International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 7, July - 2014

Design and Performance Evaluation of Shell and Tube Heat Exchanger using CFD Simulation

Santosh Kansal
Department of Mechanical Engineering
Institute of Engineering and Technology
Indore, Madhya Pardesh

Mohd. Shabahat Fateh
Department of Mechanical Engineering
Institute of Engineering and Technology
Indore, Madhya Pardesh

***Abstract*—The most commonly practiced types of heat exchanger are the shell-and-tube heat exchanger, the optimal design of which is the primary aim of this work.The present paper deals with the design of a shell and tube heat exchanger. The main objective of this paper is to verify the heat exchanger designed with the use of the Kern's method, by the use of Commercial computational fluid dynamics (CFD) software. In the present study, CFD simulation is used to study the temperature and velocity profiles through the tubes and the shell.**

Keywords:Commercial computational fluid dynamics (CFD), Shell and tube heat exchanger, Kern's method, Methonal, Brackish water

I. INTRODUCTION

Kern's method was based on experimental work on commercial exchangers with standard margins and will give a reasonably satisfactory prediction of the heat-transfer coefficient for standard designs. The prediction of pressure drop is less acceptable, as pressure drop is more affected by leakage and bypassing than heat transfer.The case-side heat transfer and friction components are correlated in a like manner to those for tube-side flow by using a hypothetical shell velocity and shell diameter.As the cross-sectional area for flow will vary across the shell diameter, the linear and mass velocities are based along the maximum area for cross-flow: that at the shell equator.The shell equivalent diameter is computed utilizing the flow field between the tubes taken in the axial direction (parallel to the tubes) and the wetted perimeter of the pipes

At least 60% of heat exchanger types are the shell-and-tube type exchanger, indicating its prevalence in the process industry [1], with applications in oil coolers and power condensers, as well as preheaters in power plants and steam generators in nuclear power plants, process applications, and the chemical industry [2]. One of the most common assumptions in basic heat exchanger design theory is that fluid can be distributed uniformly at the intake of the exchanger on each fluid side and throughout the core.

II. BASIC DESIGN PROCEDURE AND THEORY

The general equation for heat transfer across a surface is:

$$Q = UA\Delta T_{lm} \qquad (1)$$

where,

Q = heat transferred per unit time, W,
U = the overall heat transfer coefficient, W/m^2°C,
A = heat-transfer area,m^2
ΔT_m = the mean temperature difference, the temperature driving force, °C

The overall heat transfercoefficient is the reciprocal of the overall resistance to heat transfer, which is the sum of several individual resistances. For heat exchange across a typical heat exchanger tube the relationship between the overall coefficient and the individual coefficients, which are the reciprocals of the individual resistances, is given by:

$$\frac{1}{U_o} = \frac{1}{h_o} + \frac{d_o \ln(d_o/d_i)}{2k_w} + \frac{d_o}{d_i} \times \frac{1}{h_{id}} + \frac{d_o}{d_i} \times \frac{1}{h_i} \qquad (2)$$

where,

U_o = the overall coefficient based on the outside area of the tube, W/m^2°C,
h_o = outside fluid film coefficient,W/m^2°C,
h_i = inside fluid film coefficient, W/m^2°C,
h_{od} = outside dirt coefficient (fouling factor), W/m^{2o}C,
h_{id} = inside dirt coefficient, W/m^{2o}C,
k_w = thermal conductivity of the tube wall material, W/m^{o}C,
d_i = tube inside diameter, m,
d_o = tube outside diameter, m.

The magnitude of the individual coefficients will depend along the nature of the heat transfer process (conduction, convection, condensation, boiling or radiation), on the physical properties of the fluids, on the fluid flow-rates, and on the physical arrangement of the heat-transfer surface. As the physical layout of the exchanger can't be determined until the area is known the design of an exchanger is of necessity a trial and error procedure. The steps in a typical design procedure are given below:

1. Define the duty: heat-transfer rate, fluid flow-rates and temperatures.

2. Collecting together the fluid physical properties required: density, viscosity, thermal conductivity.
3. Deciding on the type of exchanger to be used (which in our case is shell and tube type).
4. Selecting a trial value for the overall coefficient, U.
5. Calculating the mean temperature difference,ΔT_{lm}:

$$\Delta T_{lm} = \frac{(T_1 - t_2) - (T_2 - t_1)}{\ln(\frac{T_1 - t_2}{T_2 - t_1})} \quad (3)$$

where,
ΔT_{lm} = log mean temperature difference,
T_1 = hot fluid temperature, inlet,
T_2 = hot fluid temperature, outlet,
t_1 = cold fluid temperature, inlet,
t_2 = cold fluid temperature, outlet.

6. Calculating the area required from equation 1:
7. Deciding the exchanger layout.
8. Calculating the individual coefficients.
9. Calculating the overall coefficient and compare with the trial value. If the calculated value differs significantly from the estimated value, substitute the calculated for the estimated value and return to step 6.
10. Calculating the exchanger pressure drop; if unsatisfactory return to steps 7 or 4 or 3, in that order of preference.
11. Optimizing the design: repeat steps 4 to 10, as necessary, to determine the cheapest exchanger that will satisfy the duty.Usually this will be the one with the smallest area [3].

A. *Theoritical Design*

We have to design a shell and heat tube heat exchanger to sub-cool methanol from 95°C to 40°C. Flow-rate of the methanol is 150 kg/h. Brackish water is being used as the coolant, with a temperature rise from 25°C to 40°C. Using the Kern's method our obtained values are mentioned in the Table 1.

TABLE I. HEAT EXCHANGER DIMENSIONS

No.	Description	Unit	Value
1	Shell diameter	mm	170
2	Tube outer diameter	mm	20
3	Tube inner diameter	mm	16
4	No. of tubes	mm	21
5	Shell/Tube length	mm	850
6	Inlet length	mm	30
7	Outlet length	mm	30

III. CFD ANALYSIS

Computational fluid dynamic study of the system starts with modelling desired geometry and mesh for modeling the domain. Generally, asimplified geometry is used for the CFD studies. Meshing helps in the discretization of the domain into small elements. Setup starts with defining the boundary, initializing conditions for the domain and then it leads to modeling the entire system domain. Finally, we analysis the results.

A. *Geometry*

The Shell and tube heat and exchanger is an assembly of twoinlet/outlet chambers, twenty one aluminum tubes with triangular pitch arrangement. In this arrangement, cooling water will flow throughtube side and methanol will flow from shell side, in opposite direction of the cooling water as shown in figure-1. This figure shows us the actual geometry of the shell and tube heat exchanger

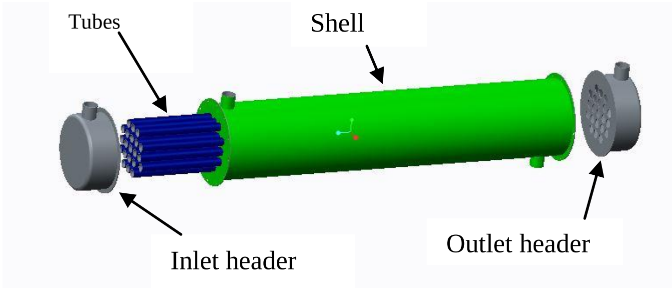

Fig. 1. Shell and tube heat exchanger

While the figure-2 shows us the fluid volume which will be occupied by the methanol and water when they are inside heat exchanger. The fluid assembly is deigned in Creo 2.0 student edition.

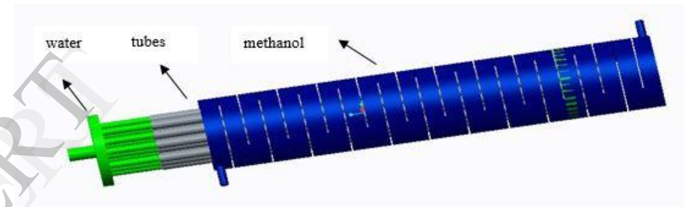

Fig. 2. Fluid assembly model of the shell and tube heat exchanger

The CFD simulation is carried out for fluid assembly model. Firstly the model's symmetric view is taken of the given geometry as the geometry is too complex and it will reduce the load on Ansys. The symmetric model is meshed in the Ansys and the whole model is divided into verysmall regions called nodes andelements.

B. *Solution*

Ansys Fluent 13.0 is used to simulate the the fluid assembly model. The boundary condition found using the Kern's method are used for the simulation.

a) Boundry Conditions: Highlight all author and affiliation lines.

TABLE II. BOUNDRY CONDITION FOR THE HEAT EXCHANGER

	BC Type	Shell	Tube
Inlet	Velocity-inlet	1.326 m/s	0.156 m/s
Outlet	Pressure-outlet	0	0
Temperature	Inlet temperature	368	298
Mass flow rate		250 kg/s	0.287 /s

IV. RESULT

The temperature distribution along the heat exchanger can be seen through side view on the plane of symmetry. The contour plots in Figure 3 shows the whole length of heat exchanger. The top most part is the inlet region and lowest part is the outlet of the shell. Whereas the left most part is the inlet and right most the outlet of the tube.The e+2 or e-2 in the contour plot figures represents the 10^2 and 10^{-2}.

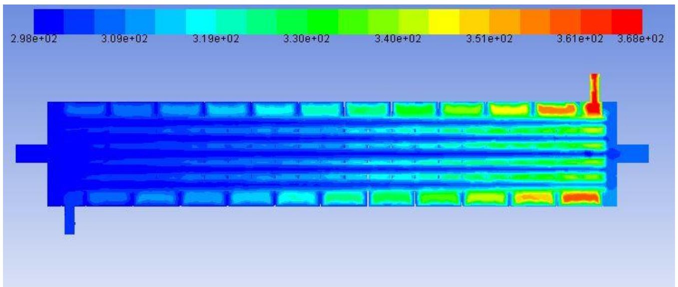

Fig. 3. Temperatue contour plot at symmetrical plane

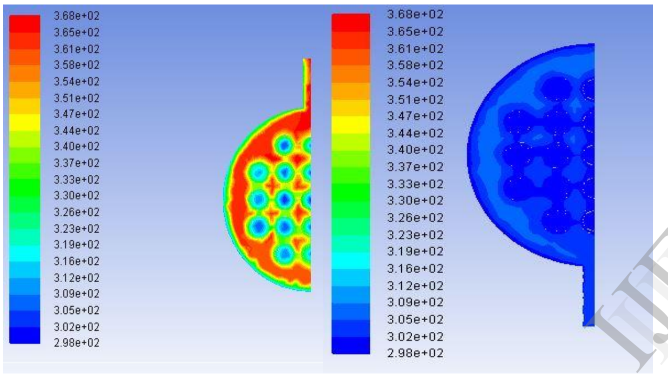

Fig. 4. Temperatue contour plot at inlet and outlet cross section

From figure-3 and figure-4 we can see the temperature profile. From the temperature profile we can see that the heat transfer is not uniform throughout the length and we can see that it is lesser near the inlet and out let of the shell: greater in middle of the heat exchanger. This is due to the use of counter flow heat exchanger.

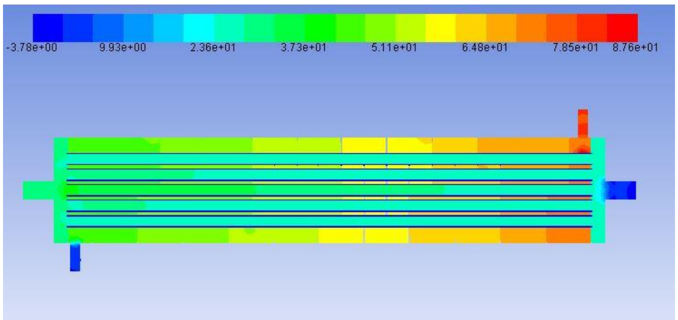

Fig. 5. Pressure contour plot at symmetrical plan

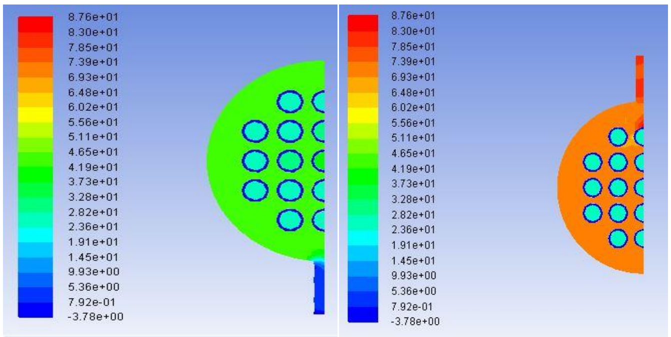

Fig. 6. Pressure contour plot at inlet and outlet cross section

From figure-5 and figure-6 we can see the pressure profile. Pressure profile is a representation of what is happening with fluid inside the shell and tubes. From the pressure profile we can see that the high pressure is building up on the inlet of the shell and the pressure decreases throughout the shell profile thus showing the decrease in the velocity of the fluid.

TABLE III. CFD RESULTS FOR TEMPERATURES AND THE OVERALL EFFECTIVENESS

Paramters	**Desired Values**		**CFD Result**	
	Inlet Section	*Outlet Section*	*Inlet Section*	*Outlet Section*
Mean Temperature of methonal (K)	368	313	368	315.53
Mean Temperature of water (K)	298	313	298	308.43
Effectiveness	0.79		0.76	

V. CONCLUSION

The practical simulation of a designed model is very expensive, hence CFD is a tool which helps to simulate the process and thus eliminating the cost and for the development of a prototype based on study. Our study shows us how effective is the Kern's method for design of a shell and tube heat exchanger using the CFD.

The assumption of plane symmetry works well for our study of most of the length of heat exchanger except the outlet and inlet regions where the rapid mixing and change in flow direction takes place. Thus improvement is expected if complete geometry is modelled in CFD. Furthermore, the enhanced wall functions are not used in this project due to convergence issues, but they can be very useful with $k-\varepsilon$ models. Our results are almost close to the condition for which we were fabricating the heat exchanger. As our desired effectiveness was too high so matching the result was a problem.

Absence of a practical model for the designed model limitation of this study. An experimental setup shall be fabricated in future for the practical comparison with the CFD results.

REFERENCES

[1] M. S. Peters, K. D. Timmerhaus and R. E. West, Plant design and economics for chemical engineers, 5thEd., McGraw-Hill, Boston (2003).

[2] S. Sadik Kakac and H. Liu, Heat exchangers: selection, rating, and thermal design, 2ndEdition, CRC Press, Boca Raton, FL (2002).

[3] R. K. Sinnott,Coulson & Richardson's Chemical Engineering, Vol. 6, 4th Ed., Elsevier Butterworth-Heinemann, Jordan Hill, 2005.

[4] H K Versteeg And W Malalasekera,"An Introduction To Computational Fluid Dynamics", 4th Ed.,Pearson Education Limited,England, 2007.

[5] Muhammad Mahmood Aslam Bhutta, Nasir Hayat, Muhammad Hassan Bashir, Ahmer Rais Khan, Kanwar Naveed Ahmad, Sarfaraz Khan,"CFD applications in various heat exchangers design: A review",Elsevier,Vol.32,1-12,2012.

[6] Ender Ozden, Ilker Tari,2010 "Shell side CFD analysis of a small shell-and-tube heat exchanger", Energy Conversion and Management Vol. 51, No. 5, pp. 1004-1014 (2010).

[7] Ansys,"ANSYS CFD-Post Tutorials", Ansys,Inc. Southpointe 275 Technology Drive Canonsburg, PA 15317, 2012.