REVIEW OF COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF SHELL AND TUBE HEAT EXCHAGER WITH HAVING DIFFERENT TYPES OF FINS

Shubham Gupta, Karan Singh Verma

Oriental college of technology, Bhopal

Abstract: **A shell and tube heat exchanger is used as industrial application because of its compact structures and effective heat transfer area through inlet metal pipe and outlet metal pipe. The importance of shell and tube heat exchanger as it provides typically higher surface area density ranging in metal pipes along with the baffles are arranged in the system to support the tubes, prevent tube vibration and sagging also it guides the flow of heat transfer coefficient directionally. A CFD (computational fluid dynamics) process is studied in this study using ANSYS fluid fluent. Designing of shell and tube heat exchanger is performed using SOLIDWORKS software which is a CAD modeling software. In this study 4 different type of shell and tube heat exchanger is designed and analyzed to find out thermal behavior of shell and tube heat exchanger. 3 different type of fins are used in this study which is spiral fins, baffles and rectangular fins on inner hot fluid pipe. Results are compared in terms of hot and cold fluid throughout the heat exchanger. And the effectiveness is calculated for each case. According to the results, heat exchanger with having baffles and rectangular fins provides better effectiveness as compared to other type of fins.**

Introduction

Heat exchangers are devices used to transfer heat between two or more liquids. These usually contain a lot of active or auxiliary fluid such as water or air which removes or absorbs heat from valuable substances such as oil, petrochemical and liquids. There are many types and designs of heat exchangers. Hot and cold liquids can be separated by a highly conductive wall (usually steel or aluminium pipe) or can be connected directly. The water can have similar or different components (such as water-liquid, carbon dioxide). Changes in both fluid parameters are considered by the manufacturer during the manufacturing process.
Heat exchanger are different from oil, electricity or nuclear reactors such as boilers. The source of the heat and the receiving medium must be liquid. Liquid is defined as anything that moves under the shear or external force, including liquids, air, and steam.
Heat exchanges are widely used in many industries such as the food, medical, scientific and medical industries, where heating or cooling is the final or intermediate phase in the preparation of beverages for further processing. They can also be used to control germs in food and medicine. There are many cases in which the use of heat exchangers is considered to be effective.
Heat exchanger is a device that converts heat between two or more liquids at different temperatures and exposure to heat. In Heat exchanger, heat dissipation occurs from very hot water to cold water. Usually the fluid portion is not changes in temperature.
Heat exchanger can be classified according to their modification methods, its operating system, fluid component, and geometric design. Several possible components are briefly described in the section. Because Heat exchanger transfers heat energy between water, it is simpler mathematics and physical bases of thermal conductivity are also provided.

All types of heat exchangers work using the same thermodynamic principles as the heat transfer method. These principles describe how heat energy is transferred to a greater degree. The three bodies connect in a heat transfer system: hot water, cold water, and a wall separating the two waters. Energy flows from hot water through a wall or block and then into cold water. Below are some thermodynamic tips to help you understand how heat exchangers work: A Heat exchanger is a device designed to be able to transfer heat from one fluid medium to another fluid medium. They support the temperature between two water at different temperatures by preventing them from mixing. Various functions of the heat exchanger, evaporators, air conditioning and refrigeration boilers, etc. The heat exchanger works in car radiators.

Heat exchanger is used in industry and engineering in varied field. The process of making heat exchangers is very complicated, since an accurate analysis of thermal conductivity, efficiency and intermediate temperature is required.

Literature review

Initially, the starting model is evaluated with the appropriate parameters and the desired results are obtained. This process is repeated for all the three cases to determine optimum result and then a optimum profile is evaluated to ensure a better heat transfer by fins and their consequences in comparison to efficiency.
Zhang, Zhe et al.(2003) work in this paper is focussing on plate type fins which is using in HE due to its compact design of this type of HE due to shape of this type of HE demand is much more in various variety of industry like aero science etc.in this study assumption has taken as fluid distribution is uniform and few authors have studied the inconsistent distribution of fluid discharge using computational fluid dynamics (CFD) simulation methods, especially the results changes in collectors and distributors on the move distribution of plate-fin heat exchangers. CFD modelling technology can provide flexibility in architecture Computational models that can be easily adjusted to a wide range various material without binding test or high-cost test benches.

CFD is able to provide a better platform where diversity in design options are available and cab also be tested with the appropriate design whatever required at low cost. The purpose of this activity is to explore influence different water flow distribution systems in plate-fins for heat transfer and optimization design of several plate-fin heat exchanger, related to flow distribution to minimize the effects of inconsistent or uneven distribution of fluid in the pipe. [1]

Ambekar, A.S.et al.(2016) Shell and tube HE is simple and compact in designed very useful in various areas of application in this type of HE, baffle has been used by which to support to the tubes, and velocity of flow also improved. Different type of baffle has been used and flower type baffle are showing most promising result in which pressure decreases by almost 30% also triple segment baffle are not adequate for proper heat transfer by providing baffle to the pipe flow deflected to its path so turbulence created therefore heat transfer increases.[2]

Tiwari et al.(2014) in this study, numerical analysis of HT and fluid flow with the new kind of fluid with nanofluid used. Nano fluid is very powerful to minimize thermal resistances which is very beneficial for industries. Nano fluid generally used where temperature and heat fluxes are not varying largely. in this study objective to use of nanofluid and showing of distribution of flow and htp and ffc of different nanofluid which has been consider as homogeneous and by the use of nanofluid heat transfer between pipes increases upto certain extent [3]

S. Y. Kim et al. (2000) Porous type of fins used in HE which is compared to louvered type fins which show comparatively good performance in pressure difference. Low porous fin are preferred in this study for better performance. reviews in study show porous fins mostly used on geothermal energy and friction factor has been calculated.in the analysis ,experiment result show friction factor is less for minimum permeable fins compared to large surface fins.[4]

Ralph L. Webb Paul Trauger et al.(1991) in this study uncommon type of fin used to increase heat transfer however, louvered fin shows a little better performance based on a pressure drop also in some other aspects . For compactness of heat exchanger especially porous fins with a high pore density preferably used. The correlation of friction coefficient with modified j-factor is also given in the design plate-fin heat exchanger. Even louvered fins have existed since then in the 1950s, only in the last 20 years great effort has been made to understand its movement events and practices of the louvre fins. In next section numerical results of the form of spiral heat exchanger are given. After all, heat exchanger was reviewed in the case of fluid and heat transfer and comparing the results of the calculations with the experiments. [5]

Flavio C.C. Galeazzo et al(2006) The main purpose of this work is to identify the effects of these components in the operation of the matching heat sink plate and to identify how to best use them and use of a plate heat exchanger is more efficient than a basic heat exchanger with the same volume, because it takes up less space. The analysis was performed using the ANSYS CFD method. Distinguished sections are calculated from the obtained results, and drawings are built between different sections and by meshing pictographic representation have given. The pictures have been studied and discussed to get the best results which the plate heat exchanger will give the best performance. In these papers, they are designed to form an internal bond depending on the temperature relative to the water and water temperature in the spiral fin heat exchanger. There is a trial setting designed to conduct heat transfer studies In addition, the phenomenon of heat transfer in a heat exchanger numerical analysis using CFD code FLUENT. [6]

Mourad Yataghene et al . (2008) Scraped surface HE study by cfd to calculate shear stress for different type of fluid and two dimensional system study has been done to simulate in cfd and comparisons done with prediction value. [7]

S Sanaye et al.(2010) in this study for compactness of the HE thermal modelling with optimal design has been done with the help of NTU method of plate fin HE and Effectiveness calculation with the help of decision variable has been presented in this study also for the verification modelling result which is compared with reported results. Here effort has made to improve effectiveness and reduce total cost. When increasing cold fluid flow length cost of investment also increases because of increasing of area of heat transfer.so here trying to reduce size of HE. [8]

Ender Ozden et al.(2010) Work in this paper focussing on CFD simulation are run with a wide range of baffles as well as in this study increased turbulence. Results of the analysis became when choosing a model an excellent example of a middle ground determined by comparing CFD effects of heat transfer coefficient, heat transferred. In two part of the baffle cut off, the effect of the baffle in the pipe for the fluid affects much more and ratio of the distance between the shells to the size of the shell during the operation of HE is checked when changing the flow rate. The inner tube movement in the engineering of the shell and tube is very difficult for the simulation and for the pressure distribution in smaller tubes. HE due to the many lost and passive routes it travels between different travel areas also for different types of HE, the importance of any loss and in different directions may be different. However, the patterns observed with the CFD simulation showed that it is a small fraction of the segments of HE in which baffle has given with proper spacing, some surrounding areas are created behind baffles. By increasing the number of baffle fins, turbulence increases, and the form of the heating. In short, communication channels may indicate presence structural weaknesses, but CFD adaptation is very useful for analysis of HE flow and heat transfer. The use of CFDs is test aids, capable of accelerating the temperature change of the shell and the tube making a plan and can improve the final quality and improving quality of HE. Recently, a change in computer technology will perform all CFD simulations of small shell and tube heat exchanger. [9]

R Misra et al.(2013) In this work with the use of MATLAB study has been done and MATLAB predicts energy saving capabilities and Global warming machine in the world. This example included influence of soil temperature gradient, surface culture, humidity contained in the design of the Earth Air tunnel heat they found that increase the height of the buried pipe and the height of the ground above the pipes.by increasing the possible cooling capacity of system in which cooling is required. CFD analysis was used to resolve change the field temperature around the horizontal burial pipe and for EATHE using grid. Temporary and Invisible Numbers an example of simultaneous heat transfer and flow was designed to reflect how heat works as well lighting the cooling of the Earth Air Tunnel HE system however model includes the effects of air turbulence. On the hot form EATHE By conviction temperature heat transfer takes place, object type and mesh size firstly determined for study. Selected as modification so that the calculation can be based on reality condition is to achieve a high degree of accuracy. This is because if the speed increases of fluid , the heat is transferred to the ground from the atmosphere over time, and these temperatures accumulate in the soil sections near the pipe above the ground are minimal Thermal conductivity. Therefore, increase the flow rate has a detrimental effect on EATHE temperature system. [10]

P.C. Mukesh Kumar et al.(2019) In this work Design of heat exchanger and heating system right now it is getting worse because of complications meeting current cooling needs. Many researcher have worked hard ways to increase heat instead of heat transfer additional method. It has been studied that temperature and humidity decreases with increase in volumetric concentration of liquid. Nusselt number confirmed to be minimum than MWCNT. A similar deviation between the Nusselt number and a sharp decrease in CFD analysis obtained data. They showed that the circumference of the circle is two of the inner tube affects the thermal and hydraulic properties of ground laminar fluid. They also indicated that flow was coming configuration provides better results compared to streaming configuration of HE.[11]

Dipankar De et al.(2017) In the study of HE everything in modelling were developed using the tools of the CATIA program. This article shows how the decrease in strength and temperature fluctuates due to a different part of the HE spiral fins has been used also for which calculation done when pressure is the same. The movement system in the inner space of the heat exchanger is continuous helical baffles are forced to be round due to the geometry of continuous helical baffles, which results in a significant increase in the average temperature of the unit to a decrease in ambient temperature. Modelling and preprocessor tool temperature converter made using the CATIA software tools with element-based parametric solid modelling system with many high-end designs and Software. In inner parts, the downward slope is much lower than conventional heat exchanger and baffle used for increase flow and turbulence. Stress on wall decreases with the amount of the entire helix angle were considered.so varied variety of fins used in study for improvement in heat transfer.[12]

Triloki Nath.et al.(2015) HCHE consists of a helical coil made of steel material and pipe inserted inside the tube on two concentric sides of cylinder, with the help of ANSYS software used for optimization of heat transfer , speed of fluid flow. removal of the running fluid coming outside which is cold water at continuous speed in the outer tube and hot water at variable speed in smaller diameter tube . We also find tangential pressure on the wall on both internal and external pipe. Fluid has considered as water for internal and external pipe. The results of the research show that the temperature, the run-speed area in the HE was similar to the literature data and It also appears from the results that Nusselt number depends on curvature factor which is necessary for the calculation which is increases with increasing coefficient of curvature. Also, the Nu was found increase and multiplication of mass and inlet speed of fluid, and increase in variable curvature. For the design of heat exchanger, have curved pipes or circulating heat exchangers, must have heat transfer and better effectiveness which is characterized by various coil modifications inside pipe of HE and fluid particles of water formed oscillatory flow inside both pipes. On the outside the speed of the pipe and the force were even greater compared with internal qualities. For increase of heat transfer fin of different variety has been used.[13]

Rahul Kharat et al. (2009) In this work analysis of HE with spiral fin are profitable because their high heat transfer coefficient and compactness in comparison straight pipes. Use of HE in many industries with development in the manufacturing industry in particular because of the price and power of heat exchangers, who it needs to be based on the correct and accurate temperature and heat transfer. Mathematical calculation were made to analysing data obtained from CFD and experimental results taking into account various influence Adjustments based on depend on functionality such as fixed coil width, pipe length, and coil inches which affects heat transfer. Evaluation of new technology with the evolve of a computational method which has its new correlation of varied temperature that occurred in study provides the exact similarity test results are within the range of 4-5% error which is very minimum as compared with numerical method. In huge number research on circular tubes has been done in which transfer of heat based on characteristics of fluid flowing inside helical tube of HE. The purpose of this activity is to study the characteristics of thermal conductivity. Fluid flows outwards of the surrounding coil and therefore forming connection based on the temperature required and heat transfer. Analyse how two pipes at heat transfer coefficient, temperature different pipe diameters when keeping two spring coil tubes has been submersed in the tube which is of coil form, outer coil spring coil in outer diameter pipe has been fitted with flow of constant fluid speed. Internal and the outer diameters of the coils are kept indiscriminately to minimize its effects hydraulic diameter on a heat transfer device heat transfer coefficient increases and decreases two tube. This result is very interesting and inconsistent. This is mainly related to the reduction in coil variation and the increase in the diameter of the pipe and its effect the diameter of the pipe is not differentiated by the strength of the coil difference. Various type to analyse heat transfer through HE with the use of fins.[14]

S.S. Pawar et al.(2014) In the study of HE through the cfd with the use of helical coil fins. In a helical coil fins, whenever there is a Newtonian fluid, there is a second outflow in addition to a large outflow, Straight currents in the fluid flow in the vertical tube are replaced by streamlines in the coils, resulting in a higher flow rate and turbulence fluid flow. This results in smaller dimensions and greater heat transfer compared to a straight pipe. As a result, spiral coil heat exchanger is often used in the production of electricity, nuclear power, refrigeration and air conditioning, heating systems, and other heating industry for heating and cooling purpose and second pressure induced by centrifugal force has the potential to increase the thermal conductivity in helical type fins. Grid formed through the dependence of a three different grid solution was studied until the strongest grid was completed. Fine meshing on small objects or three layers of prismatic boundary approx. 5000,000 cell is not possible at heavy geometry, which is beyond the computing power of CFD. In addition, with a large number of counting cells, the cost of computation will increase, as the integration and stability of the repetition will be more difficult. For various other connections have been made in various experiments, covering the outflow controls from the laminar to turbulence flow, as well as the connections that cover the outlet of the laminar only when there is Newtonian water. All of these links are compared to the work of ancient researchers and are closely related to numerical result which is obtained without CFD. In one study related to cfd analysis on the inlet of heat exchanger, the computational fluid dynamics (CFD) FLUENT system was used to predict flow of fluid under the temperature Consequences of uniform and non-uniform distribution of plate-fin heat exchanger according to system modification of plate-and-fin heat exchanger used in industry. Numerical data show a clear agreement with test result. According to the results of this study, two moderated collectors with two sub divisional components are described in this article. It has been proven that fluid distribution occurs at plate-fin temperatures the temperature change is the same if the output ratios and the same diameter of the collector are all the same.[15]

Conclusion

Shell and tube heat exchanger is analyzed in this study with using computational fluid dynamics by varying the design of fins inside of shell and tube heat exchanger. 3 type of fins are analysed which are spiral fins, baffle fins and rectangular fins along with inner tube. Here the fluid is selected as water and material of pipe is considered as copper. Hot fluid is flowing through inner tube which decreases its temperature by contacting with cold fluid throughout the pipe. Here the case of rectangular fins provided better results in terms of temperature decrease which is 355.18 K., and the effectiveness of heat exchanger is calculated and the highest effectiveness of 0.3946 is provided in case of having rectangular fins.

References

[1]Zhang, Zhe, and YanZhong Li. "CFD simulation on inlet configuration of plate-fin heat exchangers." *Cryogenics* 43, no. 12 (2003): 673-678.

[2]Ambekar, A.S., Sivakumar, R., Anantharaman, N. and Vivekenandan, M., 2016. CFD simulation study of shell and tube heat exchangers with different baffle segment configurations. *Applied Thermal Engineering*, *108*, pp.999-1007.

[3]Tiwari, Arun Kumar, Pradyumna Ghosh, Jahar Sarkar, Harshit Dahiya, and Jigar Parekh. "Numerical investigation of heat transfer and fluid flow in plate heat exchanger using nanofluids." *International Journal of Thermal Sciences* 85 (2014): 93-103.

[4]Kim, S. Y., J. W. Paek, and B. H. Kang. "Flow and heat transfer correlations for porous fin in a plate-fin heat exchanger." *J. Heat Transfer* 122, no. 3 (2000): 572-578.

[5]Webb, R.L. and Trauger, P., 1991. How structure in the louvered fin heat exchanger geometry. *Experimental Thermal and Fluid Science*, *4*(2), pp.205-217.

[6]Galeazzo, F.C., Miura, R.Y., Gut, J.A. and Tadini, C.C., 2006. Experimental and numerical heat transfer in a plate heat exchanger. *Chemical Engineering Science*, *61*(21), pp.7133-7138.

[7]Yataghene, Mourad, Jérémy Pruvost, Francine Fayolle, and Jack Legrand. "CFD analysis of the flow pattern and local shear rate in a scraped surface heat exchanger." *Chemical Engineering and Processing: Process Intensification* 47, no. 9-10 (2008): 1550-1561.

[8]Sanaye, Sepehr, and Hassan Hajabdollahi. "Thermal-economic multi-objective optimization of plate fin heat exchanger using genetic algorithm." *Applied energy* 87, no. 6 (2010): 1893-1902.

[9]Ozden, Ender, and Ilker Tari. "Shell side CFD analysis of a small shell-and-tube heat exchanger." *Energy Conversion and Management* 51, no. 5 (2010): 1004-1014.

[10] Misra, Rohit, Vikas Bansal, Ghanshyam Das Agrawal, Jyotirmay Mathur, and Tarun K. Aseri. "CFD analysis based parametric study of derating factor for Earth Air Tunnel Heat Exchanger." *Applied Energy* 103 (2013): 266-277.

[11]Kumar, P.M. and Chandrasekar, M., 2019. CFD analysis on heat and flow characteristics of double helically coiled tube heat exchanger handling MWCNT/water nanofluids. *Heliyon*, *5*(7), p.e02030.

[12] De, D., Pal, T.K. and Bandyopadhyay, S., 2017. Helical baffle design in shell and tube type heat exchanger with CFD analysis. *International Journal of Heat and Technology*, *35*(2), pp.378-383.

[13]Mishra, Triloki Nath. "modeling and CFD analysis of tube in tube helical coil heat exchanger." *International Journal of Science and Research (IJSR) ISSN (Online)* 4, no. 8 (2015): 2319-7064.

[14] Kharat R, Bhardwaj N, Jha RS. Development of heat transfer coefficient correlation for concentric helical coil heat exchanger. International Journal of Thermal Sciences. 2009 Dec 1;48(12):2300-8.

[15] Pawar, S. S., & Sunnapwar, V. K. (2014). Experimental and CFD investigation of convective heat transfer in helically coiled tube heat exchanger. *Chemical Engineering Research and Design*, *92*(11), 2294-2312.

[16] Ranganayakulu, C., & Seetharamu, K. N. (2018). *Compact heat exchangers: Analysis, design and optimization using FEM and CFD approach*. John Wiley & Sons.

[17] Lutcha J., Nemcansky J. (1990). Performance improvement of tubular heat exchangers by helical baffles, Chemical Engineering Research and Design, Vol. 68, pp. 263- 270. DOI: 10.1155/2011/839468

[18] Shinde S., Pancha M.H. (2012). Comparative thermal performance analysis of segmental baffle heat exchanger with continuous helical baffle heat exchanger using kern method, IJERA, Vol. 2, No. 4, pp. 2264-2271.

[19] W.M. Kays, A.L. London, Compact Heat Exchangers, third ed. Mc-Graw Hill, New York, 1984.

[20] B. Kim, B. Sohn, An experimental study of flow boiling in a rectangular channel with offset strip fins, International Journal of Heat and Fluid Flow 27 (2006) 514e521.

[21] Varga T., Szepesi G., Siménfalvi Z. Horizontal scraped surface heat exchanger - Experimental measurements and numerical analysis, Pollack Periodica, Vol. 12, No. 1, 2017, pp. 107–122.

[22] G.N. Xie, B. Sunden, Q.W. Wang, Optimization of compact heat exchangers by a genetic algorithm, Applied Thermal Engineering 28 (2008) 895e906.

[23] Jiang-Jiang Wang, You-Yin Jing, Chun-Fa Zhang, Optimization of capacity and operation for CCHP system by genetic algorithm, Applied Energy 87 (2010) 1325e1335.

[24] Serna M., Jimenez A. (2005). A compact formulation of the Bell Delaware method for Heat Exchanger design and optimization, Chemical Engineering Research and Design, Vol. 83, No. A5, pp. 539-550. DOI: 10.1205/cherd.03192

[25] Jiang-feng Guo, Mingtian Xu, Lin Cheng, The application of field synergy number in shell-and-tube heat exchanger optimization design, Applied Energy 86 (10) (October 2009) 2079e2087.

[26] A.R. Doodman, M. Fesanghary, R. Hosseini, A robust stochastic approach for design optimization of air cooled heat exchangers, Applied Energy 86 (2009) 1240e1245.
[27] Z. Liu, H. Cheng, Multi-objective optimization design analysis of primary surface recuperator for microturbines, Applied Thermal Engineering 28 (2008) 601e610.
[28] S. Sanaye, H. Hajabdollahi, Multi-objective optimization of rotary regenerator using genetic algorithm, International Journal of Thermal Sciences 48 (2009) 1967e1977.
[29] A.K. Gholap, J.A. Khan, Design and multi-objective optimization of heat exchangers for refrigerators, Applied Energy 84 (2007) 1226e1239.
[30] Qiu G., Sun J., Ma Y., Qu J., Cai W. Theoretical study on the heat transfer characteristics of a plain fin in the finned-tube evaporator assisted by solar energy, Int. J. Heat Mass Transf, Vol. 127, Part B, 2018, pp. 847–855.
[31] Al-Fahed S, Chamra L.M, and Chakroun W, 1998. Pressure drop and heat transfer comparison for both microfin tube and twistedtape inserts in laminar flow, Experimental Thermal Fluid Science, Vol. 18, No. 4, pp. 323–333.
[32] A. Bejan, G. Tsatsaronis, M. Moran, Thermal Design and Optimization. John Wiley and Sons Inc., 1996.