Computational Fluid Dynamics Analysis of Various types of Shell and Tube Heat Exchanger

Mr. Nikhil N. Sawant [1], Dr. R. G. Desavale [2]

[1]M.Tech - Student (Mechanical Design Engineering), [2]Prof.of Mechanical Department (RIT).

ABSTARCT

The aim of this study is to predict the performance of shell and tube heat exchanger. In this paper, the shell and tube heat exchanger was modeled and then CFD simulation has been carried out. Computational fluid dynamics is a computer-based simulation technique for examining fluid stream, heat transfer. The four different models have been developed which are 1-1 shell and tube heat exchanger with and without baffles, 1-2 shell and tube heat exchanger with and without baffles. The performance of heat exchanger is determined with the help of CFD results by varying the mass flow rate as 0.25Kg/s, 0.5Kg/s respectively. Using post-processor, the simulated results were computed i.e. temperature contours. The analysis shows that in 1-2 shell and tube heat exchanger with baffles gives large temperature difference than others so more heat transfer rate has occurred and hence the efficiency is improved. The CFD results are useful to design the shell and tube heat exchanger and to modify an existing design.

Keywords: Shell and tube heat exchanger, Baffles, Solid works software, CFD analysis.

1. Introduction

In various industrial sectors, a heat exchanger is mainly utilized to transfer heat in between cold and hot liquids respectively. For increasing the rate of heat transfer a number of investigations are examined. In various sorts of heat exchanger a shell and tube heat exchanger has been effectively utilized in a number of industrial units since shell and tube provide more area to heat exchange between different fluids respectively.

Shell and Tube Heat Exchanger

The shell and tube heat exchanger consists of curved tube bundles which are fixed in a cylinder-shaped shell with tubes are positioned parallel to the shell. Shell and tube heat exchangers are generally easy to make, and hence multi reason application plausibility when analyzed with sorts of heat exchangers. It is accounted by the investigators over 30 % heat exchangers being utilized are the shell and tube type. In this heat exchanger, one fluid flows through the tubes while other fluid flows across and along the axis of the exchanger. Important segments of shell and tube exchangers are tubes, shell, baffles, and fouling considerations. In this case, shell side fluid being used is oil and tube side fluid is water. A larger heat transfer area has been utilized in order to transfer heat more efficiently. So the excess amount of heat should be put to utilize. This is a superior way to preserve energy.

2. Computational Fluid Dynamics (CFD)

CFD is a computer-based recreation procedure for analyzing liquid stream, transfer of heat and so on. A CFD analysis is the better way to judge the performance of heat exchanger than practical performance since the experiments have cost directly proportional to the number of testing to be taken. Also, CFD produces a number of results with no adding expenses. CFD programming provides control to simulation for the liquid and gas flow, transfer of heat and mass, moving bodies, fluid-structure interaction through computer modeling. CFD software provides fabricated PC created model with sufficient information which judges the performance of exchanger.

For a particular structure, a 3-dimensional model has been developed by using solid works programming and after that meshing has done. Different materials are preferred for particular shell and tube, but in our case, we utilized aluminum material. Then different boundary conditions have been assigned by changing the mass stream rate of a fluid.

2.1 Designing Using Solid work Software

As per the various parameters, different models have been developed by using solid works software (ver.17). For performing CFD analysis four models have been created which is 1-1 shell & tube heat exchanger with and without baffles and 1-2 shell & tube heat exchanger with and without baffles respectively.

Fixed parameters	Variable Parameters
Length of tube = 208mm	Mass flow rate
Tube diameter (mm), OD = 8; ID = 6	
Number of tubes = 49, Hot & Cold Fluids	
Inlet temp. of oil and water is 70^oC & 30^0C	

Table 1- Fixed & variable parameters

Model No. 3 (1-Shell 2-Tube pass heat exchanger with baffles)

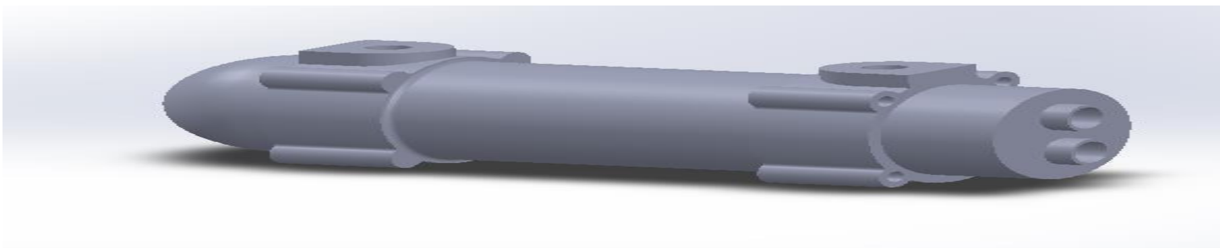


Fig.2- Assembly Model

Fig.2 (a) - 3-D Cut Section

2.1.1 Flow Simulation Procedure

Basically, there are three major steps involved in performing the analysis.

Step 1: Preprocessing

A pre-processor is used to define the geometry of the computational domain of interest & generate the mesh of control volumes (for calculation). Generally, the finer mesh in the areas of large changes gives the more accurate results. Fineness of grid also determines the calculation time needed.

Step 2: Processing

In processing, computer solves the given model as per the assigned boundary conditions.

Step 3: Post processing

After solving the model, the results will be displayed in terms of streamline, pressure, and temperature.

From the results, we can easily compare, which model will be suitable for the given application.

Model No.	No. Of tube	Shell Diameter	Iteration No.	Variation
Model 1	49	92	1.1	Inlet oil mass flow rate is 0.25 Kg/s
			1.2	Inlet oil mass flow rate is 0.50 Kg/s
Model 2	49	92	2.1	Inlet oil mass flow rate is 0.25 Kg/s
			2.2	Inlet oil mass flow rate is 0.50 Kg/s
Model 3	49	92	3.1	Inlet oil mass flow rate is 0.25 Kg/s
			3.2	Inlet oil mass flow rate is 0.50 Kg/s
Model 4	49	92	4.1	Inlet oil mass flow rate is 0.25 Kg/s
			4.2	Inlet oil mass flow rate is 0.50 Kg/s

Table 2- Iterations to be performed

SIMULATION FOR MODEL 3 (1-2 shell and tube heat exchanger with baffles)

Processing Images

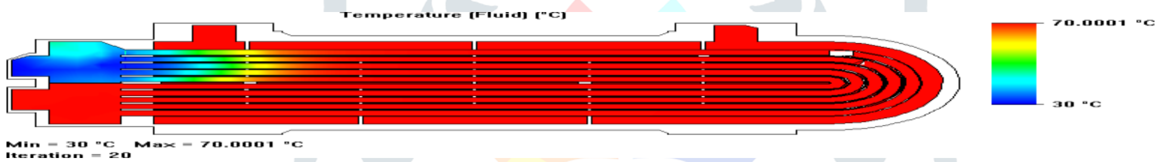


Fig.3 (a) –Plot after 20 Iteration

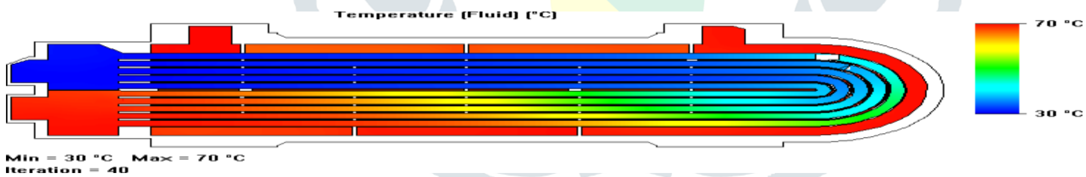


Fig.3 (b) –Plot after 40 Iteration

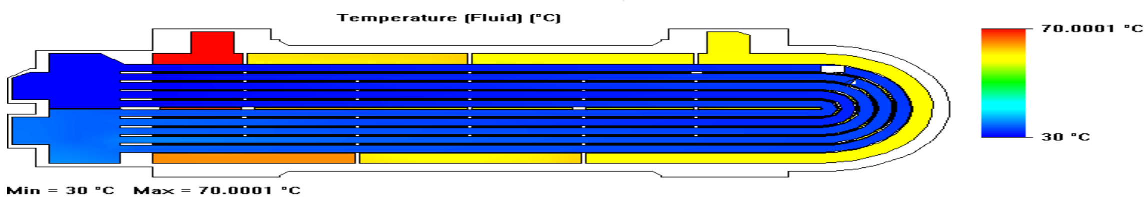


Fig.3 (c) –Plot after 60 Iteration

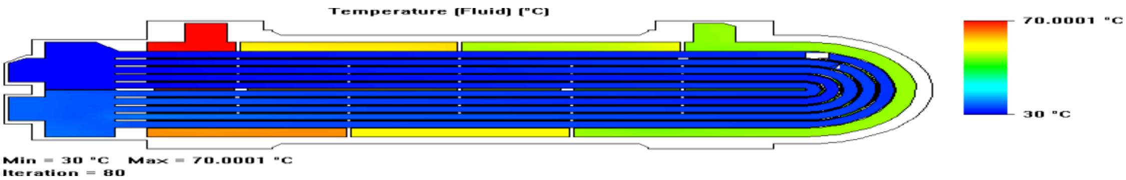


Fig.3 (d) –Plot after 80 Iteration

Above figures 3(a), 3(b), 3(c), 3(d) and 3(f) shows the temperature variations after a number of iterations which is in between 30 ^{0}C to 70 ^{0}C respectively.

ITR 3.1 - RESULTS - 0.25 Kg/s

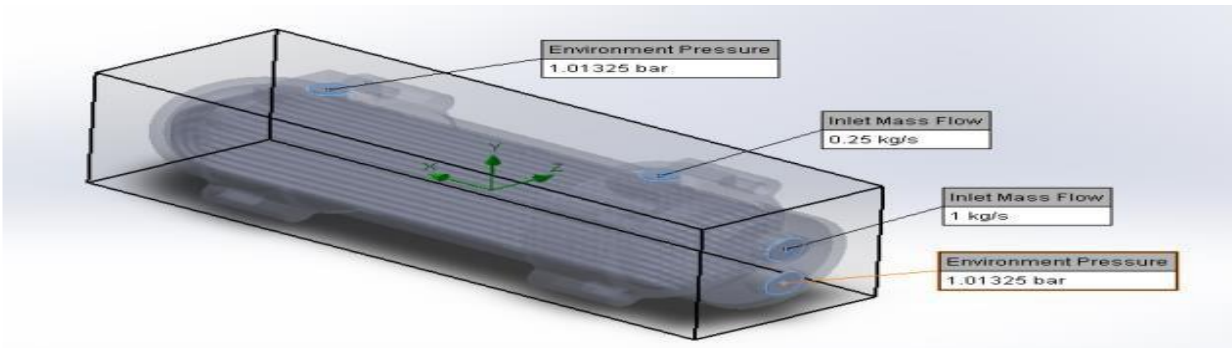


Fig.3 (e) –Boundary Condition

Figure 3(e) show different boundary conditions at the inlet and outlet of shell and tube respectively. The variation in the only inlet mass flow rate i.e. it has been varied from 0.25 Kg/s to 0.50 Kg/s respectively.

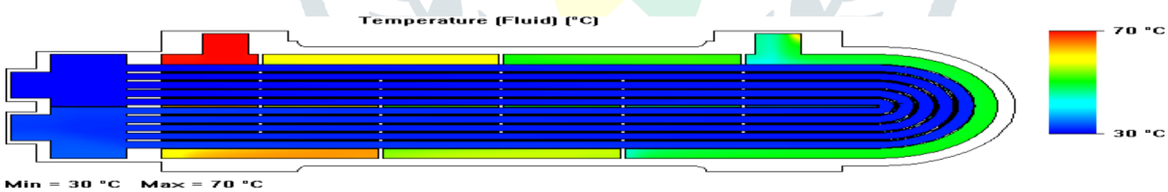
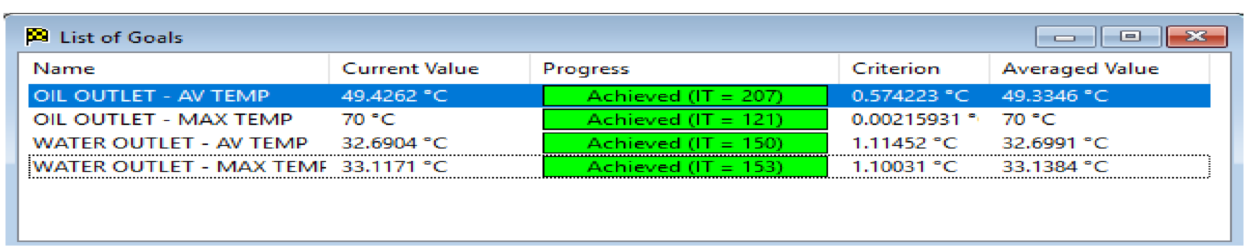


Fig.3 (f) –Preview Diagram


List of Goals

Name	Current Value	Progress	Criterion	Averaged Value
OIL OUTLET - AV TEMP	49.4262 °C	Achieved (IT = 207)	0.574223 °C	49.3346 °C
OIL OUTLET - MAX TEMP	70 °C	Achieved (IT = 121)	0.00215931 °	70 °C
WATER OUTLET - AV TEMP	32.6904 °C	Achieved (IT = 150)	1.11452 °C	32.6991 °C
WATER OUTLET - MAX TEMF	33.1171 °C	Achieved (IT = 153)	1.10031 °C	33.1384 °C

Fig.3 (g) –Temperature Result

Figure 3(g) show the different outlet temperature results for oil and water after performing a number of iterations.

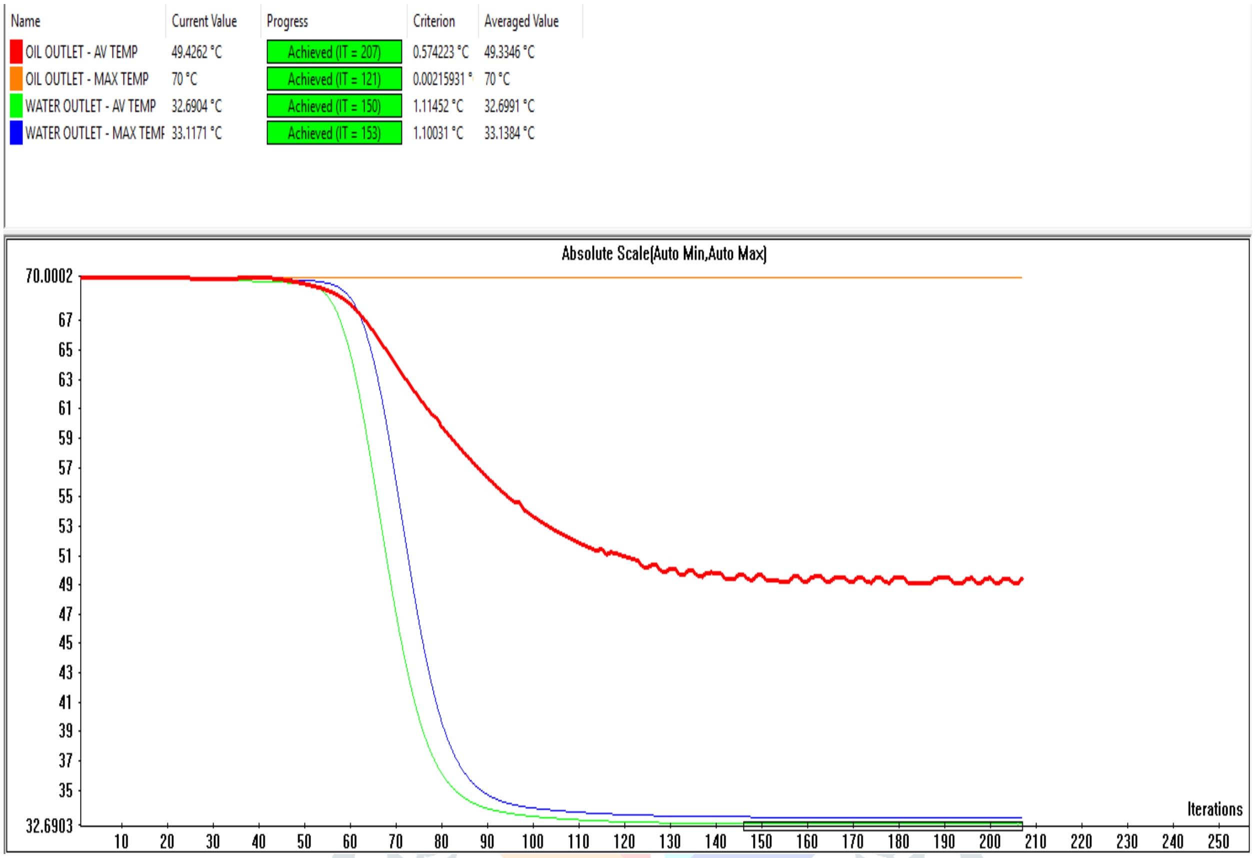


Fig.3 (h) –Graph Result

Figure 3(h) show the convergence of simulation which means after a number of iterations the graph becomes constant i.e. all the values obtained which are in a similar manner respectively.

3. Results Interpretation

Model No.	Mass Flow Rate, Kg/s	Oil outlet - Av Temp	Oil outlet - Max Temp
Model 1	0.25	57.69	58.5
	0.5	61.17	61.7
Model 2	0.25	63.9	65.47
	0.5	65.97	66.83
Model 3	0.25	49.33	70
	0.5	54.36	70
Model 4	0.25	63.62	68.45
	0.5	65.17	68.92

Table 3- Average values for all iteration

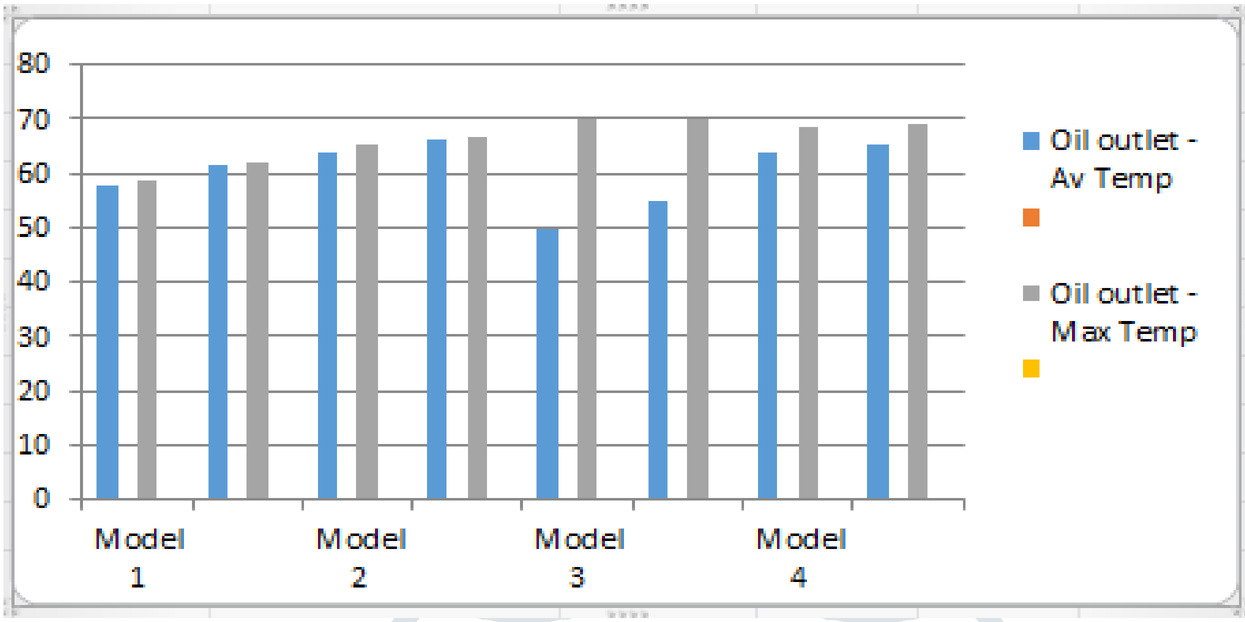


Fig.4- Average oil temperature achieved after convergence

4. Conclusion

By comparing and analyzing different proposed design (1-1 STHE with and without baffles, 1-2 STHE with and without baffles), it is seen that 1-2 STHE with baffles has better heat transfer than other because the temperature difference of oil at inlet and outlet is more so increases effectiveness of heat exchanger. After performing the analysis we come to a conclusion that as we reduce the mass flow rates of the hot fluid the rate of transfer of heat increases hence improving the efficiency of the heat exchanger.

References

[1] Bichkar, P., Dandgaval, O., Dalvi, P., Godase, R., & Dey, T. (2018). Study of shell and tube heat exchanger with the effect of types of baffles. *Procedia Manufacturing*, *20*, 195-200.

[2] Costa, A. L., & Queiroz, E. M. (2008). Design optimization of shell-and-tube heat exchangers. *Applied Thermal Engineering*, *28*(14-15), 1798-1805.

[3] Pal, E., Kumar, I., Joshi, J. B., & Maheshwari, N. K. (2016). CFD simulations of shell-side flow in a shell-and-tube type heat exchanger with and without baffles. *Chemical*

Engineering Science, *143*, 314-340.

[4] Shahril, S. M., Quadir, G. A., Amin, N. A. M., & Badruddin, I. A. (2017). Thermo hydraulic performance analysis of a shell-and-double concentric tube heat exchanger using CFD. *International Journal of Heat and Mass Transfer*, *105*, 781-798.

[5] Zhang, M., Meng, F., & Geng, Z. (2015). CFD simulation on shell-and-tube heat exchangers with small-angle helical baffles. *Frontiers of Chemical Science and Engineering*, *9*(2), 183-193.

[6] Kern, D. Q. (1950). *Process heat transfer*. Tata McGraw-Hill Education.

[7] Ambekar, A. S., Sivakumar, R., Anantharaman, N., & Vivekenandan, M. (2016). CFD simulation study of shell and tube heat exchangers with different baffle segment configurations. *Applied Thermal Engineering*, *108*, 999-1007.

[8] Arjun, K. S. (2014). *Design of shell and tube heat exchanger using computational fluid dynamics tools* (Doctoral dissertation, Manav Bharti University).

[9] Uddhage, H., Kumbhare, S., & Thakur, K. K. (2016). DESIGN AND DEVELOPMENT OF SHELL AND TUBE HEAT EXCHANGER BY USING CFD.

[10] Gao, B., Bi, Q., Nie, Z., & Wu, J. (2015). Experimental study of effects of baffle helix angle on shell-side performance of shell-and-tube heat exchangers with discontinuous helical baffles. *Experimental thermal and fluid Science*, *68*.