Design and Analysis on Performance of Chiller Barrel

Choy Hau Yan, Ang Ruo Fan
Tunku Abdul Rahman University College, Department of Mechanical Engineering,
Faculty of Engineering and Technology, Kuala Lumpur, Malaysia
Email: choyhy@tarc.edu.my

***Abstract*—Chiller barrel is considered as a shell-and-tube heat exchanger and generally applied in a water-cooled chiller. It is important to improve the performance of a chiller barrel so that the usage of electrical energy can be reduced while the quality of a product can be increased. In this project, the performance of a chiller barrel will be simulated and designing of chiller barrel being done based on the size of the baffle to improve the performance. Besides that, the flow characteristics of the fluid in the chiller barrel will be analyzed and compared by using a CFD simulation tool. Outlet temperature and pressure will be simulated while the rate of heat transfer, logarithmic mean temperature difference and overall heat transfer coefficient will be calculated. All value will be analyzed and selection the most optimum design which is the baffle with 30% of cut percentage based on the consideration of overall heat transfer coefficient and pressure gradient.**

***Index Terms*—Chiller barrel, Shell and tube heat exchanger, Heat transfer, SolidWorks flow simulation**

I. Introduction

In order to remove excess heat generated by different types of process, a chiller is needed. The chiller can produce chilled water to a process by removing the heat energy in the water that coming from a process. The chiller can be classified into two major types which are an air-cooled chiller and water-cooled chiller. The method of a water-cooled chiller to produce chilled water is using a shell-and-tube heat exchanger, which generally called as chiller barrel, to absorb the heat energy of the water through heat transfer into the refrigerant gas. [1] In this project, the performance of a chiller barrel will be analyzed. By increasing the performance, which also known as the effectiveness of heat exchanger, in a chiller system process, the cost of the electrical energy will able to be reduced as well. This project includes to simulate the performance of a chiller barrel and redesign the size of baffle and simulation of fluid flow chiller barrel through CFD simulation. [2] Designation of different types of chiller barrel will identify the temperature distribution in the chiller barrel.[3] By reducing the production cost by lowering the electrical energy for the chiller system, it also able to improve the cooling process by increasing the quality of a coolant.

Manuscript received April 5, 2019; revised February 19, 2020.

II. Literature Review

The heat transfer coefficient and the pressure drop are dependent on the baffle inclination. This is investigated by numerically modeling a small heat exchanger, the flow and temperature fields inside the shell are resolved using CFD package. A set of CFD simulations is performed for a single shell and single tube pass heat exchanger with a variable number of baffles and turbulent flow. The results are observed to be sensitive to the turbulence model selection. The best turbulence model among the ones considered is determined by comparing the CFD results of heat transfer coefficient, outlet temperature and pressure drop with the Bell–Delaware method results. [4]

In order to achieve the heat transfer enhancement, the configuration of a shell-and-tube heat exchanger was improved by installing of sealers in the shell side. The gaps between the baffle plates and shell are blocked by the sealers, which effectively reduces the short circuit flow in the shell side. The experiment results show that the shell side heat transfer coefficient of heat exchanger increased by 18.2% to 25.5%. The overall coefficient of heat transfer increased by 15.6% to 19.7% and pressure losses increased by 44.6% - 48.8% with the sealer installation, but the increment of required pump power can be neglected compared with the increment of heat flux. [5]

By referring the experimental investigation of shell and tube heat exchanger with a different type of baffles done by [6], the shell and tube heat exchanger with flower baffles and segmental baffles are designed, fabricated and tested. The heat exchanger with flower baffle gives more efficient overall performance up to 25% to 32% than the segmental baffles heat exchanger. Moreover, a reduction up to 28% of pressure drop is obtained in flower baffles by comparing with the segmental baffles. [6]

III. Proposed Methodology

According to [7], computer-aided design is needed to examine all the design parameters for a given heat load to get the economic design. In this project, SolidWorks is used to design and construct the models of the chiller barrel. SolidWorks flow simulation used to carry out the simulation of the chiller barrel which includes the heat transfer and the fluid flow characteristics.

doi: 10.18178/ijmerr.9.4.587-591

A. Construction of Models

The chiller barrel is assembled by three main parts of a heat exchanger which are shell, tubes and baffle as in Fig. 1. The hot fluid that flows through the tubes and the shell and also flows through the outer surface of the tubes. Fig. 2 shows total of 5 different cutting size are modelled which are (a) 24%, (b) 30%, (c) 36%, (d) 42% and (e) 48% of cut percentage. Table I showing the geometry of model for the design. [4]

TABLE I. GEOMETRY OF MODEL

Specification	Dimension
Length of tubes and shell	600mm
Shell inner diameter	90mm
Shell outer diameter	100mm
Tubes inner diameter	17mm
Tubes outer diameter	20mm
Number of tubes	7
Baffle inclination	0°
Baffle spacing	86mm
Baffle thickness	3mm
Number of baffles	6
Baffles cut percentages	24%, 30%, 36%, 42%, 48%

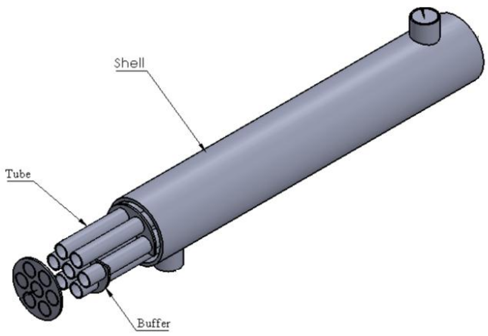

Figure 1. Geometry of the model

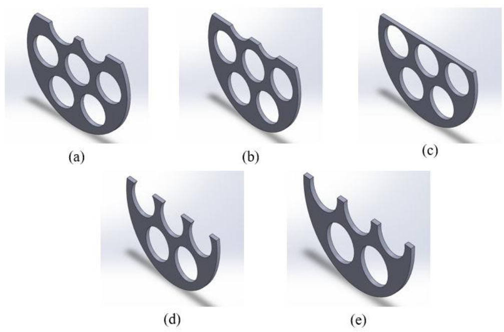

Figure 2. Different design of baffle with different cut percentage.

B. Meshing

Fig. 3 shows that mesh generated based on the global element size for the model taking into consideration of its volume, surface area and other geometric details. For the model, a combination of structured Cartesian mesh and non-structured body-fitted mesh that near the wall are generated with the total number of 324524 modes.

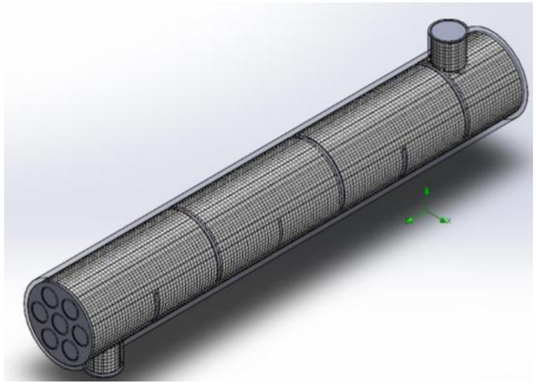

Figure 3. Meshing of the model

C. Boundary Conditions

Fig. 4 shows the boundary condition of the model. Water is chosen as the fluid that used in simulation for both hot and cold condition with turbulence flow while the other initial values such as velocity of flow and temperature of fluids are shown in Table II. The inlet velocity profile is assumed to be uniform and no-slip condition is assigned to all surfaces. [8]

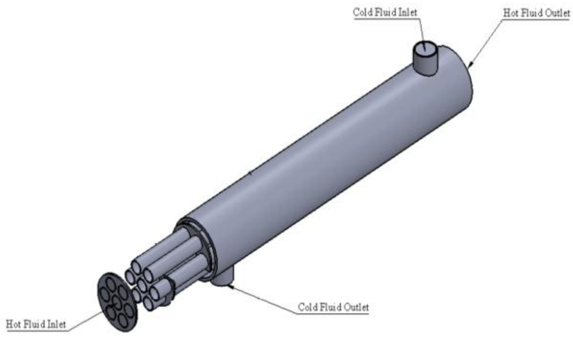

Figure 4. Boundary condition of the model

TABLE II. BOUNDARY CONDITIONS OF THE MODEL

	Inlet velocity (m/s)	Inlet temperature (K)
Hot fluid	1.594	340
Cold fluid	0.0787	300

IV. RESULTS AND DISCUSSIONS

The computational fluid dynamics analysis was done for the five different sizes of the baffles of chiller barrel where the hot water flows through the tube while the cold water flow through the shell. The comparison between the performances of those designs is done also.

A. Validation

Fig. 5 shows the results sets for simulation with fluid flow rate of 0.5 kg/s with 36% baffle cut are validated with the results from [4]. Temperatures for shell side outlet are 332K compare with 331.56K from simulation while for temperatures of tube side outlet are 339.2K and 326.02K (simulation). It is found that the outlet temperatures are matching with the literature results with a deviation of 0.13% to 3.89%. Fig. 6 shows simulation result with 36% baffle cut is 65960.4W compare with the result from the study is 66848W, the deviation between the two results only 1.33%. Therefore, simulation results can be said as validated.

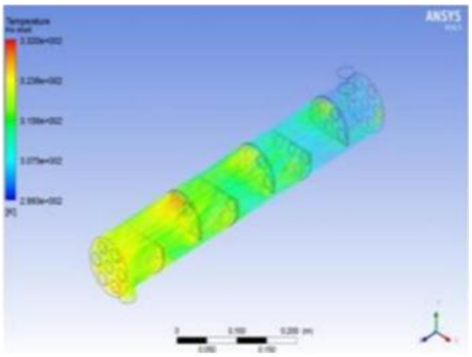

Figure 5. Temperature variation of literature result.

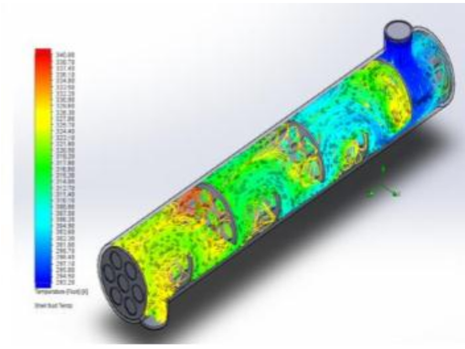

Figure 6. Temperature variation of Solidworks flow simulation

B. Temperature Variations

By setting flow rate of 0.5 kg/s, simulation for temperature variation with baffle cut 24%, 30%, 36%, 42% and 48% being done to find the output temperature for shell fluid and tube fluid. Fig. 7 to Fig. 11 show temperature variation results for different baffle cut. Simulation results for outlet temperature being shown in Table III.

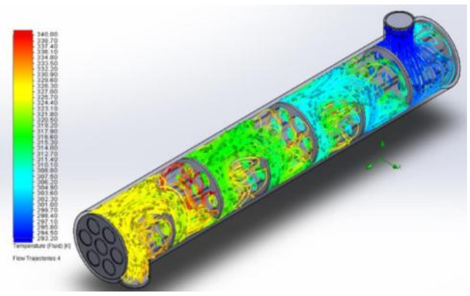

Figure 7. Temperature variation in the model with 24% baffle cut

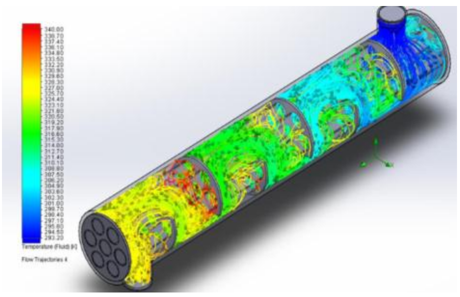

Figure 8. Temperature variation in the model with 30% baffle cut

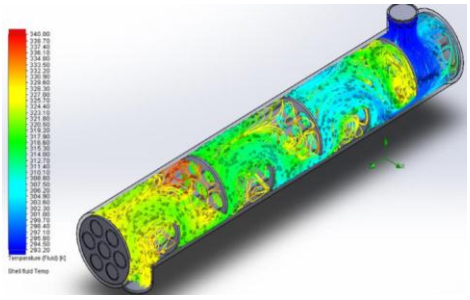

Figure 9. Temperature variation in the model with 36% baffle cut

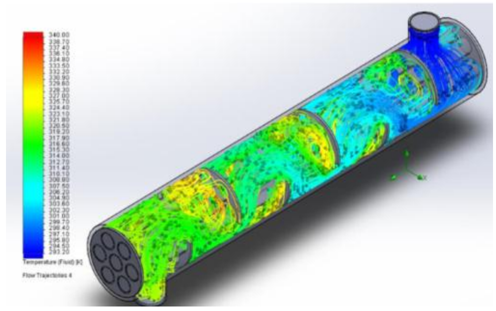

Figure 10. Temperature variation in the model with 42% baffle cut

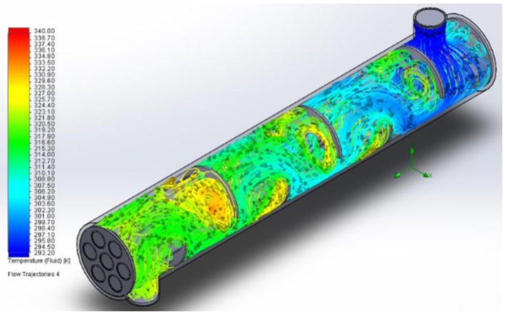

Figure 11. Temperature variation of the model with 48% baffle cut

TABLE III. TEMPERATURE VARIATION OF DIFFERENT MODELS

Baffle Cut (%)	Outlet Temperature (K)	
	Shell fluid	Tube fluid
24	334.68	325.42
30	332.58	326.02
36	331.56	326.02
42	328.66	323.23
48	328.42	323.06

C. Rate of Heat Transfer

The values of the rate of heat transfer are calculated by using equations (1) and equation (2).

$$\dot{Q} = \dot{m} \text{ x } C_f \text{ x } \Delta T \quad (1)$$

$$\dot{m} = \rho \text{ x } v \text{ x } A \quad (2)$$

$\dot{Q}$ = mass flow rate
C_f = specific heat capacity at 300K
ΔT= temperature different
ρ = density of water
A = cross-sectional area
By referring cut percentage of baffle 24%, from (1),
$\dot{m}$ = 997.01 x 0.0787 x π x 0.045^2
= 0.5 kg/s

C_f = 4181*J/kg K*, water temperature=300K, ΔT=34.68 K. Therefore, by using (2),

$\dot{Q}$ = 0.5 x 4181 x 31.54 = 72418.2 W.

Remaining value of rate of heat transfer for different size of baffle are calculated with same steps as in Table IV. Value for rate of heat transfer will use in calculation for overall heat transfer coefficient.

TABLE IV. RATE OF HEAT TRANSFER OF DIFFERENT MODELS

Baffle cut (%)	Rate of heat transfer, $\dot{Q}$ (W)
24	72481.2
30	68092.2
36	65960.4
42	59899.4
48	59397.8

D. *Logarithmic Mean Temperature Difference (LMTD)*

LMTD helps to analyze the effective heat transfer in a heat exchanger. More the value of LMTD more will be the chances for transfer of heat between two fluids. [9] Equation (3) used to calculate the values of LMTD for different sizes of baffle.

$$(\Delta T)_{in} = (\Delta T_1 - \Delta T_2) / \text{In} (\Delta T_2/\Delta T_1) \quad (3)$$

$(\Delta T)_{in}$ = Logarithmic Mean Temperature Difference

ΔT_1 = Temperature different of hot inlet and cold outlet

ΔT_2 = Temperature different of hot outlet and cold inlet

From (3), calculation for baffle size with 24% cut,

$$(\Delta T)_{in} = \frac{[(340-334.68)-(325.42-300)]}{\text{In}[(340-334.68)/(325.42-300)]}$$

$(\Delta T)_{in}$ = 12.85 *K*

For other baffle cut calculation as shown in Table V.

TABLE V. LMTD FOR DIFFERENT SIZE OF BAFFLE.

Baffle cut (%)	LMTD, $(\Delta T)_{in}$ (K)
24	12.85
30	14.82
36	15.61
42	16.58
48	16.67

E. *Overall Heat Transfer Coefficient*

Equation (4) called overall heat transfer coefficient, is defined largely by the system, and in many cases it proves to be insensitive to the operating conditions of the system. [10]

$$U = \dot{Q} / [A \times ((\Delta T)_{in})] \quad (4)$$

U = Overall heat transfer coefficient

$\dot{Q}$ = Rate of heat transfer

A = surface area

$(\Delta T)_{in}$ = Logarithmic Mean Temperature Different

From (4), calculation for baffle size with 24% cut,

$\dot{Q}$ = 72418.2W, $(\Delta T)_{in}$ = 12.85K

A = 2 x π x r x *l*

A = 7 x (2 x π x 0.01 x 0.6) = 0.25 m^2

Therefore,

$$U = \frac{72418.2}{0.26 \times 12.85} = 17.81 kW/m^2 K$$

For other baffle cut calculation as shown in Table VI.

TABLE VI. OVERALL HEAT TRANSFER COEFFICIENT OF THE MODEL

Baffle cut (%)	Overall heat transfer coefficient , U ((KW / m^2 K)
24	17.81
30	16.73
36	16.21
42	14.72
48	14.60

F. *Pressure Variation*

The pressure variations of different models were simulated and the values of pressure drop were tabulated in Table VII.

TABLE VII. PRESSURE DROP OF THE MODEL

Baffle cut (%)	Pressure variation (Pa)
24	100.23
30	71.18
36	41.03
42	27.75
48	27.41

G. *Discussions*

All the results from simulations are tabulated such as outlet temperature of both fluids and pressure variations. By using the outlet temperatures, the values of rate of heat transfer, LMTD and overall heat transfer coefficient are calculated. Outlet temperatures of the shell fluid slightly decreased as the cut percentage of the baffle increased but not much change for tube fluid. By making 36% baffle cut as the benchmark, Rate of heat transfer improve 3.23% for 30% baffle cut and 9.89% for 24% baffle cut. For baffle cut 48%, value reduce 9.95%.

From observation on LMTD, not much changes for 42% and 48% baffle cut but with reasonable value decrease for 24% and 30% baffle cut compare with 36% baffle cut. The overall heat transfer coefficient of shell fluid increase 9.87% as the baffle cut increased from 36% to 24%. There is 3.2% improve for 30% baffle cut. Pressure variation drop is 114.28% for 24% and 73.48% for 30% baffle cut but value is within an acceptable level.

Incremental of the pressure drop is caused by the increase of the cross-sectional surface area of the shell. As the cut percentage of baffle increased, size will decrease, the velocity of the flow decrease too. According to Bernoulli's Equation, as the velocity reduced, the pressure will increase. Turbulence is measured in Reynolds Numbers and it is marked by higher Reynolds Numbers and laminar flow is lower Reynolds Numbers. While increasing Reynolds Numbers positively influence thermal transfer, the pressure drop will increase through the heat exchanger, whereas the smooth flow through the heat exchanger associated with laminar flow produces a lower pressure drop, but also lower thermal efficiency.

V. CONCLUSION

In a nutshell, this study showed that with lesser baffle cut percentage, there is an improvement in rate of heat transfer and overall heat transfer coefficient. Nevertheless, as the pressure drop increased, it means that energy loss of the fluid increased too. Therefore, baffle size 30% cut percentage show optimum results in both the heat transfer coefficient and the pressure drop.

There are few limitations of using a flow simulation to determine the rate of heat transfer in a chiller barrel. First, flow simulation unable to handle phase changing of a fluid. The main fluid that passes through a chiller barrel is called as refrigerants which will occur phase changing from liquid-to-gas or gas-to-liquid. Therefore, if the fluid selected in the project is refrigerant, the results may not so accurate. Next, the flow simulation is unable to simulate solid or liquid suspensions in a fluid where the suspensions can influence the pattern or parameters of flow. For example, it cannot support the gas bubble trapped in a liquid stream that will affect the pressure drops and rate of heat transfer in a chiller barrel. In other word, it can only simulate the operation of a chiller barrel in an ideal case.

CONFLICT OF INTEREST

The authors declare no conflict of interest.

AUTHOR CONTRIBUTIONS

Mr Choy Hau Yan contributes in identified problem and give guidance on finite element analysis of simulation. Mr. Ang Ruo Fan conducted in literature review, simulation analysis and report writing. Both authors had approved the final version.

ACKNOWLEDGMENT

This research was supported by Tunku Abdul Rahman University College. We are thankful to the staff of the department of mechanical engineering who provide expertise that greatly assisted the research.

REFERENCES

[1] T. D. Eastop and A. McConkey, "Applied thermodynamics for engineering technologists," in *Proc. 5th ed, Dorling Kindersley (India) Pvt. Ltd*, South Asia. 2009

[2] F. P. Incropera, D. P. Dewitt, T. L. Bergman, and A. S. Lavine, "Foundations of heat and mass transfer," 6th ed, John Wiley & Sons. 2011

[3] A. W. Date, "Introduction to computational fluid dynamics," Cambridge University Press, Cambridge. 2009

[4] M. Irshad, M. Kaushar, and G. Rajmohan, "Design and CFD analysis of shell and tube heat exchanger," *International Journal of Engineering Science and Computing*, vol 7, no. 4, April 2017.

[5] S. Wang, "An experimental investigation of heat transfer enhancement for a shell-and-tube heat exchanger," *Applied Thermal Engineering*, vol. 29, no. 11-12, pp. 2433–2438. August 2009.

[6] P. J. Pawar, "Experimental investigation of shell-and-tube heat exchanger with different type of baffles," *International Journal of Current Engineering and Technology*, vol. 5, June 2016.

[7] A. K. Rai and M. S. Mahdi, "A practical approach to design and optimization of single phase liquid to liquid shell and tube heat exchanger," *International Journal Of mechanical Engineering And Technology*, vol. 3, no. 3, Sept- Dec 2012, pp. 378-386.

[8] K. Harika, T. Likhitha, K. Hema, and P. Pavani, "Fabrication of shell and tube heat exchanger using helical baffles based on kern's principle," *International Journal of Current Engineering and Technology*, vol. 7, no. 3, June 2017.

[9] S. Mohanty and S. U. Ahmed, " Computational modelling of counter flow heat exchanger for LMTD analysis," *International Journal fir Research in Applied Science & Engineering Technology*, vol. 6, no. IV, April 2018.

[10] J. H. Lienhard IV and J. H. Lienhard V, "A heat transfer textbook," Phlogiston Press, Cambridge, 4th ed, 2017, pp. 80

[11] A. A. Musilim, A. Nwagwo, and O. K. Uche, "Effect of baffle cut sizes on temperature and pressure drop at various mass flow rate in a shell and tube heat exchanger," *International Journal of Engineering, Science and mathematics*, vol. 6, no. 1, March 2017.

Mr Choy Hau Yan was born at Kedah, Malaysia on 17 January 1978. He obtained his master's degree in Mechanical Engineering at National University of Malaysia, Malaysia in the year of 2002. His major research interest is in the field of finite element analysis, CFD simulation and engineering design. He had extensive working experience as a lecturer at Tunku Abdul Rahman University College, Malaysia for 15 years. He has previous publication regarding finite element analysis.

Mr. Choy is currently the members of Board of Engineer of Malaysia (BEM), a qualified Certified SolidWorks Professional and a member of Technologist Professional

Mr Ang Ruo Fan has just completed his Bachelor Degree in Mechanical Engineering at Tunku Abdul Rahman University College, Malaysia in the year of Ang Ruo Fan. He partially contributed in this research under the supervision of Mr Choy Hau Yan.