THOMSON REUTERS ENDNOTE

[Negi*, 5.(1): January, 2016] ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

IJESRT

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

# NUMERICAL INVESTIGATION OF SHELL AND TUBE HEAT EXCHNGER FOR HEAT TRANSFER OPTIMIZATION: A Review

**Jaya Verma Negi*, Pankaj Negi, Dr. Satyendra Singh**

Department of Mechanical Engineering Bipin Tripathi Kumaon Institute of Technology Dwarahat, Almora, Uttarakhand, India.

## ABSTRACT

An un-baffled shell-and-tube heat exchanger design with respect to heat transfer coefficient and pressure drop is investigated by numerically modelling. The flow and temperature fields inside the shell and tubes are resolved using a commercial CFD package considering the plane symmetry. A set of CFD simulations is performed for a single shell and tube bundle and is compared with the experimental results. The results are found to be sensitive to turbulence model and wall treatment method. It is found that there are regions of low Reynolds number in the core of heat exchanger shell. Thus, k-ω SST model, with low Reynolds correction, provides better results as compared to other models. The temperature and velocity profiles are examined in detail. It is seen that the flow remains parallel to the tubes thus limiting the heat transfer. Approximately, 2/3rd of the shell side fluid is bypassing the tubes and contributing little to the overall heat transfer. Significant fraction of total shell side pressure drop is found at inlet and outlet regions. Due to the parallel flow and low mass flux in the core of heat exchanger, the tubes are not uniformly heated. Outer tubes fluid tends to leave at a higher temperature compared to inner tubes fluid. Higher heat flux is observed at shell's inlet due to two reasons. Firstly due to the cross-flow and secondly due to higher temperature difference between tubes and shell side fluid. On the basis of these findings, current design needs modifications to improve heat transfer.

KEYWORDS— Heat transfer, Shell-and-Tube Heat exchanger, CFD.

## INTRODUCTION (BAYESIAN TECHNIQUE)

Heat exchangers are one of the mostly used equipment's in the process industries. Heat exchangers are used to transfer heat between two process streams. One can realize their usage that any process which involves cooling, heating, condensation, boiling or evaporation will require a heat exchanger for these purposes. Process fluids, usually are heated or cooled before the process or undergo a phase change. Different heat exchangers are named according to their applications. For example, heat exchangers being used to condense are known as condensers; similarly heat exchangers for boiling purposes are called boilers. Performance and efficiency of heat exchangers are measured through the amount of heat transferred using least area of heat transfer and pressure drop. A better presentation of its efficiency is done by calculating over all heat transfer coefficient. Pressure drop and area required for a certain amount of heat transfer, provides an insight about the capital cost and power requirements (Running cost) of a heat exchanger. Usually, there is lots of literature and theories to design a heat exchanger according to the requirements. A good design is referred to a heat exchanger with least possible area and pressure drop to fulfil the heat transfer requirements.

**Characteristics of heat transfer for tube banks in crossflow and its relation with that in shell-and-tube heat exchanges**

By (Yongqing Wang, Xin Gu and Ke Wang)

The thermodynamics performances for tube banks in cross flow and for the shell sides of shell-and-tube heat exchangers were investigated and the relation of fluid flow and heat transfer between them were analyzed. The results indicate that the incline degree of tube does not lead to obvious change on characteristics of fluid flow and heat transfer for fluid flowing across tube banks. Under different incline degrees of tubes, the characteristics of fluid flowing across tube banks are similar concerning fluid velocity components and local heat transfer along across angles. The ratios of vertical velocity components to parallel velocity components remain about 4.2 in the banks decreasing of impacting angle in the model reduces average fluid velocity crossing tubes, which weakens heat transfer. Characteristics for fluid flowing across tube bundles in shell sides distinguish from that for tube banks in cross flow with different impacting angles. At the same mass rate, the values of parallel velocity components vary greatly. The ratios of vertical velocity component change from about 0.33-0.92 in the three types of shell sides. Many different leakage paths and bypass streams leads complicated fluid flow in shell sides of heat exchangers. The complicated flow pattern is not a simple combination of fluid flowing across tube banks with different impacting angles. The thermodynamics performance of shell side depends greatly on the vertical velocity components of fluid. The achievement in the paper can be a reference for study and development of heat exchanger.

***A numerical study on the shell-side turbulent heat transfer enhancement of shell and tube heat exchanger with trefoil-hole baffles.***

By *(Guo-yan Zhou, Jingemi Xiao and Shan_tung)*

In this paper Shell-and-tube heat exchanger with trefoil hole baffles are new type heat transfer devices and widely used in nuclear power system due ti their special advantages, with the fluid flowing longitudinally on the shell side. However, very few related academic literature are available. In order to obtain an understanding of the underlying mechanism of shell side thermal augmentation, a CFD model including inlet and outlet nozzles is proposed in the present study. Based on the RNG k-ε model, numerical investigation on shell side fluid flow and heat transfer are conducted by using commercial CFD software FLUENT 14.0. The results ahow that the fluids is fully developed after the first trefoil hole baffle. The heat transfer coefficient and pressure drop vary periodically along the axial direction. Fluid velocity increases gradually and the jet flow forms in the region near baffles. The secondary flow can decreases the thickness of the boundary layer and then enhance the heat transfer.

***The effects of flow, thermodynamic and geometrical characteristics on exergy loss in shell and coiled tube heat exchangers.***

By *(Hamed Sadighi Dizaji, Samad Jafarmadar, Mehran Hashemian)*

This work presents experimental on the effects of flow, thermodynamics and geometrical characteristics on exergy loss in the shell and coiled tubes heat exchangers have been widely studied in the the recent years. However, the effects of flow, thermodynamics and geometrical parameters on exergetic characteristics have not been explicity and experimentally studied. Hence, the main scope of the present work is to clarify the effect of shell and coil side flow rates, inlet temperatures coil pitch and coil diameter on exergy loss in shell and coiled tube heat exchangers. Both of the total exergy loss and the dimensionless exergy loss are studied.

***A Simple method to calculate shell side fluid pressure drop in a shell and tube heat exchanger***

By *(B.Parikshit, K.R Spandaana, K.N Seetharamau)*

Pressure drop predictions on the shell side of a Shell and Tube Heat Exchanger (STHK) are investigated using te concept of Finite Element Method (FEM) . In this method the shell side region is discretised into a number of elements and by taking into accounts the effect of flow pattern, the pressure drop on the shell side of a STHX is determined. The present method is simple to apply and the predictions agree reasonably well with a large number of experimental data available in the literature The range of applicability of the present method extends beyond that used by others in the literature. The earlier predictions, were restricted to tubes in the windows region, however, the predictions of the present method are extended to cases of no tubes in the windows (NTIW) region also.

***Semi numerical analysis of heat transfer performance of fractal based tube bundle in shell and tube heat exchanger.***

By *(Jian Feng Zhou, Shi-wei Wu, Chun-lei Shao)*

A bundle of topologically arranged tubes based on fractal is proposed in this work to enhance the flow of shell side fluid. The space for arranging tubes is separated into some periodic regions and the tubes are symmetrically arranged in these regions. The topological arrangement of tubes is in the radial direction starting from the shell center. Fractal treatment is applied to divide each periodic region into two smaller symmetric ones. With the alternately installed disc and doughnut baffles, the shell side flow is realized. According to the periodic characteristics of tube bundle, numerical heat transfer unit models are established and the characteristics temperature in heat exchanger are obtained using the semi numerical simulation algorithm. Comparing the results with the analytical solution to the outlet temperatures of the shell side and tube side fluids based on the Bell-Delaware method, it is revealed that, even though the number density of tubes is reduced compared to the conventional version, the new structure has a higher heat transfer efficiency due to the full uses of tubes . The fluid outflows from the tube near the shell center has a higher temperature, and the concurrent and countercurrent flows results in te temperature increasing trends of tube-side fluid as well as the different temperature decreasing trend of shell side fluid. The countercurrent results in a larger decrease of shell side fluid temperature.

**Convection heat transfer in a shell and tube heat exchanger using dheets fins for effective utilization of energy**
By *(Koichi Nakaso, Hirohi Mitani, Jun fukai)*
Convections heat transfer in a shell and tube heat exchanger using sheet fins is numerically investigated. Heat and mass transfers rate in the heat exchanger are modeled under steady state to estimate the heat transfer rate and the pressure drop for various geometries of the heat exchanger. Based on the numerical results, the Nusselt number and the pressure drop are formulated for practical applications. For convenience, similar expressions to those of conventional shell and tube heat exchanger, that is, the function of dimensionless numbers such as the Reynolds number, are derived. In these equations, the geometry of the heat exchanger, fin efficiency and contact thermal resistance are included as major factors. On formulating the equations for the overall heat transfer rate, it is found that the heat transfer coefficient for the heat exchanger with a fin does not correspond to the combination of the heat transfer coefficient of bare tube surface and the fin. This is because that heat exchange area is substantially limited especially at the narrow space between the tube and the fin. A correction factor for the substantial heat transfer area is therefore introduced. These formulated equations are helpful for installing sheet fins in manufactured heat exchanger. Using the formulated equations, effective conditions to enhance heat transfer rate by the fin are established, taking into account the increase in pressure drop.

## CONCLUSION

The heat transfer and flow distribution is discussed in detail and proposed model is compared with the experimental results as well. The model predicts the heat transfer and pressure drop with an average error of 20%. Thus the model still can be improved. The assumption of plane symmetry works well for most of the length of heat exchanger except the outlet and inlet regions where the rapid mixing and change in flow direction takes place. Thus improvement is expected if complete geometry is modeled. Moreover, SST k-ω model has provided the reliable results given the y+ limitations, but this model over predicts the turbulence in regions with large normal strain (i.e. stagnation region at at inlet of the shell). Thus the modeling can also be improved by using Reynolds Stress Models, but with higher computational costs. Furthermore, the enhanced wall functions are not used in this project due to convergence issues, but they can be very useful with k-ω models.

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 in order to achieve the better heat transfer in two ways. Either, the shell diameter is reduced to keep the outer fluid mass flux lower or tube spacing can be increased to enhance the inner fluid mass flux. Just doing this might not be enough, because it is seen that the shell side fluid after 3m doesn't transfer heat efficiently.

It is because the heat transfer area is not utilized efficiently. 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.

## REFERENCE

[1] Computational Fluid Dynamics for Chemical Engineers. sixth ed., (2010). Andersson, Andersson, Akansson, Mortensen, Sudiyo, and Wachem
[2] Cfd applications in various heat exchangers design: Areview," Applied Thermal Engineering, vol. 32, pp. 1–12, 2011 Bhutta, and Hayat,
[3] "Comparison of near-wall treatment methods for high Reynolds number backward-facing step flow," Int. J Computational Fluid Dynamics, vol. 19, 2005. Jae-Young and Afshin,

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

[4] Heat Transfer in Process Engineering. McGraw Hill, 2009.38 Cao
[5] “Comparison of near-wall treatment methods for high Reynolds number backward-facing step flow,” Int. J Computational Fluid Dynamics, vol. 19, 2005] Jae-Young and Afshin
[6] Validation of a cfd model of a three dimensional tube in- tube heat exchanger.,” 2003. Van Der Vyver and Dirker
[7] An Introduction to Computational Fluid Dynamics, The Finite Volume Method. Pearson Education Limited, 2007 Versteeg and M. W
[8] “Heat exchangers: Selection, rating, and thermal performance,” 1998. Kakac and Liu
[9] Serth, Process Heat Transfer, Principles and Applications. Elsevier Science and Technology Books,
[10] Shell side cfd analysis of a small shell-and-tube heat exchanger Ozden and Tari
[11] “Shell-side heat transfer in baffled cylindrical shell and tube exchangers- an electrochemical mass transfer modelling technique,” Int J Heat Mass Transfer, vol. 19, pp. 995–1002, 1976. Gay, and Mackle
[12] “Pressure drop on the shell side of shell-and-tube heat exchangers with segmental baffles,” Chem Eng Process, vol. 36, pp. 149–59, 1997. Gaddis
[13] “Validation of a cfd model of a three dimensional tube in- tube heat exchanger.,” 2003] Van Der Vyver and Dirker
[14] Shell and tube heat exchangers. part 1: foundation and fluid mechanics, “Heat Transfer, p. 33:799–816., 1998 Prithiviraj,
[15] “Ansys fluent theory guide.” http://www.ansys.com, 2010.