CFD Thermal Analysis of Heat Exchanger

[1]Shamsuz Zaman, [2] Dr. Sohail Bux
[1]MTech Scholar, [2] Associate Professor
[1]Department of Mechanical Engineering Agnos College of Technology, Bhopal, India
[2]Department of Mechanical Engineering Agnos College of Technology, Bhopal, India
[2] buxsohail@gmail.com

Abstract: Heat transfer is among the core features to maintaining reliability, enhancing product quality, in industrial equipment, appliances and processes. Therefore, heat exchangers of different types and sizes are used in these applications to extract the additional temperature from the process and equipment and retain the desired operating temperature. In this paper, the design process of a heat exchange device must first be theoretically investigated and its efficiency evaluated and optimized by means of computer-assisted fluid dynamics. The project considered a countercurrent heat exchanger, theoretically measured using the LMTD process thus measuring the reduction of pressure and energy consumption using the Kern method. In this analysis, the behavior of the heat transfer, mass flow rates, drops in pressure, flow rates and tube bundle currents eddies were analyzed by the CFD analysis. In terms of hot fluid cooling efficiency, theoretical and CFD results showed a difference of 1.15%. Positive correlations between the axial pressure drops were found both with the total heat exchange coefficient and with the pump capacity. Overall, the study results suggest that CFD modelling is promising for the development and implementation of heat exchangers, enabling you to test several design possibilities without constructing any rapid models.

Keywords: Heat Exchanger, CFD, Pressure, Velocity, Mass Flow Rate.

I. Introduction

Heat exchangers are among the most commonly used devices in the process industry. Heat exchangers are used to transfer heat between two process flows. Their use shows that any process involving cooling, heating, condensation, boiling or evaporation requires a heat exchanger for this purpose. Process liquids are generally heated or cooled before the method or are subject to a phase transition [1]. Diverse heat exchangers are named predictable with their application for example, heat exchangers utilized for buildup are called condensers, even as heat exchangers are called boilers. The presentation and productivity of heat exchangers are estimated by the amount of heat move utilizing the humblest heat move zone and thusly the least weight drop [2]. Proficiency can best be spoken to by figuring the whole heat move coefficient. The weight drop and zone required for a particular heat move give a rundown of the speculation expenses and vitality necessities (working expenses) of a gadget [3]. By and large there's huge amounts of writing and speculations for planning a gadget bolstered the prerequisites.

There are two type of heat exchangers:

- If the two fluids between which heat is exchanged are in direct contact with each other, the heat exchanger is in direct contact.
- If the two fluids are separated by a wall through which heat is transferred so that they never mix, contact the heat exchangers with indirect contact.

A common heat exchanger, regularly for applications with higher weights up to 552 bar, is that the cylinder group heat exchanger. Shell and cylinder heat exchanger, circuitous contact gadget. It comprises of a movement of cylinders through which one among the fluids streams. The bowl is that the holder for the bowl fluid. For the most part, it's a barrel shaped shape with a round cross area, despite the fact that shells of different shapes are used in certain applications. A shell is considered for this specific investigation, which is typically one pass shell [4]. A shell is that the most usually utilized gratitude to its minimal effort and ease and has the absolute best adjustment factor for the log-mean temperature contrast (LMTD). In spite of the fact that the cylinders can have single or various sections, there's an entry on the packaging while the contrary liquid inside the packaging streams over the funnels to be heated or cooled [5]-[10]. Fluids on the cylinder and on the shell are isolated by a cylinder plate.

II. Literature Review

Chamil Abeykoon et al. [1] in this study purpose of the theoretically examine the planning process of a tool, then to research and optimize its performance using computer-assisted fluid dynamics. For configuration purposes, a counter-current gadget was thought of and its length was hypothetically determined utilizing the LMTD strategy, while the weight drop and vitality utilization were determined utilizing the Kern technique.

Devvrat Verma et al. [2] the project consists within the planning of a tube bundle device with a spiral deflector and thus the study of the temperature and the flow of the fields inside the coat utilizing the Ansys programming apparatuses. Furthermore, the heat exchanger of contains 7 channels and a pipe dia meter of 800 mm and a length of 90 mm. The winding edge of the winding divider shifts from 00 to 20°.CFD shows how the temperature within the hull changes due to the varied angles of the spiral and thus the various flow rates.

Mohammed Irshad et al. [3] this article presents the tube bundle heat exchangers most commonly utilized during this scenario. Heat exchangers are devices widely utilized in various sectors like energy production and transportation, refrigeration and chemical process industries as they're suitable for top pressure applications. During this project, a comparison between different tube bundle heat exchangers with segment guide plates is presented.

Neeraj Kumar Nagayach et al. [4] this work provides a summary of the research work of the last decade on the expansion of heat transfer in circular and non-circular tubes. Dynamic and latent techniques are utilized to build the heat move coefficient inside the heat exchanger; Passive strategies don't require outer force, as inside the instance of dynamic strategies.

Gurbir Singh et al. [5] this text examines the tube bundle device, where predicament flows through a tube and cold water flows through that tube. PC helped liquid elements innovation, which is PC helped investigation, is utilized to reenact the heat exchanger where liquid stream and heat move play an errand. CFD disintegrates the whole gadget into discrete components to make sense of temperature inclinations, pressure circulation and speed vectors. The k-ε disturbance model is utilized for definite CFD results.

III. Methodology

The main object of the study calculated the total heat transfer coefficient.

Improve the heat transfer rate by using ANSYS CFD.

During the CFD calculations of the flow in internally ribbed tubes.

Calculated the temperature distribution and pressure inside the tube by using ansys.

Governing Equations

Conservation of mass or continuity equation: The equation for conservation of mass, or continuity equation, can be written as follows:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho \vec{v}) = S_m$$

Where

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho v_x) + \frac{\partial}{\partial r}(\rho v_r) + \frac{\rho v_r}{r} = S_m$$

S_m= mass added to the continuous phase or use of the defined sources. For geometries 2D axisymmetric, and the eq^n is given by
Where:
x = axial coordinate, r = radial coordinate, v_x = axial velocity, and v_r = radial velocity.

Table I: Design Parameters

Parameter	CFD HE 30
D_{tube}	0.02 m
D_{shell}	0.18 m
$D_{inlet,shell}$	0.0508 m
SL	0.03 m

S_T	0.03 m
S_D	0.03354 m
B_F	0.12 m
A_{flow}	0.0216 m
D_{H-T}	0.02m
D_{H-S}	0.0296m
N	16m
n	25m
L	3.12m
C_{tube}	0.01m
Baffle cut%	30

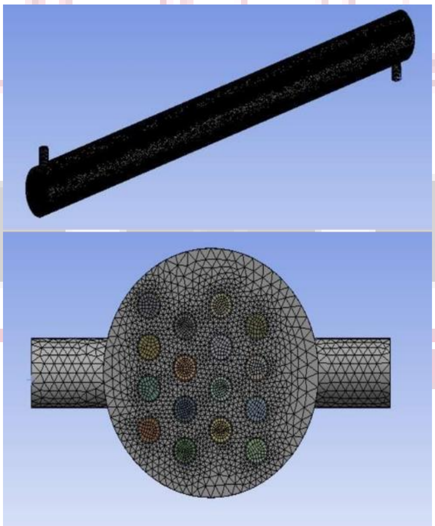

Fig.1. Meshing of Elements

Table II: Boundary conditions for HE 30 CAE boundary conditions for [1].

Boundary conditions	value
$V_{inlet\ shell}$	1.25m/s
$V_{inlet\ tube}$	0.25m/s
Shell inlet temperature	353k
Tube inlet temperature	302k

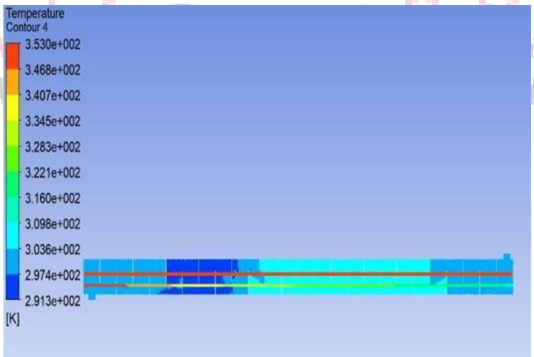

Figure 2: Temperature contours

Figure 3: Pressure contours

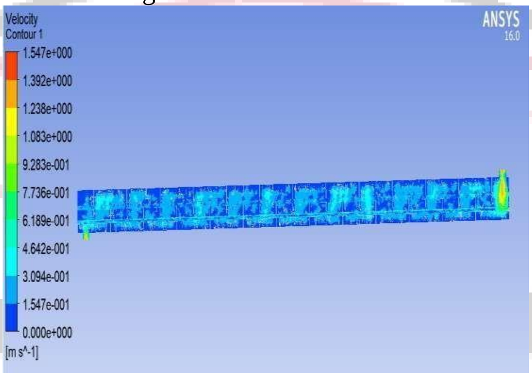

Figure 4: Velocity contours

IV. Result Analysis

The shell temperature decreases and tube temperature increase due to change in heat transfer rate. The figure 5 show the shell outlet temperature and figure 6 show the tube outlet temperature. Figure 7 shows that pressure increases with heat change.

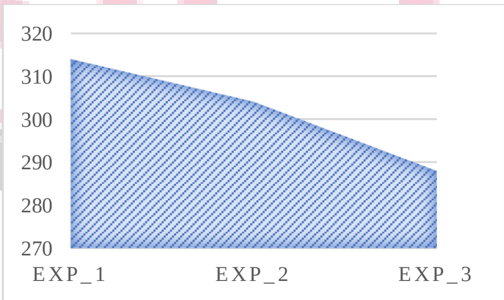

Figure 5: Shell Temperature (in K)

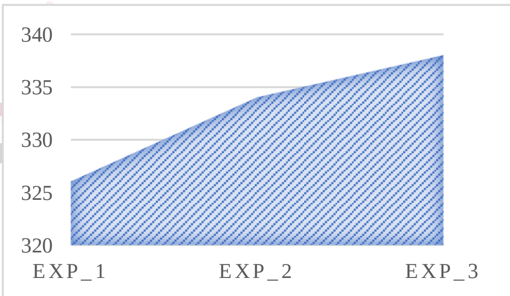

Figure 6: Tube Temperature (in K)

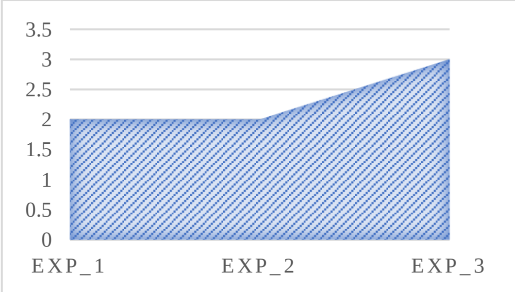

Figure 7: Pressure (in Kpa)

In ours case shell velocity decreases due to change in number of baffles. The Figure 8 and 9 show the comparative shell outlet velocity and tube outlet velocity.

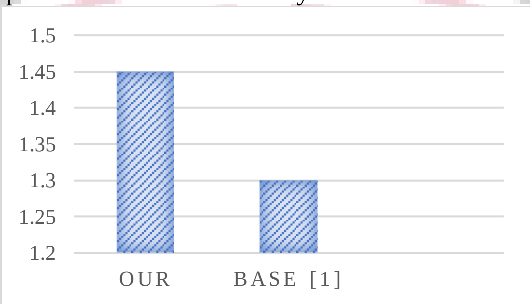

Figure 8: Shell Outlet velocity (m/s)

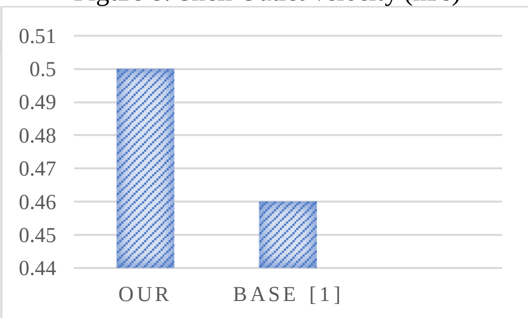

Figure 8: Tube Outlet velocity (m/s)

V. Conclusion

Based on the CFD findings, it became known that it is important to ensure optimum output by carefully selecting parameters such as the deflection ratio, the numbers of deflector and pipes and the flow rate and the layout of the conduits. " a heat exchanger tube bundle for a certain amount of time. The three Cfd simulations evaluated included the best results for the desired operation of the CASE-3 CFD model (with a diaphragm cutting rate of 35%), 16 tubes and 25 deflectors, which is well in consistent with the theoretical results. Decreasing the cutoff intensity by impact raises the coefficient of heat transfer on the box's side, but this also increases the pressure decrease.

Of course, in order to retain the necessary mass flow, the number of tubes used in a heat exchanger has an effect on tube flow. The less pipes, the higher the pressure for a certain operation would decrease. More significantly, the findings of this work align with the previously published research results and prove that the results obtained are correct.

In total, the findings of the experiment confirmed that the design and optimization of a heat exchanger by CFD modelling are promising.

Depending on the stun configuration, the heat transfer rate of the heat exchanger is complete. The architecture of the trunk is also optimized by fluent application of ANSYS for further analysis. The adjustment in the angle of deflection will also increase the total heat transfer rate and maximize the heat transfer rate overall.

References

[1] EChamilAbeykoon, "Compact heat exchangers – Design and optimization with CFD" IJHMT, Volume 146, January 2020, 118766.
[2] Devvrat Verma, Aanand Shukla "Design of Shell and Tube Type Heat Exchanger using CFD Tools" IJIRST, Vol. Issue 03 Aug. 2017.
[3] Mohammed Irshad, MohammedKaushar "Design and CFD Analysis of Shell and Tube Heat Exchanger" IJESC, Vol. 7 Issue.4, 2017.

[4] Neeraj kumarNagayach, Dr. AlkaBani Agrawal "Review Of Heat Transfer Augmentation In Circular And NonCircular Tube" IJERA ISSN-2248-9622 Volume. 2, Issue 05, September- October 2012, pp.796-802

[5] GurbirSingh, Hemant Kumar "Computational Fluid Dynamics Analysis of Shell and Tube Heat Exchanger" JCEET, Vol. 1, issue 3; Aug., 2014 pp. 66-70.

[6] Nasir Hayat, Muhammad Mahmood AslamBhutta "CFD applications in various heat exchangers design: A review" ATE, vol. 32, issue 1, pp. 1–12. January 2012

[7] S. B. Hosseini, R.H.Khoshkhoo and M. Javadi, "Experimental and Numerical Investigation on Particle Deposition in a Compact Heat Exchanger," Applied TE, volume. -115, page no.. 406-417,2017.

[8] S. Ismail and R. Velraj, "Studies on Fanning Friction (F) And Colburn (J) Factors of Offset And Wavy Fins Compact Plate Fin Heat Exchanger–A CFD Approach," NHT, volume. 56, page no. 987-1005, 2009.

[9] R.B.S. Rao, G. Ranganath and C.Ranganayakulu, "Development of colburn 'j' factor and fanning friction factor 'f' correlations for compact heat exchanger plain fins by using CFD," HMT, volume. 49, page no.. 991-1000,2013.

[10] S. Girgin, "An Investigation of Heat Transfer Performance of Rectangular Channel by Using Vortex Generators," ITU, İstanbul, 2017.