ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 3, Issue 2, March 2014

Application of CFD in STHE: A Review

Savitri Patel, D.S Patel

***Abstract-** 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 petroleum refineries, chemical plants, petrochemical plants, natural gas processing, air-conditioning, refrigeration and automotive applications. Computational Fluid Dynamics (CFD) can very useful to gain visualize the flow and temperature fields on the shell side can simplify the assessment of the weaknesses. Different turbulence models available in general purpose commercial CFD tools likes k-ε, K-ω and K-ω SST models. This literature review focuses on the importance and usefulness of k-ε model in CFD analysis and simulation.*

Index words: **Baffle inclination angle, Computational fluid dynamics, Computational modeling, Shell and tube heat exchanger, Turbulence model.**

I. 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.

II. 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.

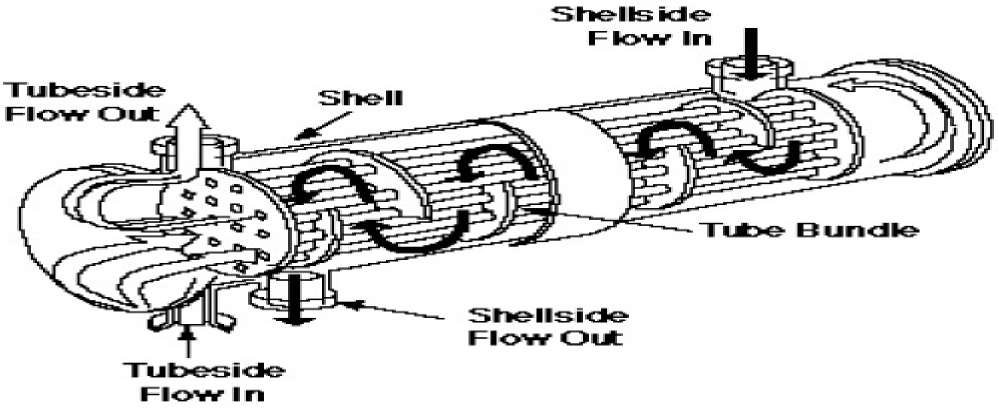

Fig.1: Shell and tube heat exchanger

III. LITERATURE SURVEY

Hamidou Benzenine et al.[1] investigate 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.

D.P.Naik et al.[2] has 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.

Sunil S. Shinde et al.[3] has studied about the performance Improvement in Single phase Tubular Heat Exchanger using continuous Helical Baffles and investigated that that the performance of tubular heat exchanger can be improved by helical baffles instead of conventional segmental baffles. The use of helical baffles in heat exchanger reduces shell side pressure drop, pumping cost, size, weight, fouling etc. as compare to segmental baffle for new installations. The helix changer type heat exchangers can save capital cost as well as operating and maintenance cost and thus improves the reliability and availability of process plant in a cost effective way. For the helical baffle heat exchangers, the ratios of heat transfer coefficient to pressure drop are higher than those of a conventional segmental heat exchanger. This means that the heat exchangers with helical baffles will have a higher heat transfer coefficient when consuming the same pumping power. It can be concluded that proper baffle inclination angle will provide an optimal performance of heat exchangers.

A.E. Zohir[4] analysis the Heat transfer characteristics in a heat exchanger for turbulent pulsating water flow with different amplitudes. 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.

Jitendra Kumar Patro[5] discussed about the experimental studies on heat transfer augmentation using TMT rods with and without baffles as inserts for tube side flow of liquids. The different results came are:

1.For same baffle spacing 8mm & 10mm inserts with baffles shows greater heat transfer coefficient & friction factor than the value we get for inserts without baffles, because of increased degree of turbulence created.

2.On the basis of R1 i.e. Performance evaluation criteria based on constant flow rate, we can say that the 10mm insert with baffle spacing (=10cm) gives the highest R1 range with the maximum value of Heat transfer coefficient around 2.46 times of the value for the smooth tube.

3.The effect of 10mm insert (without baffles) and 8mm insert with spacing = 30cm are almost equivalent on both the performance evaluation criteria R1 & f_a/f_o where f_a is friction factor for the tube with inserts and f_o is theoretical friction factor for smooth tube.

4. With decrease in baffle spacing, heat transfer coefficient increases but at the same time pressure drop also increases.

Sombat Tamna et al.[6] has the numerical heat transfer study in a square channel with zigzag-angled baffles and concluded that laminar periodic flow and heat transfer characteristics in a square channel fitted with 45° angled baffle elements in tandem, zigzag arrangements on two opposite walls have been investigated numerically. The P-vortex flows created by using the zigzag-angled baffles help to induce impingement flows on the sidewall and wall in the baffle cavity leading to drastic increase in heat transfer. The order of enhancement is about 100-600% for using the 45° zigzag-angled baffle with BR=0.05-0.30. However, as expected, the heat transfer augmentation is associated with enlarged pressure loss ranging from 1 to 18 times above the smooth channel depending on the BR and Re values. The heat transfer enhancement for the 45° zigzag-angled baffle is around 100-270% higher than smooth channel and the TEF is found to be much higher than unity and its maximum value is about 2.7 indicating much higher performance over the smooth channel.

Muhammad Mahmood Aslam Bhutta et al.[7] 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%.

S. Pethkool et al.[8] carried out experimentally augmentation of convective heat transfer in a single-phase turbulent flow by using helically corrugated tubes . Effects of pitch-to-diameter ratio (P/DH=0.18, 0.22 and 0.27) and rib-height to diameter ratio (e/DH=0.02, 0.04 and 0.06) of helically corrugated tubes on the heat transfer enhancement, isothermal friction and thermal performance factor in a concentric tube heat exchanger are examined. The experiments were conducted over a wide range of turbulent fluid flow of Reynolds number from 5500 to 60,000 by employing water as the test fluid. Experimental results show that the heat transfer and thermal performance of the corrugated tube are considerably increased compared to those of the smooth tube. The mean increase in heat transfer rate is between 123% and 232% at the test range, depending on the rib height/pitch ratios and Reynolds number while the maximum thermal performance is found to be about 2.3 for using the corrugated tube with P/DH=0.27 and e/DH=0.06 at low Reynolds number. Also, the pressure loss result reveals that the average friction factor of the corrugated tube is in a range between 1.46 and 1.93 times over the smooth tube. In addition, correlations of the Nusselt number, friction factor and thermal performance factor in terms of pitch ratio (P/DH), rib-height ratio (e/DH), Reynolds number (Re), and Prandtl number (Pr) for the corrugated tube are determined, based on the curve fitting of the experimental data.

Ender Ozden et al.[9] 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 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.

Wang Yongqing et al.[10] has carried out the Analysis of fluid flow and heat transfer characteristics by adopting numerical models with the segmental baffle, rod baffle and H-shape baffle support structures in shell-sides of shell-and-tube heat exchangers. The characteristics in the three heat exchangers were compared. Heat exchanger with the H-shape baffle support structure combines the respective characteristics of heat exchangers with cross fluid flow and longitudinal fluid flow in shell-sides. In H-shape baffle heat exchanger, fluid flows in a mixing flow pattern. The flow Pattern avoids fluid velocity and kinetic energy loss, decreases the flow dead zone, and bigger transverse flow component reserves the bigger action of segmental baffle. The flow pattern strengthens fluid turbulence. At some range of flow flux, H-shape baffle is an ideal tube support structure, which induces fluid flows in a mixing pattern and enhances heat transfer.

J.S. Jayakumara et al.[11] 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.

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 exergy 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.

Qiuwang Wang et al.[13] 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.

M.R. Salimpour [14] has carried out to study heat transfer coefficients of the shell and helically coiled tube heat exchangers. Heat exchangers with three different coil pitches were tested for counter-flow configuration. It was revealed that the empirical correlation for constant temperature boundary condition is quite in agreement with the present data in low Dean number region. From the results of the present study, it was found out that the shell-side heat transfer coefficients of the coils with larger pitches are higher than those for smaller pitches. Finally, based on the results of this study, two correlations were developed to predict the inner and outer heat transfer coefficients of the coiled tube heat exchangers.

Kevin M. Lunsford et al.[15] 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.

Huadong Li et al.[16] 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.

Huadong Li et al.[17] has performed the experiment to decide the pressure drop and local heat transfer to change in the leakage between baffles and shell on the shell side of shell and tube heat exchanger. By the means of mass transfer measurement the local mass and heat transfer co efficient are determined. Effect of leakage on pressure drop and local heat transfer shows that baffle shell leakage is slight contribution to the local heat transfer at the surfaces of the external tubes of the tube bundle, but reduces the average heat transfer for Re = 500 up to 21% and for Re =16,000 up to 17 %.The baffle tube leakage is positive for the improvement of the heat transfer and for the reduction of pressure drop. The shell side flow distribution was calculated from the measurements at the pressure distribution in the baffle compartment.

IV. COMPUTATIONAL MODELING

Computational Fluid Dynamics (CFD) provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of

- mathematical modeling (partial differential equations)
- numerical methods (discretization and solution techniques)
- software tools (solvers, pre- and post processing utilities)

CFD enables scientists and engineers to perform ‘numerical experiments’ (i.e. computer simulations) in a ‘virtual flow laboratory’**.**

Governing equations

$$\textbf{X-momentum: } \nabla.(\rho u V_r) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z}$$

$$\textbf{Y-momentum: } \nabla.(\rho v V_r) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho g$$

$$\textbf{Z-momentum: } \nabla.(\rho w V_r) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho g$$

$$\textbf{Energy: } \nabla(\rho e V_r) = -p\nabla V_r + \nabla(k\nabla T) + q\varphi \quad \text{-----------(1)}$$

In Eq.(1), φ is the dissipation function that can be calculated from

$$\varphi = \mu\left[2\left[\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2 + \left(\frac{\partial w}{\partial z}\right)^2\right] + \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)^2 + \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial z}\right)^2 + \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)^2\right] + \lambda(\nabla.V_r)^2$$

A .Turbulence Model

- A turbulence model is a computational procedure to close the system of mean flow equations.
- For most engineering applications it is unnecessary to resolve the details of the turbulent fluctuations.
- Turbulence models allow the calculation of the mean flow without first calculating the full time-dependent flow field.
- We only need to know how turbulence affected the mean flow.
- In particular we need expressions for the Reynolds stresses.
- For a turbulence model to be useful it:
  - must have wide applicability,
  - be accurate,
  - simple,
  - and economical to run.

They are used to predict the effects of turbulence in fluid flow without resolving all scales of the smallest turbulent fluctuations 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.

i. K-ε turbulence model: - The k-ε (k-epsilon) model has been implemented in most general purpose CFD codes and is considered the industry standard model. It has proven to be stable and numerically robust and has a well established regime of predictive capability. For general purpose simulation, the model offers a good compromise in term of accuracy and robustness.

ii. K-ω turbulence model: - One of the advantages of the k-ω formulation is the near wall treatment for low Reynolds number computations. The model does not involves the complex non linear damping functions required for the k-ω model and is therefore more accurate and more robust. The models assume that the turbulence viscosity is linked to the turbulence kinetic energy and turbulent frequency via the relation.

iii. K-ω SST Model: - The k- ω SST turbulence model is a two equation model. This model is known because it uses both k - ω and k - ε models.SST models stands for Shear Stress Transport model. k - ε is a high Reynolds number model thus in the near wall region k - ω model is used.Whereas,in the region away from the walls, k - ε model is used. The SST model uses a blending function whose value depends upon the distance from the walls. Near the wall, in viscous sub-layer, this blending function is one and only k - ω model is used. The regions away from the wall this function is zero and uses only k - ε model.

The standard k-ε model is a semi-empirical model based on model transport equations for the turbulence kinetic energy k and its dissipation rate ε.For steady state, k and ε are obtained from the following transport equations:

$$\partial/\partial x_i(\rho k u_i) = \partial/\ \partial x_j[(\mu + \frac{\mu_i}{\sigma_k})\partial k/\partial x_j] + G_k + G_b - \rho\varepsilon + S_k$$

$$\partial/\partial x_i(\rho \varepsilon u_i) = \partial/\ \partial x_j[(\mu + \frac{\mu_i}{\sigma_\varepsilon})\partial \varepsilon/\partial x_j] + \frac{C_{1\varepsilon}\varepsilon}{k(G_k + C_{3\varepsilon}G_b)} - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_k$$

and the turbulent viscosity is defined by the following equation:

$$\mu_t = \rho C_\mu k^2/\varepsilon$$

V. CONCLUSION

From literature review it can be concluded that,

- 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.

- It is observed that the CFD simulation results are in very good agreement with the Bell-Delaware method.
- Proper baffle inclination angle will provide an optimal performance of heat exchangers.

REFERENCES

[1] 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.

[2] 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.

[3] Sunil S. Shinde, Samir S. Joshi and Dr. S. Pavithran, “Performance Improvement in Single phase Tubular Heat Exchanger using continuous Helical Baffles”, International Journal of Engineering Research and Applications (IJERA), Vol. No.2, pp. 1141-1149, Jan 2012.

[4] 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.

[5] Jitendra Kumar Patro, “Experimental Studies on Heat Transfer Augmentation Using TMT Rods with and without Baffles as Inserts for Tube Side Flow of Liquids”, B.Tech dissertation, National Institute of Technology, Rourkela, India, 2012.

[6] Sombat Tamna, Warakom Nerdnoi, Chinaruk Thianpong and Pongjet Promvonge, “Numerical Heat Transfer Study in a Square Channel with Zigzag-Angled Baffles”, International Conference on Mechanical Engineering, Vol. No.1, pp. 1-9,Oct. 2011.

[7] 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.

[8] S. Pethkool, S. Eiamsa-ard, S. Kwankaomeng, P. Promvonge," A Turbulent heat transfer enhancement in a heat exchanger using helically corrugated tube", International Communications in Heat and Mass Transfer 38 (2011).

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

[10] Wang Yongqing, Gu Xin, Wang Ke, Dong Qiwu, “Numerical Investigation Of Shell - Side Characteristics Of H -Shape Baffle Heat Exchanger”, Elsevier Science, China, 2010.

[11] 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).

[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] 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).

[14] M.R.Salimpour, "Heat transfer coefficients of shell and coiled tube heat exchangers", Experimental Thermal and Fluid Science 33 (2009) 203–207.

[15] K.M. Lunsford, “Increasing Heat Exchanger Performance” ,Bryan Research and Engineering, Inc., Vol. No.1, pp. 1-13, March 1998.

[16] 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.

[17] 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.