International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 1 Issue 8, October - 2012

Comparative Study Of Heat Transfer Rate In Tubular Heat Exchanger

Paresh Patel, Amitesh paul
P.G.Student, Professor

Abstract

The Tubular heat exchangers are generally built of circular tubes, although elliptical, rectangular or round/flat twisted tubes have also been used in some applications. There is considerable flexibility in design because the core geometry can be varied easily by changing the tube diameter, length, and arrangement. Tubular exchangers can be designed for high pressures relative to environment and high-pressure differences between the fluids. Tubular exchangers are used primarily for liquid-to-liquid and liquid to phase change (condensing or evaporating) heat transfer applications. There are used for gas-to liquid and gas-to-gas heat transfer applications primarily when the operating temperature and /or pressure is very high or fouling is a severe problem on at least one fluid side and no other types of exchangers work.CFD analysis has been carried out for different material and on the basis of results made which one give the best heat transfer rates.

1. Introduction

One of the important processes in engineering is the heat exchange between flowing fluids, and many types of heat exchangers are employed in various types of installations, as petro-chemical plants, process industries, pressurized water reactor power plants, nuclear power stations, building heating, ventilating, and air-conditioning and refrigeration systems. As far as construction design is concerned, the tubular or shell and tube type heat exchangers are widely in use. The shell-and-tube heat exchangers are still the most common type in use. They have larger heat transfer surface area-to-volume ratios than the most of common types of heat exchangers, and they are manufactured easily for a large variety of sizes and flow configurations. They can operate at high pressures, and their construction facilitates disassembly for periodic maintenance and cleaning. The shell-and-tube heat exchangers consist of a bundle of tubes enclosed within a cylindrical shell. One fluid flows through the tubes and a second fluid flows within the space between the tubes and the shell. Heat exchangers in general and tubular heat exchangers in particular undergo deterioration in performance due to flow maldistribution. The common idealization in the basic tubular heat exchanger design theory is that the fluid is distributed uniformly at the inlet of the exchanger on each fluid side throughout the core. However, in practice, flow maldistribution is more common and significantly reduces the idealized heat exchanger performance. Flow maldistribution can be induced by the heat exchanger geometry, operating conditions (such as viscosity or density-induced maldistribution), multiphase flow, fouling phenomena, etc. Geometry-induced flow maldistribution can be classified into gross flow misdistribution, passage-to-passage flow maldistribution and manifold-induced flow maldistribution. The flows in shell-and-tube heat exchangers have only been investigated analytically [1, 2, and 3] due to their complexity. Ranjit Kumar Sahoo, Wilfried Roetzel [2] and Chakkrit Na Ranong [1] carried out an analysis of the effect of maldistribution on the thermal performance and the temperature distribution in shell and tube heat exchanger using a finite difference method. Mueller and Chiou [17] summarised various types of flow maldistribution in heat exchangers and discussed the reason leading to flow maldistibution. . Ranganayakulu and Seetharamu [19] carried out an analysis of the effects of inlet fluid flow nonuniformity on the thermal performance and pressure drop in cross flow plate-fin heat exchangers by using a finite element method. Lalot and Florent [4] used the computer code STAR-CD to study the gross flow maldistribution in an electrical heater. They found that reverse flows would occur for the poor header design and the perforated grid can improve the fluid flow distribution. However, few authors studied the fluid flow mal-distribution using the computational fluid dynamics (CFD) simulation technique, especially the effects of the configuration of header and distributor on the flow distribution in plate-fin heat exchangers. CFD simulation technique can provide the flexibility to construct computational models that are easily adapted to a wide variety of physical conditions without constructing a large-scale prototype or expensive test rigs. Therefore, CFD can provide an effective platform where various design options can be tested and an optimal design can be determined at a relatively low cost. Spiral-tube heat exchangers consist of one or more spirally wound coils fitted in a shell or designed as co axial condensers and co-axial evaporators that are used in refrigeration systems. Heat transfer rate associated with a spiral tube is higher than that for a straight tube. In addition, a considerable amount of surface can be accommodated in a given space by spiralling. Thermal expansion is no problem, but cleaning is almost impossible.

2. Results and Discussion

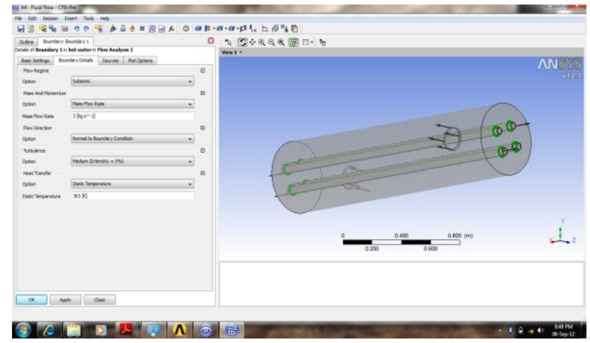
All Set-ups for the Analysis:

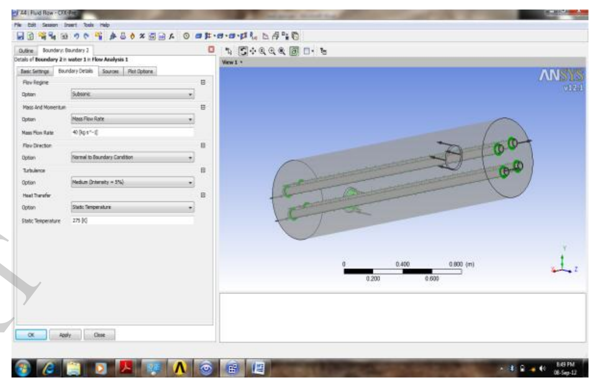
Shell and Tube heat exchanger used in this project analysis, is a single pass four tube and one shell with a parallel flow feature. During the aalysis the tube material changed basically I three types of steel, second is aluminium and third is a copper. By the aalysis the effect of material has bee examined.

3. Set-up ad boundary condition:

	Temperature	Mass flow rate	Pressure at outlet
Hot water in tube	90 c	2 kg/s in each tube	1 atm
Cold water in cell	27 c	40 kg/s in cell	1 atm

In this analysis the two types of water are passed through the shell ad tube type heat exchanger, hot water is passed with the 90 degree C with the 2 Kg/sec each tube mass flow rate and the cold water is passed through the shell with the 27 degree C with the 40 Kg/Sec mass flow rate. Both of the fluid passed through the heat exchanger at the atmospheric pressure.

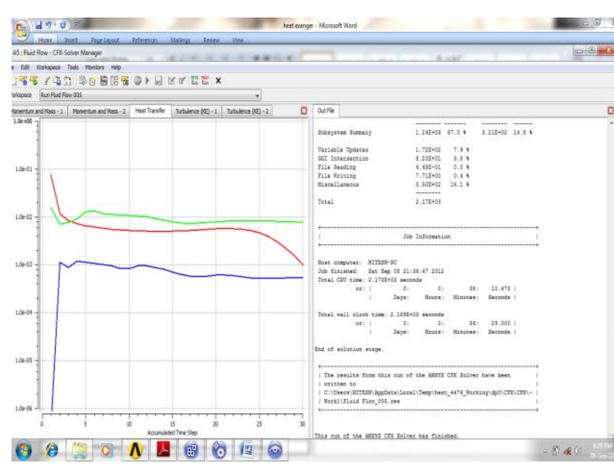




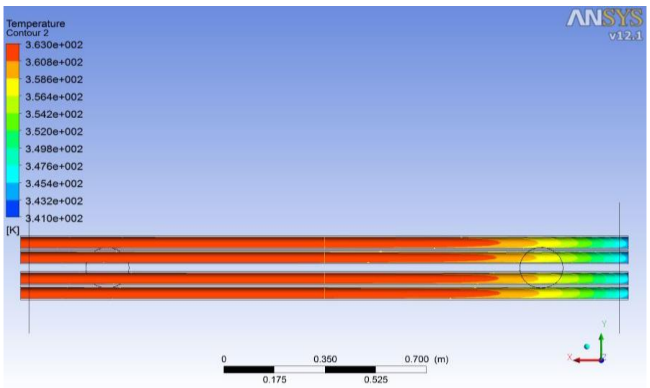
As show in the figure the fluid passed with the parallel flow heat exchanger ad the solution is ru I the ansys CFX.

4. Results of the Aluminium Tube Heat Exchanger:

Above figure is shows the aalysis progress where the set-up run for the results. Above figure shows the heat transfer progress for the hot fluid ad cold fluid.

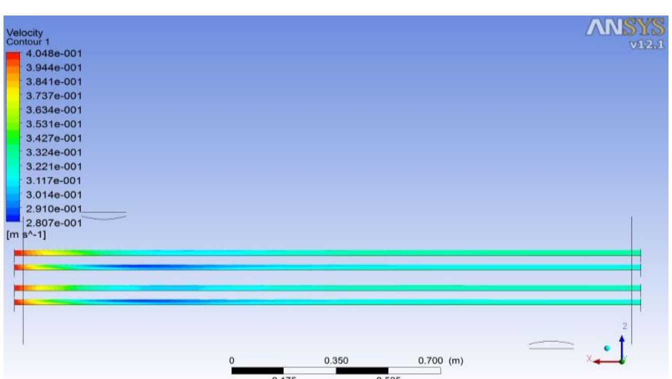


Temperature of cold water:



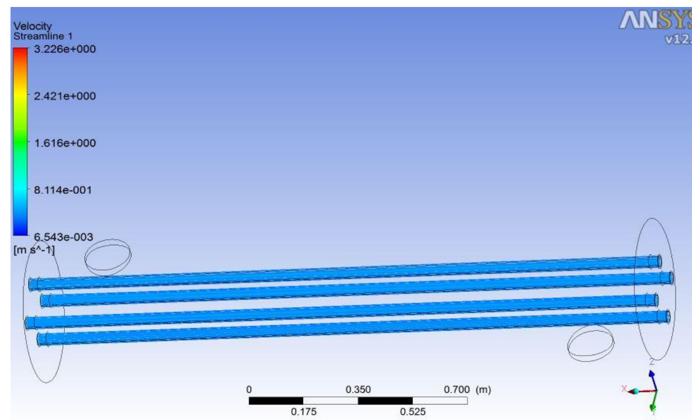
After performing the analysis of heat exchanger, the maximum temperature produced during the analysis is 363 K ad minimum temperature produced is 341 K. minimum temperature is produced at the output of the cold fluid.

Velocity of hot water tube:



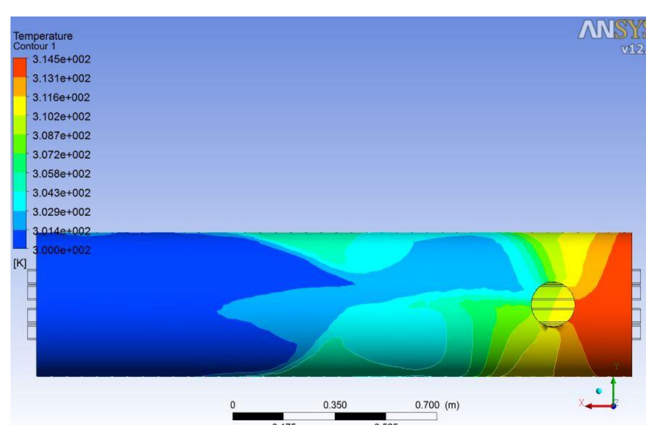
After performing the analysis of heat exchanger, the maximum velocity produced during the analysis is 0.405 m/sec ad minimum velocity produced is 0.2807 m/sec.

Velocity stream line of hot water in Tube



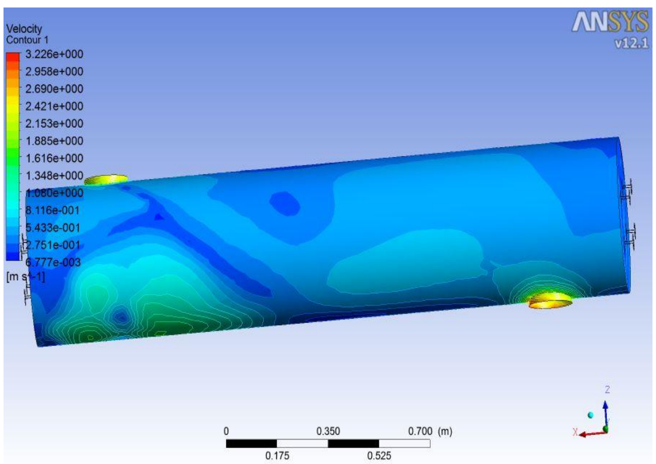
After performing the analysis of heat exchanger, the maximum velocity produced during the analysis is 3.23 m/sec ad minimum velocity produced is 0.0065 m/sec.

Temperature of cold water:



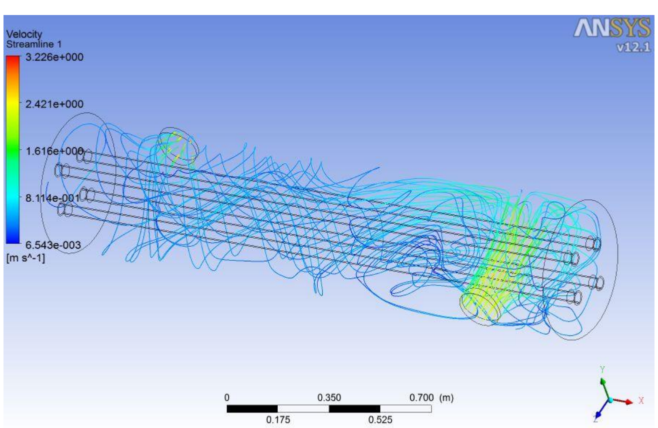
After performing the analysis of heat exchanger, the maximum temperature produced during the analysis is 314.50 K ad minimum temperature produced is 300 K.

Velocity of cold water in shell



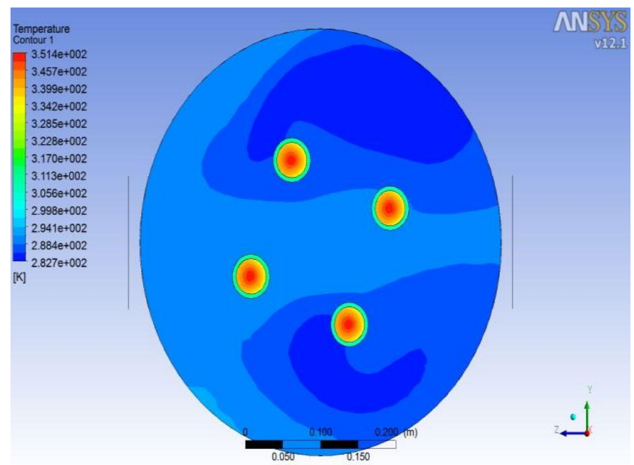
After performing the analysis of heat exchanger, the maximum velocity produced during the analysis is 3.23 m/sec ad minimum velocity produced is 0.0068 m/sec.

Velocity stream lie of cold water in shell



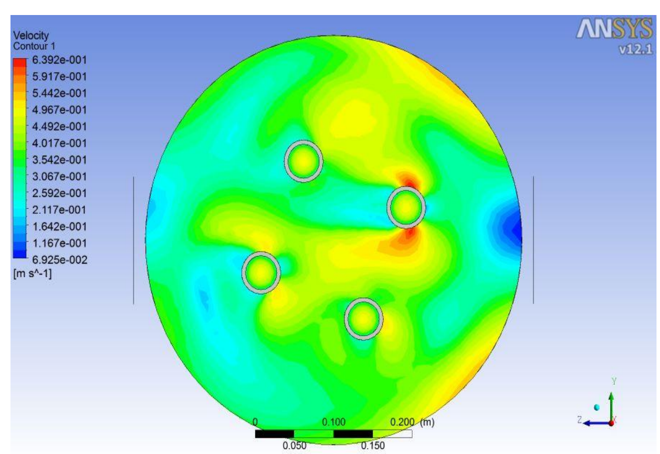
After performing the analysis of heat exchanger, the maximum velocity produced during the analysis is 3.23 m/sec ad minimum velocity produced is 0.0068 m/sec.

Temperature of cold water in shell



After performing the analysis of heat exchanger, the maximum temperature produced during the analysis is 351 K ad minimum temperature produced is 283 K.

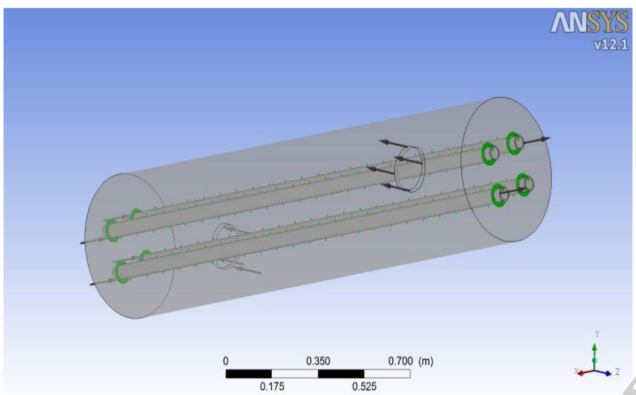
Velocity of cold water in shell



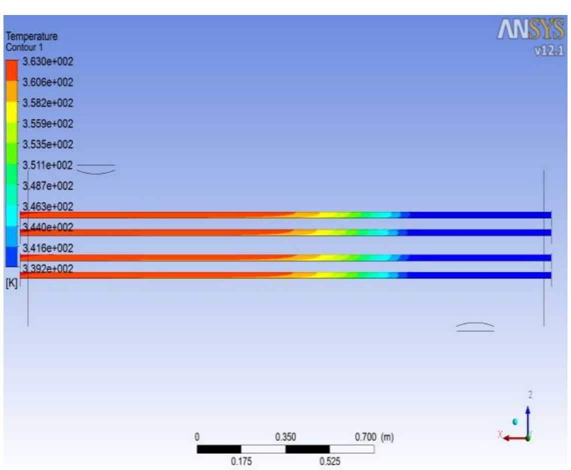
After performing the analysis of heat exchanger, the maximum velocity produced during the analysis is 0.6392 m/sec ad minimum velocity produced is 0.0693 m/sec.

Analysis with the Steel Tube material:

Different types of boundary set-up used for the analysis are generally two types of is domain physics ad another of is a boundary physics which are explained as under:

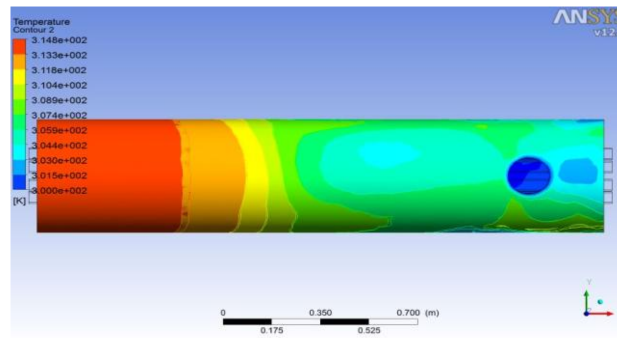


Temperature of hot water I tube:



After performing the analysis of heat exchanger, the maximum temperature produced during the analysis is 363 K ad minimum temperature produced is 339 K.

Temperature of cold water shell:



After performing the analysis of heat exchanger, the maximum temperature produced during the analysis is 316 K ad minimum temperature produced is 300 K.

Conclusion

After performing the all analysis work for the shell ad tube type heat exchanger, following observation we have observed.

	aluminium		steel		
	Hot water	Cold water	Analytical Hot water	Hot water	Cold water
Max temperature	363 k	314.5 k	363 k	363 k	314.8 k
Min Temperature	341 k	300 k	347 k	339 k	300 k

From the study of the result table as mentioned as above. After performing the calculation the fluid water the output temperature is 347 °C which is ear to the value mentioned I output temperature of ansys.

As we change the material from the aluminium to the brass, temperature difference between input temperature and output temperature.

References

[1] Wilfried Roetzel and ChakkritNa Ranong, 1999, "Consideration of maldistribution in heat exchangers using the hyperbolic dispersion model" Chemical Engineering and Processing 38, pp. 675-681.

[2] Sahoo, R.K., and Wilfried Roetzel., 2002, "Hyperbolic axial dispersion model for heat exchangers" International Journal of Heat and Mass Transfer 45, pp 1261-1270.
[3] Wilfried Roetzel , Chakkrit Na Ranong., 2000 "axial dispersion model for heat exchangers" Heat and Technology vol. 18.

[4] Yimin Xuan, and Wilfried Roetzel., "Stationary and dynamic simulation of multipass shell and tube heat exchangers with the dispersion model for both fluids" Int. J. Heat Mass Transfer. Vol. 36, No. 17,4221A231,

[5] Danckwerts, P.V., 1953, Continuous flow systems Distribution of Residence times Chemical Engineering science genie chimique Vol. 2.

[6] Lalot, S., P. Florent, Langc, S.K., Bergles, A.E., 1999, "Flow maldistribution in heat exchangers "Applied Thermal Engineering 19, pp 847-863.

[7] Prabhakara Rao Bobbili, Bengt Sunden, and. Das, S.K., 2006, "An experimental investigation of the port flow maldistribution in small and large plate package heat exchangers Applied Thermal Engineering 26, pp 1919-1926.

[8] Zakro Stevanovic, Gradimir, Ilić., Nenad Radojković, Mića Vukić, Velimir Stefanović, Goran Vučković., 2001, "Design of shell and tube heat exchangers by using CFD technique- part one: thermo hydraulic calculation" Facta Universities Series: Mechanical Engineering Vol.1, No 8, pp. 1091 – 1105

[9] Wilfried roetzel and Das, S.K., 1995 "Hyperbolic axial dispersion model: concept and its application to a plate heat exchanger" Int..J. Heat Mass Transfer. Vol. 38, No. 16, pp.3062-3076.

[10] Anindya Roy, and Das, S.K., 2001, "An analytical solution for a cyclic regenerator in the warm-up period in presence of an axially dispersive wave" Int. J. Therm. Sci. 40, pp.21-29

[11] Ping Yuan., 2003 "Effect of inlet flow maldistribution on the thermal performance of a three-fluid crossflow heat exchanger International" Journal of Heat and Mass Transfer 46, pp.3777-3787.

[12] Srihari, N., Prabhakara Rao, B., Bengt Sunden, Das, S.K., 2005 "Transient response of plate heat exchangers considering effect of flow maldistribution" International Journal of Heat and Mass Transfer 48, pp. 3231-3243.