Engineering and Technology Journal e-ISSN: 2456-3358
Volume 08 Issue 05 May-2023, Page No.-2171-2175
DOI: 10.47191/etj/v8i5.02, I.F. – 7.136

Design and simulation of Shell-and-Tube Heat Exchanger (STHE) with the Effect of Baffles using CFD-tool

Tesfaye Barza[1], Mesay Dejene[2]
[1,2]Lecturer, Department of Mechanical Engineering, Wolaita Sodo University, Ethiopia

ABSTRACT: In the current scenario, heat exchangers are used in the thermal system to maintain the working liquid's temperature. Due to their robust design and performance characteristics, the Shell and Tube Heat Exchangers (STHE) are mainly used in the generation of electricity, the cooling of hydraulic fluid, transmissions, and hydraulic power packs. This can further be improved to achieve a higher heat transfer rate. It consists of a casing with several tubes inside. The project's ideology is based on the use of baffles attached to the heat exchanger to increase the flow rate of highly viscous fluids. The paper's intended result is to determine the rate of heat transfer using hot water as the hot liquid. The theoretical analysis results obtained in this study were reasonably predicted by computational fluid dynamic (CFD) under Ansys Cfx. The simulation consists of modeling, meshing cross-sectioning, and the fluid behavior in the shell and tube of STHE are used in Ansys fluent 16.0. Finally, the effectiveness of the design parameters and contour conditions published under Ansys Cfx.

KEYWORDS: Computational Fluid Dynamics (CFD), Heat Exchanger, ANSYS Fluent

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. Our automobiles incorporate heat exchangers in the form of oil coolers and radiators [1]. Different heat exchangers are named according to their applications. For example, heat exchangers are used for condensing as condensers; in the same manner, heat exchangers for boiling purposes are called boilers. The performance and efficiency of heat exchangers are measured through the amount of heat transferred using the least area of heat transfer and pressure drop [2]. A better presentation of its efficiency is done by calculating the overall heat transfer coefficient. Pressure drop and area required for a certain amount of heat transfer, provide an insight into the capital cost and power requirements (Running cost) of a heat exchanger [3]. To build a heat exchanger following the criteria, there is usually a significant body of literature and theories. A heat exchanger with the smallest area and pressure drop to meet the needs of heat transfer is referred to as a good design. [4] Additionally, heat exchangers are devices that enable thermal energy to transfer between two or more fluids that are at different temperatures, it also has a broad range of applications, notably in the food, chemical, and process industries, including in electronics and environmental engineering. waste heat recovery (WHR); manufacturing industries; and air-conditioning, refrigeration, and space applications [5].

These are more effective than the others in many ways, such as having a high volume-to-heat transfer area ratio, being relatively simple to make, simple to maintain, and having the ability to transfer high mass flow rates. Furthermore, using baffles is simple to improve this equipment's performance. With the help of these components, the flow direction could be stated, all of the heat transfer areas were involved in the heat transfer, and due to the flow section being reduced, the velocities and turbulence were increased. This increased value led to a higher heat transfer coefficient and heat performance.

The tube setup, the type, position, and opening of the baffles, the spacing between the baffles, the location of the most extreme baffles, and additional possibilities for improving heat transfer are among the most important geometric parameters [6]. Shell and tube heat exchangers are the most common kind of heat exchanger because of their simple design and performance characteristics. These shell and tube heat exchangers do what they are supposed to do, but they can still be effectively constructed with higher heat transfer rates. Heat exchangers are often used in heating and cooling systems when controlling the system temperature is needed for a high-quality product. The effectiveness and efficiency of the heat exchanger impact the quality of the result [7].

The numerical values of the intake and exit temperature differential are used to determine the heat exchanger's efficiency. Because there are numerous different types of heat exchangers for many different applications, the construction will similarly vary greatly. Nevertheless, despite their versatility, most heat exchangers may be categorized into several basic design principles. Here, to encompass certain analysis and design approaches, we will just take into consideration the more prevalent varieties. behind it.

To transfer heat between the two fluids, one fluid flows through the tubes whereas another fluid flows over the tubes (via the shell). The arrangement of tubes is characterized as a tube bundle and can consist of several types of tubes, such as plain, longitudinally finned, etc. Correlations between the Kern approach and the Bell-Delaware approach are used in the design of tube and shell heat exchangers. In Bell's method, the effects of leakage, bypassing, and flow in the window zone are taken into consideration by using correction factors [8].

The heat-transfer coefficient and pressure drop are determined from correlations for flow across perfect tube banks. Compared to Kern's method, this method will provide more accurate predictions of the heat-transfer coefficient and pressure drop. It can also be used to investigate the impact of constructional tolerances and the use of sealing strips since this compensates for bypassing and leakage. The bell-Delaware approach is more accurate and can provide detailed information [9].

1.1 Shell and tube heat exchanger

Such a type of heat exchanger design is a shell and tube heat exchanger. It is the most common type of heat exchanger used for large chemical processes like oil refineries and is designed for higher-pressure applications. [10]. For reasons that operate thermal energy between two fluids, heat exchanger devices are used. They are widely utilized in the petrochemical, chemical, and pharmaceutical industries, in power plants, and also in homes for a wide range of applications including heating, cooling, condensation, and evaporation. The shell-and-tube heat exchanger is the variety of them that is commonly used [11]. These are significantly better than the others since they have a high volume-to-heat transfer area ratio, can carry large amounts of mass efficiently, and are reasonably simple to make. Additionally, using baffles makes it simple to improve this equipment's performance[12]. These elements allow for the direction of the flow to be guided, and the entire area of the heat transfer. The most significant geometrical parameters concern the baffles' size, shape, and orientation, as well as their spacing between one another and any other potential heat transfer improvement solutions [13].

1.2 Problem Definition

A heat exchanger requires to be simulated using physical parameters that are industry-standard and exposed to CFD research with present operating conditions of 55 and 25 degrees Celsius with hot water in the shell and cold water in the tube. To be sure that the CFD outcomes coincide with the real observations, this analysis must be performed under parallel flow conditions with some general restrictions. The analysis should also be done for different operating conditions such as parallel and counter flows, as well as different baffle geometric conditions. The objective of this project is to reveal the utility of CFD tools in thermal system design. To find the best operating conditions, the baffle conditions must also be optimized. The Preliminary Design Specifications by CFD ANSYS CFx. The Tube and Shell Heat Exchanger are designed and the requirements are given in below.

Table I. Geometric dimension of Heat exchanger

Shell Details	Dimensions
Outer diameter	142mm
Inner diameter	126mm
Length of the HE	1400mm
No. of baffles	3
Distance b/n baffles	2700mm
Baffle opening	20% (except first and last)
Tube Details	
Outer diameter	25mm
Inner diameter	22mm
Length	1200 mm
No. of tubes	13

1.2 Computational Fluid Dynamics (CFD) Simulation

To improve on the results from two of the theoretically generated models, a CFD simulation was carried out. Because of this, two models were generated and a CFD software was used to discretize the domains of each model. Based on the tube's outer diameter and baffle thickness, the best and lowest designs were determined as the appropriate designs. The outlet temperatures on the tube and shell sides were anticipated via a simulation of steady-state heat transfer [14].

1.3 Geometry

First, Heat Exchanger geometry is designed in ANSYS software.

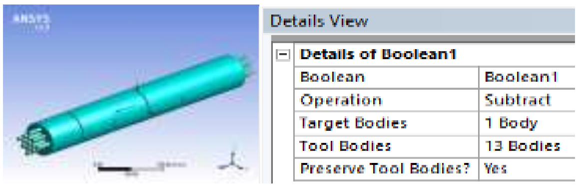

Fig 1: Exchanger geometry with a baffle design

2. METHODOLOGY

Thermal design is the methodology used in the current study for the heat exchanger's design. While designing a shell and tube heat exchanger, it ought to factor in the thermal transfer area, the number of tubes, their length and diameter, their layout, the number of shell and tube passes, the type of heat exchanger (fixed tube sheet, removable tube bundle, etc.), the tube pitch, the number of baffles, their type and size, the shell and tube side pressure drop, and the heat transfer coefficients[15].

A key requirement for a project is the methodology, which specifies the right start and finish conditions of what needs to be done. The workflow's success depends on its proper setup and execution. The methodology of this project is as follows.

2.1 CFD Methodology

The CFD Analysis was done in the following steps:

- Pre-processing Solution
- Post-processing

The model importing meshing, boundary conditions, and material property assigning are all done at the preprocessing stage. The solver settings and output settings and simulations are carried out at the solution stage [16]. The extraction of results from the saved database in the form of contour plots and tabulated values is done in the post-processing and this post-processing will be explained in the upcoming chapter. The boundary conditions used for the analysis are as follows.

Table II: Specifications

Hot liquid inlet	50◦C
Cold liquid inlet	20◦C
Mass flow rate (Hot)	0.027 Kg/sec
Mass flow rate **(cold)**	0.014 Kg/sec
Ambient temperature	30◦C
Hot and cold liquid	Water

2.2 Assumptions

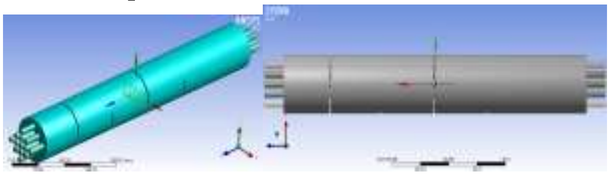

Fig 2: Heat exchanger Boolean with internal baffles

2.3 Boundary Conditions

Different boundary conditions were applied for different zones. Since it is a shell-and-tube heat exchanger, there are two inlets and two outlets. The inlets were defined as velocity inlets and the outlets were defined as pressure outlets. The inlet velocity of the cold fluid was kept constant i.e. 0.0787m/s, whereas the velocity of the hot fluid was kept constant i.e. 1.594 m/s. The outlet pressures were kept default i.e. atmospheric pressure. The hot fluid temperature at the inlet was 340k and the cold fluid inlet temperature was kept at 300k. The other wall conditions were defined accordingly. besides, the surrounding temperature was kept at 300k.

2.4 Analysis

The methodology known as computational fluid dynamics (CFD) is incredibly flexible, precise, and versatile. With industry-leading accuracy and robustness, ANSYS Fluent and ANSYS CFX provide fast results for virtually any fluid or multiphysics application. This sophisticated CFD software has the comprehensive features needed to address your design flaws currently and in the future. [17]. CFD solvers extend the limits of what is practical to improve the efficiency and performance of your product.

2.5 Governing equations

The 3-D flow through the shell-and-tube heat exchanger has been simulated by solving the appropriate governing equations. Conservation of mass, momentum, and energy using Ansys Cfx code.

Turbulence is taken care of by the shear stress transport (SST) k-w model of closure, which has a blending function that supports Standard k-w near the wall and Standard k-e elsewhere. The analysis is under the following governing equations.

$$x\ componenet:\ \rho\frac{Du}{Dt} = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho f_x$$

$$ycomponenet:\ \rho\frac{Dv}{Dt} = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{yx}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho f_y$$

$$z\ componenet:\ \rho\frac{Dw}{Dt} = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho f_z$$

Momentum equations (Non-conservation)

$$x\ componenet:\ \frac{\partial(\rho u)}{\partial t} + \nabla.\ (\rho u V) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho f_x$$

$$y\ componenet:\ \frac{\partial(\rho v)}{\partial t} + \nabla.\ (\rho v V) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho f_y$$

$$z\ componenet:\ \frac{\partial(\rho w)}{\partial t} + \nabla.\ (\rho w V) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho f_z$$

3. RESULTS AND DISCUSSION

The typical CFD contour plot outputs are used below. The post-processing tools are used to get these outputs.

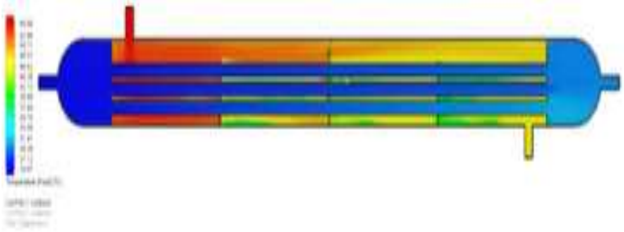

Fig 3: Interior fluid temperature (front plane cut section)

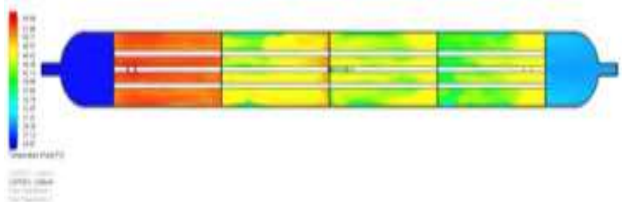

Fig 4: Interior fluid temperature (top plane cut section)

Fig 5: Interior fluid flow pattern (flow trajectories)

3.1 Effect of baffle spacing

The Tube Diameter Performance In this section, the results obtained from the rating analysis based on the Bell Delaware method, the results were estimated for a fixed baffle thickness of 0.004 m while varying the tube diameter. Variation was achieved by increasing the tube diameter from 0.0210 to 0.0240m with an interval of 0.0006 m. The effect of baffle spacing on the heat transfer rate is obtained from the analysis and tabulated below.

Table III. Effect of baffle spacing

SI.No.	1	2
Baffle spacing (mm)	230	122
T1 (K)	470	470
T2 (K)	345.15	345.15
Heat Transfer rate (W)	82.45	76.23

Fig 6: (a) and (b) Convergence of simulation (Baffle spacing of 120 mm and 240 mm).

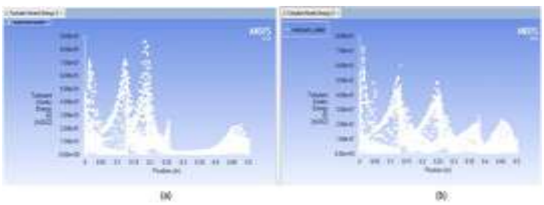

Fig 7: (a) and (b) Turbulent kinetic energy vs. Position graph for baffles of 125 mm and 250 mm

CONCLUSIONS

In this research paper, shell and tube heat exchanger analysis has been performed using Ansys CFx. This ensures by considering baffle spacing after creating a simple model. It also performed steady-state thermal analysis. The baffle spacing places a significant role in the heat transfer was determined. The main contribution of this paper was to design and simulate of Shell-and-Tube Heat Exchanger (STHE) with the effect of baffles using Ansys-CFx software, generating the high quality of Mesh (Tetrahedra) and CFx as the CFD solver. Besides, the contour plot and volume rendering are generated for a better understanding of flow through Exchanger (STHE). Finally, the air is found to have better performance found simulation is done under a steady-state phenomenon

REFERENCES

1. S. K. Katarki and A. S. Malipatil, “CFD Analysis of Shell and Tube Heat Exchanger for Heat Transfer Capabilities Abstract:,” vol. 3, no. 6, pp. 102–106, 2017.
2. S. L. T. D. V. K. Chirag Raut, “IRJET- Study and Design of Shell and Tube Heat Exchanger,” Irjet, vol. 8, no. 6, pp. 1156– 1160, 2021.
3. X.-W. Z. and W.-H. Z. B.-W. Gong, “Chinese Journal of Geotechnical Engineering,” vol. 28, no. 5, pp. 207–209, 2006, doi: 10.11779/CJGE202205.2.
4. N. Das Pal, “Designing and Performance Evaluation of a Shell and Tube Heat Exchanger using Ansys (CFD),” no. 3, 2016.
5. M. Irshad, M. Kaushar, and G. Rajmohan, “Design and CFD Analysis of Shell and Tube Heat Exchanger,” vol. 7, no. 4, 2017.
6. J. V. Kumar, M. V. S. P. Kumar, J. S. Kumar, and T. N. V. A. Kumar, “Modified design of shell and tube heat exchanger and CFD analysis,” no. 4, pp. 39–44, 2018.
7. K. P. B, B. U. Lakshmi, and A. H. Krishna, “Numerical Simulation o Shell and Tube Heat Exchanger using Ansys Fluent,” no. 11, pp. 543–545, 2019, doi: 10.35940/ijitee. K109309811S219.
8. A. N. Gemeda, “Parametric & CFD Analysis of Shell and Tube Heat Exchanger by Varying Baffles Geometry,” Ind. Eng. Lett., vol. 10, no. 3, pp. 30–42, 2020, doi: 10.7176/iel/10-3-03.
9. M. A. Alperen, E. Kayabas¸i, and H. Kurt, “Detailed comparison of the methods used in the heat transfer coefficient and pressure loss calculation of shell side of shell and tube heat exchangers with the experimental results,” Energy Sources, Part A Recover. pp. 1–20, 2019, doi: 10.1080/15567036.2019.1672835.
10. D. O. F. Shell, T. Gebru, and U. T. H. E. Guidance, “Institute of Technology.”
11. H. K. Kumar, “Design Optimization of a Shell and Tube Heat Exchanger.”
12. H. R. Jawad, “The Republic of Iraq CFD Simulation for Shell and Double Concentric Tube Heat Exchanger College of Engineering at Baghdad University in Partial Fulfillment of the Requirements for the Degree of Master of Science in Chemical Engineering,” no. January 2018, pp. 1–88, 2019, doi: 10.13140/RG.2.2.18527.30884.
13. M. Petrik and G. L. Szepesi, “Shell Side CFD Analysis of a Model Shell-and-Tube Heat Exchanger,” vol. 70, pp. 313–318, 2018, doi: 10.3303/CET1870053.
14. E. T. Idowu, T. S. Mogaji, and M. A. Akintunde, “Investigation of Tube Diameter and Baffle

Thickness Effect on Shell and Tube Heat Exchanger Performance,” Eur. Int. J. Sci. Technol., vol. 9, no. 5, pp. 14–29, 2020, [Online]. Available: www.eijst.org.uk.
15. R. Gondane and Y. M. Jibhakate, “design and cfd analysis of shell and tube heat exchanger using,”no. 3, pp. 2192–2199, 2018.
16. A. H. V. Reddy, G. R. Reddy, G. Phanindra, and K. V. Kumar, “Design and analysis of condenser using 3D modeling software,” pp. 7–15, 2018.
17. H. R. Jawad, “CFD Simulation for Shell and Double Concentric Tube Heat Exchanger,” no. March, pp. 1–88, 2018