Literature Review on Shell and Tube Heat Exchanger's Performance Prediction and Optimization Methods

Gajanan P Nagre[1], A.V.Gadekar[2]
Department of Mechanical Engineering
[1]Student of post graduate,MIT (E) Engineering, Aurangabad (MH)
[2]Professor, MIT (E) Engineering, Aurangabad (MH)

Abstract*-Shell and tube heat exchangers find wide variety of applications ranging from boilers, power plants, nuclear industry, waste heat recovery, household appliances etc. There have been various research in the field of design, development, performance prediction as well as optimization of shell and Tube heat exchangers with the help of Computational Fluid Dynamics (CFD) simulations, Analytical Methods and Experimental Procedure. In this journal, a comprehensive report on the recent research work on the shell and tube heat exchanger had been discussed.*

Keywords:Shell and Tube Heat Exchanger, CFD Modeling, Segmental Baffles, Helical Baffles

I. INTRODUCTION

Heat exchangers are simple devices that enable the heat transfer mechanism between high temperature fluids to low temperature fluids without those fluids are in direct contact. There have been various types of heat exchangers such as shell-and-tube type, tube-in-tube types, plate heat exchangers etc. Shell and tube heat exchangers find wide variety of applications ranging from boilers, power plants, nuclear industry, waste heat recovery, household appliances etc. The shell and tube heat exchangers contain shell, baffles and tubes in their assembly.

This journal summarizes the recent research on shell and tube heat exchanger design methods, experimental procedures for performance evaluations, CFD modeling approach, performance optimization techniques etc.

II. LITERATURE SURVEY

Ender Ozden [1] had conducted CFD simulations for a small heat exchangers by varying baffle spacing, baffle cut. For their study, the authors had investigated the influence of turbulence models on the prediction of heat exchanger performance by comparing the results from Standard k-ϵ, Realizable k-ϵ, Spalart-Allmaras models for the First-Order as well as Second-Order numerical discretization schemes. They concluded that the Realizable k-ϵ turbulence model with First-Order numerical discretization scheme had produced results – as compared with Bell-Delaware analytical calculations - with better accuracy for the considered operating conditions. Among the two baffle cut configurations studied by the authors, 25% baffle cut produced lesser pressure drop as compared to the 36%.

Eshita Pal [2] had investigated the impact of baffles on a short shell and tube heat exchanger. In their studies, the author had two geometrical configurations of the heat exchangers – with and without the baffles. OpenFOAM CFD solver was used for the numerical simulations. They had suggested to apply fully developed conditions at the inlet for improving the CFD simulation convergence. The pressure estimations from their CFD simulations were compared against the Bell-Delaware method and was found to be in acceptable accuracy.

There have been continuous research in enhancing the heat transfer rate from the shell-side of the heat exchangers. One such method has been the inclusion of the fins on the tubes. This method was studied by Rahul Singh [3]. The authors had compared various heat exchanger geometrical configurations – with and without fins, with and without baffles. From their CFD simulation data, they had concluded that the rectangular shaped fins provided higher heat transfer rate as compared to the circular fins.

Jian Wen [4] had designed ladder-type fold baffles for the shell and tube heat exchangers with a focus to reduce the leakage zones that typically results in the case of the helical baffles. The ladder-type fold baffles increases the spiral flow which enhances the participation of higher fluid for heat exchange with the tubes. Overall, this mechanism had resulted in shell-side heat transfer coefficient enhancement. The authors had observed nearly 22.3 – 32.6 % increase in the shell-side heat transfer coefficient.

Salim Fettaka [5] had designed the shell-and-tube heat exchanger with non-dominated sorting genetic algorithm

(NSGA-II) module in MATLAB. For this optimization design, the authors had considered the following nine variables – number of tube passes, baffle spacing, baffle cut, layout pattern, tune-to-baffle-diametrical clearance, shell-baffle diametrical clearance, tube length, tube outer diameter and tube wall thickness. The authors had applied Bell-Delaware method for the heat transfer coefficient and pressure drop of the shell-side in the optimization.

The impact of cross-flow on the heat transfer characteristics on the tube banks was investigated by Yongqing Wang [6] using numerical simulations as well as experimental methods. The authors had varied the cross-flow from 30° to 90° at an interval of 15°. In their model, there were 10 rows of tubes and each of these rows had 10 tubes that were arranged in staggered pattern with a pitch of 32 mm. The CFD simulations were performed using ANSYS FLUENT. The standard k-ϵ turbulence model was applied in their simulations. The predictions from the CFD simulations were verified against the analytical expressions by the authors.

B. Parikshit [7] had applied finite element methods for investigating the shell-side flow losses in the shell and tube heat exchangers. The model that was developed by the authors had taken in to account of the window-section as well as mid-section elements. Their model included a yaw correction to model the flow losses in the inter-baffle region. The shell-side computational volume was discretized using finite elements with 4 elements between the baffles. The stiffness matrix from each element was assembled to form the global stiffness matrix. The solution for this matric provided the flow losses in shell-and-tube heat exchanger.

The impact of baffle spacing, baffle cut and the shell diameter on the shell and tube heat exchanger performance was investigated by Avinash D Jadhav [8]. Their CFD simulation results were compared with analytical calculations such as Bell-Delaware method. They had studied two variants for baffle cuts 25% and 30%. The pressure drop on the shell-side of the heat exchanger was observed to be reduced as the baffle cut was increased and the authors had observed this for a wide range operating conditions i.e. inlet flow rate.

There have been various methods to numerically simulate the flow field in the shell and tube heat exchangers. Some researchers had modeled only the shell side for understanding the flow and thermal characteristics. Yonghua You [9] had applied the porosity and permeability concepts for modeling the flow physics of the shell-side in a shell-and-tube-heat-exchanger. In this approach, volumetric porosity was applied for the core region of the heat exchanger where the tubes were present. And, user defined functions (UDFs) were supplied for the simulations for momentum, energy and turbulent kinetic energy and its dissipation rate' governing equations. With these approaches, the authors had obtained numerical results that were in good agreement with the experimental data.

In another research, Jie Yang [10] had applied four modeling approaches for the heat exchanger simulations using CFD methods. They were of unit model, periodic model, porosity model and the whole model. In the unit model, only a section of the heat exchanger geometry was modeled while the geometry between the set of baffles were considered for the periodic model. In the case of porosity approach, the tubes were removed and the equivalent flow losses were specified in terms of porosity and permeability coefficients. The results (Nusselt number vs Reynolds Number) from these simulations were compared against the experimental data. Based on their results, the authors had concluded that the porosity approach and the whole model approach provided accurate results as compared the unit model and the periodic model.

The heat transfer performance of the heat exchanger could be enhanced by various methods. One such method is by attaching fins on the tube outer surfaces. This was studied by Jignesh M Chaudhari [11] using experimental methods. In their model, circular shaped plate-fins with a thickness of 1 mm were attached on tubes with the external diameter of 14.6 mm. The fin outer diameter was 3.43 mm. The heat transfer coefficient from the heat exchangers with and without fins on the tubes were compared. And, the authors had estimated ~17% increase in heat transfer coefficient when the fins were attached to the tubes.

Anas El Maakoul [12] had studied the shell-side heat transfer enhancement mechanisms such as helical baffles, segmental baffles and trefoil-holes on the baffles of the shell-and-tube-heat-exchangers with CFD simulations in ANSYS FLUENT. For these simulations, the authors had considered the complete geometry without any simplifications such as porosity. They had indicated that the inclusions of trefoil-holes on the baffles would lead to high velocity jets which subsequently lead to fluid energy dissipation. This mechanism would help in obtaining high heat transfer rate in the heat exchanger. The heat exchanger performance as defined by h/dp (ratio between the heat transfer coefficient and the pressure drop) was enhanced by 192% for the trefoil-holes in these studies.

The typical smooth surfaces shell and tube heat exchangers were replaced by corrugated surfaces in the experimental studies by HamedSadighiDizaji [13]. They had investigated both convex and concave corrugations on these surfaces for heat transfer enhancement. They had observed an increase of 34% - 60% in NTU for the corrugated tube and shell surfaces in the heat exchangers as compared to the smooth surfaces.

G. V. Srinivasa Rao [14] had conducted studies of shell-and-tube heat exchanger with steam on the shell side and had varied fluid on the tube side. On the tube side, CO2 and SO2 were supplied as fluid medium. The corresponding Reynolds number range was 400,000 to 650,000. Their analytical design, based on Kern method, was validated against the Dittus-Boelter correlation for the heat transfer coefficient. For the varying the tube fluid, the authors had observed no change in flow losses.

Jian-Feng Yang [15] had compared shell-and-tube-heat-exchanger configurations of combined parallel two pass (CPTSP-STHX) and combined serial two pass (CSTSP-STHX) that had helical baffles with that of shell-and-tube-heat-exchanger with segmental baffles (SG-STHX). Based on their CFD simulations in ANSYS FLUENT, the authors had observed an increase in heat transfer by 5.1% for the CPTSP-STHX and a 9.5% increase for the CSTSP-STHX configurations. They had attributed the higher flow velocity, due to the reduction in the flow area, for the improved thermal performance.

Guo-Yan Zhou [16] had studied the trefoil-hole baffles for improving the shell and tube heat exchanger performance for the nuclear industrial applications. Their study was based CFD simulations using ANSYS FLUENT. In their RANS based simulations, the authors had used RNG k-epsilon turbulence model. The results from the CFD simulations were compared against the Nusselt number that were estimated from the experimental studies. And, they had noted the difference between them were less than 20% and found to be acceptable. The trefoil-hole baffles induced periodic flow in the heat exchangers as observed by the authors.

The application of compact shell-and-tube-heat-exchanger for heat recovery in the diesel engine was investigated by Saifyl Bari [17]. Their study was conducted using experimental as well as computational simulations (CFD) in ANSYS CFX. The phase change from water to vapor in the heat exchanger was included in their simulation with the help of Thermal Phase Change Model in ANSYS CFX. The heat exchanger effectiveness from their study was in the range of 0.4 – 0.6. The authors had observed that the working fluid pressure should be varied in order to handle additional load on the heat exchangers at part loads.

Swapneel Sharma [18] had applied triangular shaped fins on a shell-and-tube heat exchanger for the application of waste heat recovery process with water as heat transfer fluid (HTF). They had considered 15 HP diesel engine for this heat recovery analysis. While comparing the finned heat exchanger against the non-finned heat exchanger, the higher pressure drop was observed for the triangular finned heat exchangers.

III. CONCLUSIONS FROM LITERATURE SURVEY

The following conclusions were drawn based on this literature survey

1) The Reynolds Averaged Navier-Stokes (RANS) approach in the CFD simulations with the Realizable k-ϵ turbulence had been providing accurate results.

2) Results obtained from the CFD simulations of the shell-tube heat exchangers were in agreement with the experimental studies. So, major performance optimization investigations could be conducted using CFD simulations. And, a validating experimental study could be conducted at the final stage. This would reduce developmental cost of the shell-and-tube heat exchangers.

3) Attaching fins on the heat exchanger tubes had resulted in heat transfer performance significantly while inducing acceptable additional flow losses.

REFERENCES

[1] Ender Ozden, Ilker Tari, “Shell Side CFD Analysis of a Small Shell-and-Tube Heat Exchanger” Energy Conservation Management, Volume 51, pp 1004-1014, 2010;

[2] Eshita Pal, Inder Kumar, Jyeshthraj B Joshi, N. K. Maheshwari, “CFD Simulations of Shell-side Flow in a Shell-and-Tube Heat Exchanger with and wihout Baffles” Chemical Engineering Science, January 2016;

[3] Rahul Singh, Divyank Dubey, Harischandra Thakur, “CFD Analysis of Shell and Tube Heat Exchanger”, International Journal for Scientific Research and Development, Volume 3, Issue 12, December 2016;

[4] Jian Wen, Huizhu Yang, Simin Wang, YulanXue, Xin Tong, “Experimental Investigation on Performance Comparison for Shell-and-Tube Heat Exchangers with Different Baffles” International Journal of Heat and Mass Transfer, Volume 84, pp 990-997, 2015;

[5] Salim Fettaka, Jules Thibault, Yash Gupta, “Design of Shell-and-Tube Heat Exchangers using Multi-objectiveOptimization” International Journal of Heat and Mass Transfer, Volume 60, pp 343-353, 2013;

[6] Yongqing Wang, Xin Gu, ZunlongJin, Ke Wang, “Characteristics of Heat Transfer for Tube Banks in Crossflow and its Relation with that in Shell-and-Tube Heat Exchangers” International Journal Heat and Mass Transfer, Volume 93, pp 584-594, 2016;

[7] B. Parikshit, K. R. Spandana, V. Krishna, T. R. Seetharam, K. N. Seetharamu, “A Simple Method to Calculate Shell side Fluid Pressure drop in a Shell and Tube Heat Exchanger” International Journal of Heat and Mass Transfer, Volume 84, pp 700-712, 2015;

[8] Avinash D Jadhav, Tushar A Koli, “CFD Analysis of Shell and Tube Heat Exchanger to Study the Effect of Baffle Cut on the Pressure drop” International Journal of Research in Aeronautical and Mechanical Engineering Volume 2, Issue 7, pp 1-7, July 2014;

[9] Yonghua You, Aiwu Fan, Suyi Huang, Wei Liu, “Numerical Modelling and Experimental Validation of Heat Transfer and Flow Resistance on the Shell side of a Shell-and-Tube Heat Exchanger with Flower Baffles” International Journal of Heat and Mass Transfer, Volume 55, pp 7561-7569, 2012;

[10] J. Yang, L. Ma, J. Bock, A.M. Jacobi, W.Liu, “A Comparison of Four Numerical Modelling Approaches for Enhanced Shell-and-Tube Heat Exchangers with Experimental Validation” Applied Thermal Engineering, January 2014;

[11] Jignesh Chaudhari, DattatrayaSubhedar, Nikul Patel, “Experimental Investigation of Finned Tube Heat Exchanger” International Journal of Innovative Research in Advanced Engineering, Volume 1, Issue 5, June 2014;

[12] A. E. Maakoul, A. Laknizi, S. Saadeddine, M. E. Metoui, A. Zaite, M. Meziane, A. B. Abdellah, “Numerical Comparison of Shell and Tube Heat Exchangers with Trefoil-hole Helical and Segmental Baffle” Applied Thermal Engineering, August 2016;

[13] H. SadighiDizaji, S. Jafarmadar, S. Asaadi, “Experimental Exergy Analysis for Shell and Tube Heat Exchanger made of Corrugated Shell and Corrugated Tube” Experimental and Thermal Fluid Science, September 2016;

[14] G. V. Srinivasa Rao, C. J. Rao, N. Haribabu, “Heat Transfer Analysis on Shell and Tube Heat Exchangers” International Journal of Research in Aeronautical and Mechanical Engineering, Volume 2, Issue 1, pp 11-26, January 2014;

[15] Jian-Feng Yang, Min Zeng, Qiu-Wang Wang, “Numerical Investigation on Shell-side Performances of Combined Parallel and Serial Two Shell-Pass Shell-and-Tube Heat Exchangers with Continuous Helical Baffles” Applied Energy, Volume 139, pp 163-174, 2015;

[16] Guo-Yan Zhou, Jingmei Xiao, Lingyun Zhu, Juntao Wang, Shan-Tung Tu, “A Numerical Study on the Shell-side Turbulent Heat Transfer Enhancement of Shell-and-Tube Heat Exchanger with Trefoil-Hole Baffles” The 7th International Conference on Applied Energy, pp 3174-3179, 2015;

[17] Saiful Bari, Shekh N Hossain, “Waste Heat Recovery from a Diesel Engine using Shell and Tube Heat Exchanger” Applied Thermal Engineering. Volume 61, pp 355-363, 2013;

[18] SwapnaneelSarma, D.H. Das, “CFD Analsysis of Shell and Tube Heat Exchanger using Triangular Fins for Waste Heat Recovery Processes” IRACST – Engineering Science and Technology: An International Journal, Volume 2, Issue 6, pp 1033-1041, December 2012;