60 | 2 |
| 23 | 5.3 | 11.25 | 46.6 | 23.9 | 70 | 2 |
| 24 | 5.3 | 10.6 | 46.8 | 23.4 | 80 | 2 |
| 25 | 5.3 | 10.1 | 47 | 23 | 90 | 2 |