ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 2, Issue 2, March 2013

CFD Analysis of Shell and Tube Heat Exchanger- A Review

[1]Hetal Kotwal, [2]D.S PATEL

Abstract-This review focuses on the various researches on CFD analysis in the field of heat exchanger. Different tur ulence models available in CFD tools i.e. Standard k-ε model, k- ε RNG model, Realizable k- ε, k-ω and RSM model in conjunction with velocity pressure coupling scheme and have been adopted to carry out the simulation. The steady increase in computing power has enable model to react for multi phase flows in realistic geometry with good resolution. The quality of the solution has proved that CFD is effective to predict the behavior and performance of heat exchanger.

Keywords; **STHE-Shell and Tube Heat Exchanger, CFD- computational fluid dynamics, CAE- Computer Aided Engineering, turbulence model**.

I. INTRODUCTION

A heat exchanger transfers energy from one fluid to another across a solid surface by convection and conduction.[1] Heat exchangers are used in power plants, nuclear reactors, refrigeration and air conditioning systems, automotive industries, heat recovery systems, chemical processing, and food industries.[3] The design of a new heat exchanger (HE) is referred to the (i) sizing problem, means it includes construction type, flow arrangement, tube and shell material, and physical size which has to meet the specified heat transfer and pressure drop.(ii) rating of existing heat exchanger.[1,4] Shell and tube heat exchanger design is based on correlations between the Kern method and Bell-Delaware method.[4,5] Kern method used for calculating shell-side pressure drop and heat transfer coefficient. And is restricted to a fixed baffle cut (25%) and cannot adequate for baffle-to-shell and tube to-baffle leakage. Kern method is not applicable in laminar flow region where shell-side Reynolds number is less than 2000. [4] Bell-Delaware method is more accurate method and can provide detailed results. It can predict and estimate pressure drop and heat transfer coefficient with better accuracy. The Bell-Delaware method is actually the rating method and it can suggest the weaknesses in the shell side deign but it cannot indicate where these weaknesses are. [5] The optimization of shell-and-tube heat exchangers requires a good knowledge of the local and average shell-side heat transfer coefficients which is complicated by a shell diameter, baffle cut, baffle spacing, tube diameter, pitch, arrangement and clearances or leakage paths. These leakages reduce the velocity in the tube bundle and, hence, the heat transfer coefficient and pressure drop. Recently, several models have been applied to analyze flow and thermal transfer but these models can only provide the integral heat transfer co-efficient. [6]

II. SHELL AND TUBE HEAT EXCHANGER

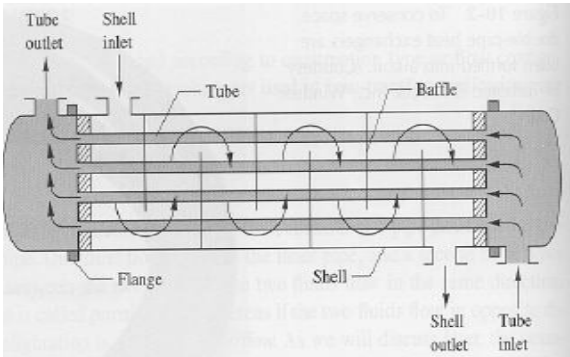

Fig.1 Shell and tube heat exchanger

Shell and tube heat exchanger consist bundle of tubes enclosed with in cylindrical shell one fluid pass through the tubes and second fluid flows between the tube and shells. The basic components of a shell and tube heat exchangers are tubes, tube sheets, shell and shell-side nozzles, tube side channels and nozzles, channel covers, pass divider, baffles etc .Most commonly used STHE have large heat transfer surface area-to-volume ratios to provide high heat

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 2, Issue 2, March 2013
transfer efficiency in comparison with others. They are mechanically rugged enough to withstand the fabrication stresses and normal operating conditions. They can be easily cleaned and the failure parts like gaskets and tubes can be easily replaced. They offer greater flexibility of mechanical features to withstand any service requirement. They are manufactured easily for a large variety of sizes and flow configurations. They can operate at high pressures and high temperature. [2] They can be employed for processes which require large quantities of fluid to be heated or cooled. [1]

III. 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. [9] 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. [11] Basic Approach to using CFD

a) Pre-processor: Establishing the model

- Identify the process or equipment to be evaluated.
- Represent the geometry of interest using CAD tools.
- Use the CAD representation to create a volume flow domain around the equipment containing the critical flow phenomena.
- Create a computational mesh in the flow domain.

b) Solver:

- Identify and apply conditions at the domain boundary.
- Solve the governing equations on the computational mesh using analysis software.

c) Post processor: Interpreting the results

- Post-process the completed solutions to highlight findings.
- Interpret the prediction to determine design iterations or possible solutions, if needed.

IV. CHOOSING A TURBULENCE MODEL

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]

V. LITERATURE REVIEW

M. Thirumarimurugan, T.Kannadasan and E.Ramasamy [1] have investigated heat transfer study on a solvent and solution by using Shell and Tube Heat Exchanger. In which Steam is taken as the hot fluid and Water and acetic acid-Water miscible solution taken as cold fluid. A series of runs were made between steam and water, steam and Acetic acid solution .The flow rate of the cold fluid is maintained from 120 to 720 lph and the volume fraction of Acetic acid is varied from 10-50%. Experimental results such as exchanger effectiveness, overall heat transfer coefficients were calculated. . MATLAB program was used to simulate a mathematical model for the outlet temperatures of both the Shell and Tube side fluids. The effect of different cold side flow rates and different compositions of cold fluid on the shell outlet temperature, tube outlet temperature and overall heat transfer coefficients were studied. Result shows that the overall effectiveness of heat exchanger was found to increase with decrease in composition of water. From the comparisons it can be said that the mathematical model developed and simulated using MATLAB and compared with the experimental values for the system is very close.**Usman Ur Rehman** [5] had investigated an un-baffled shell-and-tube heat exchanger design with respect to heat transfer coefficient and pressure drop by numerically modeling. The heat exchanger contained 19 tubes inside a 5.85m long and 108mm diameter shell. The flow and temperature fields are resolved using a commercial CFD package and it is performed for a single shell and tube bundle and is compared with the experimental results. Standard k- ε model

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 2, Issue 2, March 2013
is used first to get the flow distribution but it is not good for predicting the boundary layer separation and impinging flows. For this reason, Realizable k – ε model is used with standard and then Non-equilibrium wall functions. The non-equilibrium wall functions with Realizable k- ε model give better results than standard k- ε model. The pressure drop heat transfer still are being over predicted by almost 25%, which is probably due to y^+ values limitations at tube walls .Thus in order to avoid this and to include the low Reynolds modification SST k-ω model is also used. Because it uses both k - ε and k- ω model in the region of high and low Reynolds number respectively. SST k- ω model has provided the reliable results with the y+ limitations. Thus the modeling can also be improved by using Reynolds Stress Models, but with higher computational costs and the enhanced wall functions are not used.
The heat transfer is found to be poor because the most of the shell side fluid by-passes the tube bundle without interaction. Thus the design can be modified to achieve the better heat transfer in two ways. Either, the shell diameter is reduced or tube spacing can be increased .Thus the design can further be improved by creating cross-flow regions in such a way that flow doesn't remain parallel to the tubes. It will allow the outer shell fluid to mix with the inner shell fluid and will automatically increase the heat transfer.
Jian-Fei Zhang, Ya-Ling He, Wen-Quan Tao [7] developed a method for design and rating of shell-and-tube heat exchanger with helical baffles based on the public literatures and the widely used Bell–Delaware method for shell-and-tube heat exchanger with segmental baffles (STHXSB). The accuracy of present method is validated with experimental data. Four design cases of replacing original STHXsSB by STHXsHB are taken. In case 1 comprehensive performance is greatly improved by using tube-core with 40 degree middle overlapped helical baffles, and the pressure drop is 39% lower and 16% decrease in heat transfer area. In case 2 the usage of tube-core with 40 deg middle overlapped helical baffles can reduce the over-all pressure drop by 46% and the heat transfer area is 13% lower. In case 3 pressure drop of the heat exchanger with 40 deg middle-overlapped helical baffles is equivalent, the heat transfer area reduced by 33%. In case 4 20 deg middle-overlapped helical baffles were adopted and the pressure drop is 33% lower than that of the original unit with 10% decrease in heat transfer area. And comparison result shows that all shell and tube heat exchanger with helical baffles have better performance than the original heat exchanger with segmental baffles.
Muhammad Mahmood Aslam Bhutta, Nasir Hayat, Muhammad Hassan Bashir, Ahmer Rais Khan,Kanwar Naveed Ahmad, Sarfaraz Khani[9] , It 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%.
Žarko Stevanović, Gradimir Ilić, Nenad Radojković, Mića Vukić, Velimir Stefanović, Goran Vučković [12] has developed an iterative procedure for sizing shell-and-tube heat Exchangers according to given pressure drop and the thermo-hydraulic calculation and the geometric optimization on the basis of CFD technique have been carried out. A numerical study of three-dimensional fluid flow and heat transfer is described. The baffle and tube bundle was modeled by the 'porous media' concept. Three turbulent models were used for the flow processes. The velocity and temperature distributions and total heat transfer rate were calculated by using PHOENICS Version 3.3 code. Due to the effects of eddy-viscosity the effect of different turbulence models on both flow and heat transfer is significant. It was concluded that Chen-Kim modification of the standard k − ε turbulence model give the best agreement to the experimental data. The optimization of flow distribution is an essential step in heat exchanger design optimization because an optimal flow distribution can result in a higher heat transfer rate and lower pressure drop.
Ender Ozden, Ilker Tari [13] Has investigated 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 2kg/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 [14] the present work has been 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.

VI. CONCLUSIONS & FUTURE SCOPE OF WORK

Conventional methods used for the design and development of Heat Exchangers are expensive. CFD provide alternative to cost effectiveness speedy solution to heat exchanger design and optimization.CFD results are the integral part of the design process and it have eliminated the need of prototype .due to the development of CFD models, the use of CFD is no longer a specialist activity. It is accessible to process engineers, plant operator and manager. Further study needs to be carried out for performance optimization of shell and tube heat exchanger by varying tube & shell diameter, no. of tubes, pitch and baffle angles.CFD is still a developing art in prediction of erosion/ corrosion due to lack of suitable mathematical models to represent physical process. New flow modeling strategies can be developed for flow simulation in shell and tube heat exchanger.

REFERENCES

[1] M. Thirumarimurugan, T.Kannadasan and E.Ramasamy, Performance Analysis Of Shell And Tube Heat Exchanger Using Miscible System, American Journal Of Applied Sciences 5 (5): 548-552, 2008.

[2] K. Sudhakara Rao, Analysis Of Flow Mal distribution In Tubular Heat Exchangers By Fluent, National Institute Of Technology Rourkela ,2007.

[3] M.R. Salimpour, Heat Transfer Coefficients Of Shell And Coiled Tube Heat Exchangers, Isfahan University Of Technology, Iran, 2008.

[4] Yusuf Ali Kara, Ozbilen Gurarasa, A Computer Program For Designing Of Shell-And-Tube Heat Exchangers, Applied Thermal Engineering University Of Ataturk, Turkey, 2004.

[5] Usman Ur Rehman, Heat Transfer Optimization of Shell-And-Tube Heat Exchanger through CFD Studies, Chalmers University of Technology, 2011.

[6] Huadong Li And Volker Kottke, Effect Of The Leakage On Pressure Drop And Local Heat Transfer In Shell-And-Tube Heat Exchangers For Staggered Tube Arrangement, Inl. J. Heat Mass Transfer, Elsevier Science Ltd., 1998.

[7] Jian-Fei Zhang, Ya-Ling He, Wen-Quan Tao, A Design And Rating Method For Shell-And-Tube Heat Exchangers With Helical Baffles, Journal Of Heat Transfer, May 2010.

[8] Huadong Li And Volker Kottke, Effect Of Baffle Spacing On Pressure Drop And Local Heat Transfer In Shell-And-Tube Heat Exchangers For Staggered Tube Arrangement, Inl. J. Heat Mass Transfer, Elsevier Science Ltd., 1998.

[9] 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.

[10] Philip J Stop ford, Recent Application of CFD Modeling in Power Generation Metal and Process Industries, Second International Conference on CFD in Mineral and Process Industries, CSIRO, Melbourne, Australia, 1999.

[11] Khairun Hasmadi Othman, CFD Simulation Of Heat Transfer In Shell And Tube Heat Exchanger, University Malaysia Pahang, April 2009.

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 2, Issue 2, March 2013
[12] Žarko Stevanović, Gradimir Ilić, Design Of Shell-And-Tube Heat Exchangers By Using CFD Technique, University Of Niš, Fr, 2002.

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

[14] 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.