

Optimization of Shell and Tube Heat Exchanger for Tube Arrangements

Eldhose Lomy¹, Nidheesh P²

PG Student, Dept. of Mechanical Engineering, Toc H Institute of Science and Technology, Ernakulam, Kerala, India¹

Assistant Professor, Dept. of Mechanical Engineering, Toc H Institute of Science and Technology, Ernakulam, Kerala, India²

ABSTRACT: Shell and tube heat exchanger is the most common type heat exchanger widely used in oil refinery and other large chemical process, because it suits high pressure application. The objective of the paper is to study the different parameters affecting heat transfers of a shell and tube heat exchanger and optimize the shell and tube heat exchanger for maximum heat transfer using CFD analysis. Computational analysis mainly carried out in the case of baffle angle and tube arrangement. 5° baffle angle and 45° circular arrangement with 3 cm pitch show uniform tube arrangement and better heat transfer coefficient than other arrangements.

KEYWORDS: shell and tube heat exchanger, baffle angle, baffle cut, simulation, heat transfer rate.

I. INTRODUCTION

The main purpose of heat exchanger is the transfer of heat from one fluid to another. Among all type of heat exchanger, shell and tube heat exchanger most commonly used. . The performances of shell and tube heat exchanger mainly depend on baffle geometry and tube geometry. Also fouling consideration and the fluids used in shell and tube have large effect on heat exchanger performance. The effect of fouling is considered in heat exchanger design by including shell side tube side fouling resistance. The tube side fluid should be corrosive and have high fouling resistance. Baffle is one of the major parts of shell tube heat exchanger. Baffles are designed to direct shell side fluid across the tube bundle efficiently as possible. The most common type of baffle is the single segmental baffle which changes the direction of shell fluid to archive cross flow. Now a day's helical baffles and flower baffles are used in shell and tube heat exchanger instead of segmental baffles and got better results. There were lots of experiments conducted in baffle plate in order to increase heat exchanger performance. **M.M. Elias**[1] experimentally investigated the effect of different nano particle shape on the overall heat transfer coefficient and shell and tube heat exchanger with different baffle angle (0, 10, 20, 30, 40, and 50). Boehmite alumina used as nano particle. In all shapes cylindrical nano particle show grater over all heat transfer coefficient .20° baffle angle show better heat transfer than other angles **Arjun K.S** [2] investigated the performance of one shell and tube heat exchanger model with helical angle vary from 0 to 40° .They got an effective heat transfer hike by the impact of helical angle and helix baffle inclination angle 20° makes the best performance of shell and tube heat exchanger .**Yonghua you** [3] solved a numerical model shell and tube heat exchanger with flower baffles for Reynold number ranging from 6813 to 22326.They found that, the mean heat transfer coefficient and convective heat transfer coefficient increase with increase of Reynolds number. Also they found that over all thermal hydraulic behaviour $h_{sm} / \Delta p$ decreases with increase of Reynolds number. **Xiaoming xiao** [4] investigated the heat transfer performance of fluid with different prandel number fluid in the shell side of helical baffle side of heat exchanger. The value of 'pr' ranges from 5 to 15000 and helical tilt angle from 10° to 50°. High prandel number fluid with small helical angle reveals to be optimum choice. In this study optimization of shell and tube heat exchanger have done for maximum heat transfer. Optimization mainly on for tube arrangement and baffle angle.

II. COMPUTATIONAL DOMAIN AND BOUNDARY CONDITION

In this study optimization of shell and tube heat exchanger have done for maximum heat transfer. Optimization mainly is mainly on tube arrangement and tube angle. Below show the both front and side view of the computational domain used for the CFD analysis. Here all dimensions are in cm .Computational domain drawn based on the experimental heat

exchanger arrangement. This paper focused on CFD part. So, no need to explain about experimental part. Here, use 2 volumes in the domain .Shell volume and tube volume. Hot water flows through the tube side with initial velocity of .7 m/sec. Also cold water flow through the shell side with initial velocity of .9 m/sec. Both hot water and cold water inlets considered as velocity inlet and outlets considered as pressure outlet and no slip boundary condition used. Turbulent kinetic energy added in the boundary condition, because shell side treated as turbulent side.

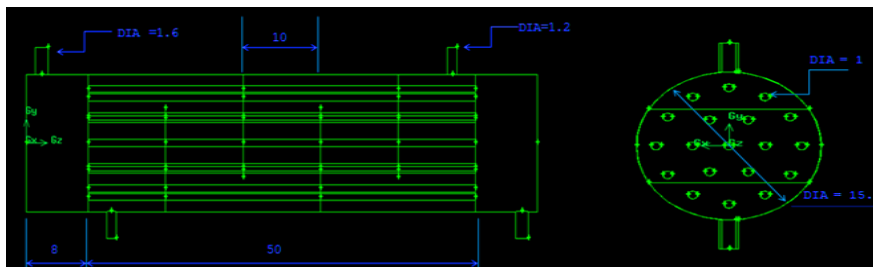


Fig 1(a) front view

fig 1(b) side view

III. EQUATIONS

1) CONTINUITY EQUATION

In fluid dynamics, the continuity equation is an expression of conservation of mass. For a steady or unsteady incompressible 3 dimensional flow, the continuity equation can be expressed in Cartesian co-ordinates as

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (1)$$

Where u, v and w are the components of velocity field in x, y and z directions respectively.

2) MOMENTUM EQUATION (NAVIER-STOKES EQUATION)

The Navier-Stokes equations are the basic governing equations for a fluid flow. It is obtained by applying Newton's Second Law of Motion to a fluid element and is also called the momentum equation. The Navier Stokes equation in scalar form for x, y and z directions are as follows.

a) Along x-direction

b) Along y-direction

$$\left[\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right] \times \rho = -\frac{\partial p}{\partial x} + \rho g_x + \mu \left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right]$$

$$\left[\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right] \times \rho = -\frac{\partial p}{\partial y} + \rho g_y + \mu \left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right] \quad (3)$$

c) Along z-direction

$$\left[\frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right] \times \rho = -\frac{\partial p}{\partial z} + \rho g_z + \mu \left[\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right] \quad (4)$$

Where p represents pressure is the density, μ is the dynamic viscosity and g is the acceleration due to gravity

3) ENERGY EQUATION

$$\rho C_p \left[\frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} + w \frac{\partial T}{\partial z} \right] = k \left[\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right] + S \quad (5)$$

Where 's' represent dissipation rate and 'k' is the thermal conductivity

IV. NUMERICAL METHODOLOGY

For the simulation of shell and tube heat exchanger, Simple algorithm is used to solve the equation related to the problem. Because, If we consider the discretised form of the Navier-Stokes system, the form of the equations shows linear dependence of velocity on pressure and vice-versa. This inter-equation coupling is called velocity pressure coupling. One of the methods is solving by SIMPLE algorithm. SIMPLE means semi implicit method for pressure linked equation.

SST K- ω turbulent model is used. The SST k- ω turbulence model [Menter 1993] is a two-equation eddy-viscosity model which has become very popular. SST k- ω model can be used as a Low Re number model without any extra damping functions. The SST k- ω model often merit it for its good behaviour in adverse pressure gradients and separating flow

V. CODE VALIDATION

The term Code Validation means, the process of affirming that the CFD algorithm used for solving a fluid problem is correct. In CFD, a given problem can be solved using a wide range of algorithms such as SIMPLE (Semi Implicit Method for Pressure linked equations), SIMPLEC (Semi Implicit Method for Pressure Linked Equations-Corrected), PISO, COUPLED etc.

The total heat transfer and temperatures of the shell and tube heat exchanger were the parameters taken for code validation. The process starts with the verification of the results obtained using the chosen algorithm, with that of the experimental results. After validating and finding that the algorithm chosen is suitable for the given case, the same algorithm is used for the optimization purpose.

| Temperature(k) | Experimental | CFD |
|-------------------|--------------|------|
| Hot water inlet | 338 | 338 |
| Hot water outlet | 323 | 322 |
| Cold water inlet | 303 | 303 |
| Cold water outlet | 315 | 315 |
| Q(KW) | 4.9 | 5.02 |

Table 1(a) mass flow rates $m_c = .09$ kg/sec
 $m_h = .07$ kg/sec

| Temperature(k) | Experimental | CFD |
|-------------------|--------------|------|
| Hot water inlet | 338 | 338 |
| Hot water outlet | 323 | 322 |
| Cold water inlet | 303 | 303 |
| Cold water outlet | 315 | 315 |
| Q(KW) | 4.9 | 5.02 |

Table 1(b) mass flow rates $m_c = .08$ kg/sec
 $m_h = .06$ kg/sec

VI. RESULT AND DISCUSSION

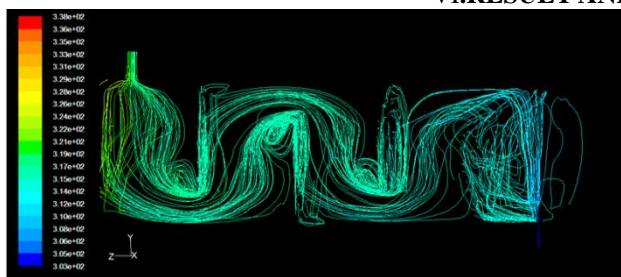


Fig2- path line through baffles ($m_h = .07$ kg/sec
 $m_c = .09$ kg/sec)

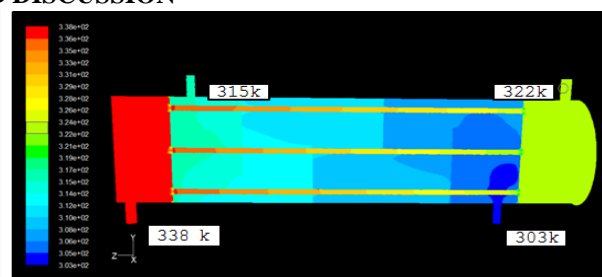


Fig 3 – Temperature contour ($m_h = .07$ kg/sec
 $m_c = .09$ kg/sec)

CFD result of first set of flow rates shown above. Here is code is validated. So, precede to the optimization steps. Optimization is mainly for baffle angle and tube arrangement. In industries, helical baffles and flower baffle used instead of segmental baffles. Compared to the segmental heat exchangers, the heat exchangers with continual helical baffles have higher heat transfer coefficients to the same pressure drop. Continual helical baffles can reduce or even eliminates the dead regions in the shell side of a shell and tube heat exchanger.

In my model, segmental baffles are used for supporting the tubes with baffle cut =25%. In previous studies, we can see that, if we provide inclination for baffles. There is an improvement of heat transfer due to increased turbulence. Also there is controlling of pressure take place. So we can eliminate the problem developed by segmental baffle like heavy pressure drop, less heat transfer performance .Baffle angle changed from 0 to 25°. From the optimization, got 5° as the optimum angle .because it shows maximum heat transfer than other angle. From the optimization of angle, there is a pattern of heat transfer and turbulent intensity. Both turbulent intensity and heat transfer are directly proportional to each other. That is 5° have maximum turbulent intensity than others. Hear first set of reading is the base model.

| Sl.no | Angle (degree) | Total heat transfer rate(W) | surface heat transfer coefficient (w/m ² k) | Turbulent intensity (%) |
|----------|----------------|-----------------------------|--------------------------------------------------------|-------------------------|
| 1 | 0 | 5021 | 635.8 | 23.2 |
| 2 | 5 | 5183 | 650.6 | 24.3 |
| 3 | 10 | 4231 | 493.6 | 16.4 |
| 4 | 15 | 5106 | 643.2 | 23.8 |
| 5 | 20 | 5058 | 634.2 | 23.3 |
| 6 | 25 | 3896 | 459.8 | 14.2 |

Table (2) baffle angle comparison (mh =.07kg/sec, mc= .09kg/sec)

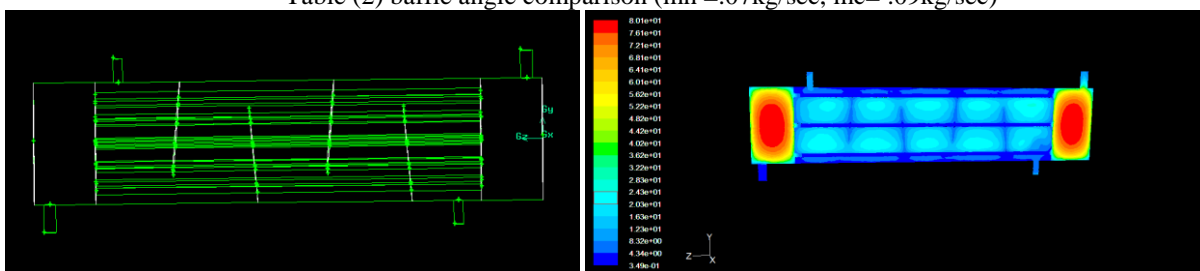


Fig 4 baffle plate arrangement and turbulent intensity contour of angle 5°

Results of second set of flow rate similar to the first set. Pressure drop of angle 5 degree is also with in the design pressure drop range. So it is used as optimum angle and goes for optimization of second parameter .That is, optimization for tube arrangement.

Heat transfer performance of shell and tube heat exchanger depends on many factors. One of the important factors is tube arrangement. Good tube arrangement provide uniform distribution of fluid across the tube bundle .ideal tube arrangement give good heat transfer performance also which limits the pressure in a design range. Commonly used tube arrangements are 1) TRANGULAR 2) SQUARE3) ROTATED SQUARE 4)_CIRCULAR. And commonly used pitch for this arrangements are 1) 15/16 INCH 2)1 INCH 3)1.25 INCH. Tube pitch means distance between the tubes.

We used about 14 arrangements based on the standard tube arrangements. One of the methods to increase heat transfer by increasing surface area. It is done by increasing no of tubes. Even though, the no of tube is increased, but there is chance of reduction in heat transfer, due to non uniformity of fluid inside the shell and reduction in turbulent intensity value. This leads to the reduction of surface heat transfer coefficient. From the turbulent intensity contour; we can see

that maximum turbulent intensity at the top of the baffle plate. So we have to arrange more tube at that sides, this will produce good contact of fluid with tubes, produce high surface heat transfer coefficient. For all arrangement angle 5 set as baffle angle.

From the arrangements, circular pitch arrangement, i.e. 45° inner circle and 45° outer circle with 1.25 inch (17 tubes) pitch show more heat transfer ($h=712 \text{ w/m}^2\text{k}$). Also this arrangement show maximum turbulent intensity than others. Pressure drop of this arrangement also with in the design range. So it is taken as the optimum tube arrangement. Circular tube arrangement (60° inner and 30° outer) with 1.25 inch pitch is the base model ($h=652.4 \text{ w/m}^2\text{k}$). Like baffle angle optimization, optimized tube arrangement show maximum turbulent intensity than others.

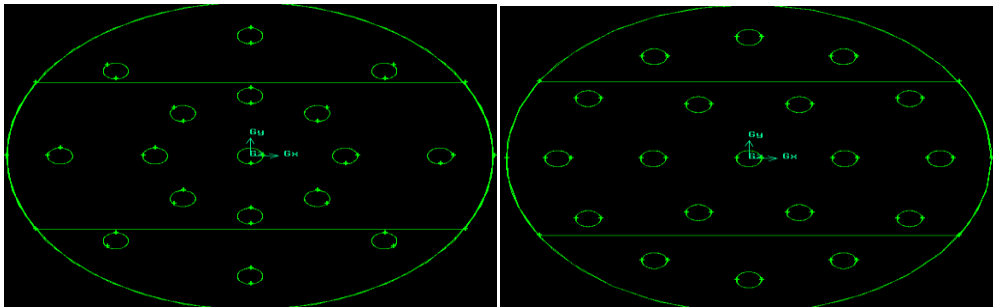


Fig 5(a) Optimized tube arrangement

Fig 5(b) Base model

VII. CONCLUSION

Optimization of shell and tube heat exchanger model for maximum heat exchanger has done using CFD analysis. After the optimization procedure, got 5° baffle angle and circular tube arrangement (45° and 45°) with 1.25 inch pitch be the optimized parameter. For both parameters, there is relation between heat transfer and turbulent intensity. Optimized model have maximum turbulent intensity than other arrangements.

REFERENCES

- [1] KEVIN M. LUNSFORD, 1998, *Increasing Heat Exchanger Performance*, Bryan Research and Engineering Technical Papers.
- [2] Simin Wang, Jian Wen, Yanzhong Li, 2008, *an experimental investigation of heat transfer enhancement for a shell and tube heat exchanger*, Applied thermal engineering
- [3] Yan Li, Xiumin Jiang, Xiang yong Huang, Jigang Jia, Jianhui Tong, 2010, *Optimization of high pressure shell and tube heat exchanger for syngas cooling in a an IGCC*, IJHM
- [4] Farhad nemati taher, kazem razmi et al. (2011), *Baffle space impact on the performance of helical baffle shell and tube heat exchangers*, applied thermal engineering.
- [5] Xiaoming Xiao, youmei xia et al. (2012), *Numerical investigation of helical baffles heat exchanger with different Prandtl number fluids*, IJHM.
- [6] Yonghua you, Wei liu et al. (2012), *Numerical modelling and experimental validation of heat transfer and flow resistance on the shell side of a shell-and-tube heat exchanger with flower baffles*, IJHM.
- [7] M.M. Elias, I.M. Shahrul, I.M. Mahbulul, R. Saidur, N.A. Rahim (2013), *Effect of different nano-particle shapes on shell and tube heat exchanger using different baffle angles and operated with nanofluid*, IJHM.
- [8] Rajiv Mukherjee, *Effectively Design Of Shell-and-Tube Heat Exchangers*.
- [9] Ye Yao, Xingyu Zhang, Yiyang Guo. 2010, *and Experimental Study on Heat Transfer Enhancement of Water- water STHX Assisted by Power Ultrasonic*, I R A Conference.
- [9] R. Shankar Subramanian, *Shell-and-Tube Heat Exchangers*.
- [10] KERN, D.Q., *Process Heat Transfer*, McGraw-Hill, Inc., New York, 1950.
- [11] Rajiv Mukherjee, *Effectively Design Of Shell-and-Tube Heat Exchangers*