

CFD Analysis of the Effect of Mass Flow Rate in a Shell and Tube Heat Exchanger Design

¹Oku Ekpenyong Nyong, ²Celestine Ebieta, ³Francis Jeremiah Okara, ⁴Eward Asu Obi, ⁵Bassey Ekpo Ene

^{1,3,5}Thermo-fluid, Combustion, and Energy System Research Group, Department of Mechanical Engineering, University of Cross River State, Calabar, Nigeria

²Department of Mechanical Engineering, University of Port Harcourt, Port Harcourt, Nigeria

⁴Department of Mechanical Engineering, University of Calabar, Calabar, Nigeria

Corresponding Author's E-mail: nyong.oku@unicross.edu.ng

Abstract - Heat exchangers are often found in the chemical process industry, but of all types of heat exchangers, the Shell and Tube Heat Exchanger (STHE) are the most used. The design of shell and tube heat exchangers are continually evolving because of it reduce production costs. A computational fluid dynamics (CFD) simulation was employed using SOLIDWORKS Flow Simulation version 2022 to model the effect of operating parameter such as the mass flow rate on the pressure drop, heat transfer coefficient and the turbulence intensity. The turbulence model adopted for the simulation was the k-ε turbulence model incorporating the boundary conditions. Three conditions were studied increasing both shell and tube side mass flow rate, maintaining either the shell or tube constant and then varying the other. The results obtained showed that an increase in mass flow rate have tremendous effect on the pressure drop, heat transfer and turbulence intensity.

Keywords: Pressure drop; Heat transfer Coefficient; Baffle; Turbulence intensity; Mass flow rate;

I. INTRODUCTION

Heat exchangers are thermal devices that are widely used for transferring heat energy from one fluid to another or transferring thermal energy between two or more fluids. Heat exchanging mechanisms can be found in many of the commonly used small devices and vehicles in our day-to-day life, there are also ideal for process industry applications according to specified characteristics, such as the inlet and outlet temperatures of fluids, flow rates and heat duty. Heat exchangers can be widely classified according to flow arrangement such as parallel flow, counter flow and cross flow and by construction such as shell-and-tube, plate, plate-fin, shell and plate, adiabatic wheel[1]. Based on the classifications by construction shell-and-tube heat exchangers (STHX) are the most commonly used type of heat exchangers due to their large heat transfer surface area-to-volume ratios compared to other common types of heat exchangers[2]. In

shell-and-tube heat exchangers, two fluids with different temperatures pass separately through the shell and the tube. The fluid on the tube side flows through the tubes, while the fluid on the shell side flows outside the tubes but inside the shell. Heat is transmitted through the tube walls from one fluid to another, either from the tube side to the shell side or the opposite [3]. Due to the constantly growing demand for energy utilization a lot of work has been carried out by scholars to enhance the performance of the heat exchanger by understanding the effect of various design parameters and the operating conditions. In more recent study computational fluid dynamics (CFD) are being utilized to study and moderate the fluid flow conditions in heat exchangers as a form of numerical prototype, saving cost and time of physical fabrication and testing. Pranita *et al.*, [3] carried out a numerical simulation using CFD to study the effect of different baffles type in a shell and tube heat exchanger to denote its effect on the model pressure drop at different flow rates. From their result they discovered that low pressure drop increases the system overall efficiency, they also discovered that an increase in flow rate increases the pressure drop of the heat exchanger. The accuracy of various turbulence models used to solve the Reynolds averaged Navier-Stokes (RANS) equations was compared by Balaji and Prakash [4] in their work. The CFD computations employ two-parameter of turbulence models such as the k-ε and k-ω model, and SST model. The obtained results agree satisfactorily with the information from the commercial experiments. This makes it possible to increase production efficiency, solve the problem of process control, and identify the ideal regime parameters for technical processes. Fetuga *et al.*, [5] carried out a study to examine the impact of tube layout configurations on the hydraulic-thermal performance of shell-and-tube heat exchangers (STHXs) as waste heat recovery systems. The STHXs are triangular (30°, STHX-T), rotated triangular (60°, STHX-RT), and combined (STHX-C). The study uses ANSYS-Fluent for a three-dimensional computational study, and the results show that the STHX-T layout has the highest overall heat transfer coefficient, with the highest pressure drop

and temperature drop. Vlahostergios *et al.*, [6] did an experimental measurement on a heat exchanger model to investigate the impact of turbulence intensity on pressure drop and heat transfer mechanism in an aircraft engine. The model, a cross-flow heat exchanger, was used to transfer heat from exhaust hot gas to cold compressed air before combustion. Results showed that increasing turbulence intensity reduced pressure drop and improved heat exchanger efficiency, reducing fuel consumption and improving thermal efficiency. Piyus *et al.*, [7] examined the impact of mass flow rate on the convective heat transfer coefficient of an incompressible fluid in turbulent flow regimes. It shows that a threefold increase in mass flow rate decreases the heat transfer coefficient by 10%, while doubling the mass flow rate will result in a 92% increase in the heat transfer coefficient. However, this increase also increases pressure drop, potentially affecting the heat transfer coefficient. Yuying *et al.*, [8] investigates the heat and mass transfer performances of a spacecrafts active fluid loop thermal control system combined with a sublimator. It found that the fluid loop mass flow rate affects the sublimator's heat and mass transfer performances, but the heat rejection ability doesn't always increase with the increase. The study also found a positive correlation between heat rejection ability and fluid loop mass flow rate.

This research work is to numerically study the effect of varying mass flow rate as it affects the fluid turbulence intensity, pressure drop and the heat transfer coefficient of the shell and tube heat exchanger design.

II. COMPUTATIONAL MODEL

2.1 Geometrical domain

For this study the shell and tube heat exchanger were modelled using SolidWorks 2022 (CAD software). The geometrical specifications used for the shell and tube heat exchanger were obtained from an experimental whose work is still in view.

Table 1: STHE model geometry

Parameter	Description
Internal diameter of Shell:	127mm
Internal diameter of Tube:	9.5mm
Number of tubes	30
Length of Tube	800mm
Length of Shell	127mm
Diameter of Baffles	123mm
Number of baffles	12
Sheet thickness	4mm
Baffle spacing	61.5 mm
Tube arrangement	Square

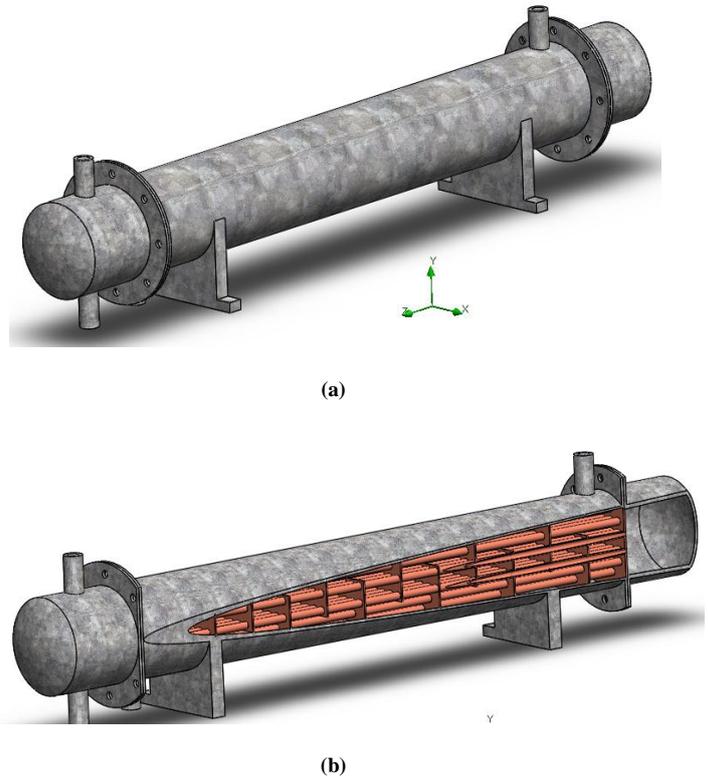


Figure 1: (a) Isometric View of the shell and tube heat exchanger, (b) Section view of the shell and tube heat exchanger

2.2 Numerical modelling

Computational fluid dynamics (CFD) is a process where computer simulations are used to analysis systems including fluid flow, heat transport, and associated phenomena like chemical reactions. CFD employs Navier-Stokes numerical techniques to depict and predict fluid flow and the fluid behaviour in various situations, computational fluid dynamics also develops a way for resolving fluid dynamics issues. For this model, the k-ε turbulence model is adopted for calculating the flow process of the fluid in the shell and tube heat exchanger. SolidWorks 2022 simulation tool was utilized for simulating the 3D model. The k-ε turbulence model was implemented for this study due to its reliability and accuracy standard in heat transfer and fluid flow study [9-11].

2.2.1 Governing equation

The Following Navier-Stokes equation, continuity, momentum, energy, k and ε models were used in the computational domain [10].

Continuity

$$\frac{\partial}{\partial x_1}(\rho u_1) = 0 \tag{1}$$

Naiver-Stokes' equation (Momentum):

$$\frac{\partial}{\partial x_1}(\rho u_1 u_k) = \frac{\partial P}{\partial x_1} + \frac{\partial}{\partial x_1}(\mu \frac{\partial u_k}{\partial x_1}) \quad (2)$$

$k - \epsilon$ turbulence model (Energy):

$$\frac{\partial}{\partial x_1}(\rho u_1 T) = \frac{\partial}{\partial x_1}(\frac{\partial T}{\partial x_1} \frac{k}{c_p}) \quad (3)$$

Turbulent kinetic energy:

$$\frac{\partial}{\partial t_i}(\rho k) + \frac{\partial}{\partial x_i}(\rho u_i k) = \frac{\partial}{\partial x_i}((\mu + \frac{\mu}{\sigma_k}) \frac{\partial k}{\partial x_i}) + S_k \quad (4)$$

Turbulence dissipation energy:

$$\frac{\partial}{\partial t_i}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho u_i \epsilon) = \frac{\partial}{\partial x_i}((\mu + \frac{\mu}{\sigma_\epsilon}) \frac{\partial \epsilon}{\partial x_i}) + S_\epsilon \quad (5)$$

Where:

$$S_k = \tau_{ij}^R \frac{\partial u_i}{\partial x_j} - \rho \epsilon + \mu_t P_B \quad (6)$$

And

$$S_\epsilon = C_{\epsilon 1} \frac{\epsilon}{k} (f_1 \tau_{ij}^R \frac{\partial u_i}{\partial x_j} + \mu_t C_B P_B) - C_{\epsilon 2} f_2 \frac{\rho \epsilon^2}{k} \quad (7)$$

The turbulent generation that is due to buoyancy force P_B is represented as

$$P_B = - \frac{g_i}{\sigma_B} \frac{1}{\rho} \frac{\partial \rho}{\partial x_i} \quad (8)$$

The boundary conditions used for modelling the fluid flow and heat transfer simulation of the studied shell and tube heat exchanger were set using SolidWorks 2022 Wizard flow simulation. The simulation analysis type was set as internal flow to exclude cavities without flow condition. Hot water enters the tube at a temperature of 70°C, and the cooled water enters the shell at the temperature of 34.2 °C, Stainless steel and copper were used as the material for the Shell and pipes, water was set as the working fluid. Three flow conditions were monitored in this research to study and visualize the effect on the pressure drop, heat transfer coefficient and the turbulent intensity. The first condition has the mass flow rate for both shell and tube side gradually increased from 0.033kg/s, 0.049kg/s, 0.066kg/s, 0.083kg/s, 0.099kg/s, 0.116kg/s to 0.133kg/s. The second condition, the mass flow rate entering the shell inlet was kept constant at 0.033kg/s while varying the mass flow rate entering the tube side from 0.049kg/s to 0.133kg/s. The third condition the mass flow rate entering the tube side was kept constant at 0.033kg/s while the varying the mass flow rate at the shell side from 0.049kg/s to 0.133kg/s. The outlets boundary condition was set at room pressure. Adiabatic wall condition was adopted at the outer wall of the

shell with a thermal boundary condition of zero heat flux, while the $k - \epsilon$ turbulence model was utilised to predict the fluid behaviour.

2.3 Mesh Study

The model was meshed using SolidWorks Automatic mesh level 4 all through the simulation. The mesh analysis was carried out at the shell side to ascertain the effect of mesh cells increment on pressure drop. As depicts in Figure 2(a) as the mesh size was increased the pressure drop along the shell side decreases up to a point where we have 325000 cells further increase in the cell sizes showed no appreciable change in the pressure drop. The mesh sizes between 325000 and 355000 were chosen to run all the conditions in the simulation. The pressure drops for all the four cases were carefully monitored and from the distribution a mesh grid with 325000 cells was chosen for all the CFD simulation due to stability in pressure drop[5]. Figure 2(b) shows the visualize cross section of the shell and tube heat exchanger model.

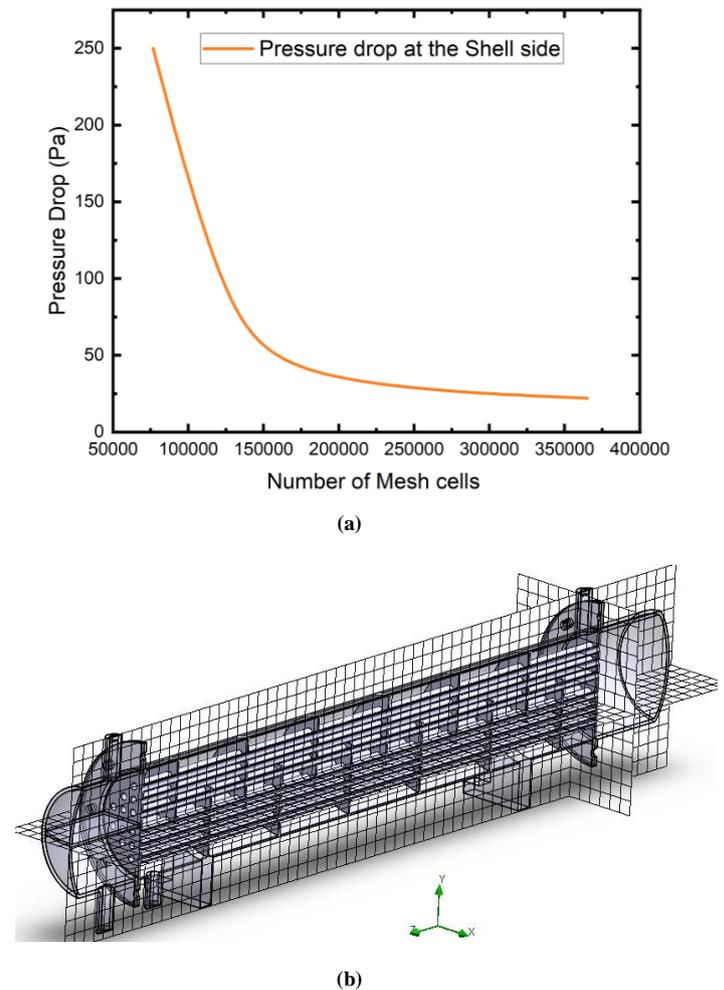


Figure 2: (a) Mesh Sensitivity analysis (b) Mesh Visualisation

III. RESULTS AND DISCUSSION

3.1 Turbulence intensity

The nature of the flow vortices of a shell and tube heat exchanger are dependent upon factors such as the mass flow rate, number of passes, number of baffles and tubes [7, 12, 13]. However, Figure 3(a-f) shows the turbulence intensity distribution along the shell side of the STHE. Turbulence intensity in the shell is greatly influenced by an increase in mass flow rate. As the mass flow rate was increased as show

in Figure 3(a-f) an increase in the turbulence intensity was observed from the mass flow rate of 0.033 kg/s to 0.113 kg/s. This attributes were seen due to the increase in velocity of the fluid causing more mixing and fluctuation in the shell side. The turbulence intensity has the highest value as shown in Figure 3(f). Nevertheless, increasing the mass flow rate increases the pressure drop as shown in Figure 3 (a-f). It is seen that an increase in mass flow rate lead to a increase in the turbulence intensity which is in agreement with literature [14, 15].

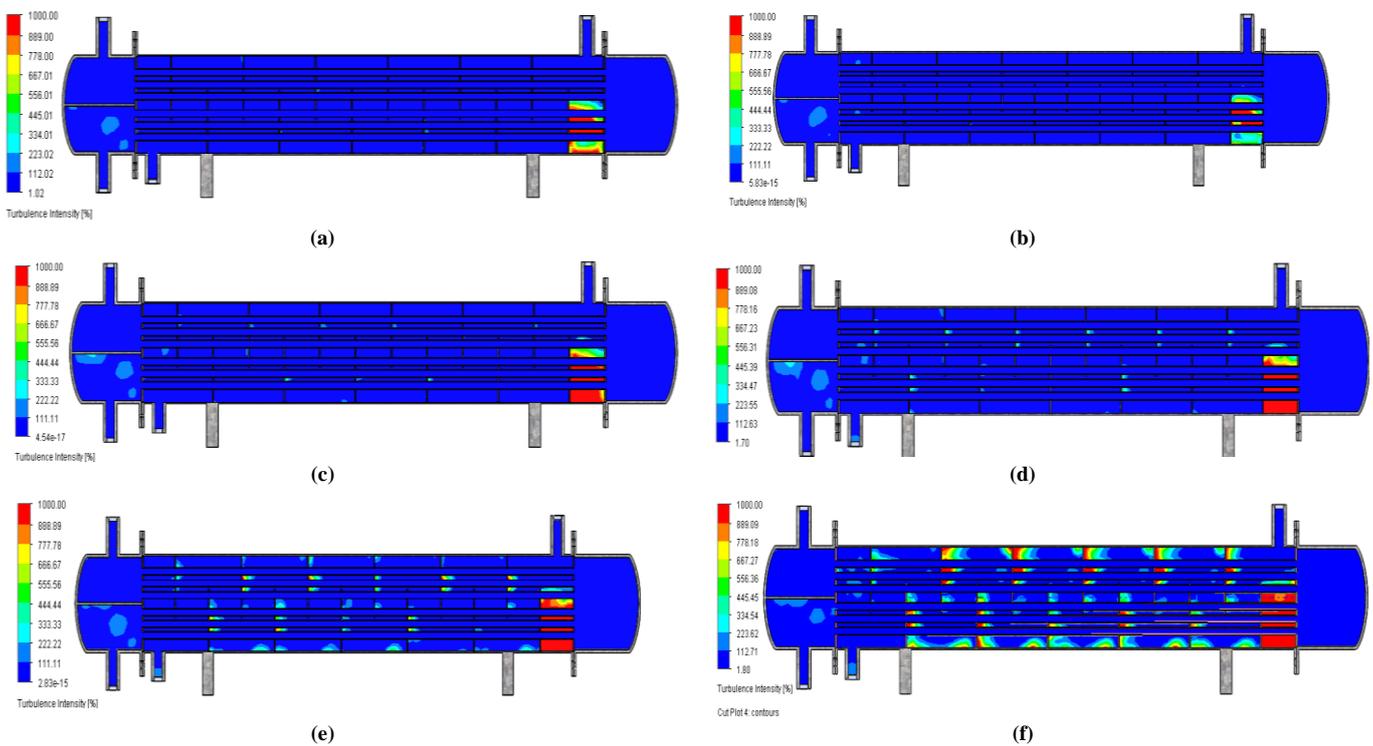


Figure 3: Shows the contours of turbulence Intensity at the shell side at same mass flow rate of (a) 0.34kg/s, (b) 0.050kg/s (c) 0.066kg/s (d) 0.083kg/s (e) 0.099kg/s (f) 0.113kg/s

3.2 Pressure drops

The pressure drop is a very important factor in the design of shell and tube heat exchanger as it directly related to operating cost and efficiency. Lower pressure drops result to lower pumping power, which in turn increases the overall system efficiency. Figure 4 demonstrates the three conditions of the simulation. Figure 4(a) where the flow rate at the shell and tube size is gradually increased from 0.033 to 0.133 kg/s. We can see that the increase in the flow rate has a tremendous effect on the pressure drop as it increases. This increased pressure drop can potentially affect the performance and efficiency of the system [2, 3]. Figure 4 (b) shows the

condition where the shell side flow rate was maintained at 0.033 kg/s while varying the tube side flow rate. We can see that in Figure 4(b) significant change in flow regime since the shell-side mass flow rate remains constant, the flow regime (laminar or turbulent) is unlikely to change. However, there is no significant change in heat transfer coefficient. The shell-side heat transfer coefficient may not change significantly, as the flow rate and velocity remain constant. Therefore, there is steady rise in the pressure drop along the tube-side of the heat exchanger. The same trend is also observed when the tube-side flow rate is kept constant while varying the shell-side flow rate. As the flow rate is increased at the shell side there is corresponding increase in the pressure drop at the shell-side.

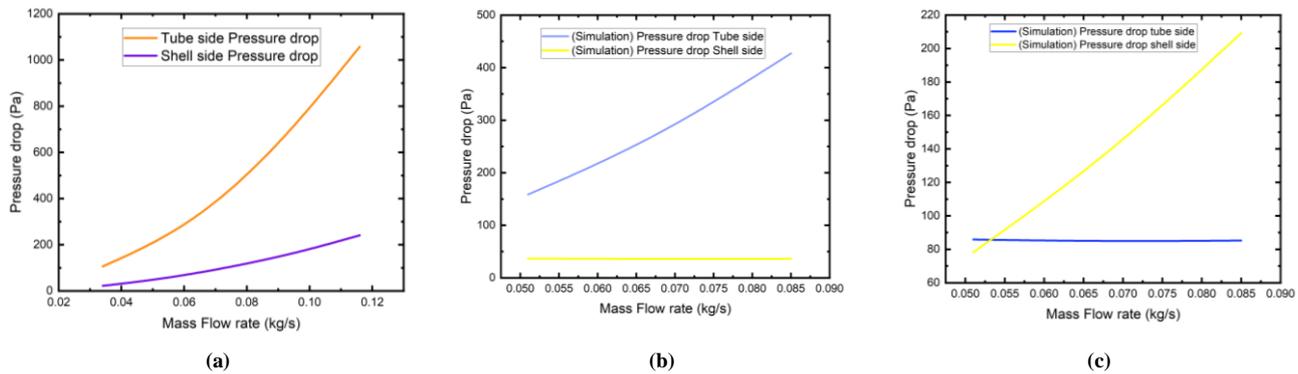


Figure 4: (a) shows increasing both shell and tube side mass flow rate (b) shell side mass flow rate maintained at 0.034 kg/s while varying mass flow rate at the tube side, (c) Tube side mass flow rate maintained at 0.034 kg/s while varying mass flow rate at the shell side

3.3 Heat Transfer Coefficient

Figure 5 shows the plot of mass flow against the heat transfer coefficient distribution of heat transfer coefficient across different flow rate. As shown in Figure 5 for all cases of mass flow rate increasing the heat transfer coefficient increases the heat transfer coefficient linearly as readily observed in other literatures [1, 2, 5, 7]. It can be seen that increased turbulence can improve the shell-side heat transfer coefficient, potentially leading to better overall heat exchanger performance.

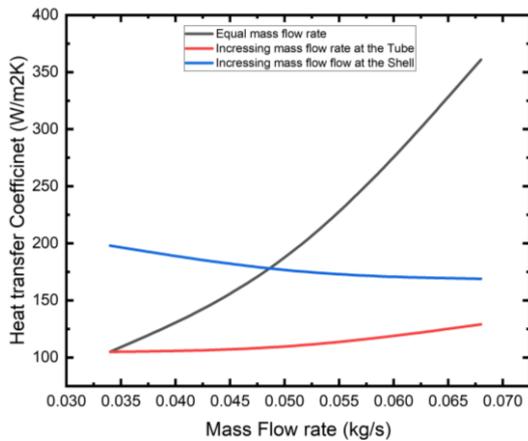


Figure 5: Effect of mass flow rate on the heat transfer coefficient

IV. CONCLUSION

In this study, a numerical model was used to study the performance of a shell and tube heat exchanger under varying flow rates using SolidWorks 2022 Wizard flow simulation. From the results obtained so far, it was we observed that increasing the mass flow rate can drastically improve the heat transfer coefficient in a shell a tube heat exchanger but would lead to increasing pressure drop incurring pumping cost. Mass flow rate influences turbulence intensity in the shell side of a heat exchanger. Higher shell-side mass flow rate can lead to increased turbulence, enhancing heat transfer and mixing. Increased turbulence can equally improve the shell-side heat

transfer coefficient, potentially leading to better overall heat exchanger performance. Higher shell-side mass flow rate can result in increased pressure drop across the shell side, potentially affecting system performance and efficiency.

REFERENCES

- [1] C. Abeykoon. Compact heat exchangers–Design and optimization with CFD. International Journal of Heat and Mass Transfer. 146 (2020) 118766.
- [2] Y. Li, X. Jiang, X. Huang, J. Jia, J. Tong. Optimization of high-pressure shell-and-tube heat exchanger for syngas cooling in an IGCC. International Journal of heat and mass transfer. 53 (2010) 4543-51.
- [3] P. Bichkar, O. Dandgaval, P. Dalvi, R. Godase, T. Dey. Study of shell and tube heat exchanger with the effect of types of baffles. Procedia Manufacturing. 20 (2018) 195-200.
- [4] D. Balaji, L. Prakash. CFD analysis of a pressure drop in a staggered tube bundle for a turbulent cross flow. International Advanced Research Journal in Science, Engineering and Technology. 3 (2016) 35-40.
- [5] I.A. Fetuga, O.T. Olakoyejo, S.M. Abolarin, J.K. Gbegudu, A. Onwuegbusi, A.O. Adelaja. Numerical analysis of thermal performance of waste heat recovery shell and tube heat exchangers on counter-flow with different tube configurations. Alexandria Engineering Journal. 64 (2023) 859-75.
- [6] Z. Vlachostergios, D. Missirlis, M. Flouros, C. Albanakis, K. Yakinthos. Effect of turbulence intensity on the pressure drop and heat transfer in a staggered tube bundle heat exchanger. Experimental Thermal and Fluid Science. 60 (2015) 75-82.
- [7] P. Sabharwall, V. Utgikar, F. Gunnerson. Effect of mass flow rate on the convective heat transfer coefficient: analysis for constant velocity and constant area case. Nuclear technology. 166 (2009) 197-200.
- [8] W. Yuying, L. Jindong, N. Xianwen, Q. Zhong, M. Jianyin, W. Lu, et al. Influences of mass flow rate on heat and mass transfer performances of water

- sublimator combined with fluid loop. Chinese Journal of Aeronautics. 32 (2019) 888-94.
- [9] D.C. Wilcox. Reassessment of the scale-determining equation for advanced turbulence models. AIAA journal. 26 (1988) 1299-310.
- [10] A.A.A. Arani, R. Moradi. Shell and tube heat exchanger optimization using new baffle and tube configuration. Applied Thermal Engineering. 157 (2019) 113736.
- [11] F.R. Menter. Turbulence modeling for engineering flows. Ansys, Inc. (2011).
- [12] S.H. Abdulmalek, M.K. Assadi, H.H. Al-Kayiem, A.A. Gitan. Effect of tube diameter on the design of heat exchanger in solar drying system. IOP Conference Series: Materials Science and Engineering. IOP Publishing 2018. p. 012028.
- [13] O. Nyong, D. Igbong, C. Ebieto, V. Akpan. FLOW PREDICTION ON THE EFFECT OF BAFFLE CUTS IN A SHELL AND TUBE HEAT EXCHANGER DESIGN.
- [14] N.T. Basse. Turbulence intensity scaling: A fugue. Fluids. 4 (2019) 180.
- [15] M.A. Jamil, T.S. Goraya, M.W. Shahzad, S.M. Zubair. Exergoeconomic optimization of a shell-and-tube heat exchanger. Energy Conversion and Management. 226 (2020) 113462.

Citation of this Article:

Oku Ekpenyong Nyong, Celestine Ebieto, Francis Jeremiah Okara, Eward Asu Obi, & Bassey Ekpo Ene. (2024). CFD Analysis of the Effect of Mass Flow Rate in a Shell and Tube Heat Exchanger Design. *International Research Journal of Innovations in Engineering and Technology - IRJIET*, 8(8), 215-220. Article DOI <https://doi.org/10.47001/IRJIET/2024.808023>
