CFD Analysis of Shell and Tube Heat Exchangers –A review

Dilip S Patel[1], Ravindrasinh R Parmar[2], Vipul M Prajapati[3]

[1] Associate Prof., Department of Mechanical Engineering S.K.Patel college of Engg. Visnagar, Gujarat, India
[2] Student of ME, Department of Mechanical Engineering S.K.Patel college of Engg. Visnagar, Gujarat, India
[3] Assistant Prof., Department of Mechanical Engineering S.K.Patel college of Engg. Visnagar, Gujarat, India

---***---

Abstract - ***This review focuses on the various researches on CFD analysis in the field of heat exchanger. Shell and tube heat exchanger is an indirect contact type heat exchange as it consists of a series of tubes, through which one of the fluids runs. They are widely used in power plant, chemical plants, petro-chemical plants and automotive applications. Different turbulence models available in general purpose commercial CFD tools likes k-ε model, K-ω model and K-ω SST models. Different CFD code available like CFX, FLUENT. The quality of the solution has proved that CFD is effective to predict the behavior and performance of a wide variety of heat exchanger.***

Key Words: *Shell and tube heat exchanger, CFD, turbulence model, computational modeling, Fluent.*

1.INTRODUCTION

Heat exchangers are devices used to transfer heat energy from one fluid to another. Typical heat exchangers experienced by us in our daily lives include condensers and evaporators used in air conditioning units and refrigerators. Boilers and condensers in thermal power plants are examples of large industrial heat exchangers. There are heat exchangers in our automobiles in the form of radiators and oil coolers. Heat exchangers are also abundant in chemical and process industries.
There is a wide variety of heat exchangers for diverse kinds of uses, hence the construction also would differ widely. However, in spite of the variety, most heat exchangers can be classified into some common types based on some fundamental design concepts. We will consider only the more common types here for discussing some analysis and design methodologies.

2. SHELL AND TUBE HEAT EXCHANGER

A shell and tube heat exchanger is a class of heat exchanger designs. It is the most common type of heat exchanger in oil refineries and other large chemical processes, and is suited for higher-pressure applications. As its name implies, this type of heat exchanger consists of a shell (a large pressure vessel) with a bundle of tubes inside it. One fluid runs through the tubes, and another fluid flows over the tubes (through the shell) to transfer heat between the two fluids. The set of tubes is called a tube bundle, and may be composed of several types of tubes: plain, longitudinally finned, etc.
Shell and tube heat exchanger design is based on correlations between the Kern method and Bell-Delaware method
-In Bell's method the heat-transfer coefficient and pressure drop are estimated from correlations for flow over ideal tube-banks, and the effects of leakage, bypassing and flow in the window zone are allowed for by applying correction factors. This approach will give more satisfactory predictions of the heat-transfer coefficient and pressure drop than Kern's method; and, as it takes into account the effects of leakage and bypassing, can be used to investigate the effects of constructional tolerances and the use of sealing strips. Bell-Delaware method is more accurate method and can provide detailed results

In Kern's method-is based on experimental work on commercial exchangers with standard tolerances and will give a reasonably satisfactory prediction of the heat-transfer coefficient for standard designs. The prediction of pressure drop is less satisfactory, as pressure drop is more affected by leakage and bypassing than heat transfer. The shell-side heat transfer and friction factors are correlated in a similar manner to those for tube-side flow by using a hypothetical shell velocity and shell diameter.

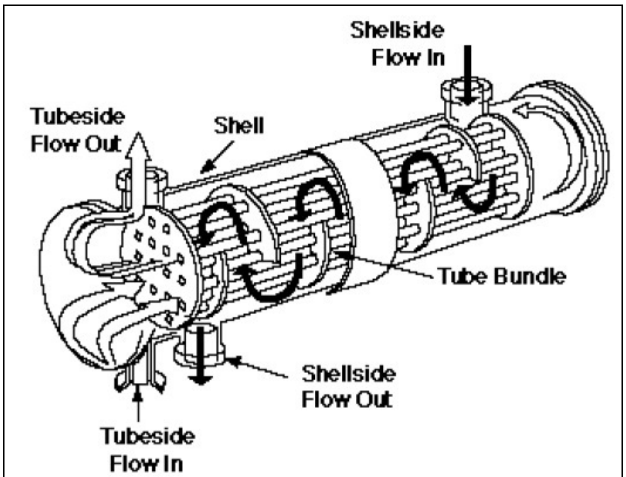

Fig.1: Shell and tube heat exchanger

3.COMPUTATIONAL FLUID DYNAMICS

CFD is useful for studying fluid flow, heat transfer; chemical reactions etc by solving mathematical equations with the help of numerical analysis.CFD resolve the entire system in small cells and apply governing equations on these discrete elements to find numerical solutions regarding pressure distribution, temperature gradients. This software can also build a virtual prototype of the system or device before can be apply to real-world physics to the model, and the software will provide with images and data, which predict the performance of that design. More recently the methods have been applied to the design of internal combustion engine, combustion chambers of gas turbine and furnaces, also fluid flows and heat transfer in heat exchanger. The development in the CFD field provides a capability comparable to other Computer Aided Engineering (CAE) tools such as stress analysis codes. Basic Approach to using CFD

4. TURBULENCE MODELS

Turbulence arises due to the instability in the flow. Turbulent flows contain a wide range of length, velocity and time scales and solving all of them makes the costs of simulations large. Therefore, several turbulence models have been developed with different degrees of resolution. There are several turbulence models available in CFD-software including the Large Eddy Simulation (LES) and Reynolds Average Navier- Stokes (RANS). There are several RANS models available depending on the characteristic of flow, e.g., Standard k-ε model, k- ε RNG model, Realizable k- ε, k-ω and RSM (Reynolds Stress Model) models.

5. LITERATURE SURVEY

Muhammad Mahmood Aslam Bhutta et al.[1] focuses on the applications of Computational Fluid Dynamics (CFD) in the field of heat exchangers. It has been found that CFD employed for the fluid flow mal-distribution, fouling, pressure drop and thermal analysis in the design and optimization phase. Different turbulence models such as standard, realizable and RNG, k – ε, RSM, and SST k - ε with velocity-pressure coupling schemes such as SIMPLE, SIMPLEC, PISO and etc. have been adopted to carry out the simulations. Conventional methods used for the design and development of Heat Exchangers are expensive. CFD provides cost effective alternative, speedy solution and eliminate the need of prototype, it is limited to Plate, Shell and Tube, Vertical Mantle, Compact and Printed Circuit Board Exchangers but also flexible enough to predict the fluid flow behavior to complete heat exchanger design and optimization involving a wide range of turbulence models and integrating schemes the k - ε turbulence model is most widely employed design and optimization .The simulations results ranging from 2% to 10% with the experimental studies. In some exceptional cases, it varies to 36%.

Qiuwang Wang et al.[2] has investigated a combined multiple shell-pass shell-and-tube heat exchanger (CMSP-STHX) with continuous helical baffles in outer shell pass has been invented to improve the heat transfer performance and simplify the manufacture process. The CMSP-STHX is compared with the conventional shell-and-tube heat exchanger with segmental baffles (SG-STHX) by means of computational fluid dynamics (CFD) method. The numerical results show that, under the same mass flow rate M and overall heat transfer rate Q_m, the average overall pressure drop ΔP_m of the CMSP-STHX is lower than that of conventional SG-STHX by 13% on average. Under the same overall pressure drop ΔP_m in the shell side, the overall heat transfer rate Q_m of the CMSP-STHX is nearly 5.6% higher than that of SG-STHX and the mass flow rate in the CMSP-STHX is about 6.6% higher than that in the SG-STHX. The CMSP-STHX might be used to replace the SG-STHX in industrial applications to save energy, reduce cost and prolong the service life.

K.Mohammadi et al.[3] has studied in the vertical baffle orientation seems more desirable in the intermediate baffle spacing zones particularly for low viscous shell fluids. Same trend seems to apply for highly viscous shell fluids at low Reynolds number. The baffle orientation has a significant influence on the shell side pressure drop and heat transfer of shall and tube heat exchanger. The advantage of the horizontal baffle orientation over the vertical has been found in the inlet and outlet zone of heat exchanger for all investigated Prandtl numbers. Simulation results for the inlet region show that the horizontal baffle orientation produces up to 20% higher pressure drop than the pressure drop in vertical baffle orientation. The result also show that the Nusselt number for horizontal baffle orientation is approximately 15% to 52% higher than the nusselt number vertical orientation.

Hari Haran et al.[4] has compared result of C and ANSYS and getting an error of 0.0274in effectiveness. By using ANSYS process thermal analysis in less time and our analysis report also almost accurate. In this paper using theoretical formulae design a model of a shell and tube heat exchanger using Pro-e and done the thermal analysis by using ANSYS software and comparing the result that obtained from ANSYS software and theoretical formulae. Simplification of theoretical calculation use C code for calculating the thermal analysis of a counter flow of shall and tube heat exchanger.

Huadong Li et al.[5] has investigated local heat transfer and pressure drop for different baffle spacing in the shell and tube heat exchangers with segmental baffles. The distributions of the local heat transfer coefficients on each tube surface were determined and visualized by means of mass transfer measurements. The determination of the shell-side flow distributions are allowed by the local pressure measurements. For same Reynolds number, the pressure drop and average heat transfer are increased by an increased baffle spacing which can increase the heat transfer coefficient in the whole baffle compartment due to the reduction of the percentage of the leakage stream and due to the higher flow velocity through the baffle opening and the local heat transfer coefficient distribution for individual tube is slightly affected by the baffle spacing.

Ender Ozden et al.[6] has worked on the design of shell and tube heat exchanger by numerically modeling in particular the baffle spacing, baffle cut and shell diameter dependencies of heat transfer coefficient and pressure drop. The flow and temperature fields are resolved by using a commercial CFD package and it is performed for a single shell and single tube pass heat exchanger with a variable number of baffles and turbulent flow. The best turbulent model among the one is selected to compare with the CFD results of heat transfer coefficient, outlet temperature and pressure drop with the Bell-Delaware method result. By varying flow rate the effect of the baffle spacing to shell diameter ratio on the heat exchanger performance for two baffle cut value is investigated. Three turbulence models are taken for the first and second order discretizations to mesh density. By comparing with the Bell-Delaware results the k-ε realizable turbulence model is selected as the best simulation approach. By varying baffle spacing between 6 to 12,and the baffle cut values of 36% and 25% for 0.5 and 2 kg/s flow rate, the simulation results are compared with the results from the kern and Bell-Delaware methods. It is observed that the CFD simulation results are very good with the Bell-Delaware methods and the differences between Bell-Delaware method and CFD simulations results of total heat transfer rate are below 2% for most of the cases.

Apu Roy, D.H.Das [7] has carried out with a view to predicting the performance of a shell and finned tube heat exchanger in the light of waste heat recovery application. Energy available in the exit stream of many energy conversion devices such as I.C engine gas turbine etc goes as waste, if not utilized properly. The performance of the heat exchanger has been evaluated by using the CFD package fluent 6.3.16 and the available values are compared with experimental values. By considering different heat transfer fluids the performance of the above heat exchanger can also be predict. The performance parameters of heat exchanger such as effectiveness, overall heat transfer coefficient, energy extraction rate etc, have been taken in this work.

D.P.Naik et al.[8] did an assessment of counter flow shell and tube heat exchanger by entropy generation minimization method. The design variables which are used for the shell and tube heat exchanger are tube inside diameter, tube outside diameter, number of tubes, baffle spacing and tube pitch etc. The analyses of these design parameters are very important for the better performance of shell and tube heat exchanger. Shell and tube heat exchanger performance has improved significantly by minimization of entropy generation number considering the various design variables. As the mass flow rate of shell side fluid increases, the entropy generation number increases. Therefore we can reduce the entropy generation number by reducing the mass flow rate of cold fluid by optimization. If we change tube side area heat exchanger effectiveness also change.

Hamidou Benzenine et al.[9] has investigated numerically to study a turbulent flow of air through a rectangular section. Two baffles were introduced into the field to produce vortices, to improve the mixture and thus, the transfer of heat. The numerical results obtained by the finite volume method, are validated and presented to analyze the dynamic behavior of a turbulent flow using the low Reynolds number model. The highest disturbance is obtained upstream second baffle. This study showed that the undulation of the baffles induced with an improvement on the skin friction of about 9.91 % in the case of α=15°, more than 16% in the other cases. Concerning the pressure loss the undulation of the baffles was insured improvements starter from 10, 43% in all cases compared with the baffles of plane form. The investigation was carried for four cases of slopes for the corrugated baffles going from 0° up to 45°, with a step equal to 15°. It may be concluded that the purely vertical use of the waved baffles (α=0) in the geometry studied, ensures the optimal size of the zone of recirculation and thus necessary time for guarantee the improvement of heat exchange. Also this case ensures us a very high velocity in the exit of the channel, measures more than four times the reference velocity, and most significant is to reduce less the action flow induces on the pressure losses.

Kevin M. Lunsford et al.[10] has analyzed to increase the heat exchanger performance and suggested increasing heat exchanger performance through a logical series of steps. The first step considers if the exchanger is initially operating correctly. The second step considers increasing pressure drop if available in exchangers with single-phase heat transfer. Increased velocity results in higher heat transfer coefficients, which may be sufficient to improve performance. Next, a critical evaluation of the estimated fouling factors should be considered. Heat exchanger performance can be increased with periodic cleaning and less conservative fouling factors. Finally, for certain conditions, it may be feasible to consider enhanced heat transfer through the use of finned tubes, inserts, twisted tubes, or modified baffles.

A.E. Zohir [11] The analysis of the Heat transfer characteristics in a heat exchanger for turbulent pulsating water flow with different amplitudes has been carried out. The effect of pulsation on the heat transfer rates, for turbulent water stream with upstream pulsation of different amplitudes, in a double- pipe heat exchanger for both parallel and counter flows, with cold water on the shell side, was investigated. The heat transfer coefficient was found to increase with pulsation, with the highest enhancement observed in the transition flow regime. The heat transfer coefficient was strongly affected with pulsation frequency, amplitude and Reynolds number. In the counter flow, the enhancements in heat transfer rates are somewhat greater than that in the parallel flow. The heat transfer coefficient was found to increase with pulsation, with the highest enhancement observed in the transition flow regime. The results showed that an enhancement in relative average Nusselt number of counter flow up to 10 times was obtained for higher amplitude and higher pulsation frequencies. While, an enhancement in relative average Nusselt number of parallel flow up to 8 times was obtained for higher amplitude and higher pulsation frequency. The maximum enhancements in the heat transfer rates were obtained at Reynolds number of 3855 and 11570

Simin Wang et al. [12] has investigated that the shell-and-tube heat exchanger was improved through the installation of sealers in the shell-side. They are cheap, firm and convenient to install. Sealers effectively decreases the short-circuit flow in the shell-side and decrease the circular leakage flow. The original short-circuit flow then participates in heat transfer, which intensifies the heat transfer performance inside the heat exchanger. The results of heat transfer experiments show that the shell-side heat transfer coefficient of the improved heat exchanger increased by 18.2–25.5%, the overall coefficient of heat transfer increased by 15.6–19.7%, and the energy efficiency increased by 12.9–14.1%. Pressure losses increased by 44.6–48.8% with the sealer installation, the energy utilization improves, which is of significance of the optimum design to the shell-and-tube heat exchanger. The sealers are a solution settling the puzzle of the effect of baffle-shell leakage flow in tube-and-shell heat exchangers. The heat transfer performance of the improved heat exchanger is increased, which is a benefit for optimizing of heat exchanger design.

J.S. Jayakumar et al.[13] has established that heat transfer in a helical coil is higher than that in a corresponding straight pipe. However, the detailed characteristics of fluid flow and heat transfer inside helical coil is not available from the present literature. This paper brings out clearly the variation of local Nusselt number along the length and circumference at the wall of a helical pipe. Movement of fluid particles in a helical pipe has been traced. CFD simulations are carried out for vertically oriented helical coils by varying coil parameters such as (i) pitch circle diameter, (ii) tube pitch and (iii) pipe diameter and their influence on heat transfer has been studied. After establishing influence of these parameters, correlations for prediction of Nusselt number has been developed. A correlation to predict the local values of Nusselt number as a function of angular location of the point is also presented.

6.CONCLUSIONS

CFD provides cost effective alternative, speedy solution and eliminate the need of prototype. The literature review focus on the analysis of various parameters which influence on the performance of the STHE. It has been observed that computational modeling is one of the efficient techniques to study these type of heat elements. The parameters like tube and shell diameter, number of tubes, pitch and baffle angles are the important one to be worked upon. A detailed analysis using the CFD simulation will be worthy to be carried out. An heat exchanger used in the KLTPS Pandro power plant has been taken for the further study in the proposed research work.

REFERENCES

1. Muhammad Mahmood Aslam Bhutta, Nasir Hayat, Muhammad Hassan Bashir, Ahmer Rais Khan, Kanwar Naveed Ahmad, Sarfaraz Khan5, CFD Applications In Various Heat Exchangers Design: A Review, Department Of Mechanical Engineering, University Of Engineering & Technology, Applied Thermal Engineering, 2011.

2. Qiuwang Wang, Qiuyang Chen, Guidong Chen, Min Zeng,"Numerical investigation on combined multiple shell-pass shell-and-tube heat exchanger with continuous helical baffles",International Journal of Heat and Mass Transfer 52 (2009).

3. K.Mohammadi, W.Heidemann, H.Muller – Steinhagen “Numerical Investigation Of The Effect Of Baffel Orientation And Baffle Cut On Heat Transfer And Pressure Drop Of A Shall And Tube Heat Exchanger” ResearchGate , august 2006.

4. Hari Haran , Ravindra Reddy and Sreehari “ Thermal Analysis of Shall and Tube Heat Exchanger using C and Ansys” International Journal of Computer Trends and Technology-volume 4 issue 7 –July 2013.

5. Huadong Li, Volker Kottke ,” Effect Of Baffle Spacing On Pressure Drop And Local Heat Transfer In Shell-And-Tube Heat Exchangers For

Staggered Tube Arrangement", International Journal of Heat Mass Transfer, Elsevier Science , Germany, 1998.

6. Ender Ozden, Ilker Tari ,"Shell Side CFD Analysis of A Small Shell And Tube Heat Exchanger" ,Middle East Technical University, 2010.

7. Apu Roy, D.H.Das, CFD Analysis Of A Shell And Finned Tube Heat Exchanger For Waste Heat Recovery Applications, National Institute Of Technology, 2011.

8. D.P. Naik and V.K. Matawala, "An assessment of counter flow shell and tube heat exchanger by entropy generation minimization method", World Journal of Science Technology, Vol. No. 2, pp. 28-32, 2012.

9. Hamidou Benzenine, Rachid Saim , Said Abboudi and Omar Imine, "Numerical analysis of a turbulent flow in a channel provided with transversal waved baffles",International Journal of thermal science, Vol. No. 17, pp. 1491-1499, 2013.

10. K.M. Lunsford, "Increasing Heat Exchanger Performance", Bryan Research and Engineering, Inc., Vol. No.1, pp. 1-13, March 1998.

11. A. E. Zohir, "Heat Transfer Characteristics in a Heat Exchanger for Turbulent Pulsating Water Flow with Different Amplitudes", Journal of American Science, Vol. No. 8, pp. 241-250, 2012.

12. Simin Wang, Jian Wen and Yanzhong Li, "An experimental investigation of heat transfer enhancement for a shell-and-tube heat exchanger", Applied Thermal Engineering, Vol. No. 29, pp. 2433-2438, 2009.

13. J.S. Jayakumara, S.M. Mahajania, J.C. Mandala, Kannan N. Iyer, P.K. Vijayan, "CFD analysis of single-phase flows inside helically coiled tubes" ,Computers and Chemical Engineering 34 (2010).