



e-ISSN: 2319-8753 | p-ISSN: 2347-6710

IJRSET

International Journal of Innovative Research in
SCIENCE | ENGINEERING | TECHNOLOGY

INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

Volume 12, Issue 7, July 2023

ISSN INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA

Impact Factor: 8.423

9940 572 462

6381 907 438

ijirset@gmail.com

www.ijirset.com

Computational Fluid Dynamics Analysis of Shell and Tube Heat Exchanger- A Review

Aviral Verma¹ Dr. Nitin Tenguria²

Research Scholar, Department of Mechanical, Sagar Institute of Research and Technology, Bhopal, India¹

Head of Dept., Department of Mechanical, Sagar Institute of Research and Technology, Bhopal, India²

ABSTRACT:- Consider the shell and tube heat exchanger, in which cold water flows over a tube with hot water flowing through it. The simulation of the heat exchanger's fluid flow and heat transfer is done using a method called computational fluid dynamics. To determine the temperature gradients, pressure distribution, and velocity vectors, CFD resolves the complete heat exchanger into discrete parts. For reliable CFD results, the k-model for turbulence is used. By adjusting the mass flow rate of fluid of 2L/min and 3L/min, which is controlled by a rota metre, the temperature variations are calculated from experiment for parallel and counter flow, and the temperature fluctuations are detected by the sensors connected at the inlets and outlets of the tube. The model geometry is meshed using the ANSYS 13.0 pre-processor GAMBIT after the solid geometry has been created in the SOLID WORKS software. The simulated results, such as temperature contours, pressure contours, and velocity vectors, are generated using the post processor Fluent. The objective of the project is design of shell and tube heat exchanger with helical baffle and study the flow and temperature field inside the shell. The heat exchanger contains 7 tubes of outer diameter 20 mm and a 600 mm long shell of inner diameter 90 mm. The helix angle was varied from 00 to 200. The simulation shows how the pressure varies in shell due to different helix angle and flow rate.

KEYWORDS: CFD, HE, ANS, S&T

I. INTRODUCTION

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. Heat exchanger is required in process which involves cooling, heating, condensation, boiling or evaporation.

Heat exchangers devices used for transferring thermal energy between two fluids. They are widely used within the petrochemical, chemical and pharmaceutical industries, power stations and also in the households for different applications, such as heating, cooling, condensation and evaporation process. The most common used type of them is the shell-and-tube heat exchangers. These have a lot of advantageous properties compared to the others: have a high ratio of volume and heat transfer area, relatively easy to manufacture, easy to clean and repair, and able to transfer high mass flow rates. Furthermore, easy to increase the performance of these equipment with the usage of baffles. With these components, the flow direction could be guided, the whole heat transfer area is involved in the heat transfer and due to the decreased flow section, the velocities and the turbulence will be higher, and this increased value will cause a higher heat transfer coefficient and heat performance.

The most important geometric parameters are the tube arrangement, the type, the orientation and the opening of the baffles, the space between the baffles, the position of the most extreme baffles and any other heat transfer improvement possibilities used the Kern's procedure to determine the external heat transfer coefficient, examined different segmental baffle types (single, double, triple and flower). investigated the effect of the usage of different turbulence model, while used the number of transfer unit method with different tube layout. The other way to make artificial route to the shell side fluid to create scraped blades or made helical baffles for the more favorable flow arrangement. On the other hand, these baffles will increase the additional costs: the manufacturing and cleaning of these appliances more difficult than the simple ones, and the pressure drop will be also much higher, which requires a more powerful pump. Further, the analytical calculation is harder and shows a higher difference from the reality. The difficulty is caused by the different values of velocity in the different direction and the difficulty of the heat transfer processes.

The locations of the baffles inside the shell-and-tube HEs investigated the heat transfer of a two passes tube side and a one pass shell side heat exchanger with segmental baffles. In that study they showed that the performance is strongly depending on the number of baffles, baffle size, distance between baffles, the shape of the baffle and the first and last baffle position to the inlet and outlet nozzle of the shell side. continued this study, they investigated the effect of the segment type baffles. They tested two baffles spacing and six baffles orientation angles and showed that the decreased baffle spacing causes a higher performance, and the maximum angle of 180° between two adjacent baffles will cause the highest performance. investigated two different type of baffles: the sector-shaped plain baffles and the ladder-type fold baffles. Both baffles run the fluid much more tightly, than the segment type baffle, so the heat transfer at these cases will higher.

The disadvantages of these type are the increased pressure drop and the manufacturing time and costs. Applying CFD (Computational fluid dynamics) software becomes more and more important to investigate the heat transfer processes. Nowadays, the most investigated baffle type is the helical baffles. Especially at this appliance has the greatest importance of the numerical simulations, because of the difficulties at the manufacturing and maintenance. Due to the high performance computers and the high speed computing softwares the heat exchanger constructions can be investigated. Without the expensive manufacturing and long investigation time the equipment can be comparable to each other. The aim of this study to determine the reliability of the empirical correlations.

1.1 Shell and tube heat exchangers

Shell and tube heat exchangers consist of a series of tubes. One set of these tubes contains the fluid that must be either heated or cooled. The second fluid runs over the tubes that are being heated or cooled so that it can either provide the heat or absorb the heat required. A set of tubes is called the tube bundle and can be made up of several types of tubes: plain, longitudinally finned, etc. Shell and tube heat exchangers are typically used for high- pressure applications. This is because the shell and tube heat exchangers are robust due to their shape. Heat exchangers are widely used in industry both for cooling and heating large scale industrial processes. The type and size of heat exchanger used can be made to suit a process depending on the type of fluid, its phase, temperature, density, viscosity, pressures, chemical composition and various other thermodynamic properties.

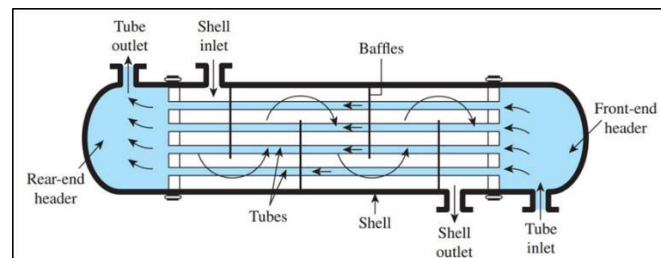


Figure 1.1: Shell and tube heat exchanger

1.2 Computational Fluid Dynamics (CFD)

CFD is a sophisticated computationally-based design and analysis technique. CFD software gives you the power to simulate flows of gases and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid-structure interaction and acoustics through computer modelling. This software can also build a virtual prototype of the system or device before can be apply to real-world physics and chemistry to the model, and the software will provide with images and data, which predict the performance of that design.

Computational fluid dynamics (CFD) is useful in a wide variety of applications and use in industry. CFD is one of the branches of fluid mechanics that uses numerical methods and algorithm can be used to solve and anal y se problems that involve fluid flows and also simulate the flow over a piping, vehicle or machinery. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. More accurate codes that can accurately and quickly simulate even complex scenarios such as supersonic and turbulent flows are ongoing research. Onwards the aerospace industry has integrated CFD techniques into the design, R & D and manufacture of aircraft and jet engines. 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 (Figure 1.2). Furthermore, motor vehicle manufactures now routinely predict drag forces, under bonnet

air flows and surrounding car environment with CFD. Increasingly CFD is becoming a vital component in the design of industrial products and processes.

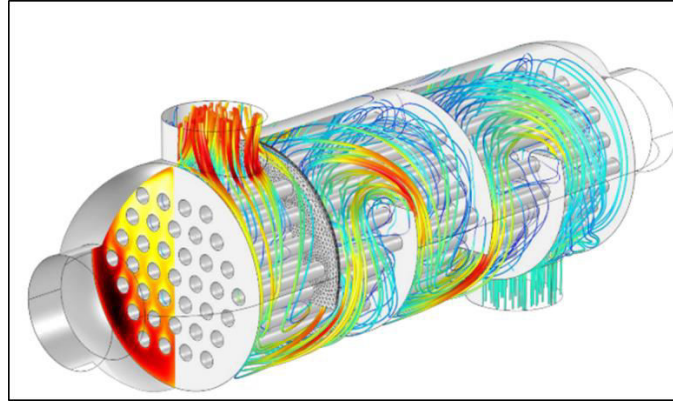


Figure 1.2: Fluid flow simulation for a shell and tube exchanger

II. REVIEW OF LITERATURE

E. J Fernandes and S. H. Krishanmurthy (2022) in the research paper, baffle spacing and its effect on heat transfer was analyzed using CFD analysis and compared these results with the theoretical analysis. The Design and modelling of the heat exchanger was modelled using PTC Creo parametric and using ANSYS Fluent CFD analysis was carried out considering copper, aluminum, and steel as the materials.

Results stated that baffle spacing place a significant role in the heat transfer rate in case of shell and tube heat exchangers. The selection of the material has a significant impact on heat transfer rate in case of shell and tube heat exchangers. Results concluded that shell and baffles made of copper has performed better as compared to stainless steel and aluminum. However, due to high cost, copper cannot be used for all parts of shell and tube boiler but can be used as baffles. Also, copper has moderate corrosion resistance but tends to discolor easily. In these cases, aluminum would be the next best option as it is lightweight.

Pushparaj Singh and Mukesh Kumar Gautam (2020) research paper analyzed the performance of heat exchanger by using experimental and Computational fluid dynamics so we categories these work in two different phase. Analysis has been perform in two different phases, in first phase we prepare one setup of shell and tube type heat exchanger with brass tube for shell for cold water and straight copper tube of 500 mm for hot water due to its good thermal conductivity. ANSYS was for the purpose of modelling and analysis. Results concluded that the CFD result show better result as compare to the experimental result.

Vivek Singh Parishar et.al (2019) research paper presented the designing and comparative analysis of possible model of the heat exchanger to enhance the performance of the device. The straight tube was replaced by a helical tube in the heat exchanger and used 10, 12, 14 number of helical baffles with 50% baffle cut. Total ten models have been developed. CFD analysis simulation was done on ANSYS FLUENT 18.

Results stated that with increasing the number of turns the hot fluid temperature difference increase till 6-Turns and in 7-Turns the temperature difference in hot fluid again decrease.

Máté Petrik and Gábor L. Szepesi (2018) research paper numerically investigated a model-size shell-and-tube heat exchanger with horizontal baffles and compared to the measured values using the commercial software SC-Tetra V11. Primary objective of research work was to analyze the effect of the type of the baffles, the space between them and the baffle cut to the real heat transfer coefficient and the necessary material quantity and compared the simulated heat transfer coefficients with analytically calculated ones based on the literature.

Results stated that usage of baffles in the shell side increases the overall heat transfer coefficient in every case. The empirical correlations in the literature can provide good preliminary results and can narrow the range of the geometric parameters to achieve a construction with a lower cost.

Pramod Deshmukh et.al (2016) in the research paper, counter flow analysis of helical coil tube was performed where the contours of temperature of the tube were calculated and plotted using ANSYS FLUENT 15.0. Copper was chosen as the metal for the construction of the helical tube. The fluid flowing through the tube was taken as water. First law of thermodynamics concept plays a vital role in helical coil tube heat exchanger.

The CFD results when compared with the experimental results from different studies were well within the error limits. The simulation was carried out for water to water heat transfer characteristics and different inlet temperatures were studied. Characteristics of the fluid flow were also investigated for the constant temperature and constant wall heat flux conditions.

Praveen Kumar Kanti et.al (2016) research paper aimed to perform a CFD analysis of shell and tube heat exchanger using FLUENT 15.0 after modelling the heat exchanger with CFD Design moduler 15.0 in order to determine the effectiveness and temperature rise in the heat exchanger, for counter flow with copper as inner material and stainless steel as outer material.

The comparison of Stainless Steel and Galvanized Iron reveal that Stainless Steel is a better outer material when coupled with inner material of Copper for a heat exchanger. The effectiveness and efficiency of Stainless Steel was better than Galvanized Iron.

Beigzadeh et.al (2015) research paper reported experimental and Computational Fluid Dynamics (CFD) investigations on heat transfer and flow characteristics of a shell and helically coiled tube heat exchanger. The experiments were carried out using a helically coiled tube, which was placed in a cylindrical shell. Hot and cold water were used as the process fluids on the tube and shell side, respectively. The CFD modeling technique was employed to describe the experimental results, fluid flow pattern, and temperature profiles as well as dead zones in the heat exchanger.

Quantitative predicted results of CFD modeling show a good agreement with the experimental data for temperature. The effect of the coil pitch on heat transfer rate was numerically studied and it was found that the heat transfer coefficient intensifies with an increase in coil pitch. The average turbulent kinetic energy (k) for the old coil tube and twice coil pitch heat exchanger was computed as 2.9×10^{-3} and $3.3 \times 10^{-3} \text{ m}^2/\text{s}^2$, respectively. This indicates an increase of about 14% in flow turbulent kinetic energy. Nusselt numbers were compared with those estimated using published correlation and a mean relative error (MRE) of 14.5% was found between the experimental and predicted data. However, a good agreement was obtained in lower shell Reynolds numbers (lower than $Re=200$).

Gurbir Singh and Hemant Kumar (2014) in the research paper, the shell and tube heat exchanger was considered in which hot water is flowing inside one tube and cold water runs over that tube. Computational fluid dynamics technique which is a computer based analysis was used to simulate the heat exchanger involving fluid flow, heat transfer. The solid geometry is made in SOLID WORKS software and then imported into GAMBIT which is the pre-processor of the ANSYS 13.0 for meshing the model geometry. Using the post processor FLUENT, the simulated results are computed i.e. temperature contours, pressure contours and velocity vectors.

Analysis stated that there was a difference between temperatures values computed from the experiment and the simulation by ANSYS 13.0. CFD helps to design the heat exchanger by varying the different variables very easily otherwise it is very difficult if done practically. CFD models or packages provides the contours and data which predict the performance of the heat exchanger design and are effectively used because it has ability to obtain optimal solutions and has work in difficult and hazardous conditions.

III. CONCLUSION

The entire research in past shows that authors tried to analyze structure with differet lateral load resisting members under lateral loading condition. But none of them shows variations observed due to effect of different positioning of CFD.

REFERENCES

1. Patricia Anne D. Cruz, Ed-Jefferson E. Yamat, Jesus Patrick E. Nuqui and Allan N. Soriano, [Computational Fluid Dynamics (CFD) analysis of the heat transfer and fluid flow of copper (II) oxide-water nanofluid in a shell and tube heat exchanger], Digital Chemical Engineering 3 (2022) 100014.
2. Máté Petrik and Gábor L. Szepesi, [Shell Side CFD Analysis of a Model Shell-and-Tube Heat Exchanger], CHEMICAL ENGINEERING TRANSACTIONS VOL. 70, 2018.

3. Pramod Deshmukh, Vikram D Patil and Baviskar Devakant, [Cfd Analysis Of Heat Transfer In Helical Coil Tube In Tube Heat Exchanger], International Journal Of Innovations In Engineering Research And Technology [IJERT] ISSN: 2394-3696 VOLUME 3, ISSUE 1, JAN.-2016.
4. Vivek Singh Parihar, Shrikant Pandey, Rakesh Kumar Malviya and Palash Goyal, [CFD Simulation of Helical Shell and Tube Heat Exchanger Using Optimization Techniques], Journal of Engg. Research Online First Article, DOI:10.36909/jer.11985, 2019.
5. Shubham Sharma, Shalab Sharma, Mandeep Singh, Parampreet Singh, Rasmeet Singh, Sthitapragyan Maharana, Nima Khalilpoor and Alibek Issakhov, [Computational Fluid Dynamics Analysis of Flow Patterns, Pressure Drop, and Heat Transfer Coefficient in Staggered and Inline Shell-Tube Heat Exchangers], Hindawi Mathematical Problems in Engineering Volume 2021, Article ID 6645128, 10 pages.
6. R S Anand, Shibin David, M. Gajendiran and Kingston Stanley, [Design and analysis of shell and tube heat exchanger for low temperature applications using CFD], IOP Conf. Series: Materials Science and Engineering 912 (2020) 042056.
7. Praveen Kumar Kanti, Karthika.U.P, Sabeer Ali, Sanath Kumar.N and Shyam Chandran C, [CFD analysis of shell and tube heat exchanger], International Journal of Engineering Research, Volume No.5 Issue: Special 6, pp: 1129 - 1254 20 May 2016.
8. Gurbir Singh and Hemant Kumar, [Computational Fluid Dynamics Analysis of Shell and Tube Heat Exchanger], Journal of Civil Engineering and Environmental Technology, Volume 1, Number 3; August, 2014 pp. 66-70.
9. R. Beigzadeh, A. Parvareh and M. Rahimi, [Experimental and CFD Study of the Tube Configuration Effect on the Shell-Side Thermal Performance in a Shell and Helically Coiled Tube Heat Exchanger], Iranian Journal of Chemical Engineering Vol. 12, No. 2 (Spring 2015), IACHe.
10. Shubham Gupta and Karan Singh Verma, [Computational Fluid Dynamics Analysis Of Shell And Tube Heat Exchanger With Having Different Types Of Fins], International Journal for Research Trends and Innovation, 2022 IJRTI | Volume 7, Issue 2 | ISSN: 2456-3315.
11. Pushparaj Singh and Mukesh Kumar Gautam, [A Comparision Of Shell And Tube Type Heat Exchanger By Experimental And Cfd Analysis], International Journal of Scientific Development and Research (IJS DR), November 2020 IJS DR | Volume 5, Issue 11, ISSN: 2455-2631.
12. Nikhil N. Sawant and Dr. R. G. Desavale, [Computational Fluid Dynamics Analysis of Various types of Shell and Tube Heat Exchanger], Journal of Emerging Technologies and Innovative Research (JETIR), 2019 JETIR May 2019, Volume 6, Issue 5, ISSN-2349-5162.
13. A.Vennila, A.M. Fevija Dhas, Karthick Mahadev Shivpuje and Gokulanaath S., [Parametric Study of a Shell and Tube Heat Exchanger using CFD], International Journal of Engineering Research & Technology (IJERT), ISSN: 2278-0181, Vol. 6 Issue 08, August - 2017.
14. Erica Jacqueline Fernandes and Sachidananda Hassan Krishanmurthy, [Design and analysis of shell and tube heat exchanger], Int. J. Simul. Multidisci. Des. Optim. 13, 15 (2022).
15. Basma Abbas Abdulmajeed and Hawraa Riyadh Jawad, [Analysis of Shell and Double Concentric Tube Heat Exchanger Using CFD Application], Journal of Engineering, Number 11 Volume 25 November 2019.



SJIF Scientific Journal Impact Factor

Impact Factor: 8.423



INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

 9940 572 462  6381 907 438  ijirset@gmail.com



www.ijirset.com

Scan to save the contact details