

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

Design and Performance Evaluation of Shell and Tube Heat Exchanger for Methylene Di Chloride

Nilima Khare¹, R.R. Arakerimath², Gurunath Gore³

PG Student, Heat Power Engineering, G.H Rasoni College of Engineering and Management, Pune, India¹

Professor, Department of Mechanical, G.H Rasoni College of Engineering and Management, Pune, India²

Inventor Engineer, Consulting Engineers PVT.LTD, India³.

ABSTRACT : Heat exchanger exchanges the thermal energy between two or some times more than two fluids between or it can be used for transfer the heat between a solid surface and flowing fluid. There are some heat exchangers which transfers heat between fluid and solid particulates at different temperatures with maximum heat transfer rate and minimum investment is known as heat exchanger. The shell and tube heat exchanger with straight tubes and double pass is selected to study among different types of heat exchanger. In different industries Heat exchangers has many application .In present study the shell and tube heat exchanger with straight tubes and double pass is under study . Design and performance Evaluation of a Shell and Tube Heat exchanger according to the requirement of sponsored industry. Designing the heat exchanger based on Kern's technique and evaluating the performance by changing the Cold fluid, flowrate and temperature of cold fluid In current chemical industry they want to design the single shell and double tube pass heat exchanger for Methylene Di-chloride. In this methylene Di -chloride is a hot fluid .We analyze the effect of two different type of cold fluids on the performance of shell and tube heat exchanger i.e. water and methanol, by varying the flow rate and temperature The effect of above parameter on heat transfer rate and pressure drop is to be studied.

I. INTRODUCTION

Heat exchanger transfers heat from one fluid to another fluid. Heat exchangers are used in chemical industries and process industries. Heat exchangers that are observed by us in our day to day life are evaporators and condensers used in refrigeration and air conditioning units. Condensers and Boilers are the examples of large industrial heat exchangers. Heat exchangers are termed as per their applications. Heat exchangers used for condensing the fluid are named as condensers. Also heat exchangers used for vaporizing liquids are known as evaporators. [6] Heat exchanger performance is calculated on the basis of minimum area of heat transfer and pressure drop through heat exchanger. A presentation of efficiency is done by computing over all heat transfer coefficient for the heat exchanger. Area required for a definite amount of heat transfer and pressure drop provides an idea about the total cost of a heat exchanger and power requirements. For designing the heat exchanger there is a lot of literature is obtainable since lot of work is done on the heat exchanger previously. For a better performance the heat exchanger design must consider smallest amount of probable pressure drop and heat transfer areas to accomplish the requirements of the heat transfer [9].

II. LITERATURE REVIEW

Aiket al.[7] the paper is based on as the length of the shell and tube heat exchanger decrease then effect of it on the cost, manufacturing time and important factor is space can be reduced.Sandeep K. Patel et.al [3] In this paper the tube length, tube layout Tube pitch ratios, as well as baffle spacing ratio were found to be important design parameters

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

which has a direct effect on pressure drop and causes a conflict between the effect uniformity and total cost. Sunil S. Shinde et al. [4] In this paper baffle inclination angle will provide an optimal performance of heat exchangers. P.S. Gowthaman et.al [2] the pressure drop in segmental baffle is more as compare helical baffle.

Shell and tube type heat exchanger design using kern's technique [1]

Kern's technique is formed on work on heat exchangers which are used in commercial industries.

III. METHODOLOGY REVIEW

In the present study to get the better cooling, we vary the three parameters like temperature of the coolant, flow rate and type of coolant. MDC is the bi product of the chemical process which we want to cool and convert the vapor into liquid. so in this study the in late temperature of MDC is difficult to change because the boiling point of MDC is 40 °C . under the vacuum. in current industry the process is carried out under the vacuum. MDC is supply to the heat exchanger in vapor phase so the in let temperature of MDC is fix hence we vary the flow rate of MDC . and the temperature of cold fluid .first we used water as a coolant to cool the MDC and next we use the coolant methanol.

Parameters :Type of coolant : water , methanol Temperature of cold fluid: 20 °C 15° C 10° C Flow rate of MDC: 0.0611,0.0672, 0.0733For water, 0.0611, 0.0733 for Methanol.

Boundary conditions:

The desired mass flow rate and temperature values are assigned to the inlet nozzle of the heat exchanger. The shell inlet temperature is set to 313 K. Zero gauge pressure is assigned to the outlet nozzle, in order to obtain the relative pressure drop between inlet and outlet. The inlet velocity profile is assumed to be uniform. No slip condition is assigned to all surfaces. The zero heat flux boundary condition is assigned to the shell outer wall, assuming the shell is perfectly insulated outside. Since the tube side flow is easy to resolve, the present study is concentrated on the shell side flow. After modeling the tubes as solid cylinders, the constant wall temperature of 293 K is assigned to the tube walls.

Governing equations :

Governing equations of the flow are modified according to the condition of the simulated case. Since problem assume to be steady ,time dependent parameters are dropped from the equations. The resulting equations are conservation of mass $\nabla \cdot (\rho V) \dots (16)$

$$\text{X-direction momentum : } \nabla \cdot (\rho u V) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} \dots (17)$$

$$\text{Y-direction momentum : } \nabla \cdot (\rho v V) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho g \dots (18)$$

$$\text{Z-direction momentum : } \nabla \cdot (\rho w V) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} \dots (19)$$

$$\text{Energy : } \nabla \cdot (\rho e V) = -\rho \nabla \cdot V + \nabla \cdot (K \nabla T) + q + \phi \dots (20)$$

Equation (5) ϕ is dissipation function that can be calculated from

$$\phi = \mu \left[2 \left[\left(\frac{\partial u}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial y} \right)^2 + \left(\frac{\partial w}{\partial z} \right)^2 \right] + \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^2 + \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right)^2 + \lambda (\nabla \cdot V) \right] \dots (21)$$

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijrset.com

Vol. 8, Issue 5, May 2019

Mesh selection :

Mesh generation is performed using Gambit. The surfaces of the model are meshed using tetragonal elements. The shell volume is meshed using tetragonal elements. Two different mesh sizes are used in the 5 baffle case: elements; finer mesh with approximately 6,000,000 elements

Turbulence model:

Since the flow in this study is laminar , laminar effects should be taken into account using turbulence modeling. The choice of turbulence model is very critical in CFD simulations However, there is no universal criterion for selecting a turbulence model. The turbulence model used in one study may not work in a different study. It is advisable to try a few different turbulence models. In this study, the Spalart–Allmaras and two different k–ε turbulence models are tried. In the Spalart–Allmaras turbulence model, only one turbulence equation is solved, therefore it is the least expensive model in ANSYS Fluent considering the computational effort. Two types of k–ε turbulence model are tried: standard and realizable. The transport equations, the viscosity calculation method, and the constants used in the model are the main differences between these models The Spalart–Allmaras model is a one-equation model that solves a modeled transport equation for the turbulent viscosity k-ε turbulence model is used for current stimulations and governing equations for steady state problems are as follows In which component X_j represents component in corresponding direction (x ,y, z) ε equation is

$$\rho \frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + G_{1\varepsilon} + \frac{\varepsilon}{k} (G_K - G_{3\varepsilon} G_b) - c_{2\varepsilon} \rho \frac{\varepsilon^2}{k}$$

$$\rho \frac{Dk}{Dt} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon$$

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

With the constant values of

$$C_{1\varepsilon} = 1.44 \quad C_{2\varepsilon} = 1.9 \quad \sigma_k = 1 \quad \sigma_\varepsilon = 1.2$$

Turbulent viscosity in above equation calculated from

Table No. 1 CFD results forwater

Type of cold fluid	Temperature of cold fluid(°C)	Flow rate of hot fluid(MDC) (kg/s)	Tube out let side temp (K)	shell out side let temperature (K)	Heat transfer coefficient (shell side) (W/m ² K)	Pressure drop of shell side (Pa)	Total heat transfer rate (shell side) (kw)
Water	20	0.0611	296.3	312.8	4.52	0.65	69.60
	20	0.0672	297	312.8	4.52	0.8	84.62
	20	0.0733	297	312.8	4.52	1.03	86.18
	15	0.0611	292.9	312.7	4.52	0.93	59.36
	15	0.0672	293.5	293.5	4.52	0.98	67.84
	15	0.0733	293	292.7	4.52	0.99	68.62
	10	0.0611	289.5	312.7	4.52	0.78	40.69
	10	0.0672	289.5	312.6	4.52	0.89	51.22
	10	0.0733	288.9	312.7	4.52	0.87	51.20

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

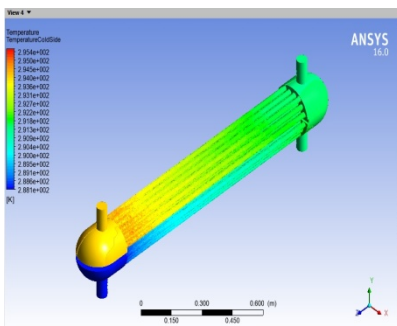
Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

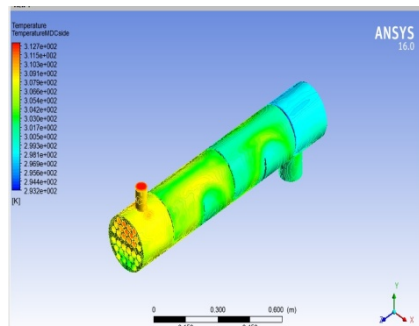
When we are using water as a cold fluid then at 10°C we got the maximum cooling of hot fluid but in actual practice it is very difficult to maintain the 10°C temp of water. So we go for the another solution we used methanol as a cold fluid:

Table No. 2 CFD results for Methanol :

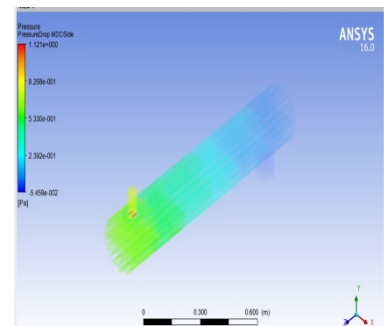
Type of cold fluid	Temperature of cold fluid(°C)	Flow rate of hot fluid(MDC) (kg/s)	Tube out let side temp (K)	shell out side let temperature (K)	Heat transfer coefficient (shell side) (W/m ² K)	Outlet Pressure of shell side (Pa)	Total heat transfer rate (shell side) (kw)
Methanol	20	0.0611	298.4	312.8	4.42	0.68	63.1512
	20	0.0733	298.6	312.8	4.42	0.69	86.3596
	15	0.0611	294.4	312.7	4.42	1.07	52.6884
	15	0.0733	295.4	312.7	4.42	1.06	60.951
	10	0.0611	290.6	312.7	4.42	0.68	34.0737
	10	0.0733	292.6	312.8	4.417616	1	41.6173



Temperature profile of cold fluid (fig.1)

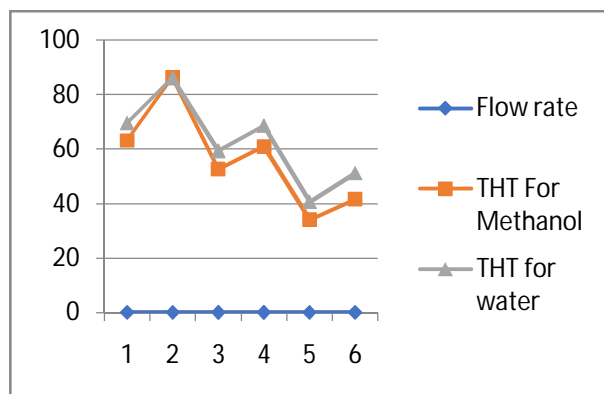


Temperature profile of hot fluid (fig.2)



Pressure profile (fig.3)

3.7 Graphs :



Graph no 1 Flow Rate Vs Total Heat Transfer Rate

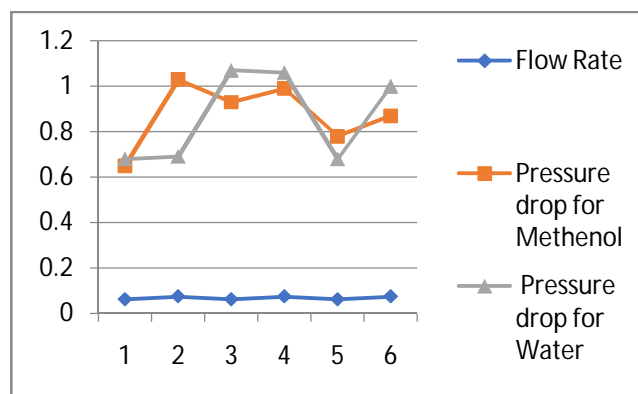
International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

If we compare the CFD results for total heat transfer rate vs flow rate It observe that it gives the maximum heat transfer rate in case of water over the methanol As flow rate increase heat transfer rate is increase but as temperature decreases heat transfer rate decreases.



Graph No. 2 Flow Rate Vs Pressure Drop

If we compare the pressure drop of shell side when the temperature is 20⁰C if we use water as cold fluid it gives the less pressure drop as compare to methanol but, if the temperature is decreases in case of water the pressure drop is increase by large amount and again decrease the temperature pressure drop is decrease .incase of methanol as the temperature decreases the pressure drop is also decrease

IV. CONCLUSION

In this study industrial problem is considered for cooling methylene di chloride by using shell and tube heat exchanger with straight tubes and double pass. The basic dimensions of the single shell and double tube heat exchanger are calculated. The present study focuses on thermal design only have shown. The method is used for the calculation is kern's method. The process involved is the iterative process. Only the final iteration of the calculation is represented. As coolant is assigned to the tube-side because it is corrosive and The MDC is low temperature and Low pressure gas assign it to the shell side. Selecting the segmental baffle for increase the turbulence. Selecting the baffle cut as 25% it gives the best result. Selecting the triangular pitch arrangement As per calculation the number of tube is 30 and the required area for the heat exchanger is 3m². I have done the model of heat exchanger in inventor as per design calculation. After getting experimental results and CFD results it observed that when we supply the cold fluid as a water it 10⁰ C gives the high temperature drop but in actual practice it is very difficult to supply the water at 10⁰ C and if we supply water at 10⁰ C and if there is any leakage occurs then it get blast in reactor which is very dangerous that's why we move towards other fluid i.e. Methanol if we are using methanol it gives the satisfactory results at 10⁰ C inlet temp. of methanol it give the outlet temp of shell side is 17⁰ C .and also give the satisfactory pressure drop.

REFERENCES

- [1] ThundilKaruppa Raj, Srikanth Ganne. Shell side numerical analysis of a shell and tube heat exchanger considering the effects of baffle inclination angle on fluid flow using CFD journal of science direct.
- [2] P.S. Gowthaman. Analysis of segmental and Helical Baffle in shell and tube heat heatexchanger . International Journal of Current Engineering and Technology
- [3] Sandeep K. Patel, Professor Alkesh M. Mavani. Shell and tube heat exchanger thermal design with optimization of mass flow rate and baffle spacing Journal of International Journal of advance engineering research and Studies.
- [4] Sunil S. Shinde, Samir S. joshi , Dr. S. Pavithran. Performance Improvement in Single phase Tubular Heat Exchanger using continuous Helical Baffles. journal of International Journal of Engineering Research

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

- [5] Analysis study of shell and tube heat exchanger for clough company with reselect different parameters to improve the design by Ammar Ali Abda, SamahZakiNajib
- [6]. <http://www.wcr-regasketing.com>. "Heat exchanger applications" <http://www.wcr-regasketing.com/heatexchanger-applications.htm>, 2010.
- [7]LimEngAik "Computational Fluid Dynamics Analysis of Shell and-double Concentric-tube Heat Exchanger." Science direct.
- [8] Usman Ur rehman heat transfer Optimization of Shell-and-Tube Heat Exchanger through CFD Studies journal of science direct.
- [9] Arjun K.S. and Gopu K.B. Design of Shell and Tube Heat Exchanger Using Computational Fluid Dynamics Tools journal of Research Journal of Engineering Sciences.
- [10] Coulson & Richardson's Chemical Engineering Volume 6 FourthEdition Chemical Engineering Design R. K. Sinno