

Study of The Thermal Performance of a Shell and Tube Heat Exchanger

(A Computational Fluid Dynamics Approach)



Aness Rashedi (9422221125)

A thesis submitted in partial fulfillment of the requirements for the degree of
Bachelor of Science in Chemical Engineering
at
Bahá'í Institute for Higher Education (BIHE)

Supervised by
Soroush Ahmadi

2020

Abstract

Heat exchangers are used in various industries to exchange thermal energy between hot and cold streams. Thereby, heat exchangers are an invaluable tool for processes integration to minimize the need for external sources of energy. There are numerous types of heat exchangers, but shell and tube heat exchangers (STHXs) are by far the most ubiquitous type in the market. Baffles are an integral part of the STHX design, used in order to increase the thermal efficiency, and support the tube bundle from flow-induced vibrations.

The existence of baffles entails several adjustable design parameters that can affect the pressure drop and heat transfer rate drastically. The number of baffles, baffle cut, baffle spacing, and baffle orientation are just a few key parameters that can be fine-tuned for optimization purposes. Evidently, the experimental modification of all these variables is tedious and costly, and a computational approach must be used.

In this project, Ansys Fluent is employed as a computational fluid dynamic (CFD) code to perform an in-depth investigation of various design elements of baffles. Also, experimental data on an available STHX is collected and compared with our modeling results to ensure the validity of our model.

The number of baffles is changed from zero to ten, and the resulting pressure drop (the main drawback) and heat transfer rate (the main benefit) of each case are compared. It was found that four baffles are the optimum number of baffles, whereas only three baffles were present in the experimental unit. We then considered three baffle cut orientations: up-down, left-right, and clockwise but the changes in performance were marginal. We concluded the baffle cut orientation is not a notable design variable, and the default up-down orientation is acceptable. Finally, the baffle cut was modified that resulted in significant enhancement of the thermal performance of the device.

Overall, CFD enabled us to improve the thermal efficiency of an STHX by ten percent, and, at the same time, decrease its pressure drop to one third. This approach has been proven effective and can be scaled up to large industrial STHXs.

Acknowledgments

I would like to sincerely gratitude the supervisor of this project, Soroush Ahmadi, who accompanied me patiently along the way and guided me in the technical aspects of this thesis. This project would have been difficult, if not impossible, without the support of one of my dear peers, Sahar Bayani. I would also like to express my gratitude and appreciation to the Chemical Engineering Department of BIHE that supported me in the education process and also provided vital computational resources for this project. Lastly, I express my sincere thanks to my family, especially my parents, who have been nothing but supportive; I am grateful for their love, prayers, caring and sacrifices for educating and preparing me for the future.

Table of Contents

Abstract.....	2
Acknowledgments	3
1. Introduction	5
1.1 Type of Heat Exchangers.....	5
1.2 Shell and Tube Heat Exchanger (STHX).....	6
1.2.1 Baffles.....	8
1.3 Scope of Work.....	10
2. Review of Previous Work.....	12
2.1 Experimental Studies	12
2.2 Computational Fluid Dynamics Studies	13
3. Methodology:.....	17
3.1 Heat Transfer Calculation in STHXs.....	17
3.2 Governing Equations of a CFD Study	25
3.2.1 Continuity Equation (Conservation of Mass).....	26
3.2.2 Conservation of Momentum	26
3.2.3 Energy Equation	27
3.2.4 Turbulence Modeling.....	27
3.3 Geometry	29
3.4 Discretization of the 3D Model (Meshing)	30
3.5 Ansys Fluent Configuration (Setup and Solution).....	34
4. Result and Discussion	39
4.1 Study the Effect of Shell Outlet	39
4.2 Comparison Between Experimental and Simulation Data (Validation)	39
4.3 Comparison between our results and well-known correlations. (Verification).....	44
4.4 Temperature Distributions	46
4.5 Study the Effect of Number of Baffles	50
4.6 Study the Effect of Baffle Cut Orientations	54
4.7 Final Suggestions on the Design Modification	56
5. Conclusion.....	57
6. Recommendations and Suggestions for Future Work	58
References	59

1. Introduction

In many industries, ranging from food processing to large-scale oil and gas refineries, heat exchangers are widely used to interchange thermal energy between hot and cold streams to conserve energy.¹ Heat exchangers are designed to allow heat to flow between two streams without direct contact or merging the fluids.²

1.1 Type of Heat Exchangers

The widespread application of HXs resulted in countless types of heat exchangers that differ in construction and type of operation. As can be seen in Figure 1, HXs can be classified into five general categories, namely plate, tabular, extended, regenerative, and proprietary.¹

An exhaustive description of all types of HXs is beyond the scope of this project. Therefore, we are going to focus only on the tabular category and briefly introduce its subcategories. Tubular HXs are by far the most common category, that encompasses double pipe, shell and tube, and spiral tube HXs.¹

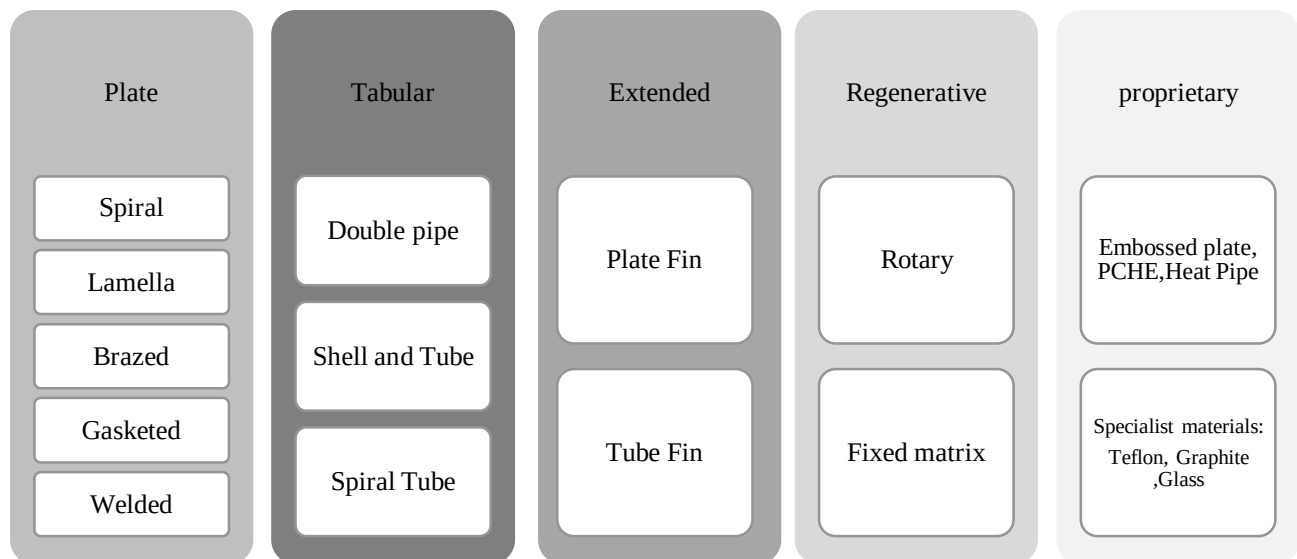


Figure 1.Types of heat exchangers¹

The simplest type of tubular HXs, consisting of two concentric tubes, is called double pipe HX. The schematic design of a double pipe HX is shown in Figure 2.a.³ Due to its simplicity in design, they are widely used for small-scale heat transfer applications, additionally, Since the transmission rate of heat is comparably small, double pipe HXs are preferred for temperature-sensitive compounds, such as pharmaceuticals and polymers to avoid decomposition or degradation. Double pipe heat exchangers

can also handle high temperature and pressure streams. The main drawback of this type of exchangers is the cost per surface area due to its low efficiency and contact time.⁴

Spiral Tube HX is constituted of a spiral coil that is located in a large diameter shell. One fluid passes through the spiral pipe and transfers heat to the fluid in the shell-side. The heat transfer rate in the spiral tube is considerably higher than that of a straight tube, leading to a more efficient exchange of heat. On the shell-side, however, the complexity of the spiral tube hinders the possibility of using baffles to further improve the heat transfer rate.⁴ The schematic design of this class of tubular HXs can be found in Figure 2.b.⁵

Another type of tubular HXs is the shell and tube (STHX).³ STHXs are by far the most ubiquitous class of exchangers in the market. This equipment is composed of a number of tubes, known as a tube bundle, positioned in a larger vessel (shell). The exchange of heat occurs through the tube wall between the shell-side and the tube-side fluids. As shown in Figure 2.c,⁶ the flow in the shell-side is conventionally disturbed by inserting some obstacles, known as baffles.⁴ In the next section, we will describe shell and tube heat exchangers in more detail.

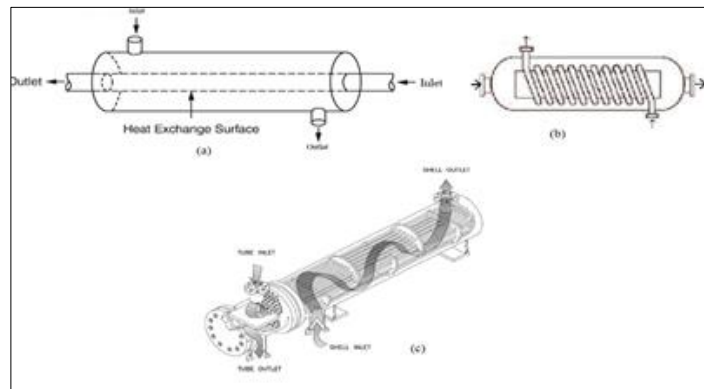


Figure 2.(a) double pipe heat exchanger³, (b) spiral tube heat exchanger, and⁵ (c) shell and tube heat exchanger⁶.

1.2 Shell and Tube Heat Exchanger (STHX)

Shell and tube HXs are overwhelmingly flexible in design; a wide range of parameters, including number of tubes, number of baffles, baffle cuts, tube and shell diameters, flow direction, and etc., can be modified to suit a wide range of applications. They can cope with high pressure and temperature flows, and, provided that the design is optimized, they are highly efficient.

Due to its industrial significance, Tubular Exchanger Manufacturers Association, TEMA for short, has established a set of standards for STHXs that are widely accepted and followed in the industry. Currently, the 10th edition of these standards is available online for \$650. We are not going to give a detailed explanation of these standards, but instead, focus on basic components and configurations of STHXs.

The number of tube passes inside the shell is one of the basic concepts that can increase the thermal efficiency of an STHX. Increasing the number of passes brings the fluids into contact with each other several times, exchanging heat each time one medium passes over the other. As shown in Figure 3, we can have both odd and even numbers of tube passes.⁷

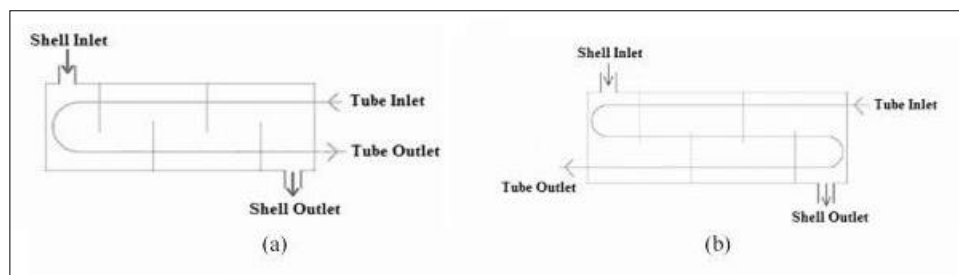


Figure 3. (a) even, and (b) odd number of tube passes in STHXs.⁷

Shell and tube heat exchangers can also be distinguished based on the flow direction. There are two flow configurations, namely co-current and counter-current, illustrated in Figure 4.² In co-current, the inlets of the hot and cold fluids are on the same side, leading to parallel passage of the flow in the STHX. In this case, the driving force for heat transfer, which is the local temperature difference between hot and cold fluids, starts at a very large number at the inlet, and gradually declines towards the end. In counter-current, on the other hand, hot and cold currents are moving in the opposite directions, keeping a relatively constant local temperature difference between the cold and hot streams across the heat exchanger length.²

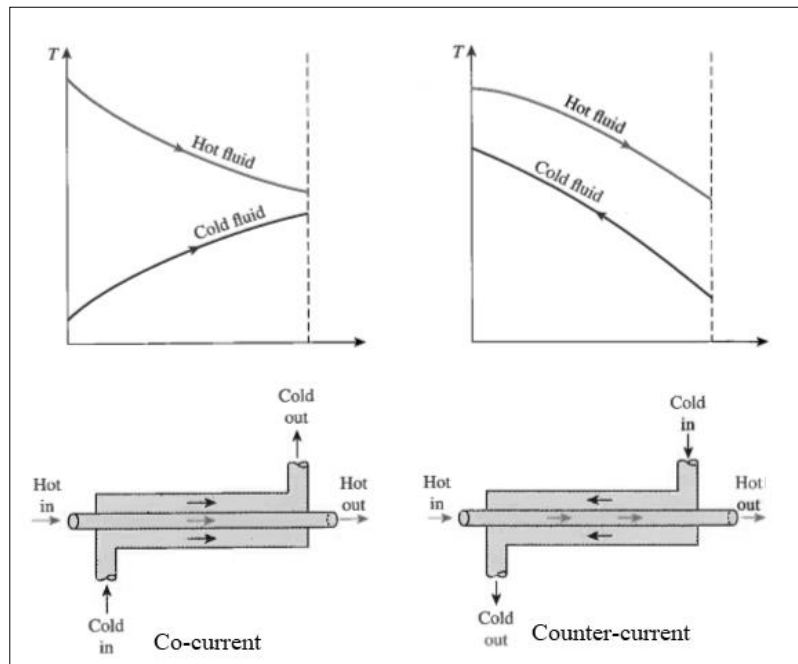


Figure 4. Co-Current and Counter-Current²

1.2.1 Baffles

Baffles are panels that are used in various equipment, such as STHXs, chemical reactors, and mixers, in order to enhance the motion of the fluid in a desired way. In other words, baffles increase the fluid path line, direct flow in a desired path, and enhance turbulence in the system. In STHX, in particular, baffles are an integral part of the design both for increasing the thermal efficiency, and also supporting the tube bundle from severe vibrations caused by fluid flow.⁸

Depending on their structure, baffles are divided into several types, the most commonly used being segmental, double segmental and helical baffles. The design of these common baffle types is shown in Figure 5.⁸ We narrowed down the scope of our simulation and analysis to segmental baffle type to be consistent with the design of our available experimental device.

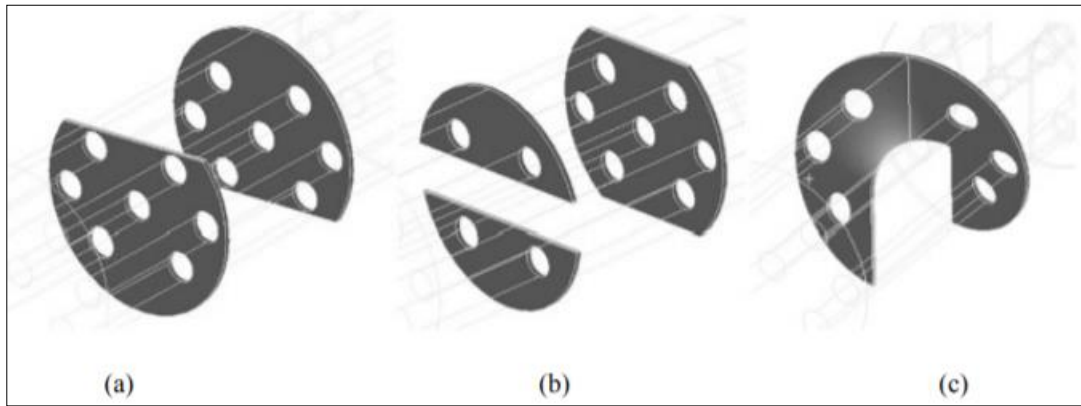


Figure 5. Common baffle types. (a) Segmental baffles, (b) Double segmental baffles, (c) Helical baffles.⁸

The diameter of baffles is generally equal to the diameter of the shell and they are occasionally welded to the shell to minimize undesired bypassing between the baffle and the shell. To allow the shell-side fluid to move along the heat exchanger, some portions of the baffles are trimmed, known as baffle cut. The proper design of the baffle cuts can lead to heat transfer enhancement by creating irregularities in the movement of the fluid and imposing additional turbulence in the system, as shown in Figure 6.c. If the baffle cut is too small (Figure 6.a) or too large (Figure 6.b), the efficiency of the baffles will decline.⁹

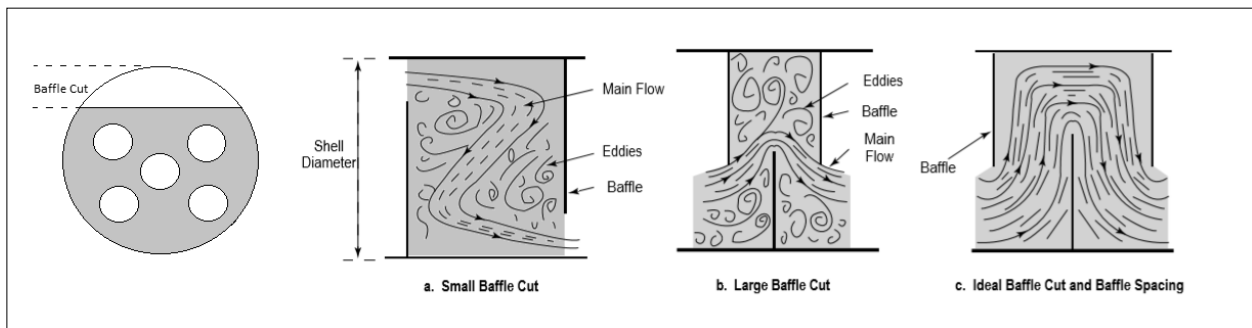


Figure 6. Schematic diagram of the flow movement in the shell when baffle cut is (a) small, (b) large, and (c) ideal.⁹

To summarize, the existence of baffles adds additional degrees of freedom in the design of the already-flexible STHX. Therefore, a large number of parameters can be adjusted to optimized the design of the heat exchanger. Just to name a few, the type of baffles, number of baffles, baffle spacing, baffle cut, diameters of both shell and tubes, number of tubes, number of tube passes, the direction of the flow, and many other parameters, are all adjustable and affect the overall thermal performance and efficiency of the heat exchanger.

1.3 Scope of Work

Evidently, experimental modifications in the design of STHX are tediously long and costly, and almost all STHXs are far from ideal design. To cope with this major issue, Computer-aided design (CAD) and Computational Fluid Dynamics (CFD) play an important complementary role to analyze heat exchangers in more depth and optimize the design within feasible time and cost constraints. In this project, we aimed at two major objectives.

First, to model the exact design of the small educational demonstration STHX unit we have in the laboratory, using the CFD approach (Ansys Fluent) to investigate the device in more depth. For instance, the probe thermometers on the experimental device only measure the inlet/outlet temperatures, whereas, with CFD, we can plot the temperature distribution along the heat exchanger and provide additional in-depth insights about the flow. Note that in the experimental setup, we were able to change the flow direction to demonstrate both co-current and counter-current flows. Therefore, both of these flow configurations were also modelled and compared in our simulations.

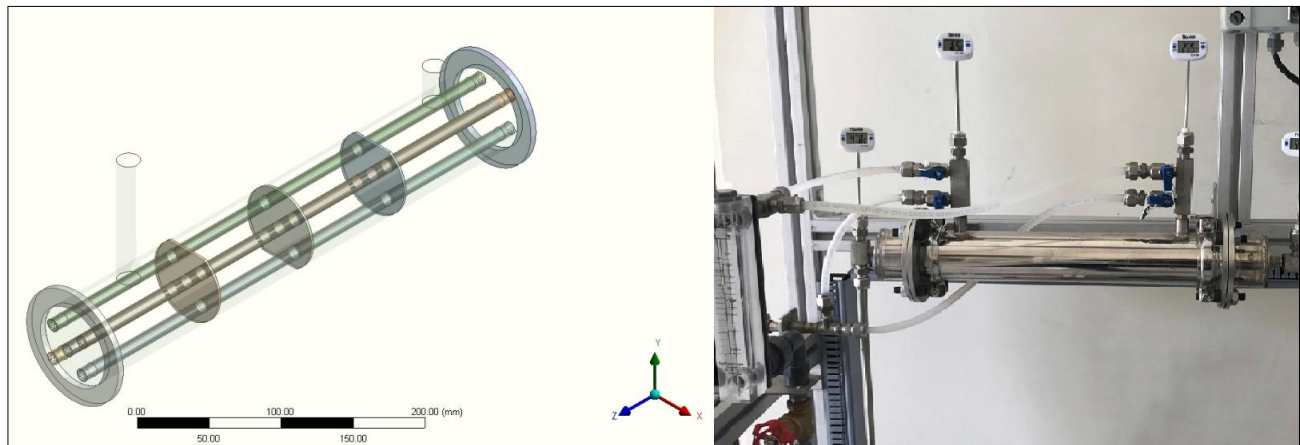


Figure 7. The model we created in Ansys (left), and the experimental setup of the STHX (right).

This objective was set to validate our simulation model with accessible experimental data. In addition, in the coming semesters, students of the Heat Transfer course can study the generated velocity and temperature contours and animations, prior to performing the lab session, in order to get acquainted with STHX.

The second objective was to use the flexibility of computer simulations to change some design aspects of this demonstration STHX unit to improve its efficiency. Please note that the full investigation of all design parameters on the heat transfer rate, although feasible, was out of the scope, and time limits

of this undergraduate project. Therefore, we selected three parameters to study: (1) the number of baffles, ranging from 0 to 10, (2) the orientation of the baffle cut (top-bottom, left-right, and clockwise), and (3) the baffle cut size. A number of analyses were then performed to examine the gains and losses of each modification, and finally, proposing some informed suggestions.

2. Review of Previous Work

2.1 Experimental Studies

In 1949, Donohue¹⁰ began to study existing experimental data, collected by other researchers, to find a correlation of convective heat transfer coefficients in heat exchangers. He evaluated the two main factors of pressure drop and heat transfer in the baffled and un-baffled shell-side of STHXs. Finally, he provided a correlation for the heat transfer coefficients of the shell-side by modifying the existing equation for tubes, proposed by Sieder and Tate¹¹ in 1936.

In 1950, Kern¹², based on experimental data, proposed a correlation for estimation of heat-transfer coefficients of shell-side in heat exchangers. This method is a simple and straightforward approach with reasonable accuracy. Nevertheless, Kern did not consider leakage effects, and more importantly, did not take into account any variation in the baffle cut (fixed at 25%).

In 1947, a long research plan was started at the University of Delaware to perform a comprehensive study on the shell-side design of STHXs. This project was continued for 16 years, until 1963, and was sponsored by ASME (American Society Mechanical Engineers), TEMA, and many other companies and institutes. A variety of design elements were considered in the Bell-Delaware method¹³ to encompass almost all possible STHX designs.

The research continued, and in 1997, Kim and Aicher¹⁴ conducted a more extensive study of the effect of geometric parameters on the rate of heat transfer in STHXs. In this work, 32 different heat exchangers were tested, each of which had distinct geometric parameters such as the number of tubes, shell diameter, length of the heat exchanger, tube pitch, and nozzle diameter. By superposing the Nu number in the longitudinal and nozzle region, they proposed a new correlation for heat transfer coefficients. In the same year, other aspects of the study were developed; Gaddis and Gnielinski¹⁵ examined the effects of segmental baffles on pressure drop in STHXs and presented a method based on the equations for calculating pressure drop in an ideal tube. In 1998, Li and Kottke¹⁶ used the analogy between heat and mass transfer to determine local heat transfer coefficients in STHXs by measuring chemical, coupled color reaction, and absorption.

In 2004, a computer program was developed by Kara and Gararas¹⁷ to examine the design of a variety of heat exchangers with U-shaped tubes and segmental baffles and different numbers of passes. This cannot be considered a computational technique because it mainly relied on simplistic experimental

correlations to find a preliminary design to meet the desired heat duty within a given allowable pressure drop. Using simple models allowed the program to examine many thousands of configurations to search for the optimized design. In order to gradually complete the methods for calculating pressure drop and heat transfer, which are the main goals of heat exchangers, in 2005 Ayub¹⁸, based on real data taken over several years, developed a method to calculate the heat transfer coefficient for shell-side of segmental STHXs. The results were compared with other known methods, and finally, due to its simplicity and accuracy, it was recognized as a practical method for engineers in this field to design STHXs.

2.2 Computational Fluid Dynamics Studies

CFD techniques developed over the years, side by side of the experimental methods, so much so that in 1973, Imperial College began a highly intuitive program based on the prediction of a two- and three-dimensional flow pattern by solving Navier-Stokes equations. However, this program was significantly costly and was only able to model simple systems.¹⁹

In 1974, Launder Spalding introduced a two-equation model known today as k-epsilon to show the function and properties of turbulent currents that have a high Reynolds number. This model was very successful and could cover very turbulent currents with a lot of eddies. CFD has evolved to overcome a wide range of challenges such as computational limitations, complex geometry, and slow convergence. Nowadays, CFD plays a vital role in engineering and scientific fields, and it is recognized as a complementary tool for experimental methods.²⁰ Some of the applications of CFD in studying heat exchangers are discussed below.¹⁹

Prithiviraj and Andrews²⁰⁻²² in a series of papers in 1998 and 1999 established a 3D model, on the basis of a distributed resistance method to model a baffled STHX. They simplified the fluid as a porous material with specific surface permeability, obtaining reasonable pressure drop and temperature distribution with up to 15% deviation from experimental data.

In 2009, Wang et al.²³ investigated the idea of continuous helical baffles in a multi-pass STHX. The CFD model of this idea was modelled in Ansys Fluent and was compared with a common segmental baffle counterpart. The velocity distributions of these two models are shown in Figure 8. They showed that the pressure drop of their proposed model was 13% lower than the conventional segmental model,

while the overall heat transfer was improved. This idea has attracted significant attention, and has led to many other CFD and experimental studies.

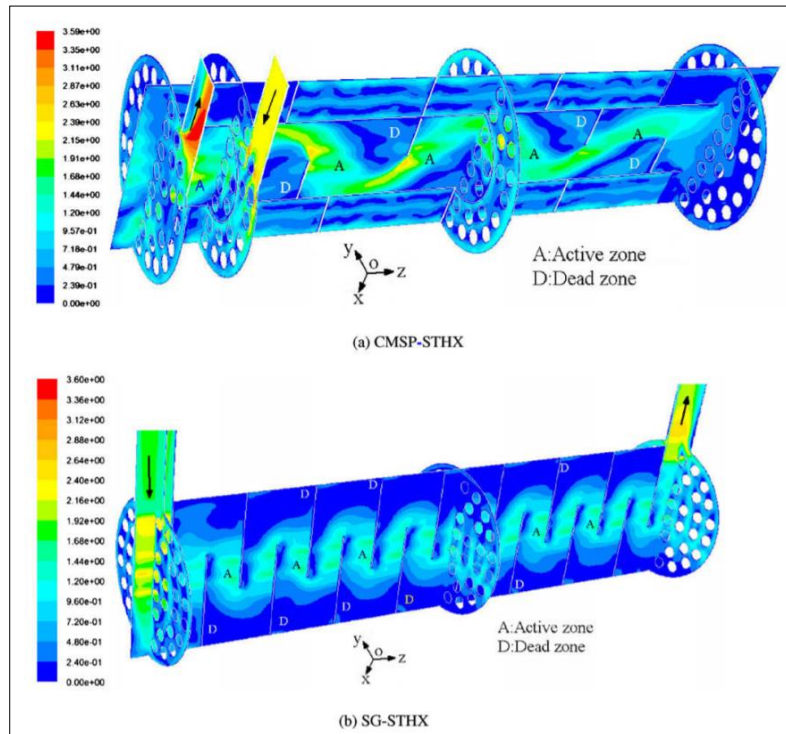


Figure 8. Comparison of velocity distributions of (a) a combined multiple shell-pass shell-and-tube heat exchanger (CMSP-STHX) with (b) the conventional shell-and-tube heat exchanger with segmental baffles (SG-STHX).²³

In 2010, Ozden and Tari²⁴ used two turbulence models (K-epsilon and Spallart-Allmaras) for STHXs, and by changing the number of baffles from 6-12, the baffle spacing, and baffle cuts in the heat exchanger, they calculated heat transfer rates, and compared with the estimated values obtained by hand calculation using the log mean temperature difference (LMTD) method. They concluded that the Bell-Delaware correlation was able to predict the heat transfer coefficients accurately.

Using the streamlines of the flow in the shell, they noticed when a smaller number of baffles are used, more recirculation regions are formed in between the baffles, lowering the heat transfer rate. By increasing the number of baffles, they were able to fix this problem and enhance the heat transfer. This shows the power of CFD to pinpoint the cause of the non-optimal performance of an SHTX, followed by quick implementation of suggested modification to fix the problem.

Zeng and Zhang²⁵, also investigated the heat transfer performance of an STHX using Fluent. They noted that for a decrease in tube length to shell diameter ratio (L/D) from 6.0 to 1.5, keeping the Reynolds number the same, the fluid flow mal-distribution and pressure drop were increased sharply, diminishing the overall performance of the exchanger.

A more rigorous examination showed that the inlet/outlet regions are responsible for most of the fluid mal-distribution. To fix this problem, they proposed a novel structure of multi-parallel-channel (MPC) for inlet and outlet of the STHX, as illustrated in Figure 9. MPC was shown to be extremely effective in optimizing the fluid flow distribution and also decreasing the pressure drop.

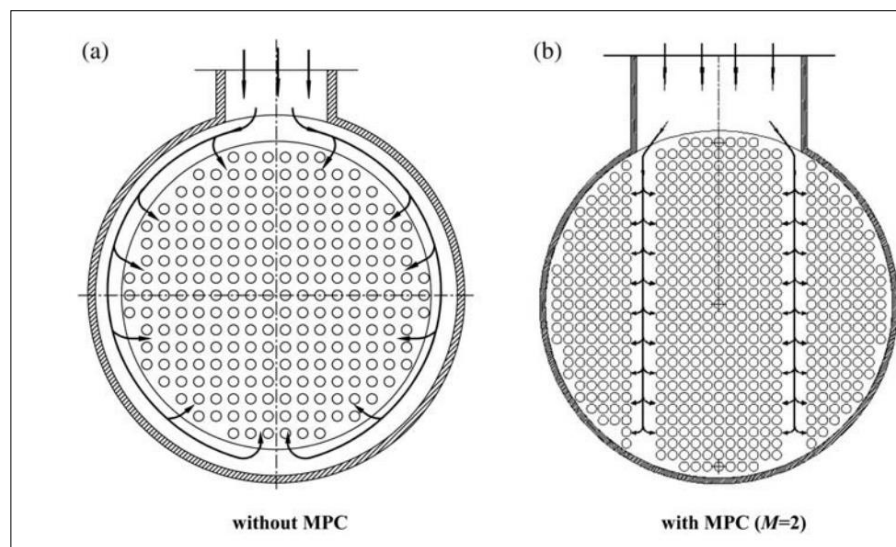


Figure 9. Scheme of shell-side fluid flow of inlet/outlet section (a) without, and (b) with two multi-parallel-channels (MPC).²⁵

In 2016, Eshita Pal et al.²⁶ conducted a CFD study in Open Foam software to examine the shell-side flow of an STHX model in the presence or absence of baffles at different mass flow rates. In this paper, short heat exchangers with baffles, where the ratio of length (L) to hydraulic diameter (D_h) was smaller than 7, were modelled. They found out that in these STHXs the flow near the nozzle notably affects the overall performance of the device, such that conventional correlations for heat transfer estimation would not be applicable.

In one of their analyses, they compared 3 heat exchangers with different numbers of tubes and shell diameter. The shell diameter of heat exchangers in Figure 10.A are 150, 300, and 600 mm, respectively. Figure 10.A.1 is the reference heat exchanger with 19 tubes. To see the effect of the number of tubes with unchanged tube diameter, 61 tubes were located in a larger shell, shown in Figure 10.A.2. The

velocity contour shows that the increase in the number of tubes resulted in an uneven penetration of inlet flow in the tube bundle, mostly moving towards the shell wall. This effect is also shown in the streamlines of Figure 10.B. On the other hand, where the tube diameter was increased in a considerably larger shell, as illustrated in Figure 10.A.3, the flow penetration was enhanced. Therefore, it was concluded that the number of tubes should be limited, in order to enhance the flow movement in the tube bundle in an STHX.

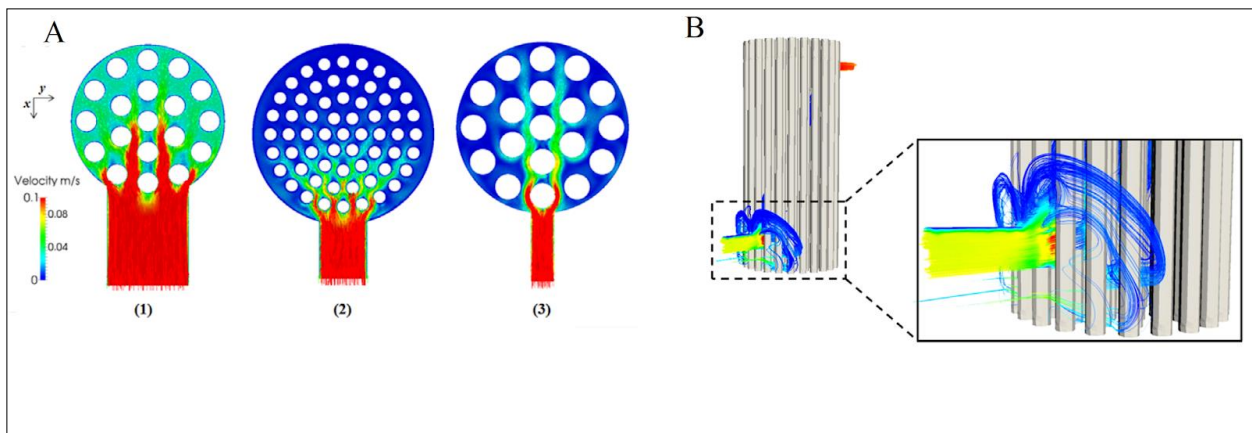


Figure 10. (A) Velocity vectors of the three systems: (1) hx150; (2) hx300; (3) hx600. (B) Flow leakage in hx300 (streamlines from inlet).²⁶

Aslam Bhutta et al.¹ did a complete literature survey, reviewing the applications of CFD in the design of heat exchangers. They studied fouling, pressure drop, mal-distribution of flow, and ultimately the optimize the design. In addition, the vast majority of the results obtained by CFD analysis were in good agreement with experimental results, which proves the effectiveness of this tool for predicting the performance and behavior of a wide range of heat exchangers.

3. Methodology:

In this project, we employed a hybrid approach to study STHXs, with the focus on the baffles. We had access to an educational demonstration unit, enabling us to collect a wide range of experimental data. This information was then utilized to validate our simulation results. The simulation part of this project was conducted in ANSYS Fluent, as a commercial CFD code.

CFD is a commonly applied tool in engineering for modeling fluid phenomena and generating solutions for fluid motion, either with or without solid interaction. Generally speaking, CFD numerically solves mathematical models of a wide range of physical phenomena, including laminar/turbulent fluid flow, conjugate heat transfer, reaction kinetics, single- or multi-phase systems, to obtain desirable parameters within the systems of interest.

The simulation procedure consists of five steps, managed in Ansys Workbench: geometry, mesh, setup, solution and result, respectively. First, a 3D model of the STHX is designed in Design Modeler, according to the dimensions of our experimental unit. Then, it has been discretized and divided into smaller elements, known as mesh. This step was conducted in Ansys Meshing that provides a wide range of features to flexibly discretize our model and refine critical regions of the system. In the next step, the desired physical models and their related parameters, such as boundary/initial conditions and type of fluid, are determined. This model is then solved using implemented mathematical algorithms in Ansys Fluent. Finally, the results are transferred to CFD-Post in order to perform the analyses and draw conclusions.

With this brief overview of the methodology, we elaborate on the steps in the following sections. we first introduce the concepts of overall heat transfer coefficient and log mean temperature difference (LMTD) method. These are common hand calculation methods used to analyze heat exchangers. Then, we change our focus to the more advanced and sophisticated approach that Ansys Fluent uses to realistically model a heat exchanger.

3.1 Heat Transfer Calculation in STHXs

There are two main types of problems in STHX design: rating and sizing. In the rating problems, one is concerned with the heat transfer rate for prescribed flow rates, inlet temperatures, and geometrical information of the instrument. In sizing problems, however, the type and size of the equipment are to

be determined for a given inlet/outlet temperature. In this thesis, we are dealing with a rating problem, since the instrument is pre-built.

The rate of absorbed heat by the cold fluid (Q_C) and that which is released by the hot fluid (Q_H) can be calculated by the below equations. Based on the law of conservation of energy, for a well-isolated system, these two must be equal ($Q_C = Q_H = Q$).

$$Q = \dot{m}_H C_p (T_{H,i} - T_{H,o}) = \dot{m}_C C_p (T_{C,o} - T_{C,i}) \quad (1)$$

However, in rating problem, we don't have the exit temperatures. Thus, calculation of Q requires a different approach based on heat transfer modes.

Heat transfer occurs in three modes, known as conduction, convection, and radiation. In a heat exchanger, because we are dealing with relatively low temperatures, radiation is negligible. The other two modes are present in a heat exchanger: conduction is mainly noticeable in the wall of the tubes that separate two fluids, and convection occurs in both shell- and tube-side fluids. The schematic diagram of a tubular heat exchanger is shown in Figure 11.

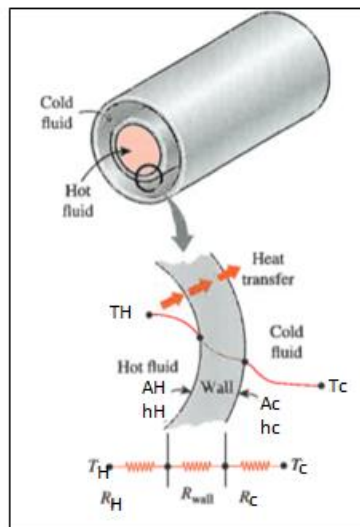


Figure 11. Heat transfer modes in a heat exchanger. ²

The main challenge is to compute convective heat transfer coefficients on both shell and tube sides (h_c , h_H). We are going to later discuss strategies by which h_c and h_H are calculated in this section.

Nonetheless, after obtaining h_C and h_H , a new parameter must be defined in order to combine all modes of heat transfer into a single parameter, known as “overall heat transfer coefficient (U)”. The concept of U originates from the resistance network analysis and is related to the inverse of total resistance. It can be shown that for an STHX, U can be written as:

$$\frac{1}{UA_s} = R_{\text{total}} = \frac{1}{h_H A_H} + \frac{1}{h_C A_C} + \ln \frac{D_C/D_H}{2\pi kL} \quad (22)$$

Assuming $R_{\text{wall}} = \left(\ln \frac{D_C/D_H}{2\pi kL} \right) \approx 0$ and $A_s = A_C = A_H$ we can reduce equation 1 to equation 2

$$\frac{1}{U} \approx \frac{1}{h_H} + \frac{1}{h_C} \quad (23)$$

To calculate the amount of heat transfer using U, we need to define log mean temperature difference (LMTD or ΔT_{lm}) as shown below.

$$\Delta T_{\text{lm}} = \frac{\Delta T_1 - \Delta T_2}{\ln \left(\frac{\Delta T_1}{\Delta T_2} \right)} \quad (24)$$

For Co-Current we have $\Delta T_1 = T_{H,\text{in}} - T_{C,\text{in}}$ and $\Delta T_2 = T_{H,\text{out}} - T_{C,\text{out}}$

For Counter-Current we have $\Delta T_1 = T_{H,\text{in}} - T_{C,\text{out}}$ and $\Delta T_2 = T_{H,\text{out}} - T_{C,\text{in}}$

ΔT_{lm} = Log mean temperature difference (K)

$$\dot{Q} = UA_s \Delta T_{\text{lm}} \text{ (W)} \quad (25)$$

The main challenge in thermal analysis (rating problem) of an STHX is how to obtain the convective heat transfer coefficients of both shell- and tube-side fluids (h_C, h_H). The conventional approach to estimate these coefficients is to employ the dimensionless Nusselt number ($Nu = \frac{hD}{k}$). There are a number of empirical relations that relates Nu to two other dimensionless numbers, describing the flow properties, namely Reynolds ($Re = \frac{VD}{\nu}$) and Prandtl ($Pr = \frac{\mu C_p}{k}$) numbers. Here, we will describe the most accurate and renowned relations to obtain h for both shell- and tube-side fluids.

Tubes, in general, have a significantly simpler design and their Nu correlations have been well-established. Colburn equation is a very simplistic relation to obtain Nu number but its accuracy, with up to 25 percent error, is not desirable for the purpose of this work.²⁷ This error can be reduced to less

than 10 percent by using a more complex equation known as Gnielinski (1976)²⁸ that uses the friction factor (f) of a smooth tube that was developed by Petukhov in 1970.²⁹

$$\text{Petukhov : } f = (0.790 \ln \text{Re} - 1.64)^{-2} \quad 3000 < \text{Re} < 5 * 10^6 \quad (2)$$

$$\text{Gnielinski: } \text{Nu} = \frac{(f/8)(\text{Re}-1000)\text{Pr}}{1+12.7(f/8)^{0.5}(\text{Pr}^{2/3}-1)} \quad 3000 < \text{Re} < 5 * 10^6, 0.5 \leq \text{Pr} \leq 2000 \quad (3)$$

Shell-side analysis, on the other hand, is more complex due to the chaotic motion of the fluid and the presents of internal, such as tubes and baffles, as described earlier.

There are a number of different methods that are widely known for calculation of heat transfer coefficient of the shell-side fluid, and, in this review, we are going to introduce three of them: Donohue, Kern, and Bell-Delaware methods. The rationale behind this selection is that Donohue is the first research on shell-side calculations, Kern offers a simple yet widely accepted route, and finally, Bell-Delaware is the most complex, and renowned method.

In 1949, Donohue¹⁰ published one of the first correlations for shell-side heat transfer coefficient calculations of an STHX with baffles. The main difference between Donohue method with tube-side calculations is the requirement to calculate 2 mass velocities: G_b that is the mass velocity of the shell-side moving parallel to the tubes, and G_c when the fluid is moving across the tubes.

$$G_b = \frac{\dot{m}_s}{\frac{\pi}{4}(f_B D_s^2 - N_{bt} d_o^2)} \quad (4)$$

In which, $\dot{m}_s$ is the mass flow rate of the shell-side fluid, f_B is the fraction of the baffle window (also known as baffle cut), D_s and d_o are the shell and tube diameters, and finally the N_{bt} is the number of tubes in the baffle cut. Note that in our case, because the baffle cut is so small and the number of tubes is only 5, no tube was located in the baffle cut and $N_{bt} = 0$.

$$G_c = \frac{\dot{m}_s}{P_B D_s \left(1 - \frac{d_o}{p_T}\right)} \quad (5)$$

In which $\dot{m}_s$ is again the mass flow rate of the shell-side fluid, P_B denotes for the baffle spacing, D_s and d_o the shell and tube diameters, and p_T is the length of the tube pitch. The geometric average of these to mass velocities ($G = \sqrt{G_b G_c}$) is used in the Re number term of the Nu number that is shown below:

$$\text{Nu}_o = \frac{h_o d_o}{k} = 0.2 \left(\frac{d_o \sqrt{G_b G_c}}{\mu} \right)^{0.6} \text{Pr}^{1/3} \quad (6)$$

The second method that we are going to describe here is the Kern method. In 1950, based on experimental work on commercial heat exchangers with standard tolerances, Kern¹² offered a reasonably accurate correlation to predict the shell-side heat-transfer coefficient. First, we calculate the Re number based on the mass flow rate ($\dot{m}$). However, we still need to define the equivalent diameter (D_e) and cross section area (A_c).

As shown in Figure 12, for a rectangular arrangement of the tubes in the tube bundle (similar to our experimental model), the equivalent diameter and area can be described as:

$$D_e = \frac{4(\frac{\sqrt{3}}{4}P_t^2 - \frac{\pi d_o^2}{8})}{\frac{\pi d_o^2}{2}} \quad (7)$$

$$A_c = \frac{D_s C_t B}{P_t} \text{ in which } B = \frac{L_t}{N_b + 1} \quad (8,9)$$

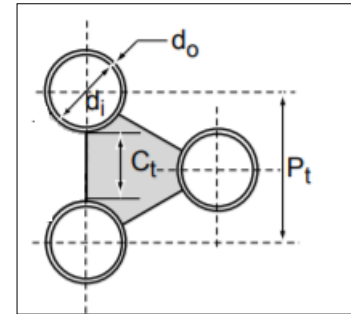


Figure 12. Geometric parameters of rectangular arrangement of the tubes.¹²

Next, we can calculate the Re number using the below equation:

$$Re = \frac{\dot{m} D_e}{A_c \mu} \quad (10)$$

After obtaining the Re number, the rest of the calculation is similar to the general approach; first, we obtain the Nu number from a proposed correlation ($Nu=f(Re, Pr)$), followed by calculation of h_c from the definition of Nu number. A simple equation to calculate Nu number is shown in equation 11.

$$Nu = 0.36Re^{0.55}Pr^{1/3} \quad (11)$$

The third and most comprehensive approach is the Bell-Delaware method,¹³ which has been introduced in the previous work section (2.1). The calculation starts with the estimation of an “Ideal” case, and then, 5 correction factors are multiplied to modify the heat transfer coefficient on the shell-side. These correction factors constitute:

- Leakage of the fluid between tube/baffle and baffle/shell gaps;
- Baffle configuration;
- Bypass of the flow between the tube bundle/shell gap;
- Unequal inlet/outlet baffle spacing;

e) The adverse temperature gradient in the case of laminar flow.

The shell-side heat transfer coefficient (h_s) can be calculated by the below equation:

$$h_s = h_i (J_c J_l J_b J_s J_r) \quad (12)$$

The h_i is the ideal case that is defined as the heat transfer coefficient for pure cross-flow in an ideal tube bank. All other J factors are correction factors: J_c for the segmental baffle window, J_l for the baffle leakage, J_b for the bypass effects, J_s for unequal inlet/outlet baffle spacings, and finally J_r for laminar flow effects of the adverse temperature gradient.

The foremost step in the calculation of shell-side heat transfer coefficient is the ideal tube bank coefficient (h_i) from the below relation:

$$h_i = j_i c_p \left(\frac{\dot{m}_s}{A_s} \right) \left(\frac{1}{Pr} \right)^{2/3} \left(\frac{\mu_s}{\mu_{s,wall}} \right)^{0.14} \quad (13)$$

Where j_i is the Colburn j-factor, c_p denotes for the specific heat of the fluid, $\dot{m}_s$ is the fluid mass flow rate, A_s is the crossflow area that has to be calculated separately, Pr is the Prandtl number of the shell-side fluid, and finally, μ_s and $\mu_{s,wall}$ are the viscosities of the shell-side fluid and the wall, respectively.

Aside from the thermodynamic properties of the fluid and its flow rate, one must carefully calculate A_s and j_i , which also requires the calculation of Re_s beforehand. In the below section, the route to obtain these parameters are described in detail.

A_s is to be determined first using the inner diameter of the shell (D_s), the clearance between adjacent tubes by subtracting the tube outer diameter from the tube pitch ($C = P_T - d_o$), and the baffle spacing (B). The relation is shown below:

$$A_s = \frac{D_s}{P_T} C * B \quad (14)$$

The Colburn j-factor depends on the Re number and some constants provided in Table 1. Re number in the shell (Re_s) is estimated by:

$$Re_s = \left(\frac{\dot{m}_s}{A_s} \right) \frac{D_e}{\mu_s} \quad (15)$$

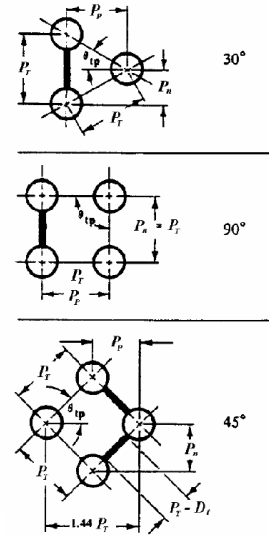
in which the equivalent diameter for the triangular pitch is:

$$D_e = \frac{4 \left(\frac{P_T^2 \sqrt{3}}{4} - \frac{\pi d_o^2}{8} \right)}{\frac{\pi d_o}{2}} \quad (16)$$

Once the Re_s is calculated, depending on the layout angle of the tubes, one needs to get 4 constants, a_1 to a_4 , from the below table.

Table 1. Constants for calculation of the Colburn j-factor based on the Re number. ¹³

Layout Angle	Reynolds Number	a_1	a_2	a_3	a_4	b_1	b_2	b_3	b_4
30°	10 ⁵ –10 ⁴	0.321	–0.388	1.450	0.519	0.372	–0.123	7.00	0.500
	10 ⁴ –10 ³	0.321	–0.388			0.486	–0.152		
	10 ³ –10 ²	0.593	–0.477			4.570	–0.476		
	10 ² –10	1.360	–0.657			45.100	–0.973		
45°	<10	1.400	–0.667			48.000	–1.000		
	10 ⁵ –10 ⁴	0.370	–0.396	1.930	0.500	0.303	–0.126	6.59	0.520
	10 ⁴ –10 ³	0.370	–0.396			0.333	–0.136		
	10 ³ –10 ²	0.730	–0.500			3.500	–0.476		
90°	10 ² –10	0.498	–0.656			26.200	–0.913		
	<10	1.550	–0.667			32.00	–1.000		
	10 ⁵ –10 ⁴	0.370	–0.395	1.187	0.370	0.391	–0.148	6.30	0.378
	10 ⁴ –10 ³	0.107	–0.266			0.0815	+0.022		
45°	10 ³ –10 ²	0.408	–0.460			6.0900	–0.602		
	10 ² –10	0.900	–0.631			32.1000	–0.963		
	<10	0.970	–0.667			35.0000	–1.000		



$$a = \frac{a_3}{1+0.14(Re_s)^{a_4}} \quad (17)$$

The j_i can then be obtained by the below equation:

$$j_i = a_1 \left(\frac{1.33}{P_T/d_o} \right)^a (Re_s)^{a_2} \quad (18)$$

Based on the design of our STHX, we were able to safely ignore many of the correction factors as they were not implemented in the design; J_l for the baffle leakage, J_b for the bypass effects, J_s for unequal inlet/outlet baffle spacings, and J_r for laminar flow effects of the adverse temperature gradient, were all ignored and assumed to be 1.

We only need to consider the J_c for the segmental baffle window.

$$J_c = 0.55 + 0.72F_c \quad (19)$$

As can be seen in the above equation, the correlating parameter is F_c , defined as the fraction of tubes between baffle tips in the cross-flow, which is $F_c = 1 - 2F_w$, where F_w is defined as :

$$F_w = \frac{\theta_{clt}}{360} - \frac{\sin \theta_{clt}}{2\pi} \quad (20)$$

In which, θ_{clt} is the angle of the baffle cut intersection with the inside shell, as shown in the below figure, and can be obtained from:

$$\theta_{ctl} = 2 \cos^{-1} \left\{ \frac{D_s}{D_{ctl}} \left[1 - 2 \left(\frac{B_c}{100} \right) \right] \right\} \quad (21)$$

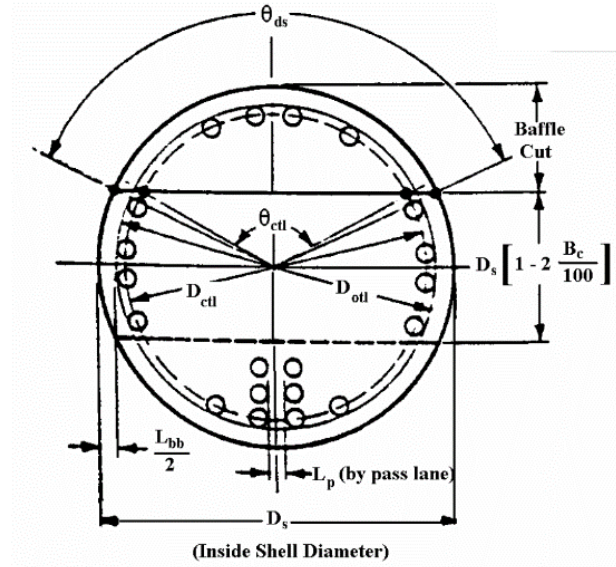


Figure 13. Illustration of required parameters for calculation of J_c in the Bell-Delaware method. ¹³

B_c is the baffle cut percentage, calculated from the height of the baffle cut (L_{bch}): $B_c = \left(\frac{L_{bch}}{D_s} \right) * 100$.

D_{ctl} , as shown in Figure 13, is the diameter of the circle from the center through the centers of the outermost tubes, estimated as $D_{ctl} = D_s - d_o$.

J_c correction factor is smaller than one when the baffle cut is large, it reaches 1.0 for baffle cuts of about 25 percent and can even become larger than 1.0 for baffle cuts smaller than 25 percent.

In our case that the baffle cut was only 7.7 percent, the J_c was approximately 1.2 which is reasonable.

Although Bell-Delaware is the industry standard for STHX calculations, it can only give accurate estimation within the design parameters it was optimized for. Our STHX unit is significantly small and is mainly designed for the educational demonstration that does not obey some criteria that guarantee the accuracy of Bell-Delaware method as listed below:

- the approximate ratio of minimum D_s to d_o should generally be about 15. In our case, this ratio is only 6.3 ($\frac{D_s}{d_o} = 6.3$).
- Minimum baffle spacing must be equal to 20% of the shell diameter, but it can maximum should not exceed the shell diameter (in our unit, the baffle spacing is 1.67 times of the shell diameter.)
- The recommended baffle cut is estimated based on the ratio of baffle spacing and the shell diameter ($0.2 < \frac{B}{D_s} < 1.0$) and ranges from 20 to 35 percent (the baffle cut is only 7.7 percent in our unit.)
- The number of tube passes must be determined based on the number of tubes in the shell. The minimum number of tubes per pass is approximately 8, and the accuracy of Bell-Delaware will decrease if this rule is violated (in our model we only had 5 tubes.)

Hence, even though the Bell-Delaware method is the most accepted method for STHX calculations, it does not necessarily provide accurate estimates for our small STHX unit. Nonetheless, we are going to compare it alongside other methods.

These hand calculations are complementary tools to our CFD analysis for verification purposes.

3.2 Governing Equations of a CFD Study

In this section, we are going to discuss the foundation of CFD studies that does not require any prior experimental data. The cornerstone of any mathematical modelling is the fundamental equations that describe physical phenomena in the language of mathematics. In CFD, these set of mathematical statements, also known as governing equations, are the continuity, momentum, and energy equations upon which all of fluid dynamics is described. The application of conservation of mass and momentum laws results in a set of partial differential equations (PDEs) known as Navier-Stokes equations.

Ansys Fluent software generally solves the continuity and momentum equation for any model that includes fluid, but if a system of interest includes other phenomena, we add those equations to the governing equations. Since our modeling involves heat transfer, the conservation of energy equation has also been considered. Moreover, due to the turbulent nature of the flow in our heat exchanger (high Reynolds number), the turbulence models are also added to our set of governing equations. Below, the mathematical equations used in Ansys Fluent are elaborated.

3.2.1 Continuity Equation (Conservation of Mass)

The first equation that is solved in Ansys Fluent is the continuity equation.³⁰

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m \quad (26)$$

The continuity equation is the direct outcome of applying total mass balance on a control volume. The first term represents the changes in density (ρ) over time. In the second term, ρ is density, and $\vec{v}$ is the flow velocity. ∇ (nabla) is the gradient vector and is defined as $(\frac{\partial}{\partial x} + \frac{\partial}{\partial y} + \frac{\partial}{\partial z})$ in Cartesian coordinates, and it denotes the inputs/outputs flow on the control volume. Finally, S_m is to consider any added mass to the continuous phase from the dispersed second phase.

In this project, we have solved for the steady-state condition of our heat exchanger and there was no second dispersed phase. Thus, the continuity equation was reduced to:

$$\nabla \cdot (\rho \vec{v}) = 0 \quad (27)$$

3.2.2 Conservation of Momentum

The second fundamental law for any fluid dynamics simulations is the conservation of momentum.³⁰

One way to describe force is the rate of change of momentum by time ($F = \frac{dP}{dt}$). By applying Newton's second law ($F = m \cdot a$) on a control volume, we can obtain the below equation.

$$\frac{\partial}{\partial t} (\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\tau) + \rho \vec{g} + \vec{F} \quad (28)$$

Where ρ is the fluid density, $\vec{v}$ is the fluid velocity, p is the pressure, τ represents the shear stresses, ρg accounts for the weight force, and finally, $\vec{F}$ denotes any additional body forces, such as friction. For a steady-state and non-accelerating frame, the components of the momentum balance can be described as the below set of equations. Note that body forces are neglected and the weight (gravitational force) is only added to the y- component of the momentum.

momentum equations for x, y, z:²⁴

$$\text{x-momentum: } \nabla \cdot (\rho u \vec{v}) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} \quad (29)$$

$$\text{y-momentum: } \nabla \cdot (\rho v \vec{v}) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho g \quad (30)$$

$$\text{z-momentum: } \nabla \cdot (\rho w \vec{v}) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} \quad (31)$$

As one can see, x-, y-, and z- components of the velocity are interconnected and cannot be solved independently. They also depend on the continuity equation. Hence, for any fluid dynamics simulation, these 4 equations, widely known as Navier-Stokes equations, must be solved together.

3.2.3 Energy Equation

Apart from the essential Navier-Stokes equations, several other equations might also need to be considered based on the physics of the model. Arguably, the most common auxiliary model is the energy equation. In a heat exchanger, as the name also suggests, the variant of temperature is of paramount importance and we need to consider the energy balance. In Ansys Fluent, the energy balance is modeled as below:³⁰

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot (\vec{v}(\rho E + p)) = \nabla \cdot (k_{eff} \nabla T - \sum h_j \vec{J}_j + (\tau_{eff} \cdot \vec{v})) + S_h \quad (32)$$

Where, on the right hand of the equation, K_{eff} is the effective conductivity, $\vec{J}_j$ is the diffusion flux of species j, and S_h represents the heat of chemical reaction. In this project, there is no reaction, so S_h is zero. Also, the accumulation term $\frac{\partial}{\partial t}(\rho E)$ is also zero since we are only interested in the steady-state results.

3.2.4 Turbulence Modeling

Whenever the ratio of inertia forces to viscous forces (Reynolds number) surpasses a certain value, the fluid layers are no longer parallel (laminar), they mix, and some degree of randomness is observed in the fluid path-line. This chaotic-in-nature phenomenon is known as turbulence. In a turbulent flow, part of the fluid kinetic energy goes into rotational movement and eddies are generated. Eddies have no exact definition and they only exist in certain time and region; Their energies are cascading from the largest eddy to the smallest one. At the final stage of this cascade, friction or viscous forces leads to the dissipation of the rotational kinetic energy of the smallest eddies.

Although eddies are random, they are still governed by Navier-Stokes equations. However, accounting for the full spectrum of the energy cascade directly requires extremely fine meshes (micron level) and time steps. It is evident that the Direct Numerical Solution (DNS) of turbulent flows is

impractical, and turbulence must be accounted for statistically. Several turbulence models have been developed over years with different degrees of accuracy and computational cost.

The choice of turbulence model is very important in CFD simulation, but there is no universal criterion for this choice.²⁴ Unquestionably, more complex models enhance the accuracy of our modeling, but the exponential increase in the computational cost forces us to accept the trade-off and employ a reasonably simple model.

Ansys Fluent software offers various models for modeling turbulent flows, starting from a simplistic one-equation model, followed by popular two-equations models, and up to some state-of-the-art but computationally expensive models, such as the seven-equation model of Reynolds Stress and even hybrid models such as SAS, DES, and LES. The extensive explanation of all available turbulence models is exhaustive and out of the scope of this work, and we only focus on our method of choice. The two-equation k- ϵ turbulence model is by far the most utilized turbulence models in the literature, owing to its power to reproduce experimental data, yet being computationally efficient.

3.2.4.1 k- ϵ and k- ϵ RNG Turbulence Models

The k- ϵ method has two parameters, kinetic energy (k) and the rate of dissipation of energy (ϵ). The k- ϵ equations in specific sections of modeling have a series of shortcomings and limitations, which were investigated in several stages and by different mathematical methods. One of the well-accepted modifications of the original k- ϵ model is using a mathematical technique called the renormalized group (RNG).

This method is based on the Navier-Stokes equations, recalculates the equations that are solved for k- ϵ , so a series of terms are added and constant terms are changed to remove a number of limitations.²⁴

$$\frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon + S_k \quad (33)$$

$$\frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} + S_\epsilon \quad (34)$$

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \quad (35)$$

Where G_k is velocity gradient, G_b is buoyancy gradient, S_k and S_ϵ are user-defined terms.²⁴

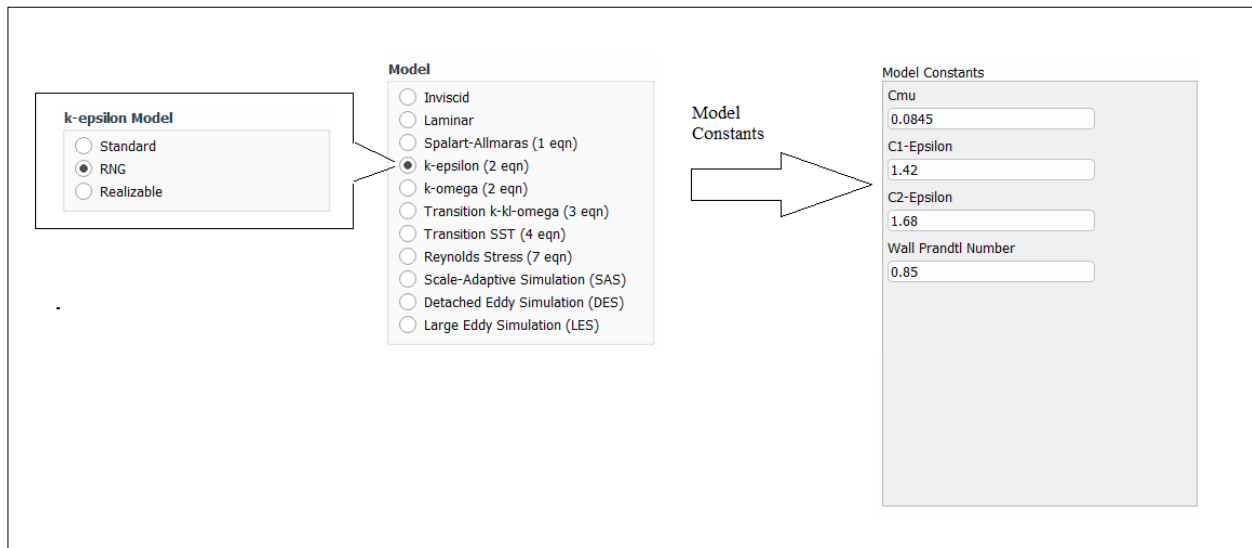


Figure 14. The screenshots of turbulence models embedded in Ansys Fluent and k- ϵ RNG model constants.

After this brief introduction to the general mathematical models used in CFD, we describe the process of implementation of these ideas on our heat exchanger analysis in Ansys Fluent. This process starts with creating the geometry that bounds our computational domain, followed by the discretization of this domain into smaller elements (mesh), and then solving the governing equations for each of the created meshes, in an iterative way.

3.3 Geometry

In this project, Ansys Design Modeler has been used for creating the 3D representation of our STHX. First, some 2-D sketches were drawn according to the dimensions of the laboratory STHX model, as shown in Table 2.

Table 2. The dimensions of our 3D model based on the design of the experimental unit

Shell and Tube heat exchanger components	size
Shell length	400mm
Shell inner diameter	63mm
Tube outer diameter	10mm
Tube inner diameter	8mm
Tube length	420mm
Number of tubes	5
Number of baffles	3
Shell inlet length	30mm
Shell outlet length	100mm

These sketches were then converted to 3D geometries, using the extrude tool. The outcome of our modeling is shown in Figure 15 from different angles.

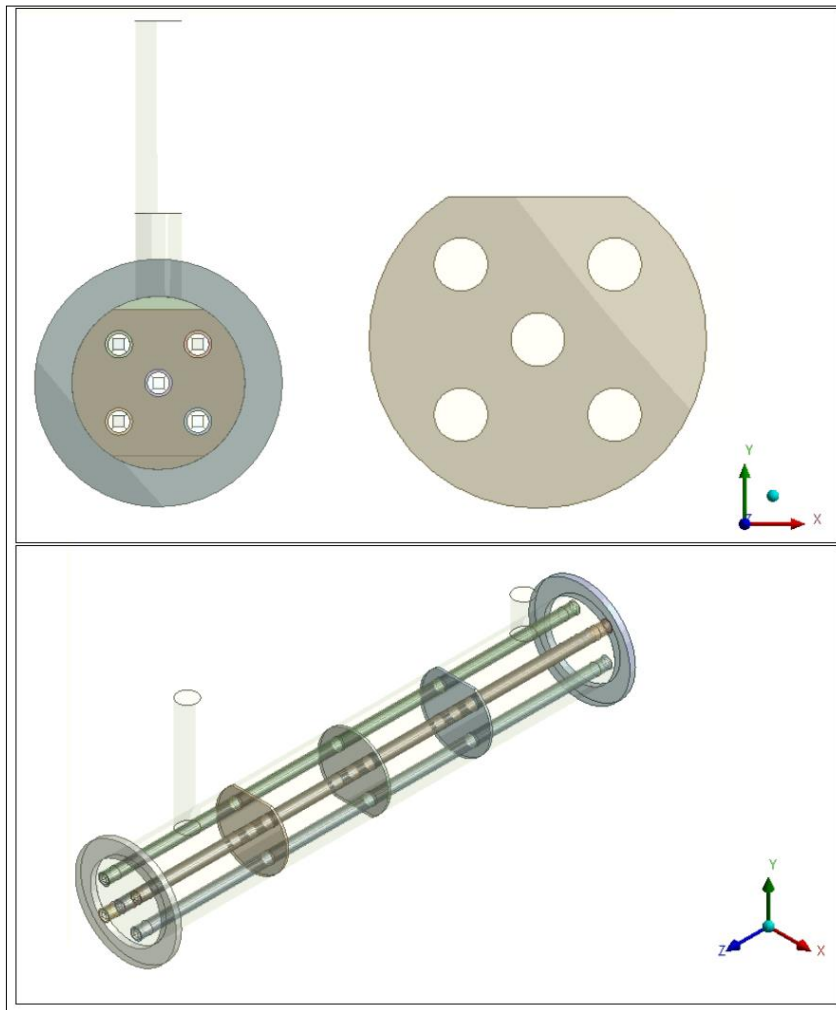


Figure 15. Different viewing angles of the 3D STHX created in Ansys Design Modeler

3.4 Discretization of the 3D Model (Meshing)

After successfully creating the geometry, we need to discretize the 3D model. Discretization or meshing is an integral part of any CFD simulation in which we divide a complex geometry into small and simple units that fluid properties can be averaged over each element. Treating continuous differential equations as a discrete set of algebraic equations can be accomplished by 3 methods: finite volume (FVM), finite difference (FDM) and finite element methods (FEM). FEM and FVM are integral schemes that minimize the discretization error. FDM, on the other hand, is a differential scheme, based on Taylor series expansion that entails more numerical errors and is also more

challenging to deal with curvature. One of the most advanced simulation software that uses CFD method to model complex geometries is Ansys Fluent. Ansys Fluent uses FVM method to integrate and transform partial differential equations (PDE) into a system of linear algebraic equations.

The way a model is discretized tremendously impacts both the accuracy and the convergence of the simulation. Generally, two aspects of the mesh must be taken into consideration: the size of the mesh, and its quality. The finer the mesh, the better the accuracy, and the more uniform the mesh, the better convergence can be achieved. However, more computational power is needed to perform the simulation on a finer mesh. We are obligated to employ advanced meshing techniques to only refine the mesh where the information is sensitive, such as boundary layers or narrow channels.

Ansys Meshing is the default and recommended environment to create a reliable mesh for any geometry. This program provides different meshing features such as local refinement, inflation, and edge/surface sizing to enhance the quality of the mesh.

In a heat exchanger, heat is transferred from the inner surface of the tube to the cold water adjacent to the outer tube wall. It is crucial to refine the mesh near the walls to capture the boundary layer on both sides of the tube walls, where high velocity and temperature gradients are expected. Based on this consideration, the inflation feature is used. On another note, one of the best ways to discretize the flow in the tubes is the O-grid technique, since the structural square mesh is numerically more desirable. The O-grid technique and the inflation features are both illustrated in Figure 16 and Figure 17.

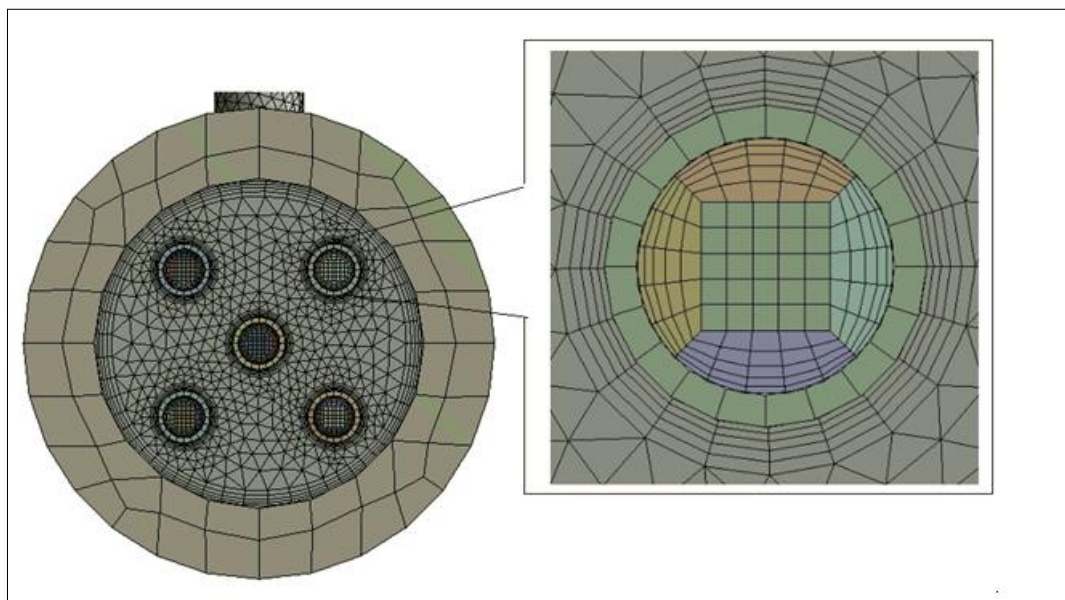


Figure 16. Orthographic projection (Front view) of our proposed discretization techniques for the STHX.

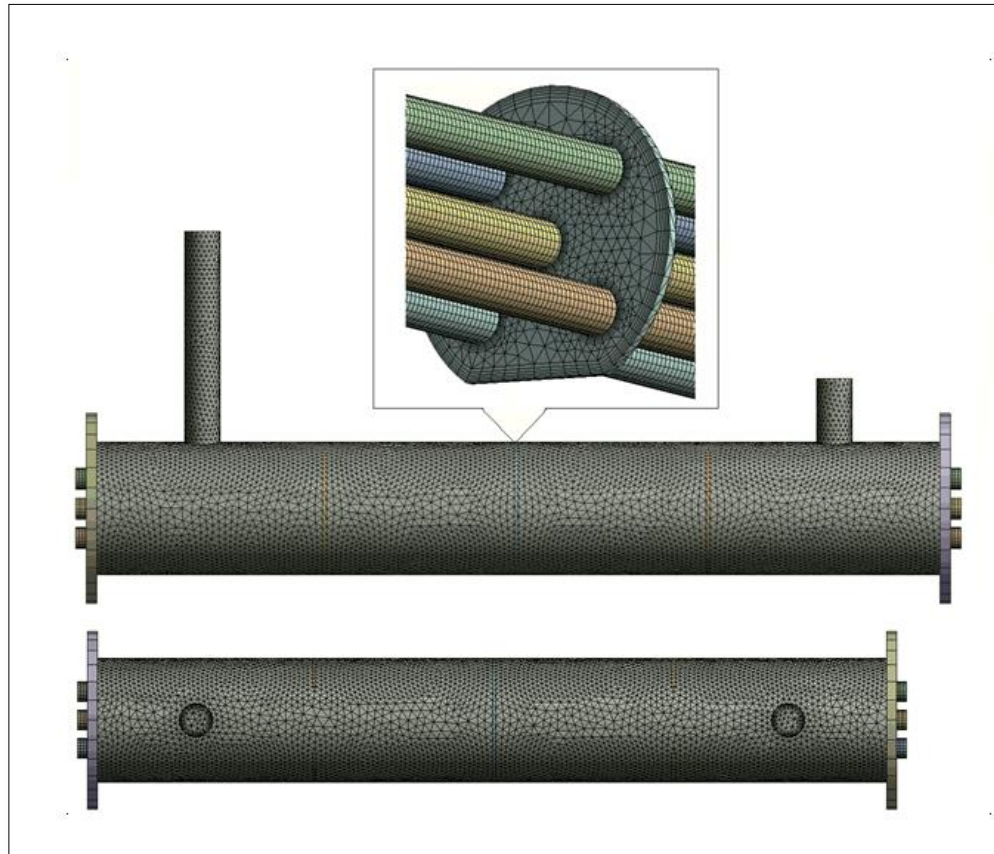


Figure 17. Side view of the generated mesh.

The defeature element size was also set to $5 * 10^{-5}m$ to ensure no coarse mesh is allowed in the process of meshing.

Overall, after fine-tuning the mesh, we ended up with 1,206,345 elements, which was found to be on the edge of feasibility based on available computing resources.

As mentioned earlier, the quality of the mesh is as important as its fineness. To check the quality of the mesh, Ansys Meshing provides different metrics. Three mesh metrics including, element quality, aspect ratio and skewness are reported below.

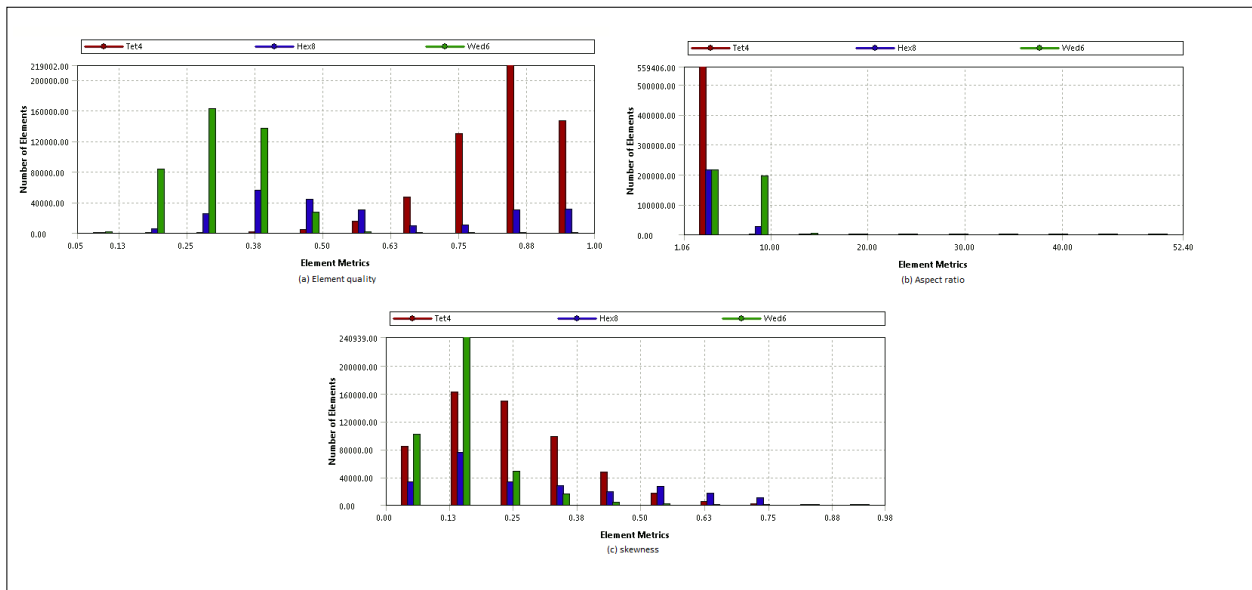


Figure 18. Three main mesh quality criteria including (a) element quality, (b) aspect ratio, and (c) skewness. Element quality is the first thing to check among mesh quality criteria provided by Ansys Meshing; it ranges from zero to one and the closer it is to one, the better the mesh quality is. The average element quality of our mesh is about 0.6. By looking at the histogram, one can see that the majority of lower quality meshes (<0.5) belong to the wedge category (green bars). This indicates that using the inflation feature near the boundaries has lowered the average quality. This was a trade-off we chose to model the boundary layers more accurately by sacrificing some degree of the element quality.

Aspect ratio is the ratio of the maximum to the minimum edge length of a mesh. For a cubic or a regular tetrahedron, in which all four faces are equilateral triangles mesh, the aspect ratio is one. In practice, we may encounter irregular geometries and meshes with significantly larger aspect ratios. In Ansys, the aspect ratios of below 100 are acceptable. In our case, the average aspect ratio is 3.7 and its maximum is 52 which is perfectly acceptable for a CFD study.

Skewness is a criterion for comparing the shape of the mesh and the shape of two equilateral meshes. Ideally, its value should be close to zero, and reaching any value above 0.95 is not acceptable and can lead to instability in further stages of the simulation. The average value of skewness should be kept below 0.33 for a smooth convergence.³⁰ In our mesh, the average skewness was 0.27, with very few meshes close to 0.95. Note that the deviation from the ideal scenario was based on an informed decision to use inflation feature near the walls and not due to the negligence.

Analyzing the mesh metrics gave us enough confidence to move on to the next stage and set up the physics of the model in Ansys Fluent. On one last note, Ansys Meshing and Ansys Fluent are well designed and they can interact seamlessly. It is extremely useful to use “Name Selection” and name all the surfaces (inlets/outlets and walls) and volumes (water in shell/tube and baffles) in the Ansys Meshing before transferring the mesh into Ansys Fluent. Choosing proper names for all entities will help Fluent to automatically detect all the boundaries and cell types, accelerating the setup procedure.

3.5 Ansys Fluent Configuration (Setup and Solution)

In this section, the setup of the physics of the model is explained in the Ansys Fluent. This is the core of the simulation, and we try to provide detailed information and screenshots so that one can follow the steps and reproduce the results.

In the very first step, we encounter the Fluent Launcher, as shown below. It is advised to activate the “Double Precision” option to store more decimal places in order to minimize the accumulation of round-up errors, at the expense of additional RAM storage requirements. The processing option was also changed from serial to parallel. We had access to a 12-core CPU, so we changed the number of processors to 12 to utilize its full power.

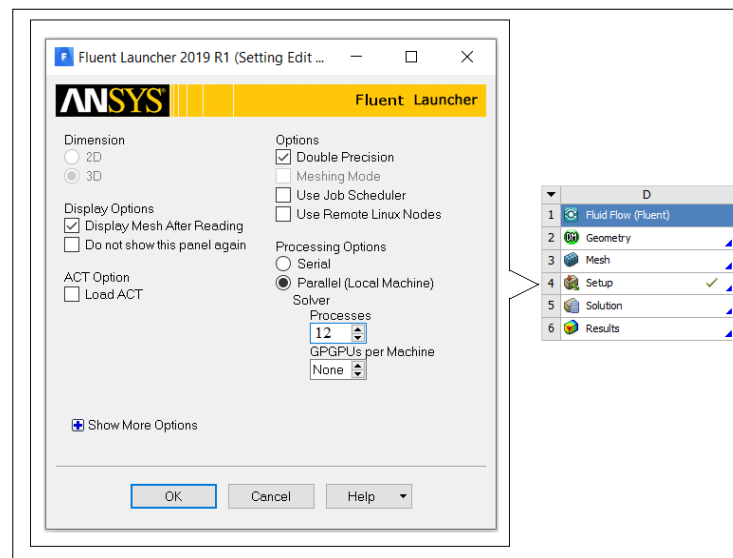


Figure 19. Fluent Launcher in Ansys Workbench.

Ansys Fluent has categorized its setup into 7 subsections, and its solution section into 8 parts. The “General” section is the first step in which the type of solver, steady/unsteady, and gravity can be defined.

Ansys Fluent's includes two types of solvers, pressure-based and density-based. The pressure-based is a more general solver and suitable for a wider range of models, such as incompressible fluids, and has better convergence than the density-based. Density-based, on the other hand, is utilized for high speed, sonic or supersonic compressible fluids. We selected the pressure-based solver.

Regarding steady or transient simulation, we have selected the steady-state model. Transient simulations can help us to obtain the time evolution of a system when a change is introduced to the system or the start-up period is important to consider. In our modeling, however, we are mainly interested in the steady-state condition of the heat exchanger and how changing the design parameters can affect the outcome.

Gravity is not defined in Fluent by default. In many cases, the forces exerted by the fluid motion are significantly larger than the one caused by gravity. In other cases, such as boiling, natural convection, and multiphase flows, gravity must be considered. In heat exchangers, we do not expect any significant variation in the results with or without modeling the gravity. However, it does not add any significant complication in the governing equations to hold us back from activating it. Therefore, gravity, with the gravitational acceleration of -9.81 in the y-direction was added.

The next subcategory of the setup in Fluent is the “Model”. Here, we configure the physics. The energy equation and the RNG $k - \varepsilon$ turbulence models were selected, as argued in the previous section.

By a closer look at the governing equations, one notices that a number of constants are present in the mathematical models that have physical meanings. For example, density and viscosity, are needed for Navier-Stokes equations, and specific heat and thermal conductivity are part of the energy balance equation. Therefore, based on the physics considered, some values have to be assigned to these constants. This is the next stage of Fluent setup, called “Materials”. By default, a single value is assigned to these parameters, but we can choose from a list of models to consider their temperature dependency. The experimental data for water is available in almost any source. Therefore, the safest way to model these parameters is piecewise-linear. The required parameters for the temperature range of interest were extracted from the heat transfer book ³¹ and shown below. Also, the solid material was defined as stainless steel 304 to be consistent with our experimental setup.

Table 3. Temperature dependency properties of water included manually in the Ansys Fluent.³¹

	Temperature T (°C)	Density ρ ($\frac{kg}{m^3}$)	Specific Heat C_p ($\frac{J}{kg.K}$)	Thermal conductivity k ($\frac{W}{m.K}$)	Dynamic viscosity μ ($\frac{kg}{m.s}$)	Prandtl Number Pr
1	15	999.1	4185	0.589	1.138×10^{-3}	8.09
2	20	998.0	4182	0.598	1.002×10^{-3}	7.01
3	25	997.0	4180	0.607	0.891×10^{-3}	6.14
4	30	996.0	4178	0.615	0.798×10^{-3}	5.42
5	35	994.0	4178	0.623	0.720×10^{-3}	4.83
6	40	992.1	4179	0.631	0.653×10^{-3}	4.32
7	45	990.1	4180	0.637	0.596×10^{-3}	3.91
8	50	988.1	4181	0.644	0.547×10^{-3}	3.55
9	55	985.2	4183	0.649	0.504×10^{-3}	3.25
10	60	983.3	4185	0.654	0.467×10^{-3}	2.99

Then for stainless steel 304:

Table 4. Stainless steel properties used for baffles and tube walls.

Material of STHX	Density ρ ($\frac{kg}{m^3}$)	Specific Heat C_p ($\frac{J}{kg.K}$)	Thermal conductivity k ($\frac{W}{m.K}$)
Stainless steel 304	7500	500	16.2

Note that the software cannot understand what different parts of the computational domains and meshes are composed of. In other words, we need to differentiate solid/fluid zones. If the parts are named unambiguously, Fluent automatically assigns the solid/fluid zones in the next step of the setup, namely “Cell Zone Conditions”. We just need to ensure the assignment has been done correctly and fix any potential mismatch.

As explained earlier, the governing equations are all partial differential equations (PDEs). The particular solution of all differential equations can only be obtained by knowing the boundary conditions. The next subsection of setup in Fluent, “Boundary Conditions”, is exactly for this purpose. The shell/tube inlet temperatures and flow rates (velocity), the outlet pressures, and all of the walls (no-slip boundary condition) must be carefully designated. Note that the outlet pressures were put

zero-gauge Pascal, equal to the atmospheric pressure. In reality, the pressure at the outlet would be different than atmospheric but the unavailability of the pressure reading in the experimental setup oblige us to make this assumption. It is worth noting that for an incompressible (pressure-independent) fluid like water, we do not expect any meaningful variation in the results, caused by our assumptions.

To this point, the “Setup” section is finished, and we need to move to the “Solution” section. In the Solution section, we are mainly concerned with the solution methods, setting of the convergence criteria, and initializing the iterative procedure to start the calculations. Generally, in a CFD study, we are dealing with a system of non-linear and coupled governing equations. There are different mathematical algorithms in Ansys Fluent to deal with this numerical challenge.

The software provides two types of algorithms, segregated and coupled. In a segregated algorithm, the individual governing equations are solved separately to define the solution variables such as velocity, temperature, and pressure³⁰ whereas in a coupled algorithm, momentum and continuity equations are solved simultaneously. Using the coupled algorithm, in comparison with the segregated algorithm, improves the rate of solution convergence significantly. As a result, depending on the model as well as the amount of system memory, we used the coupled method in this project.

After selecting the algorithm for the solution, we are required to set some convergence criteria. By default, the residuals of continuity, x-, y-, z-velocity, energy, and k and ε are considered as convergence criteria to reach the minimum values of 10^{-6} for energy and 0.001 for all others. The default convergence criteria can be seen in the figure below.

Equations			
Residual	Monitor	Check Convergence	Absolute Criteria
continuity	☒	☒	0.0001
x-velocity	☒	☒	0.001
y-velocity	☒	☒	0.001
z-velocity	☒	☒	0.001
energy	☒	☒	1e-06
k	☒	☒	0.001
epsilon	☒	☒	0.001

Figure 20. The default convergence criteria in Ansys Fluent.

Besides from the residuals, we can also add some system-specific parameters as convergence criteria in the “Report Definition” subsection. We defined the parameters that we wanted to obtain for the

follow-up analysis of the system. These parameters were the heat transfer rate, pressure drop and shell/tube fluid outlet temperatures.

Finally, we need to do the initialization to fully prepare the model for the run. It should be noted that ANSYS Fluent uses the iterative method to numerically solve the governing differential equations for each mesh. Obviously, the initial guess of this iterative method is far from the final values but having the initial guess as close to the converged solution as possible, can significantly decrease the required number of iterations to reach convergence. At this point, one should spend some time to set the initial temperatures of the shell and tube and their velocity vectors close to the experimental data using the “Patch” option. Note that the initial condition does not alter the final converged solution and it is just a good practice to decrease computational time.

We can go to the final “Run Calculations” subsection and start and continue the simulation up to the point that all convergence criteria are met. Note that the convergence may not be obtainable. This can be due to the low quality of the mesh, numerical scheme used for the solution, relaxation factors, or setting some extreme convergence criteria. One need to perform a diagnostic investigation to improve the solution and reach a converged solution.

4. Result and Discussion

4.1 Study the Effect of Shell Outlet

Before extracting any data for validation and follow-up analyses, it is essential to ensure no numerical problem is present in the setup. Thus, we performed a preliminary calculation, and, with the aid of velocity vectors and contours, searched for possible problematic artifacts. It was evident that at the outlet of the shell, we had an issue. As shown in Figure 21.a, the length of the outlet pipe was too short such that the fluid was not fully developed, causing the “reverse flow” to occur at the outlet. Any circular fluid motion near the outlet results in adverse pressure gradient and “reverse flow” which must be avoided. To tackle this issue, the shell outlet length was increased 3 folds of that of the shell inlet, allowing the fluid to be fully developed. The achieved uniform flow at the outlet of the shell is shown in Figure 21.b; the velocity vectors are all upward and the velocity contour is fully green, without any indication of reverse flow (blue spots).

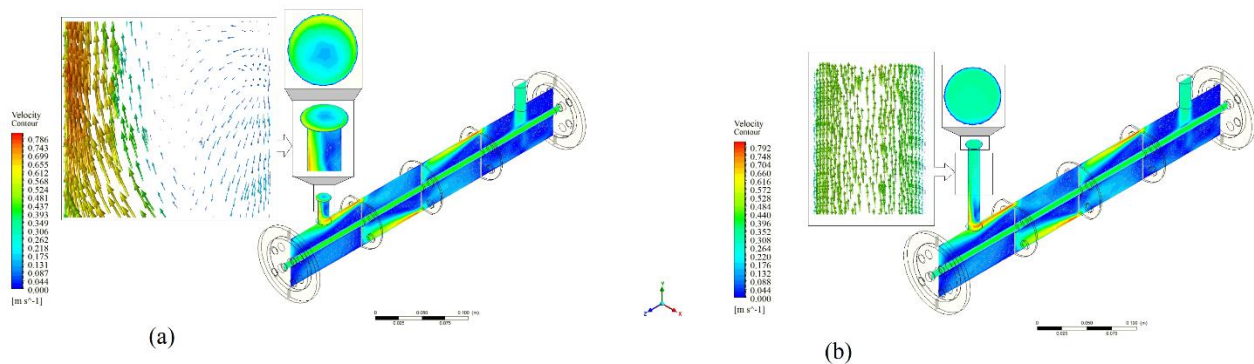


Figure 21. Velocity contour and vectors showing the reverse flow at shell outlet (a) and the proposed solution (b).

4.2 Comparison Between Experimental and Simulation Data (Validation)

For any simulation results to be considered credible in the decision-making process of design, we need to be confident about the accuracy of the results. There are two assessment techniques through which the level of uncertainties and errors are determined: verification and validation. Comparing the simulation results with analytical solutions or well-known empirical relations are the verification assessment, whereas examination of the simulation results with physical reality through experimentation is called validation assessment.

As previously discussed, this project was designed based on the availability of an experimental STHX unit. The schematic design of this unit is shown in Figure 22. We conducted a series of experiments on the STHX, by changing the flow arrangement (co- and counter-currents) and the volumetric flow rates of hot and cold fluids. In each experimental run, we waited for the system to reach steady-state condition, and then read the inlet/outlet temperatures for both fluids. Overall, the results of 12 experiments, as shown in Table 5, were collected and used for analysis. Note that each run is labelled, and herein forwards, the labels are going to be used to refer to each case.

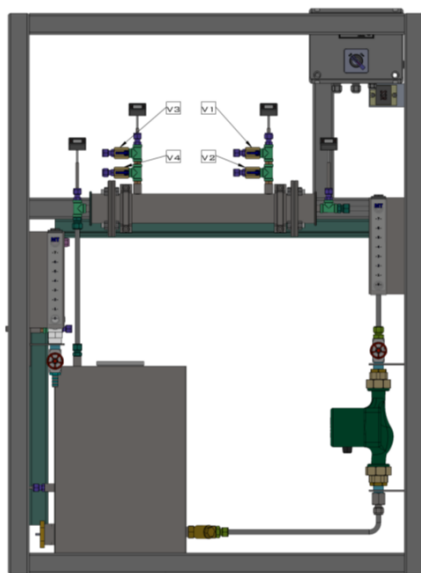


Figure 22. Schematic design of the available STHX unit.

Table 5. 12 selected runs to be performed on the STHX unit.

Flow Arrangement	Label	Flow rate in Shell (LPM)	Flow rate in Tube (LPM)	Velocity at Shell Inlet (m/s)	Velocity at Tube Inlet (m/s)
Counter-current	A	3.0	5.0	0.2	0.3
	B	4.0	5.0	0.3	0.3
	C	5.0	5.0	0.3	0.3
	D	3.0	3.0	0.2	0.2
	E	3.0	4.0	0.2	0.3
	F	3.0	5.0	0.2	0.3
Co-current	A'	3.0	5.0	0.2	0.3
	B'	4.0	5.0	0.3	0.3
	C'	5.0	5.0	0.3	0.3
	D'	3.0	3.0	0.2	0.2
	E'	3.0	4.0	0.2	0.3
	F'	3.0	5.0	0.2	0.3

The temperature readings from the 4 thermometers located at inlets and outlets are the raw data and need to be processed. The heat that was released from the tube-side fluid (Q_H), and the heat absorbed by the shell-side fluid (Q_C) can be calculated based on the mass flow rates (kg/s), specific heat (kJ/kg. K), and inlets/outlets temperatures as shown below.

$$Q_H = \dot{m}C_p(T_{t,in} - T_{t,out})(W)$$

$$Q_C = \dot{m}C_p(T_{sh,out} - T_{sh,in})(W)$$

Theoretically, Q_H must be equal to Q_C to obey the law of conservation of energy. However, in practice, this does not hold, and we expect some deviation between these two values. This can be due to a number of factors including poorly insulated shell walls that allow some heat loss to the surrounding. To get a sense of the scale of this inaccuracy, the relative error ($\left|\frac{Q_C - Q_H}{Q_H}\right| \times 100$) was calculated. The raw temperature data, as well as the calculated heats and the relative errors of all 12 experiments, are shown in Table 6.

Table 6. Experimental data collected for validation studies

Label	$T_{sh,in}$ (°C)	$T_{t,in}$ (°C)	$T_{sh,out}$ (°C)	$T_{t,out}$ (°C)	Q_C (W)	Q_H (W)	%error (-)
A	16.5	58.1	25.8	51.7	1940.7	2300.0	15.6
B	16.6	56.8	24.4	49.5	1627.7	2600.0	37.4
C	16.6	56.9	24.3	48.5	1606.8	2888.3	44.4
D	16.5	57.8	23.4	49.6	1439.9	2802.4	48.6
E	16.5	58.6	23.7	51.1	2003.3	2578.9	22.3
F	16.5	55.5	24.5	49.5	2782.4	2063.1	34.9
A'	16.9	55.9	24.3	49.2	1544.2	2303.0	32.9
B'	17.1	56.8	22.3	49.2	1085.1	2090.6	48.1
C'	17.4	57.2	20.3	49.5	2100.0	2647.6	20.7
D'	16.6	57.5	25.5	52.6	1857.3	1684.9	10.2
E'	16.8	55.5	24.8	51.1	2225.9	1512.9	47.1
F'	16.8	55.3	24.3	49.2	2608.5	2097.5	24.4

The average relative error of 36% is high. We believe the sources of this inconsistency are as follows. First of all, the outer wall of the shell did not have proper thermal insulation. This leads to the exchange of energy with ambient air and the occurrence of considerable heat loss. Secondly, the hot water was constantly circulating in the instrument, but the size of the hot water reservoir tank was small, leading to severe fluctuations in the inlet temperature. Consequently, it was challenging to ensure the steady-state condition was achieved. Thirdly, the cold-water supply was the tap water; although the inlet

temperature was adequately constant, the flow rate would experience some inevitable fluctuations occasionally. Lastly, the experiment was conducted only once. In an ideal situation, one should repeat the experiment at least three times and try to reproduce similar results every time to be confident about the validation data. A plot with error bars would then be possible to show the uncertainty in a reported measurement.

In this project, we decided to use Q_H as the validation data of our simulations. This choice was based on the logic that the poorly insulated shell wall was the primary source of error whereas the tubes are fully surrounded by the water in the shell and could not lose any unaccounted heat. Therefore, it was concluded that the released heat by the water in the tube (Q_H) was more reliable.

In the next step, we performed the CFD simulation of the STHX unit with similar inlet temperatures and flow rates as the experimental runs and obtained the outlet temperatures, the amount of the transferred heat, and the relative errors. The results of our simulation are reported in Table 7.

Table 7. Simulation data for the equivalent conditions for our 12 experimental runs.

Label	$T_{sh,in}$ (°C)	$T_{t,in}$ (°C)	$T_{sh,out}$ (°C)	$T_{t,out}$ (°C)	Q_C (W)	Q_H (W)	%error (-)	$Q_{average}$ (W)
A	16.5	58.1	29.6	49.9	2744.8	2816.3	2.5	2780.5
B	16.6	56.8	27.3	47.9	2970.2	3059.7	2.9	3014.9
C	16.6	56.9	25.9	47.1	3238.0	3348.1	3.3	3293.0
D	16.5	57.8	27.7	46.0	2343.2	2416.4	3.0	2379.8
E	16.5	58.6	29.1	48.8	2622.1	2690.5	2.5	2656.3
F	16.5	55.5	28.7	47.9	2556.3	2622.7	2.5	2589.5
A'	16.9	55.9	28.9	48.4	2494.3	2312.6	7.9	2403.4
B'	17.1	56.8	26.7	48.0	2658.6	2711.0	1.9	2684.8
C'	17.4	57.2	27.2	48.1	3056.8	2808.6	8.8	2932.7
D'	16.6	57.5	26.0	45.6	2003.9	2172.2	7.8	2088.0
E'	16.8	55.5	26.6	46.4	2073.4	2237.5	7.3	2155.4
F'	16.8	55.3	27.3	47.7	2231.3	2357.1	5.3	2294.2

One clear advantage of simulation compared to experiment is evident by comparing the relative errors between Q_C and Q_H . The average error in our simulation is 4.5 percent. This is due to the easy control of the conditions in a computer simulation study. For example, ideal insulation of the outer shell wall was achieved by simply defining the wall boundary condition as an insulated surface. Inlet

temperatures and flow rates are also fixed flawlessly. The only source of this 4.5 mismatch is numerical and discretization errors.

Unlike the experimental results, for which the Q_H was selected as the reference values, we took the average of the Q_C and Q_H of our simulation results for comparison. This averaging can reduce the already-small numerical errors and gives us an unbiased final value.

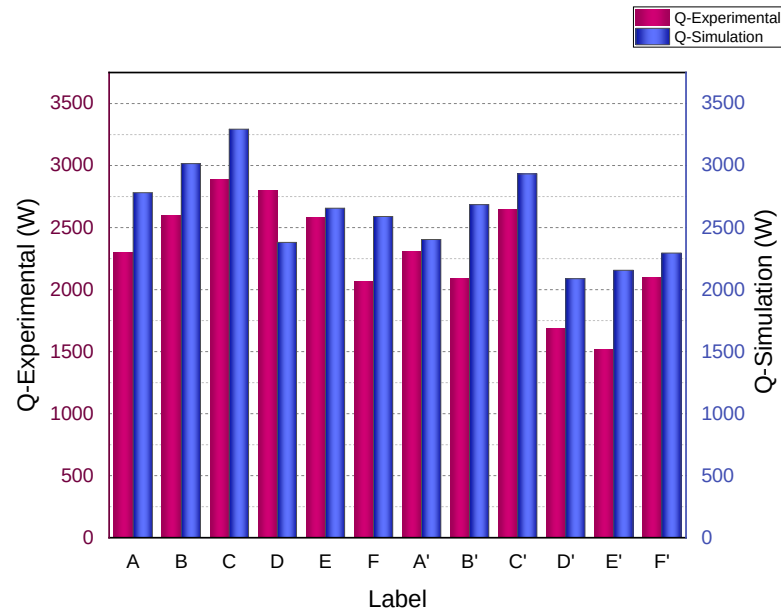


Figure 23. Comparison between exchanged heat in experimental and simulation studies.

Figure 23 is drawn for comparison of experimental and simulation data. In the x-axis, labels are shown that represent the different operating conditions (please refer to Table 6). On the y-axis, the amount of heat transfer, in the unit of Watts, is plotted for experimental (pink) and simulation (purple) studies. Note that the obtained results from experimental and computational techniques are comparable but not identical. We have already mentioned the possible sources of error in our experimental setup. Our computational model, however, showed higher levels of precision and consistency but still we should consider the numerical and modeling errors that are an inseparable part of any CFD study; the Navier-Stokes equations and the law of conservation of energy are “exact” in nature and their analytical solutions must represent the physics. However, computational limitations force us to use approximate turbulence models and numerical solution by discretization of the geometry. Hence, one should be careful when arguing the accuracy of CFD simulations to reproduce experimental results. One thing

that is obvious, though, is that the precision of CFD is much higher because of the easy control of operating conditions (boundary conditions, flow rates, and insulated surfaces).

No matter whether experimental or computational results has higher accuracy, the obtained amounts of heat transfers are on the same scale and our CFD model is working probably. Just to give some perspectives, an improper setup could easily result in orders of magnitude inconsistencies. Also, note that counter-current flows are generally speaking more efficient compared to equivalent co-currents, because of their more uniform temperature distribution in the STHX. This qualitative trend is evident in both studies, which is another reason to trust in the outcomes.

4.3 Comparison between our results and well-known correlations. (Verification)

As described in the methodology (section 3.1), we have selected three well-known correlations for the estimation of the convective heat transfer coefficient on the shell-side (h_C). Using Donohue, Bell-Delaware, and Kern correlations, we have plotted the expected h_C of our operating conditions, as shown in Figure 24. The first obvious observation is that these methods are not in agreement with each other. This can partially be explained by the fact that different aspects of the baffled shell are considered in each method. We, however, believe that the main cause of these inconsistencies is the unconventional dimensions of our STHX. Many rules of thumbs in heat exchanger design are violated in the construction of our unit; we had only five tubes in the shell while the minimum number of tubes are suggested to be eight, the baffle spacing should have been lower than the shell diameter, and the shell to tube diameter must have been larger than 15 whereas in our model it was only 6.3. Therefore, it is evident that we cannot rely on these correlations to estimate the h_C , and, subsequently, the overall heat transfer coefficient (U), since it is very unlikely that an STHX with our parameters have been included in the dataset from which these correlations are suggested.

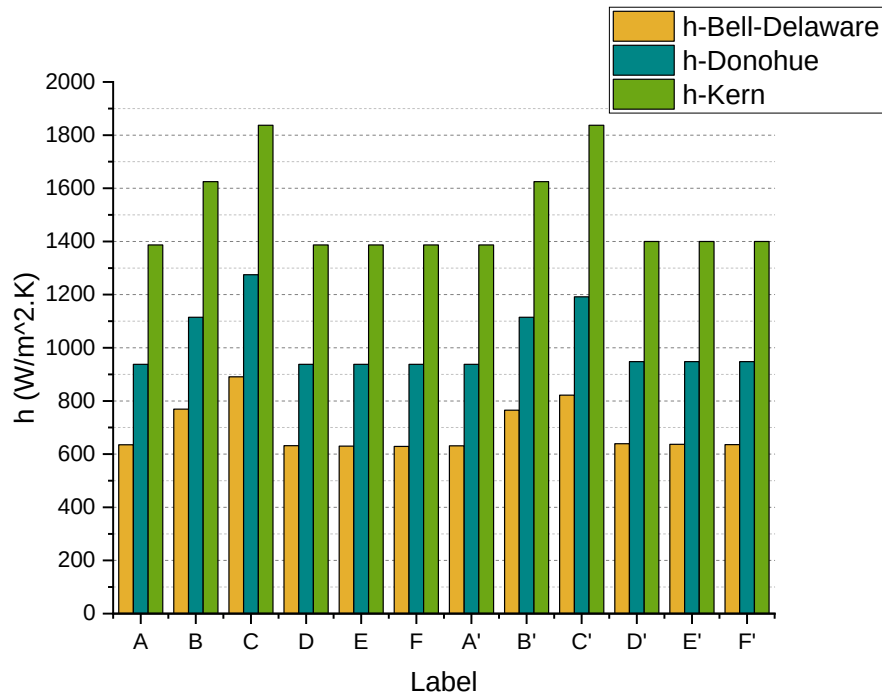


Figure 24. Comparison between shell-side convective heat transfer coefficient calculated by Bell-Delaware, Kern, and Donohue correlations.

For the sake of completeness, we have calculated the corresponding U of all of these 3 correlations and compared them with the experimental and computational (CFD) results in Table 8, in ascending order from left to right.

Table 8. Comparison between overall heat transfer coefficient using five different approaches

Label	$U_{\text{Bell-Delaware}}$ $\text{W/m}^2 \cdot \text{K}$	U_{Donohue} $\text{W/m}^2 \cdot \text{K}$	U_{Kern} $\text{W/m}^2 \cdot \text{K}$	$U_{\text{Experimental}}$ $\text{W/m}^2 \cdot \text{K}$	$U_{\text{Simulation}}$ $\text{W/m}^2 \cdot \text{K}$
A	505	680	889	1086	1434
B	587	769	981	1268	1580
C	655	841	1054	1426	1705
D	434	560	693	1322	1272
E	476	633	810	1182	1370
F	501	680	889	1027	1420
A'	503	680	889	1167	1360
B'	584	769	981	1012	1447
C'	617	805	1054	1232	1592
D'	437	563	697	800	1148
E'	480	638	815	750	1217
F'	506	685	894	1070	1282

As described earlier, the estimated overall heat transfer coefficients are neither consistent among correlation-based calculation, nor with experimental and simulation results. Bell-Delaware, although widely recognized in the industry, gives the most underestimated values, followed by Donohue and Kern. Simulation results, on the other hand, are giving higher values than the experimental results but they are closer to the experimental data compared to the correlation-based estimations.

Although the baffle cut is fixed at 25% in the Kern method, it has the most similarity with our experimental results on a side note. In the last analysis in this project, we increased the baffle cut and noticed that Kern estimations became significantly closer to our simulation results. This study can be considered as a piece of evidence that supports our hypothesis regarding the fact that the unconventional design of our small STHX is the main cause of this poor estimation of well-known correlation-based models.

4.4 Temperature Distributions

One of the main rationales behind simulation studies is that, once the model is successfully prepared, we have access to an extensive amount of data that otherwise would have been impossible to obtain. As far as heat exchangers are concerned, temperature distribution is among the most important information. Experimentally, inlets/outlets temperatures are obtained and then the temperature profile in the heat exchanger is approximated linearly, using log mean temperature difference (LMTD). It is technically difficult, if not impossible, to probe the temperature in both shell and tube all the way in the device. Nonetheless, in our simulation, we obtained more than a million temperature points (as many as the meshes) and we could easily plot a more realistic temperature distribution.

As illustrated in Figure 25, both co- and counter-current flow arrangement for the same flow rates (3LPM for both hot and cold fluids), are compared. On the left image, the co-current flow arrangement is plotted. We can see that for lower Z values, indicating closer to both inlets, the temperature difference between hot and cold fluids is notably high, resulting in the highest local heat transfers. As we approach the outlets, however, the temperature difference decreases and the efficiency decreases. On the other hand, for counter-current, we do not have severe variation between hot/cold fluids temperatures, and the temperature profile is rather constant. This is why usually counter-currents are preferred.

Another observation is that the tube temperature experiences a very flat line close to the inlet section. This area is mainly before the junction of the shell inlet/outlet. This plateau signifies a “dead-zone” and the fact that no heat transfer occurs in this region. Obviously, the fluids are in contact even in this region, but the circular motion of the shell-side at the back of its junctions significantly reduced the heat transfer coefficients.

From this figure, we can also learn that the tube-side fluid can correctly be approximated as a linear function while the shell-side fluid experiences marked fluctuations, with some step-like jumps near the baffles. This might introduce significant deviations in LMTD results and must be taken into account whenever possible.

The driving force of heat transfer in an STHX is the local temperature difference between hot and cold fluids. Therefore, heat transfer continues until the fluids reach the same local temperature. As can be seen in Figure 25, the local temperature difference never came close to the equilibrium condition. This is mainly due to the small surface area for heat transfer in our demonstration unit. This observation signifies that by changing the design of our unit, we could have further exchanged considerable amounts of heat.

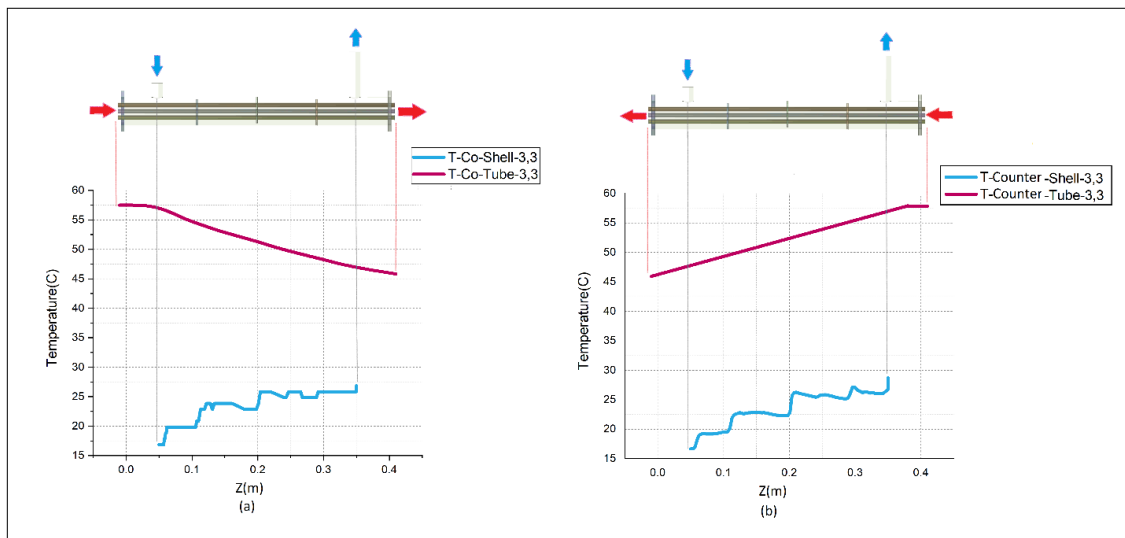


Figure 25. Temperature plot-shell & tube with flow rate of cold=3LPM and flow rate of hot=3LPM (a) Co-Current and (b) Counter -Current

Another study that can be done is the effect of flow rates on heat transfer and temperature profiles. As we have already noted, counter-current is the preferred arrangement and we narrowed down our second analysis to changing the flow rates of the counter-current system only.

Logically, we have 2 inlets and 2 temperature profiles to study. This results in 4 permutations; so that we can study the effect of the

1. cold flow rate on the cold fluid temperature (C-C),
2. cold flow rate on the hot fluid temperature (C-H),
3. hot fluid flow rate on the hot fluid temperature (H-H),
4. hot fluid flow rate on the cold fluid temperature (H-C).

Note that for C-H and H-C cases, the pattern is easier to speculated; while we change the heat transfer rate by manipulating the opposite flow rate and consequently the overall heat transfer coefficient (U), the ratio of temperature differences would be directly proportional to the ratio of heat transfers as shown below:

$$\frac{Q_{Case1}}{Q_{Case2}} = \frac{m_{same} \times C_p \times \Delta T_{Case1}}{m_{same} \times C_p \times \Delta T_{Case2}} \rightarrow \frac{Q_{Case1}}{Q_{Case2}} \propto \frac{\Delta T_{Case1}}{\Delta T_{Case2}}$$

Conversely, for H-H and C-C cases, the flow rates are no longer identical ($\dot{m}_{same}$). Therefore the expected temperature difference is not obvious and it highly depends on the interplay between competing ratios of $\frac{Q_{Case1}}{Q_{Case2}}$ and $\frac{m_{Case1}}{m_{Case2}}$ as shown below:

$$\frac{Q_{Case1}}{Q_{Case2}} = \frac{m_{Case1} \times C_p \times \Delta T_{Case1}}{m_{Case2} \times C_p \times \Delta T_{Case2}} \rightarrow \frac{Q_{Case1}}{Q_{Case2}} \propto \frac{m_{Case1}}{m_{Case2}} * \frac{\Delta T_{Case1}}{\Delta T_{Case2}} \rightarrow \frac{\Delta T_{Case1}}{\Delta T_{Case2}} \propto \frac{Q_{Case1}}{Q_{Case2}} * \frac{m_{Case2}}{m_{Case1}}$$

Considering this effect, the cases of C-H and H-C are plotted in the same figure, Figure 26, and the C-C and H-H cases are categorized differently as shown in Figure 27.

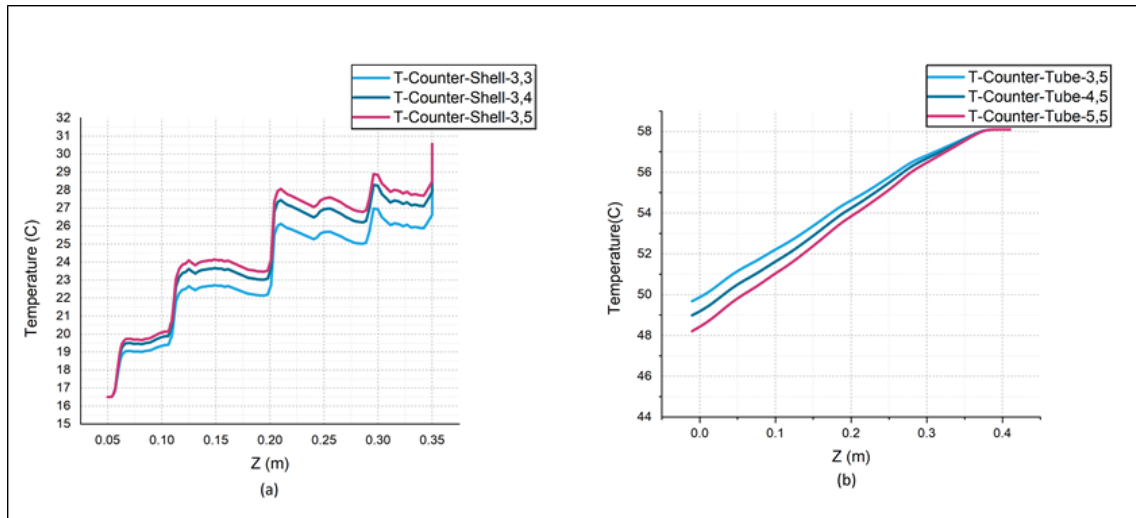


Figure 26. Temperature plot of counter- current system for systems which (a) the flow rate of cold fluid=3 LPM while the flow rate of hot fluid=3,4,5 LPM (H-C), and (b)the flow rate of cold fluid=3,4,5 LPM while flow rate of hot fluid=5 LPM (C-H).

To make sense of the plots on Figure 26, we only need to know the ratio of heat transfers.

As described earlier, in STHXs we mainly define the overall heat transfer coefficient (U) by the convective heat transfer coefficients (h_H and h_C). By changing any flow rate, we increase the Reynolds number and subsequently the Nusselt number and its corresponding convective heat transfer coefficients (h).

Consequently, In the cases plotted in Figure 27, we expect that any increase in flow rate must result in a rise in the heat transfer rate. Since $\left(\frac{Q_{Case1}}{Q_{Case2}} \propto \frac{\Delta T_{Case1}}{\Delta T_{Case2}}\right)$, the outlet temperature should diverge more from the inlet temperature to correspond to a higher ΔT , provided that the inlet temperatures are identical.

In both C-H and H-C cases, one can see that this fact holds perfectly. The shell outlet temperatures of 3-3, 3-4, and 3-5 are in ascending order, providing more deviation from the inlet temperature. Also, the tube outlet temperature of 3-5, 4-5, and 5-5 cases are in a descending order providing a higher deviation from the inlet temperature.

In Figure 27, you can see the different scenarios where the effect of the flow rate on the same fluid temperature difference is studied. On the left-hand graph, the C-C case is illustrated, and on the right-hand one, the H-H case is plotted.

As discussed earlier, we cannot just consider the ratio of the heat transfers to rationalize the pattern of the outlet temperatures. In fact, both flow rates and heat transfers have to be considered simultaneously $\left(\frac{\Delta T_{\text{Case1}}}{\Delta T_{\text{Case2}}} \propto \frac{Q_{\text{Case1}}}{Q_{\text{Case2}}} * \frac{m_{\text{Case2}}}{m_{\text{Case1}}}\right)$. The quantitative comparison showed that between two consecutive runs, the heat transfer has increased by 11% while the rise in flow rate was about 33%. Since the effect of flow rate is more pronounced than the change in heat transfer rates, the pattern would change completely.

More specifically, although the fluid gains higher amounts of thermal energy, its contact time of particles decreases more markedly, leaving the system more quickly with a lesser change in their temperature.

The observed pattern, as shown below, is therefore meaningful.

$$T_{C,\text{out}}(\text{at 5LPM}) < T_{C,\text{out}}(\text{at 4LPM}) < T_{C,\text{out}}(\text{at 3LPM})$$

$$T_{H,\text{out}}(\text{at 5LPM}) > T_{H,\text{out}}(\text{at 4LPM}) > T_{H,\text{out}}(\text{at 3LPM})$$

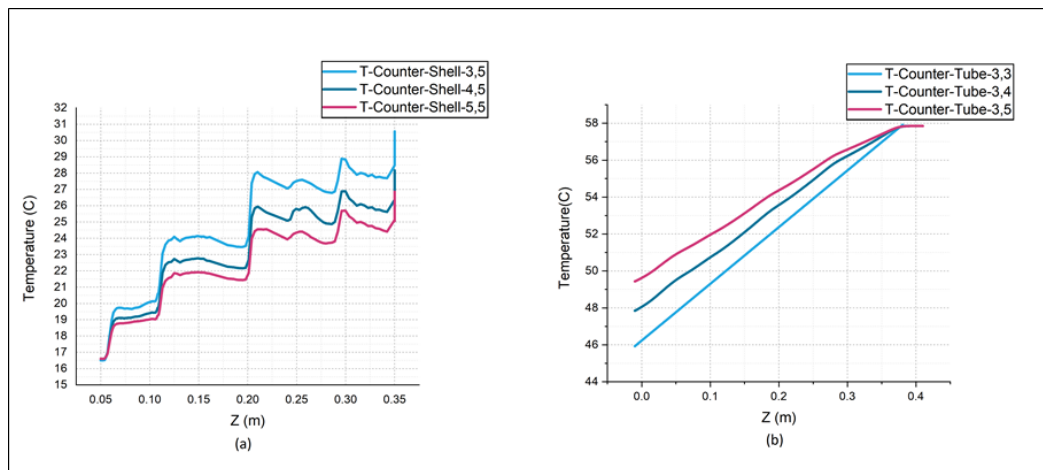


Figure 27. Temperature plot counter-current (a)flow rate of cold fluid=3,4,5 LPM - flow rate of hot fluid=5 LPM in shell-side (C-C), and (b) flow rate of cold fluid=3 LPM- flow rate of hot fluid=3,4,5 LPM in tube side (H-H).

4.5 Study the Effect of Number of Baffles

In the discussion of heat exchangers, reducing pressure drop and increasing heat transfer are of central interest and importance. Increasing the number of baffles in STHXs can be advantageous and results in an increase in turbulence in the fluid, and also the fluid path length, but at the same time it increases the pressure drop.

Therefore, the argument of the number of baffles is an optimization problem. One must choose the number of baffles somehow that the gain in the amount of heat transfer is achieved at a reasonable expense of an inevitable rise in pressure drop.

In an experimental study, changing the number of baffles is frustratingly time-consuming and costly. Nonetheless, in computer modeling, one can quickly change the number of baffles and perform a complete study. The ease of changing the design is the second benefit of CFD studies, besides the above-mentioned availability of “Full” solution data.

Therefore, we decided to change the number of baffles from zero (un-baffled) to 10 baffles with equal spacing. Note that after 5 baffles, we only divided the distance between shell inlet and outlet into equal distances to avoid inserting baffles in the dead zones at two ends of the device. For each case, the currents are in a counter-current arrangement and the flow rates are fixed to only focus on the effect of the number of baffles. Also, note that the baffle cuts were not altered from the original design of the experimental device.

The velocity contours of all eleven cases are shown in Figure 29 and the comparison between the main benefit (Q) and drawback (pressure drop) of baffles are plotted in Figure 28.

On the x-axis, we have the number of baffles, which ranges from zero to ten. On the two vertical axes, Q and pressure drop are shown in units of W and Pa, respectively. According to the diagrams and contours below, we see that the heat transfer is at its lowest when we don't have any baffle, and at most when we increase the number to 10.

This upward trend is expected but the main point here is that its slope is far from linear. After four baffles, the improvement in heat transfer becomes marginal. Pressure drop, on the other hand, is increasing with at a constant pace, from 0 to 10 baffles.

Here, we can conclude that either 4 or 5 number of baffles, as long as the baffle cut does not change, could have resulted in a better performance, compared to the device that was manufactured with only 3 baffles.

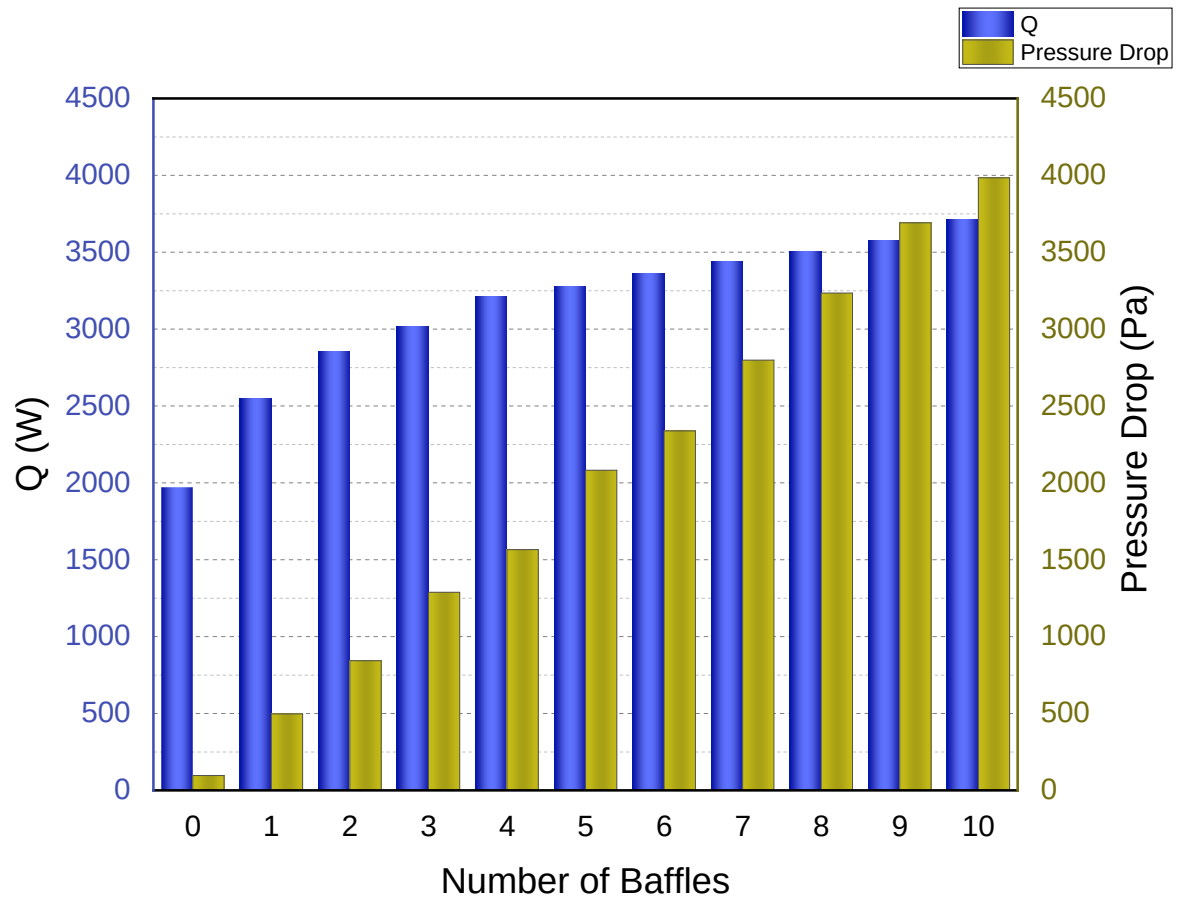


Figure 28. Comparison between heat transfer and pressure drop of 11 distinct numbers of baffles.

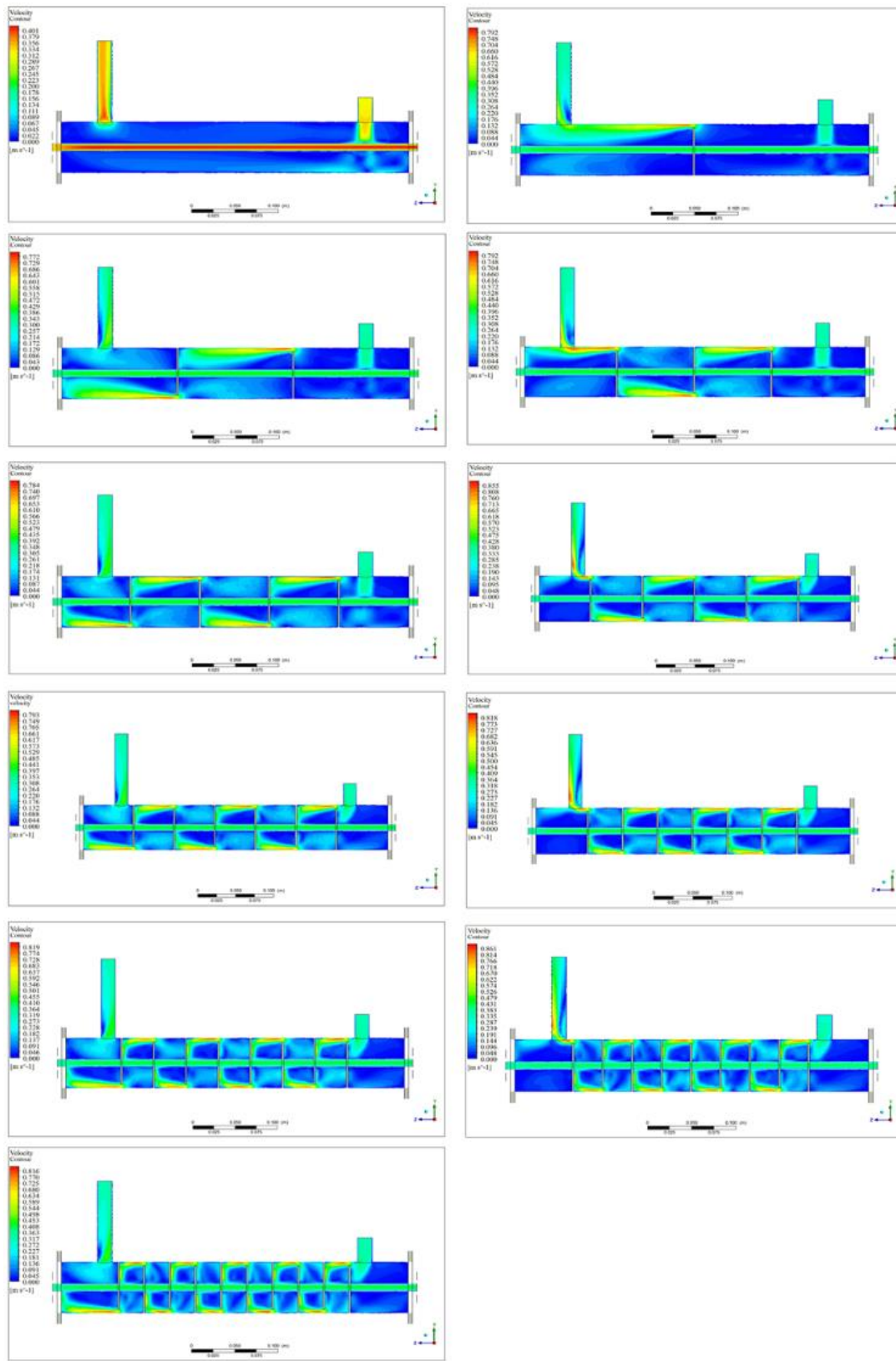


Figure 29. The velocity vectors of 11 cases with different numbers of baffles.

4.6 Study the Effect of Baffle Cut Orientations

In an STHX, baffles are one of the most significant design parameters that can be altered to optimize the performance of the device. Not only we can change the number of baffles, we can also change the baffle cuts, baffle orientations, or baffle type. In this section, we decided to modify the orientations of the baffles to see if we can further improve the thermal performance of the STHX unit. Baffle cuts were originally located in an up-down manner.

We compared this original orientation with left-right, and clockwise orientations. Since we have already shown that 4 baffles perform closest to the optimum conditions, and also to have a full clockwise rotation with 90-degree angles between the baffle cuts, we modeled this analysis with 4 baffles. The results of this investigation are shown in Figure 30.

In this diagram, the x-axis represents the baffle directions, and the two y-axes represent the Q , expressed in W, and the pressure drop, denoted by the Pa unit. According to the diagram, the heat transfers in the two up-down and clockwise models are almost the same, while in the left-right direction more heat transfer was achieved. Pressure drops are increasing in the order of up-down and left-right and clockwise directions. Note that the scales of the y-axes are not drastic, and the variations are minute. We can conclude that baffle orientations, at least in this case, could not lead to significant gains and the default up-down arrangement is already reasonable.

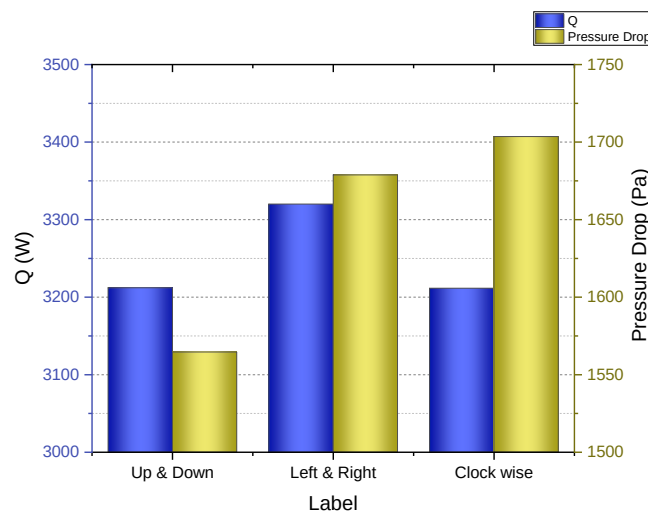


Figure 30. Q and pressure drop of the shell-side fluid with different baffle cut orientations.

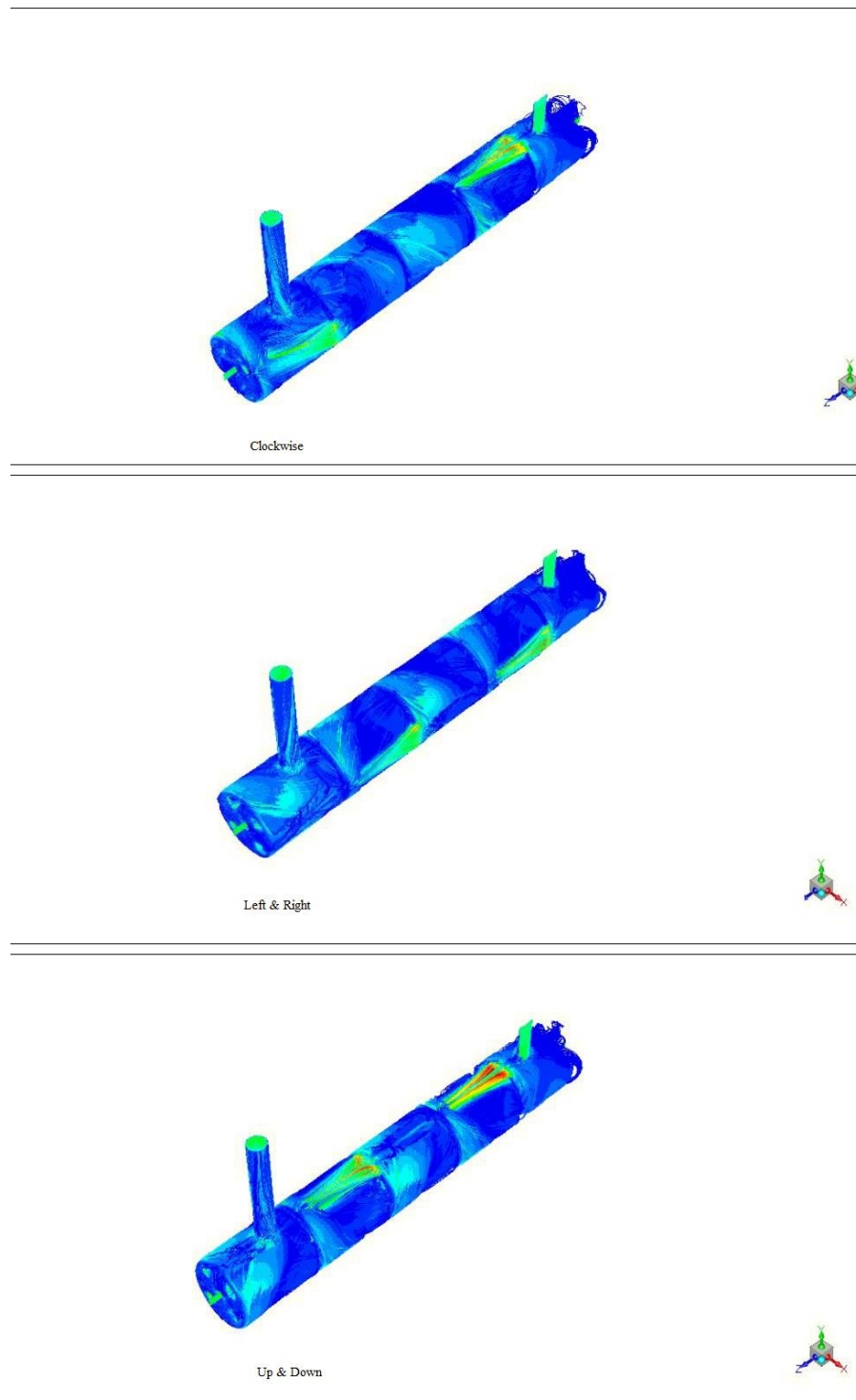


Figure 31. Streamlines of the shell-side fluid (up-down, left-right, and clockwise), colored by the velocity.

4.7 Final Suggestions on the Design Modification

As a final analysis of this thesis, we are suggesting an alternative design for this system. So far, it has been noted in the methodology section that neither the number of baffles (baffle spacing) nor the baffle cut is selected based on the conventional rules. Therefore, we increased the baffle cut to the point that it was touching the tubes (19.2% baffle cut), and also decreased the baffle spacing to 38 mm, based on the observed velocity contours, to minimize the dead zones (blue regions of the contour). This showed that we could increase the number of baffles to 8 instead of 3 for a 19.2% baffle cut. The comparison between heat transfer rate and pressure drop of original and our optimized design are shown in Table 9. Also, the fluid velocity fields of these cases are illustrated in Figure 32 for clarification.

Table 9. Comparing Q and ΔP of the original manufactured design without suggested modifications

	Original Design (3 # of Baffles & 7.7% baffle cut)	CFD Optimized Design (8 # of Baffles & 19.2% baffle cut)
Q shell (W)	2971.6	3268.8
Pressure Drop shell (Pa)	1287.4	383.2

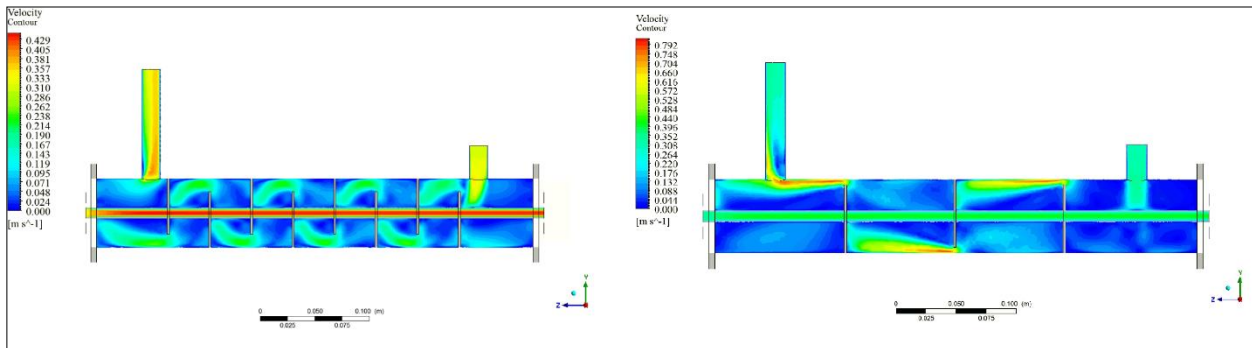


Figure 32. Contours of the fluid flow in the original (right) and our suggested design (left).

Based on the results shown in Table 9, we were able to increase the heat transfer rate by 10% while decreasing the pressure drop significantly, to less than one-third of the original model.

5. Conclusion

Shell and tube heat exchanger (STHX) is one of the most commonly used equipment in process integration, with baffles being an integral part of it. Baffles are flexible in design, and their experimental fine-tuning is not feasible in practice. Here, we used Ansys Fluent to simulate an already-constructed STHX demonstration unit, focusing on baffle optimization.

To achieve this goal, we changed three design elements, namely the number of baffles, baffle cut orientation, and baffle cut size. Altering the number of baffles from zero to ten showed us that four baffles were the optimum choice, maximizing the net gain in the thermal efficiency. In the second study, three baffle cut orientations, namely up-down, left-right, and clockwise, were compared which did not illustrate any substantial gain. Lastly, we noticed from the velocity contours that the baffle cut size was improperly small in the original design, leading to an overwhelming amount of dead-zone regions throughout the shell and unnecessary pressure drops. We were able to enhance the thermal performance of the device by increasing the baffle cut size, and at the same time, decrease its pressure drop to one third.

Moreover, CFD enabled us to plot the temperature distribution and contours, velocity vectors and contours, and pathlines along the heat exchanger in order to provide additional in-depth insights about the flow. In the coming semesters, students of the Heat Transfer course can use this information to get acquainted with the equipment, prior to performing the lab session.

In conclusion, CFD enabled us to improve the thermal efficiency of an STHX by fine-tuning some selected baffle characteristics, and also provided useful educational resources for the Heat Transfer lab session.

6. Recommendations and Suggestions for Future Work

For students who want to explore topics related to this project in the future, the below ideas can be considered.

- First of all, this project demonstrated that Ansys Fluent, and CFD in general, is capable of reproducing experimental results within reasonable precision and accuracy. Employing similar modeling techniques, one can simulate heat exchangers with fluids other than water, such as oil products.
- Also, one can look at industrial large-scale models. Modeling large tube bundles made of hundreds of tubes like the Figure 33.b, increasing the number of both tubes and shell passes, and utilizing more complicated baffle types, such as helical baffles (Figure 33.a)³², can all be noted as potential studies.

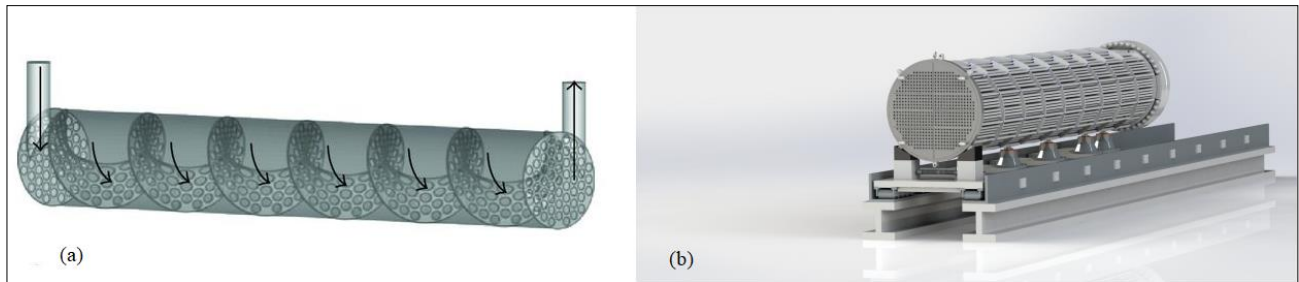


Figure 33. (a) helical baffle and (b) a large industrial STHX.

Source of the photo of Figure 33.b:

<https://academy.autodesk.com/portfolios/tube-bundle-insertion-system>

- Another recommendation is to expand on the mathematical models of this project and simulate more challenging physics. This may include the addition of boiling, evaporation, and condensation models to simulate specific types of heat exchangers that are inseparable parts of many units in chemical engineering, such as reboilers and condensers as integral parts of distillation columns (Figure 34)³³.

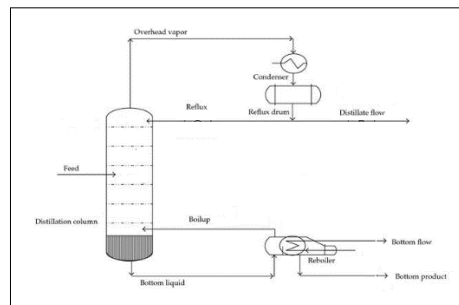


Figure 34. Schematic design of a distillation column

References

1. Aslam Bhutta, M. M. *et al.* CFD applications in various heat exchangers design: A review. *Appl. Therm. Eng.* **32**, 1–12 (2012).
2. Cengel, Y. A. & Ghajar, A. J. Heat exchangers. in *heat and mass transfer: fundamentals and applications* 647–694 (McGraw-Hill Education, 2015).
3. Toledo, R. T., Singh, R. K. & Kong, F. Heat Transfer BT - Fundamentals of Food Process Engineering. in (eds. Toledo, R. T., Singh, R. K. & Kong, F.) 135–182 (Springer International Publishing, 2018).
4. 1Jay J. Bhavsar, 2V K. Matawala, 3S. Dixit. Design and Experimental Analysis of Spiral Tube Heat Exchanger. *Int. J. Mech. Prod. Eng.* **1**, 37–42 (2013).
5. Andrzejczyk, R. & Muszyński, T. Performance analyses of helical coil heat exchangers. The effect of external coil surface modification on heat exchanger effectiveness. *Arch. Thermodyn.* 137–159 (2016).
6. Parish, R. A. & Rhea, R. A. Chapter 6 - Mechanical Equipment. in (eds. Parish, R. A. & Rhea, R. A. B. T.-P. D. and D. (Third E.) 112–133 (Gulf Professional Publishing, 2012).
7. Erdogan, A., Colpan, C. O. & Cakici, D. M. Thermal design and analysis of a shell and tube heat exchanger integrating a geothermal based organic Rankine cycle and parabolic trough solar collectors. *Renew. Energy* **109**, 372–391 (2017).
8. Bichkar, P., Dandgaval, O., Dalvi, P., Godase, R. & Dey, T. Study of Shell and Tube Heat Exchanger with the Effect of Types of Baffles. *Procedia Manuf.* **20**, 195–200 (2018).
9. Mukherjee, R. [Engineers I. L. (India)]. Effectively design shell-and-tube heat exchangers. (1998).
10. Donohue, D. A. Heat Transfer and Pressure Drop in Heat Exchangers. *Ind. Eng. Chem.* **41**, 2499–2511 (1949).
11. Sieder, E. N. & Tate, G. E. Heat Transfer and Pressure Drop of Liquids in Tubes. *Ind. Eng. Chem.* **28**, 1429–1435 (1936).
12. Kern, D. Q. *Process Heat Transfer*. (McGraw-Hill, 1950).
13. Bell, K. *Final report of the cooperative research program on shell and tube heat exchangers*. (University of Delaware, Engineering Experimental Station, 1963).
14. Kim, W.-K. & Aicher, T. Experimental investigation of heat transfer in shell- and- tube heat

- exchangers without baffles. *Korean J. Chem. Eng.* **14**, 93–100 (1997).
15. Gaddis, E. S. & Gnielinski, V. Pressure drop on the shell side of shell-and-tube heat exchangers with segmental baffles. *Chem. Eng. Process. Process Intensif.* **36**, 149–159 (1997).
 16. Li, H. D. & Kottke, V. Visualization and determination of local heat transfer coefficients in shell-and-tube heat exchangers for in-line tube arrangement by mass transfer measurements. *Heat Mass Transf.* **33**, 371–376 (1998).
 17. Ali Kara, Y. & Güraras, Ö. A computer program for designing of shell-and-tube heat exchangers. *Appl. Therm. Eng.* **24**, 1797–1805 (2004).
 18. Ayub, Z. H. A new chart method for evaluating single-phase shell side heat transfer coefficient in a single segmental shell and tube heat exchanger. *Appl. Therm. Eng.* **25**, 2412–2420 (2005).
 19. Khalil, E. E. & Akademia Baru, P. CFD History and Applications. *CFD Lett.* **4**, 2012 (2012).
 20. Prithiviraj, M. & Andrews, M. J. Three-Dimensional Numerical Simulation of Shell-and-Tube Heat Exchanger. Part II: Heat Transfer. *Numer. Heat Transf. Part A Appl.* **33**, 817–828 (1998).
 21. Prithiviraj, M. & Andrews, M. J. Three Dimensional Numerical Simulation of Shell-and-Tube Heat Exchangers. Part I: Foundation and Fluid Mechanics. *Numer. Heat Transf. Part A Appl.* **33**, 799–816 (1998).
 22. Prithiviraj, M. & Andrews, M. J. Comparison of a Three-Dimensional Numerical Model with Existing Methods for Prediction of Flow in Shell-and-Tube Heat Exchangers. *Heat Transf. Eng.* **20**, 15–19 (1999).
 23. Wang, Q., Chen, Q., Chen, G. & Zeng, M. Numerical investigation on combined multiple shell-pass shell-and-tube heat exchanger with continuous helical baffles. *Int. J. Heat Mass Transf.* **52**, 1214–1222 (2009).
 24. Ozden, E. & Tari, I. Shell side CFD analysis of a small shell-and-tube heat exchanger. *Energy Convers. Manag.* **51**, 1004–1014 (2010).
 25. Zeng, W.-L. & Zhang, Z.-G. Problems involved in the shell-side performance of a longitudinal-flow shell-and-tube heat exchanger with decrease in L/D and its solutions. *Int. J. Low-Carbon Technol.* **7**, (2012).
 26. Pal, E., Kumar, I., Joshi, J. B. & Maheshwari, N. K. CFD simulations of shell-side flow in a shell-and-tube type heat exchanger with and without baffles. *Chem. Eng. Sci.* **143**, 314–340 (2016).

27. Cengel, Y. A. & Ghajar, A. J. heat and mass transfer: fundamentals and applications. in *heat and mass transfer: fundamentals and applications* 471–517 (McGraw-Hill Education, 2015).
28. Abraham, J. P., Sparrow, E. M. & Tong, J. C. K. Heat transfer in all pipe flow regimes: laminar, transitional/intermittent, and turbulent. *Int. J. Heat Mass Transf.* **52**, 557–563 (2009).
29. Petukhov, B. S. Heat Transfer and Friction in Turbulent Pipe Flow with Variable Physical Properties. in (eds. Hartnett, J. P. & Irvine, T. F. B. T.-A. in H. T.) vol. 6 503–564 (Elsevier, 1970).
30. ANSYS Fluent Theory Guide. (2013).
31. Cengel, Y. A. & Ghajar, A. J. *heat and mass transfer: fundamentals and applications*. (McGraw-Hill Education, 2015).
32. Ji, S., Du, W., Wang, P. & Cheng, L. Numerical Investigation on Double Shell-Pass Shell-and-Tube Heat Exchanger with Continuous Helical Baffles. *J. Thermodyn.* **2011**, 839468 (2011).
33. Minh, V. T. & Abdul Rani, A. M. Modeling and Control of Distillation Column in a Petroleum Process. *Math. Probl. Eng.* **2009**, 404702 (2009).