Improving the efficiency of spiral heat exchanger based on pressure drop

Cite as: AIP Conference Proceedings **2233**, 020030 (2020); https://doi.org/10.1063/5.0001350

Talha Omar Bashir Varvani, and Abdulkareem Sh. Mahdi Al-Obaidi

AIP Conference Proceedings **2233**, 020030 (2020); https://doi.org/10.1063/5.0001350 **2233**, 020030

Improving the Efficiency of Spiral Heat Exchanger Based on Pressure Drop

Talha Omar Bashir Varvani[1], Abdulkareem Sh. Mahdi Al-Obaidi[1,a)]

[1]Taylor's University Lakeside Campus, Selangor, Malaysia

[a)] corresponding author: abdulkareem.mahdi@taylors.edu.my

Abstract. A Spiral Heat Exchanger (SHE) is a compact double-coiled heat exchanger with spiral tubing. It is widely known as one of the most efficient heat transfer systems. One of the issues observed on the SHE performance is that it has internal leakage defects from the mixing of the cold and hot fluids and gasket leakages due to an imbalance of pressure and temperature. Therefore, the aim of this project is to numerically analyze the parameters that affect the SHE performance using CFD numerical solutions. The aim is to come up with better alternatives to further improve the efficiency. Once the simulated results have been compiled, they shall be compared with those results obtained from the real existing data, and the improved efficiency will be derived from the observation that the simulated results are better. In this paper, the effect of a number of variable parameters are analyzed. These parameters are inlet and outlet pressures for both hot and cold fluids. Numerical simulation aims to find the best combination of the parameters, which give better performance compared with the existing one. Based on the results achieved from the data attained from the company, it was noted that the pressure drop reduces when there is an increase in cold input pressure from 480 kPa to 510 kPa while decreasing the hot input pressure from 520 kPa to 490 kPa based on the range of data. In conclusion, the pressure drop becomes better only at the hotter fluid region by 99.4 % while for the colder region it reduces by 50%. Therefore, physical design change is recommended to the SHE to establish better results.

INTRODUCTION

Spiral Heat Exchanger (SHE) is a type of heat exchanger used to transfer heat energy from one fluid to another. This particular heat exchanger as shown in Figure 1 [1], is made fully by welding of a cylindrical tube containing two concentric coiled-shape spiral flow plates inside with a gap between each spiraled coil [2]. There is a non-contact transfer of heat between the two fluids and two inlets and two outlets in a SHE for hot and cold fluid and the process can be done with liquid-liquid, gas-gas and liquid-gas.

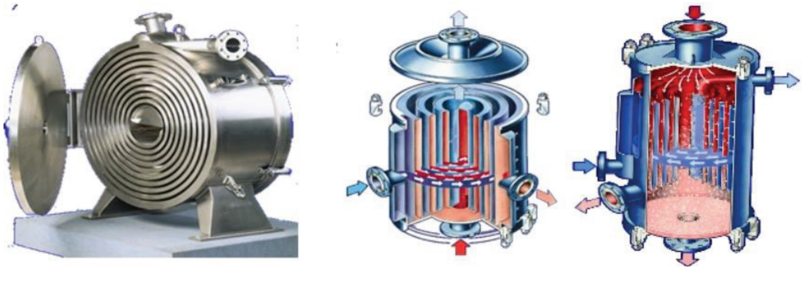


FIGURE 1. SHE flow process [1]

SHE is one of the best designed and constructed heat exchangers based on its heat transfer effectiveness [3] and its capability to reduce costs and fouling [2]. In addition to that, numerical investigation conducted using ANSYS

13th International Engineering Research Conference (13th EURECA 2019)
AIP Conf. Proc. 2233, 020030-1–020030-12; https://doi.org/10.1063/5.0001350
Published by AIP Publishing. 978-0-7354-1992-6/$30.00

FLUENT for the heat transfer and pressure drop using Reynolds number (Re) in relation with the Nusselt number (Nu) show high turbulent flow in SHE at lower velocity compared to other types of heat exchangers [4].

The SHE is used in various industries such as heating and refrigeration, air conditioning, power stations, etc. Therefore, it is one of the important technologies. As it is seen in Figure 2, the heat exchangers are used in regeneration flow or recuperator flow and depend on which process flow it is used, the design and construction is implemented. There are two types of spiral heat exchangers, namely, spiral tube heat exchanger (STHE) and spiral plate heat exchanger (SPHE). The STHE is a coil assembly of plates for two fluids in a long shell tube made for optimized heat transfer efficiency and space. For the SPHE, it is made of two long plates concentric to spiral flow passage channeling maximum heat transfer and ease of access.

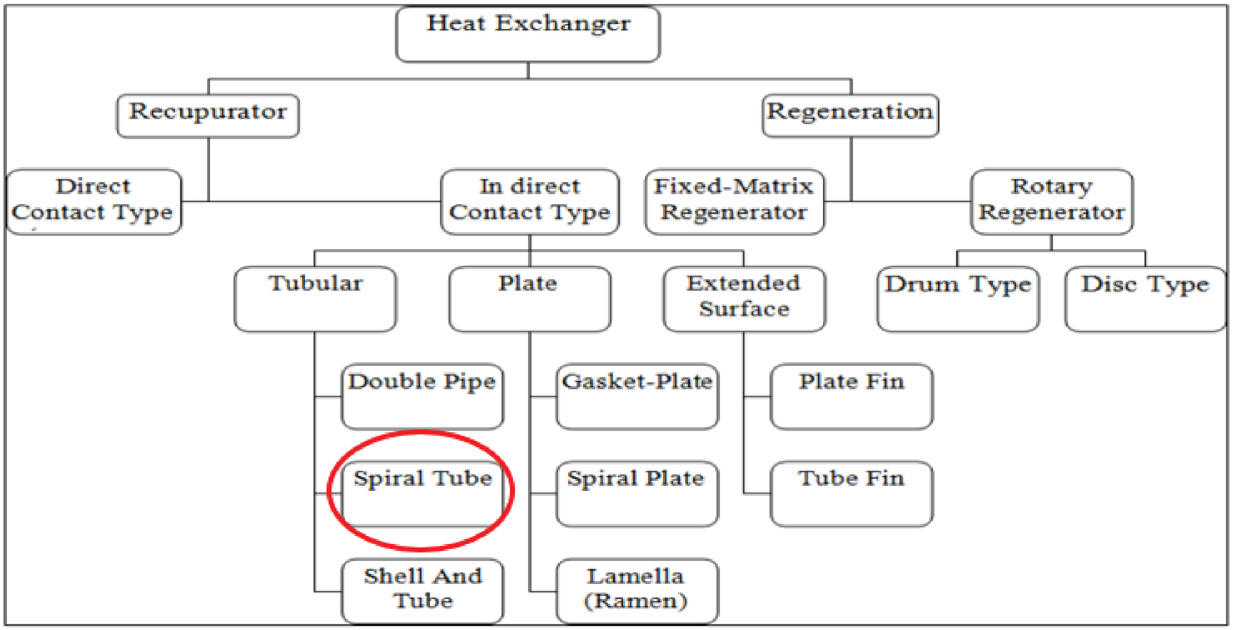


FIGURE 2. Types of heat exchangers [5]

SHE uses gaskets internally above and below the end-connections and for pipeline connections. The gaskets as shown in Figure 3 are O-rings made of carbon form usually graphite with a metallic stainless-steel surface in between the graphite. The gaskets are used for different type of fluids with different specification of size and design.

FIGURE 3. Types of gaskets [6]

With working experience gained during the industrial training at a company and from observations, it was found that an existing SHE used suffers with two problems; internal and external. The external problem comes with leaking gaskets with time for the SHE as well as the pipeline connected to it. The problem occurs due to the unbalanced pressure and temperatures causing sudden leakage or causing the bolts and nuts to loosen and then fluids leak. The internal problem arises with the imbalance of pressure from the fluids within the SHE, which causes the mixing of the two fluids thus results in not getting the required temperature and pressure values. The external problem is overcome by stopping the SHE and changing the required gaskets, while the internal problem is only solved by further heating or cooling the fluid to achieve the required temperature and pressure before continuing the next process. The current solutions to overcome these problems are costly, time and energy consuming.

This research paper is focusing specifically on how to address above-mentioned problem to improve the efficiency of that SHE. It was found, based on the literature reviewed, that the gasket leakages and internal mixing of cold and hot fluid in the SHE are not explicitly discussed. Since most of the previous research is based on the experimental analysis of SHE and its design and construction, knowledge achieved from industrial training is applied to contribute in solving these problems. In addition, previous research is based on gasket leakages for different heat exchangers thus that knowledge is applied to the SHE. Accordingly, the objectives of this paper are to identify, analyze and evaluate the factors causing the leakage of the fluids using computational fluid dynamics (CFD) and then propose an engineering solution to improve the efficiency of the current SHE used by the company by improving its overall performance. The overall performance will be improved by reducing the pressure drop in the SHE. This will help in overcoming the above-mentioned problems. In addition to that, improving the efficiency will result in saving cost of running the heat exchange and its maintenance.

LITERATURE REVIEW

In this section, the literature review is arranged into three parts; first is for the heat transfer and pressure drop, second is for gaskets leakages and the last part the literature reviewed is summarized with focusing on the research gap and motivation.

Heat Transfer and Pressure Drop

FaJiang *et al*. [7] experimentally investigated heat transfer and flowing resistance of air flow over spiral finned tube heat exchanger (SFTHE) with 13 different spiral tubes with same outer diameter but different fin pitch, fin height, transverse and longitudinal pitch with a range of Reynolds number (Re). The aim was to obtain the heat transfer Nusselt number and flowing resistance Euler number in relation with the variables applied. It was concluded after experimentally deriving the equations related to the heat transfer and flowing resistance that the heat transfer Nusselt number increases with the increasing Re, fin pitch and transverse pitch. However, it decreases with the increasing longitudinal pitch and fin height. Also, for the flowing resistance Euler number, although increases with the increment in fin height, it decreases with the increasing Re, fin pitch, transverse and longitudinal pitch.

Nguyen and San [8] developed a mathematical model to investigate the heat transfer performance of a spiral heat exchanger (SHE). This method used two dimensionless temperature fluids to evaluate heat transfer effectiveness using number of transfer units (NTU), a change in number of turns and ratio of capacity rates. It was concluded that as NTU value increases, the heat transfer effectiveness increases too reaching maximum whereby it then reduces slightly. Also, as the number of turns increases, the effectiveness increases till the SHE effectiveness reaches the effectiveness of a counter-flow heat exchanger. In addition, the effectiveness values are same despite the change in number of turns when the ratio of capacity rate is zero.

Tang *et al.* [4] numerically investigated on gas flow heat transfer and pressure drop in shell side of spiral-wound heat exchanger (SWHE). Simulation of different gases such as nitrogen, methane and ethane as working fluids in CFD having boundary conditions to achieve the results for heat transfer coefficient and pressure drop related to radial angle, winding angle and Reynolds number (Re). Based on findings, with an increase in Re, there is a higher heat transfer coefficient and an increase in Nusselt number, hence, this indicates that heat flux and pressure are independent of each other. Lastly, the friction factor is also affected by the Re, which is shown in both simulated and experimental results.

Fan *et al*. [9] studied a comprehensive failure analysis on internal leakages of heat exchanger plates operated in a power plant. The analysis showed that the causes of internal leakage failure were bowl-like perforations on cross contact points between zigzag peaks on neighboring plates while pitting, corrosion and fretting work was instigated during perforation formation.

Mostafazade and Afshin [10] investigated numerically the thermo-hydraulic characteristics of Nusselt number and friction factor of flow in a shell side spiral wound heat exchanger (SWHE) at different Reynolds number considering six parameters; start factor, tube outside diameter, number of tubes in first layer, number of layers, longitudinal pitch and radial pitch. Also, as shown in Figure 4, the turbulence model is varied to attain the difference in result simulations for Nusselt number at various Reynolds number. The simulation results indicate that the number of tubes in first layer does not have any influence in the flow characteristics. Also, when there is an increase in Reynolds number or number of layers or decrease in radial pitch, the Nusselt number and friction factor is augmented. It was concluded that large longitudinal and radial pitches aren't recommended in the designing of heat exchangers as it takes more space and weight.

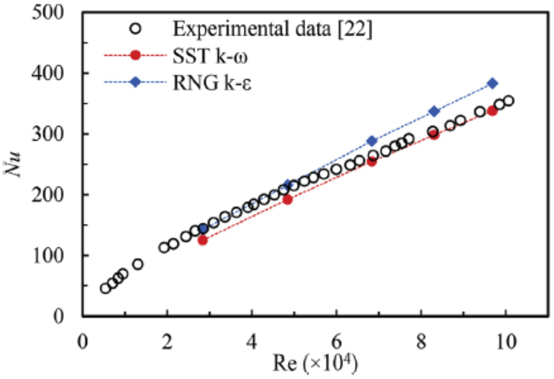


FIGURE 4. Validation of Nusselt number for different turbulence models [8]

Li *et al*. [11] established mathematical model using ANSYS CFX to explore the pressure drop and heat transfer characteristics of condensation flow with three hydrocarbons methane/ethane/propane mixtures in a spiral pipe condition under static and heaving conditions. The variables set were for mass flux, vapor quality, heat flux, saturation pressure, hydraulic diameter, curvature diameter and inclination angle. It was concluded that methane/ethane mixture is the best mixture as it has the best heat transfer and lower pressure drop. For static condition, both heat transfer and pressure drop increase with the rise in vapor quality and mass flux and drop in saturation pressure and hydraulic diameter while show no change with inclination angle, curvature diameter and heat flux. For the heaving condition, both heat transfer and pressure drop weaken with the rise in vapor quality.

Gaskets Leakage

Gao *et al*. [12] developed a combined numerical CFD solution and experimental calculation using reverse heat load method (RHLM) approach to measure the heat leakage through a refrigerator door gasketed region. It was concluded that the heat leakage increases if the system is fully operational due to the temperature difference between the ambient environment and the freezer.

Feng *et al*.[13] developed a model for non-metallic gaskets to predict the leakage rate of gasket sealing connections for compressed non-asbestos fiber gaskets and flexible graphite gaskets reinforced with tanged metal sheet. Using the laminar flow rate and molecular flow rate equations for different gases, the coefficient of leakage rate formula were obtained for each type and size of gasket. It was concluded that the overall leakage rate is the sum of laminar and molecular flow rate and the leakage rate is affected by change in pressure.

Yang *et al*. [14] investigated the failure analysis of O-ring gaskets in a hydraulic system using a series of microscopic and macroscopic analysis methods to identify root problems. The analysis concluded that the mismatch of sizes between O-rings and the groove of metal union is the primary reason which causes surface damage after initial compression strain and deformation. Also, the pressure exerted during assembly was high which can be a cause of failure.

Al-Kayiem *et al.* [15] conducted an experimental study to enhance the thermal performance of double pipe heat exchanger (DPHE) by imposing turbulence in the annular flow using artificial roughening. . Two cases of rib's pitch to height ratios, equal to 10 and 15 and three height to hydraulic diameter, equal to 0.0595, 0.083, and 0.107 have been studied within Reynolds number range of 2900 to 21000 in the cold annulus. From the results obtained, it is recommended to use ribs installed on the inner surface of the annulus ribs to hydraulic diameter in the range of 0.06 ± 0.005. Also, it is recommended to investigate further parameters to explore further on the influencing of the ribs on the hydrothermal performance of the DPHE.

Abid *et al*. [16] studied the strength and sealing performance of gasketed bolted flanged pipe joint using 3D nonlinear finite element analysis (FEA) numerical solution with respect to internal pressure, axial and thermal loading. The solutions resulted in concluding that the strength of the joint was affected by axial and thermal loading while the sealing performance of the joint was affected slightly by pressure and partially by axial and thermal loading.

Tahrour *et al.* [17] simulated the conjugate conduction-convection of heat transfer problems in eccentric annular finned tube heat exchangers using CFD. The heat transfer coefficient was evaluated over fin surfaces, as well as the fin efficiency and the pressure drop. This study aimed to determine the optimum tube position in the circular fin that maximizes heat dissipation and minimizes pressure drop. It was found that for small fin spacings, the eccentric annular finned tube is more efficient than the concentric one. Among the cases examined, the average heat transfer coefficient of the eccentric annular-finned tube, for a tube shift St =12 mm and a Reynolds number Re = 9923, was 7.61% greater than that of the concentric one. This gain is associated with a 43.09% reduction in pressure drop

Summary and Research Gap

The reviewed literatures show how the heat transfer effectiveness is calculated using experimental and numerical methods and how the pressure drop changes with certain variables. These literatures also show the relationship of Reynolds number (Re) with Nusselt number. These findings will later help during case study in numerical solutions for variable parameters related to heat transfer and pressure drop to achieve the best simulation possible to improve the efficiency of the SHE. Also, these journals discuss the causes for gasket leakages in different technologies from initial deformation and straining to cracks and tear. Based on this knowledge, it will help to arise solution for the gasket leakages in SHE.

Based on the industrial training experience and the review journals, it is noted that the problems faced by the gaskets leakages in SHE and the pressure imbalance within has not been researched before, therefore based on the knowledge achieved from the journals, parameters used to investigated the efficiency of SHE will undergo case study to analyze the problem. Therefore, the objective of this project by the end of the year is to identify, analyze and evaluate the factors causing the leakage of the fluids using computational fluid dynamics (CFD). The simulated solutions will be interpreted based on the graphs and plots achieved to analyze the balance of pressure in the SHE and to show the reduction in leakage during heat transfer by comparison of current results from company with that of the achieved simulated results. After that, propose engineering solutions based on the input parameters and fluid flow properties to increase the efficiency of the gasket to overcome the above-mentioned problems.

METHODOLOGY

In this paper, CFD simulations is the main tool used to achieve the improvement of the SHE efficiency. The simulations are designed through the ANSYS fluent software. The aim is to attain the effect of some investigated parameters by simulation to generate the results, analyse them to find the optimal values of these parameters which result in improving the SHE efficiency.

The research methodology commences with 3D geometrical drawings and meshing of the SHE. It then progresses to applying boundary conditions whilst designing and running the CDF simulations. Thereafter, results are obtained and analysed from which some conclusions are drawn. The research methodology has been constructed based on the industrial experience gained at a company whereby part of the training included learning about an existing and working SHE, its process, the problems faced and how the company tackles these problems to increase the efficiency of the SHE. The results obtained through the project will then be compared to the input data utilised by the company to conclude which method gives the best efficiency.

The research aims to investigate the effect of pressure balance within the SHE based on certain flow parameters such as inlet and outlet pressure, mass flow rate, fluid properties, etc. This investigation, in return, will also establish the negligible factors which even if changed will have no effect on the results. Based on reviewed journals and research conducted, there are a number of standard governing equations, such as continuity equation, energy equation, and Naiver-Stoke equations. These are considered as the backbone of the numerical software. Due to the limitations of space of this paper, these are not shown or discussed, hence the generated results will be directly used for the purpose of the analysis and attaining the objectives of the current research.

Mathematical Modelling

For this project, the design equations for the calculation of SHE are modelled by considering a differential cross section whereby two of the common approaches are used to analyse the heat transfer process; LMTD approach and $\varepsilon - NTU$ approach [18]. However, for this project, the equations shown will not be used to address the efficiency of the SHE as the main focus will be the CFD simulations.

Equations (1) and (2) show the heat loss and gain in the cold and hot fluids which can be used to calculate the effectiveness, ε as shown later in Eq. (4) after calculating the maximum heat transfer shown using Eq. (3). It is seen from Eq. (5) that the effectiveness of any heat exchanger can be expressed as a function of its number of transfer units and capacity ratio.

The heat loss, heat gain, maximum heat transfer, and effectiveness are expressed using Eqs. (1) to (4) respectively.

$$Q_h = \dot{m}_h C_{p.h}\left(T_{h_i} - T_{h_o}\right) \tag{1}$$

$$Q_c = \dot{m}_h C_{p.c}\left(T_{c_i} - T_{c_o}\right) \tag{2}$$

$$Q_{max} = C_{min}\left(T_{h_i} - T_{c_i}\right) \tag{3}$$

$$\varepsilon = \frac{Q}{Q_{max}} = \frac{C_{p.h}\left(T_{h_i} - T_{h_o}\right)}{C_{min}\left(T_{h_i} - T_{c_i}\right)} = \frac{C_{p.c}\left(T_{c_i} - T_{c_o}\right)}{C_{min}\left(T_{h_i} - T_{c_i}\right)} \tag{4}$$

In general, the effectiveness of any heat exchanger can be defined as:

$$\varepsilon = f\left(NTU, \frac{C_{min}}{C_{max}}\right) \tag{5}$$

where, *NTU* is the number of heat transfer unit, expressed as

$$NTU = \frac{UA}{C_{min}} \tag{6}$$

The other approach for the heat exchanger design parameter calculations is using logarithmic mean temperature difference (LMTD) equation [19]. This equation deals with the temperature inlet and outlet for both cold and hot fluids as shown in Eq. (7). The LMTD approach helps to identify the heat flux in the heat exchanger system as shown in Eq. (8) relating to the heat transfer effectiveness and exchange area.

$$LMTD = \frac{\left(T_{h_o} - T_{c_i}\right) - \left(T_{h_i} - T_{c_o}\right)}{\ln \frac{\left(T_{h_o} - T_{c_i}\right)}{\left(T_{h_i} - T_{c_o}\right)}} \tag{7}$$

$$Q = U \times Ar \times LMTD \tag{8}$$

Geometry and Meshing

At first, the 3D-model for the SHE is drawn in SolidWorks, whereby the fluid flow is later constructed using the Boolean method. Once the model for fluid flow is established, the model of the SHE is put up to mesh to create finer finite elements to allow better simulation results [20]. The meshing will be chosen based on type, inflation element quality, orthogonal quality, skewness and number of elements and nodes. The model is undergone different meshing sizes to establish the right size of face sizing method using grid independence test. Based on Figure 5, the grid independence test shows that the meshing size of 15 mm is sufficient to be used for the simulations. Once the meshing is set to a certain size, the meshing is updated and the surface boundaries are named based on face selection for inlets, outlets and wall. The model of SHE is transferred to ANSYS fluent to add the boundary conditions and research data to be collected.

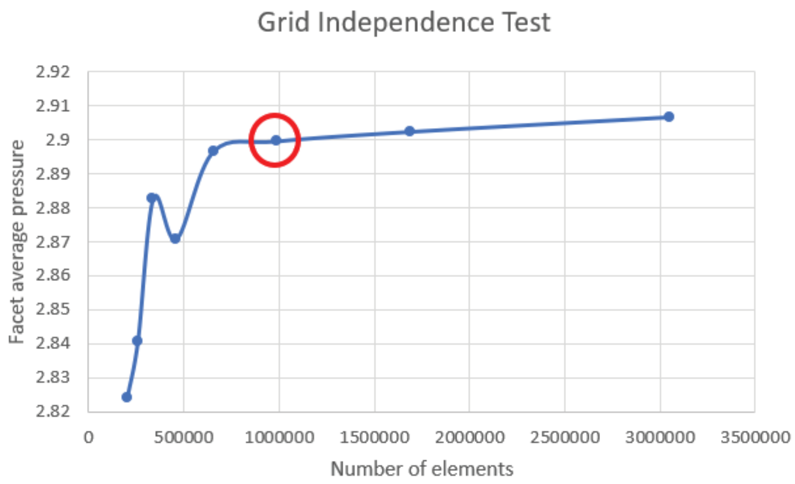


FIGURE 5. Grid Independence Test

Boundary Conditions Setup and Solution

The boundary conditions are set in ANSYS fluent to allow the related variables to be changed for CFD simulation. The fluid properties and specific viscous flow model will be set with changing variable of parameters to analyse the SHE with the same flow model. For the fluid properties, the materials are edited to create two new fluids; CPO and RBDPO. These fluids are created by inserting the properties such as density, kinematic viscosity, heat capacity and conductivity that were attained from the existing real SHE. The variables input will be the inlet and outlet velocity and pressure for both; cold and hot fluids. The iterations are set to run for 5000 iterations to let the simulation converge. The parameters will be changed in such a way that all will be independent of another variable. This way, the simulation is conducted and updated to achieve the required data results showing the effect of change in the investigated parameter of the SHE.

Results Compilation and Comparison with Current Company's Data

The obtained results data from simulation for pressure drop is compared with the existing real data to analyse the differences and explain the changes made. The aim of this approach is to achieve the optimum results from the simulated solution to show the improvement in overall efficiency based on the performance and pressure drop of the SHE. After comparison, the research conclusions are drawn to finalize the design.

RESULTS AND DISCUSSION

The numerical simulation of SHE is conducted using ANSYS fluent for the operational conditions achieved from the company for the two working fluids. Cold fluid is Crude palm oil (CPO) and hot fluid is Refined bleached deodorized palm oil (RBDPO). Also, the parameters are changed to understand the effect of each change in the investigated parameters to the overall efficiency of the SHE. This is done to reduce the pressure losses in the SHE during process. The parameter of focus is pressure drop within the SHE for the cold and hot fluids.

Table 1 shows the input operational conditions range achieved from the company used for ANSYS numerical simulations. From the range of values achieved from the company, the input operational conditions are the average values of each data. Also, each section is numbered according to the sections shown in graph to show the average value of pressure at each section.

Figure 6 shows the pressure contours for the company's original input data, for the whole system of SHE the maximum pressure exerted is of 540 kPa while the minimum pressure exerted is 476.8 kPa. From the pressure values exerted, it is seen that the maximum pressure is exerted at the hot inlet section while the minimum pressure exerted is at the cold outlet. In addition, for the cold working fluid, the pressure exerted is consistent of around 489.4 kPa throughout the whole SHE apart from the inlet and outlet sections.

TABLE 1. Input operational conditions achieved from the company.

No.	Section	Pressure (kPa)	Temperature (K)	Velocity (m/s)
1	Cold Inlet	450 – 510	358.15 – 418.15	2 – 5
2	Hot Inlet	490 – 550	493.15 – 553.15	2 – 5
3	Cold Outlet	450 – 510	468.15 – 528.15	2 – 5
4	Hot Outlet	490 – 550	388.15 – 448.15	2 – 5

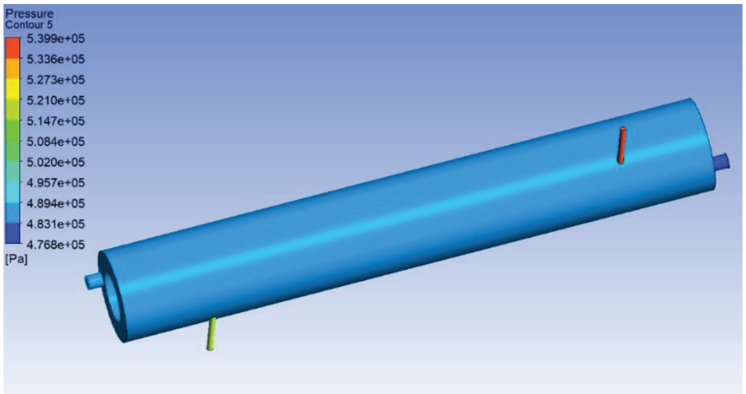


FIGURE 6. Overall system pressure contour

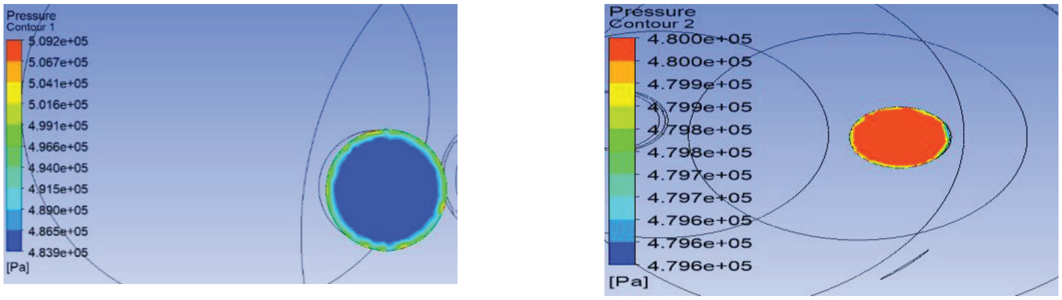
Figure 7 shows pressure contour for each individual section. It is seen that the hot fluid has a steady flow of pressure for both inlet and outlet. The pressure is steady at 539.9 kPa for hot inlet and 520 kPa for hot outlet. Although the pressure for the hot inlet has a constant value, the color change at the surface shows that the hot inlet section has a vigorous reaction. However, for the cold fluid inlet and outlet, there is a sudden increment of pressure from the center of the section to the boundary surface. The cold inlet has a maximum pressure of 509.2 kPa with a minimum of 483.9 kPa while the cold outlet has a maximum pressure of 480 kPa with a minimum of 479.6 kPa.

Also, from the individual sections of inlets and outlets in Figure 7, it is clearly seen that the maximum pressure of both inlets is higher than the maximum pressure at the outlets. Nevertheless, the minimum pressure for both the inlets is also higher than the minimum pressure of the outlets. This clearly shows that the average pressure exerted in the SHE is higher at the inlets than the outlets. In addition, the pressure exerted at the inlets for both the cold and hot fluid are higher than the input operational conditions while the outlets have the same value as the input operational conditions. For the cold inlet, the actual pressure exerted from the operational condition is increased by 40 kPa whereas for the hot inlet, it is increased by 20 kPa from the operational conditions.

Based on the input operational conditions achieved from the company, the actual input data from Table 1 and the resulted original data achieved from the CFD simulations as shown in Table 2 can be compared to validate the design of the SHE used. For the colder region, the inlet and outlet section of the actual data is 480 kPa while for the simulated data it is 479.988 kPa and 479.99 kPa respectively. For the hotter region, the inlet and outlet section of the actual data is 520 kPa while for the simulated data it is 518.22 kPa and 518.426 kPa respectively. It is shown that the values do not vary by a lot and that they have negligible change. This proves that the design is valid to be used for simulations to achieve the required results. The solutions for each parameter was obtained from ANSYS fluent and using CFD-post the following average results were calculated as shown in Table 2. The table also shows the effect of variable pressure of each section on the overall SHE.

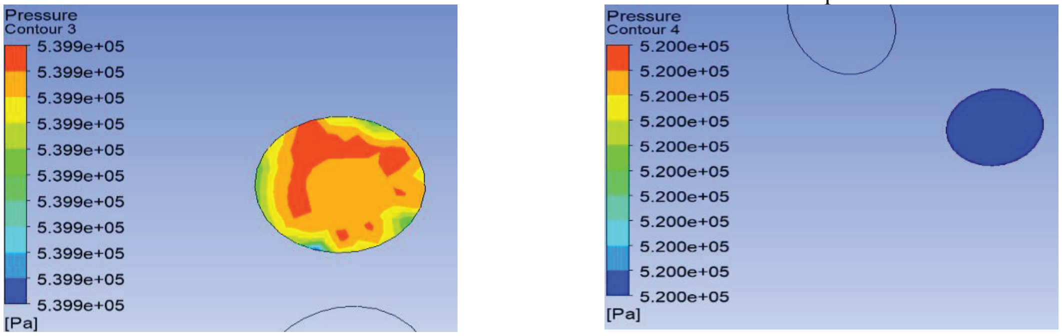
Based on the average results shown in Table 2, it is seen that however the values of pressure for all sections are kept constant while changing one of the section's pressure, the average resulted pressure changes. This shows that the pressure causes an effect to the performance of the SHE. It is seen based on the Table 2 and Figure 8 that as the cold inlet pressure increased from 480 kPa to 510 kPa, the average resulted pressure for both hot inlet and outlet also slightly increased causing no change in average pressure for the cold outlet. This is due to the heat transferring process of convection. Since the inlet for cold fluid has high pressure exerted, due to convection, the hot fluid will also have high pressure exerted. This change is also experienced when the cold outlet is increased, the average resulted hot inlet and outlet also augments. Also, the increase in pressure for hot inlet causes just the resulted hot outlet to increase as well leaving the rest of the sections not varied.

In addition, based on Figure 9 and Table 2, the cold inlet and outlet reduction in pressure causes the average resulted pressure in hot inlet and outlet to change. This means that the contact region between the hot and cold fluid is thin to let the heat escape from the cold section to go towards the hot section. Also, based on FIGURE 9, the hot inlet and outlet are related such that any sudden decrease in pressure input causes the corresponding section to have a decrease in resulted pressure.



Cold inlet pressure contour

Cold outlet pressure contour



Hot inlet pressure contour

Hot outlet pressure contour

FIGURE 7. Individual section pressure contours

TABLE 2. Averaged results for pressure.

	Sections			
Change made in pressure	Cold inlet (kPa)	Hot inlet (kPa)	Cold outlet (kPa)	Hot outlet (kPa)
Original company conditions	479.988	518.227	479.990	518.426
Increase 30 kPa in cold inlet	509.998	519.032	479.991	519.127
Increase 30 kPa in hot inlet	479.989	541.051	479.990	520.000
Increase 30 kPa in cold outlet	479.991	519.016	509.998	519.142
Increase 30 kPa in hot outlet	479.987	520.000	479.989	542.006
Decrease 30 kPa in cold inlet	449.998	517.023	479.989	517.265
Decrease 30 kPa in hot inlet	479.996	490.000	479.996	513.378
Decrease 30 kPa in cold outlet	479.992	517.620	449.984	517.877
Decrease 30 kPa in hot outlet	479.995	512.759	479.996	490.000

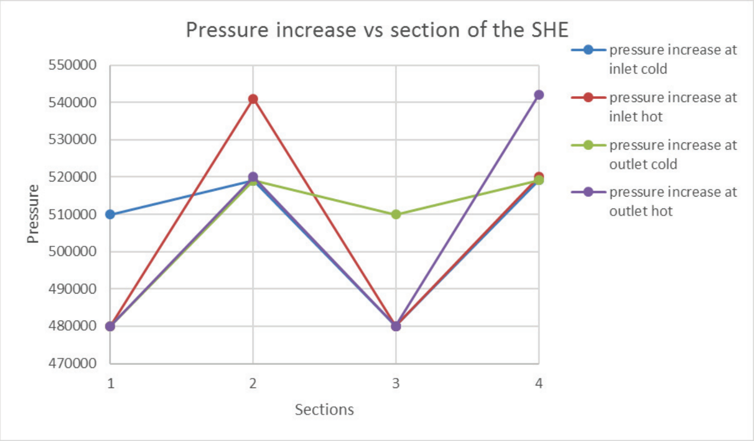


FIGURE 8. Pressure increase in sections

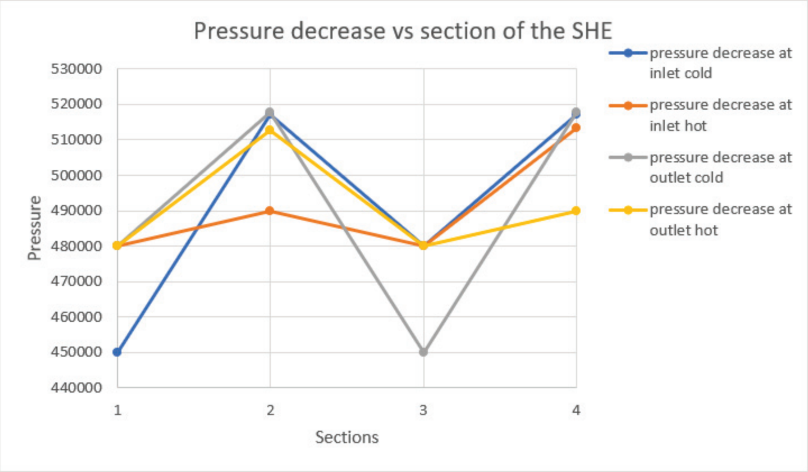


FIGURE 9. Pressure decrease in sections

The average results shown in Table 2 are used to calculate the SHE efficiency by using pressure drop to allow optimum input data to be achieved. Based on knowledge from industrial training, the pressure drop should be lesser to have optimum results and efficiency on the working SHE. This will be done by comparing the input operational conditions achieved from the company with the changed values in parameters. The pressure drop is calculated by subtracting the inlets and outlets of each hot and cold fluid to achieve the difference. The following equations (9) will be used to achieve the pressure drop for the cold and hot region of the SHE.

$$Pressure\ drop_{cold/hot} = P_{outlet} - P_{inlet} \tag{9}$$

Therefore, the pressure drop for cold and hot fluid is calculated as shown below in the Table 3. Table 3 shows the pressure drop in every change of cold and hot fluid pressure. From the table there is no pressure drop at the cold region when the hot inlet increases by 30 kPa and for every change in the pressure for hot inlet and outlet, the pressure drop at the cold region is little. In addition, the pressure drop is lesser for the hot region when there is a change in the cold inlet and outlet. This means that the pressure change in any section is correlated to the other sections in such a way that any change in pressure in a specific section will cause an effect to the overall performance.

Based on the results obtained in Table 3, to improve the overall efficiency of the SHE or to reduce pressure drop or imbalance, the input pressure for the cold inlets and outlets should be increased to reduce the hot fluid pressure drop while decreasing the input pressure for hot inlets and outlets to reduce the cold fluid pressure drop. Due to this change, the leakages caused due to pressure imbalance will be reduced to minimal rate therefore; the overall efficiency of the SHE will improve.

TABLE 3. Pressure drop in hot and cold region of the SHE.

Change made in pressure	$Pressure\ drop_{cold}$ (Pa)	$Pressure\ drop_{hot}$ (Pa)
Original company conditions	2	199
Increase 30 kPa in cold inlet	-30007	95
Increase 30 kPa in hot inlet	1	-21051
Increase 30 kPa in cold outlet	30007	126
Increase 30 kPa in hot outlet	2	22006
Decrease 30 kPa in cold inlet	29991	242
Decrease 30 kPa in hot inlet	0	23378
Decrease 30 kPa in cold outlet	-30008	257
Decrease 30 kPa in hot outlet	1	-22759

After achieving the resulted pressure values, a final simulation is conducted with the input cold pressure of 510 kPa and hot pressure of 490 kPa. The following Table 4 shows the resulted values of pressure achieved from the simulation to give the best possible solution. Based on Table 4, the pressure drop for the cold region is 3 Pa while for the hot region is calculated to be 1 Pa.

TABLE 4. Average results for better efficiency.

Section	Average resulted pressure (kPa)
Inlet cold	501.840
Outlet cold	501.843
Inlet hot	490.000
Outlet hot	490.001

The percentage change in pressure drop to show increase in efficiency in the new simulated results is done by using the following equation (10) that shows the change in pressure drop over the actual pressure drop used by the company. Based on the calculations, the pressure drop reduces in the hotter region by 99.4 % while increases in the colder region by 50 % compared to the current data of the company

$$\%\ change\ in\ Pd_{cold/hot} = \frac{Pd_{old} - Pd_{new}}{Pd_{old}} \times 100\% \tag{10}$$

$$\%\ change\ in\ Pd_{hot} = \frac{199\ Pa - 1\ \mathrm{Pa}}{199\ Pa} \times 100\% = 99.4\ \%$$

$$\%\ change\ in\ Pd_{cold} = \frac{2\ Pa - 3\ \mathrm{Pa}}{2\ Pa} \times 100\% = -50\ \%$$

CONCLUSION

In this paper, the SHE was simulated using CFD fluent to acquire the effect of pressure based on different sections. The results were obtained and accumulated to calculate the pressure drop of cold and hot region of the SHE. Based on the analysis of results, it was concluded that the pressure change affects the overall performance of the SHE and that the pressure drop can be minimized if the cold inlet and outlet pressure input increases while having low pressure input for both the hot sections.

Therefore, based on the range of operational conditions achieved from the company, a proposed simulation is done whereby the inlet and outlet cold input pressure is set to 510 kPa while the hot inlet and outlet input pressure is set to 490 kPa. From the proposed simulated results, it can be concluded that the pressure drop of the hot fluid becomes completely 1 Pa showing lesser pressure loss in the hot region of the SHE compared to actual company's results giving 199 Pa pressure drop. However, for the cold region, the pressure drop becomes 3 Pa which is better than the other simulated results but not better than the current company's resulted pressure drop of 2 Pa. All in all, the efficiency of pressure loss in the hotter region increases by 99.4 % while reduces in the colder region by 50 %. Based on the results, the company should focus on the physical design of the colder region to further enhance the efficiency of the SHE using the simulated input data for cold and hot inlet and outlet.

List of Symbols

Ar = Area of focus, m^2
C_{max} = Maximum specific heat capacity, J/kg
C_{min} = Minimum specific heat capacity, J/kg
$C_{p.h}$ = Specific heat capacity for hot fluid, J/kg
$C_{p.c}$ = Specific heat capacity for cold fluid, J/kg
$LMTD$ = Logarithmic mean temperature difference, ℃
NTU = Number of transfer units
Q = Heat transfer, J
T_{c_i} = Temperature of cold inlet, ℃
T_{c_o} = Temperature of cold outlet, ℃
T_{h_i} = Temperature of hot inlet, ℃
T_{h_o} = Temperature of hot outlet, ℃
U = Overall heat transfer coefficient
ε = Effectiveness of the heat exchanger

REFERENCES

1. Spiral Heat Exchangers - Vacuum Pump Manufacturers & Suppliers in Mumbai, India | Refrigerator Compressor. (2019). Retrieved from https://www.erplindia.com/spiral-heat-exchangers/.
2. J. Khorshidi and S. Heidari, Advances In Chemical Engineering And Science 06, (2016).
3. D. Nguyen and J. San, Heat Transfer Engineering 37, (2016).
4. Q. Tang, G. Chen, Z. Yang, J. Shen and M. Gong, Science China Technological Sciences 61, (2017).
5. J Bhavsar, V Matawala & S Dixit, Design and experimental analysis of spiral tube heat exchanger. Mechanical and Production Engineering, v1(1), (2013).
6. "Full Face Flange Gasket for Ductile Iron Flanged Piping", Maloney Technical Products, (2019). [Online]. Available: https://www.maloneytech.com/full-face-flange/.
7. H. FaJiang, C. WeiWu and Y. Ping, Energy Procedia 17, (2012).
8. D. Nguyen and J. San, Applied Thermal Engineering 76, (2015).
9. Z. D. Fan et al., Eng. Fail. Anal., vol. 96, (2018).
10. A. Mostafazade Abolmaali and H. Afshin, Int. J. Therm. Sci., vol. 139, (2018).
11. S. Li, Y. Jiang, W. Cai, H. Zhang, and F. Li, vol. 129, (2019).
12. F. Gao, S. Shoai Naini, J. Wagner, and R. Miller, Int. J. Refrig., vol. 73, (2016).
13. Feng at el., Nonmetallic Gaskets *," vol. 15, (2005).
14. X. L. Yang, Z. G. Yang, and Q. Ding, Eng. Fail. Anal., vol. 79, (2017).
15. Al-Kayiem, H., Ekhwan, A. B. & Muhi, L. N., J. Eng. Sci. Technol., vol. 12 (2017).
16. M. Abid, D. H. Nash, S. Javed, and H. A. Wajid, Int. J. Press, vol. 168, (2018).
17. Tahrour et al., J. Eng. Sci. Technol., vol. 10 (2015).
18. D. Nikhil, S. Shriramshastri, C. & A Sawant, International Journal of Trend in Research and Development 4, (2017).
19. P. Guha and V. Unde, Int. J. Eng. Res., vol. 3, (2015).
20. R. Durand, B. Pantoja-Rosero, & V. Oliveira, 158, (2019).