International Journal of Research and Review
www.ijrrjournal.com E-ISSN: 2349-9788; P-ISSN: 2454-2237

Research Paper

Optimization of the Inlet Conditions of a Shell and Tube Heat Exchanger

Beneth C. Chukwudi, Martins B. Ogunedo

Department of Mechanical Engineering, Imo State University, Owerri.

Corresponding Author: Beneth C. Chukwudi

ABSTRACT

This research aimed at optimizing the input conditions of a shell and tube heat exchanger. To achieve this, a CFD analysis was run on the model using Solid Works flow simulation tool in order to generate a series of responses for 20 input variables. The result obtained from the CFD analysis was used to generate an optimization model for maximizing the efficiency of the unit. The result of the CFD operation revealed that when exchange of heat in a shell and tube heat exchanger, is between same fluid type, the efficiency of the unit would largely be influenced by the input variables associated with a lower fluid capacity. The optimized input conditions was found to be 2.5 kg/s, 0.5 kg/s, 0.2 m/s and 0.34 m/s for the cold water inlet mass flow rate, hot water inlet mass flow rate, cold water linear velocity, and the hot water linear velocity respectively. The efficiency of the unit as a result of the optimal values of the inlet condition rose to 86.11% from 4.43% at initial conditions.
Keywords: Optimization, CFD, Objective function, Mass flow rate, Velocity.

INTRODUCTION

A heat exchanger has been defined by [1] as a thermal device that enables the exchange of heat between a solid and a fluid or between two or more different fluids. The shell and tube heat exchanger consists of tubes which could also be cylindrical or rectangular; mounted in a shell. The shells could also be cylindrical, rectangular or arbitrary in shape. Shell and tube heat exchangers are mainly used in process industries conventional and nuclear power stations, steam generators etc. Its relative large ratios of heat transfer area to volume and ease of cleaning gives it an edge over the other types of heat exchangers. Input conditions such as mass flow rate, velocity and temperature would have an effect on the efficiency of the unit. Therefore, it is necessary to ensure that the device is designed to perform at optimal conditions. This can be achieved by carrying out a Computational Fluid Dynamics (CFD). Researchers such as [2–5] allude to the fact that CFD is a veritable tool in carrying out optimization of thermal systems. Hence, this study aims to optimize the inlet conditions of a shell and tube heat exchanger using the CFD feature in Solid Works flow simulation software. The sole objective of the study is to develop an objective function that would optimize efficiency of the unit.

METHODOLOGY

Material setting

Materials assigned to the shell and tubes are given in table 1 and 2 respectively.

Table 1: Shell side material setting

Parameter/Item	Value/material type
Material	Stainless steel
Density	8000 kg/m^3
Length	0.16 m
Diameter	0.91 m
Thermal conductivity	204 W/mK

Table 2: Tube side material setting

Parameter/Item	Value/material type
Material	Aluminum
Density	2700 kg/m^3
Length	0.16 m
Diameter	0.35 m
Thermal conductivity	28 W/mK

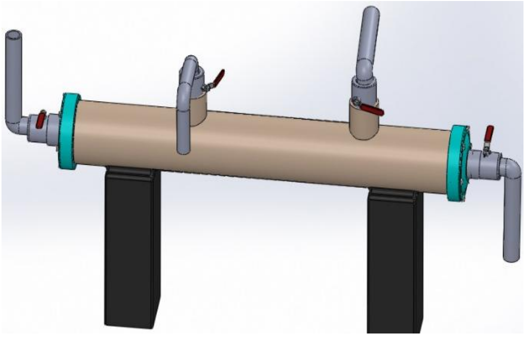

Figure 1: 3D model of Shell and Tube Heat Exchanger assembly.

Fluid subdomains

The heat exchange is between hot and cold water. Cold water was allocated to flow through the tube, while hot water was allocated to flow through the shell.

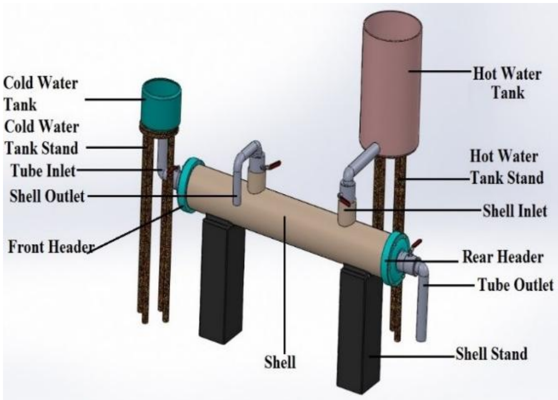

Figure 2: 3D model of assembled view of the Shell and Tube Heat Exchanger setup.

Boundary conditions

Tables 3 and 4 present the boundary condition the CFD operation was subjected to.

Table 3: Cold water inlet boundary condition.

Parameter	Value
Temperature	15 °C
Mass flow rate	1 kg/s
Linear velocity	1 m/s
Reynold number	>10000
Dynamic viscosity at 15 °C	1.1373 mPa.s
Fluid density	1000 kg/m^3
Fluid thermal conductivity	0.606 W/mK
Environmental pressure	1.0135 N/m^2

Table 4: Hot water inlet boundary condition.

Parameter	Value
Temperature	100 °C
Mass flow rate	0.5 kg/s
Linear velocity	0.05 m/s
Reynold number	>10000
Dynamic viscosity at 15 °C	0.2814 mPa.s
Fluid density	961.6 kg/m^3
Fluid thermal conductivity	0.606 W/mK
Environmental pressure	1.0135 N/m^2

CFD domain

This refers to the region within which the computer will carry out all CFD operations. The heat exchanger has a shell surface area of 0.455 m^2 and a tube surface area of 0.1762 m^2. Table 5 gives details of the size and conditions of the computational domain.

Table 5: Size and condition of the CFD domain.

Coordinate	Size	Condition
X min	-0.12024 m	Full symmetry
X max	0 m	Half symmetry
Y min	-0.1203599 m	Full symmetry
Y max	0.2402599 m	Full symmetry
Z min	-0.361221 m	Full symmetry
Z max	0.361221 m	Full symmetry

Figure 3 shows the 3D image of the CFD domain. It is seen that the domain occupies half of the figure. This is to enable a quicker response time while the software carries out iterative calculations.

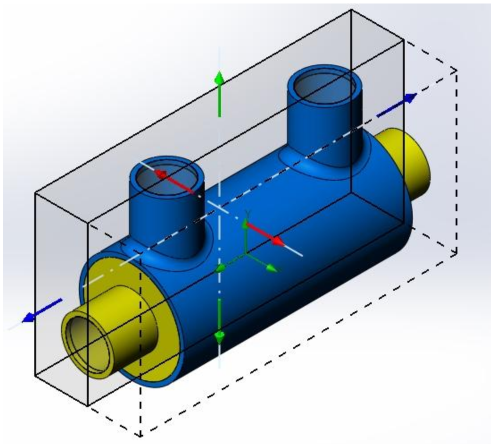

Figure 3: Computational domain of shell and tube heat exchanger.

Model predictors, constraints and objective function development

The CFD operation was designed to have the input conditions such as mass flow rates of shell and tube fluids, velocity of shell and tube fluids as the predictors. The constraints of the operation are the initial boundary conditions of the shell and tube. The lower temperature would be known as the cold water inlet temperature, while the higher temperature is the hot water inlet temperature. The fluids flow in a counter flow array. The background variable is the efficiency of the system, because it is expected that changes in the predictors would affect the efficiency. Hence, in developing an objective function, the efficiency is set as a function of the model predictors, i.e. $\eta = \boldsymbol{f(\dot{m}_s, \dot{m}_t, v_s, v_t)}$.The CFD operation was a total of 20 runs, with the predictors being varied in order to determine their effects on the efficiency of the system. This was achieved by varying the inlet mass flow rate of the first five runs while keeping the other inlet variables constant at boundary conditions. The mass flow rate was varied between 1 to 3 kg/s. The same procedure was applied to the inlet variables of hot water mass flow rate, cold water linear velocity and hot water linear velocity while varying their inputs at 0.25 to 0.45 kg/s, 0.1 to 0.8 m/s, and 0.1 to 0.5 m/s respectively. An analysis of the CFD result, led to the development of the objective function.

Model equations

The governing equations used to run the CFD which relates the design variables to other parameters and constraints are as follows:

Overall heat transfer coefficient: This is a function of all the fouling resistances, individual heat coefficient and surface efficiency of the tubes. The fouling resistances are as a result of the build-up of dirt film on the heat exchanger surfaces. The tube overall heat transfer coefficient is determined as follows.

$$\frac{1}{U_o} = \frac{1}{\frac{1}{h_T}+\frac{1}{h_s}+\frac{L}{K}+R_i+R_o} \quad (1)$$

Where R_i, R_o, are the inside, and outside fouling resistances factors.

$R_i = 0.0001$, $R_o = 0.0002$, [6]

K = Thermal conductivity of tube wall material.

L = Length of heat exchanger.

Heat Load: Assuming no phase change exists, the heat duty of the heat exchanger is expressed as:

$$q = (\dot{m}C_p)_t(T_{t,o} - T_{t,i}) = (\dot{m}C_p)_s(T_{s,i} - T_{s,o}) \quad (2)$$

Cp = 4.185 kJ/kgK,

$\mathbf{T_{t,o}} =$ **outlet tube side fluid temperature**

$\mathbf{T_{t,i}} =$ **inlet tube side fluid temperature**

$\mathbf{T_{s,i}} =$ **inlet shell side fluid temperature**

$\mathbf{T_{s,o}} =$ **outlet shell side fluid temperature**

Tube side Reynolds number: The tube side Reynold number classifies the flow regime of the fluid in the tube. It is quantified using the equation 3.

$$Re_t = \frac{\rho_t v_t d_i}{\mu_t} \quad (3)$$

Tube side heat transfer coefficient (h_t): This is expressed by equation 5a.

$$h_t = \frac{Nu \times K}{L} \quad (4)$$

Where Nu = Nusselts number

$$Nu = 0.023 \times Re^{0.8} \times Pr^{0.33} \times \left(\frac{\mu}{\mu_w}\right)^{0.14} \quad (5)$$

K = thermal conductivity of water = 0.606 W/mK

$$Pr = Prandtl\,number = \frac{\mu \times C_p}{K}$$

Shell side heat transfer coefficient (h_s): The heat transfer coefficient of the shell side is expressed using the equation put forward by [2].

$$h_s = \frac{0.023 \times Re^{0.8} \times Pr^{0.33} \times \left(\frac{\mu}{\mu_w}\right)^{0.14} \times K}{L} \quad (6)$$

Shell side cross sectional flow area: The shell side cross sectional area is expressed as:

$$A_s = 0.7855(D_s^2 - d_0^2 N_t) \quad (7)$$

Shell side linear velocity: Equation 8 gives an expression for the velocity at which the hot fluid flows.

$$v_s = \frac{\dot{m}_s}{A_s \rho_s} \quad (8)$$

Shell side Reynolds number: The shell side Reynold number classifies the flow regime of the fluid in the shell. It is quantified using the equation 9.

$$Re_t = \frac{\rho_s v_s D_s}{\mu_s} \quad (9)$$

Size of heat exchanger: This is determined based on the heat load of the heat exchanger. It is expressed as:

$$A = \frac{q}{U_o \Delta T_m} \quad (10)$$

Efficiency of the heat exchanger: The efficiency of the heat exchanger is evaluated using equation.

$$\eta = \frac{T_{s,i} - T_{s,o}}{T_{s,i} - T_{t,o}} \quad (11)$$

RESULTS

CFD result of boundary conditions

The CFD temperature contour plot of the heat exchanger and the variation of the fluid temperatures in the heat exchanger at initial conditionare given in figures 4 and 5 respectively.

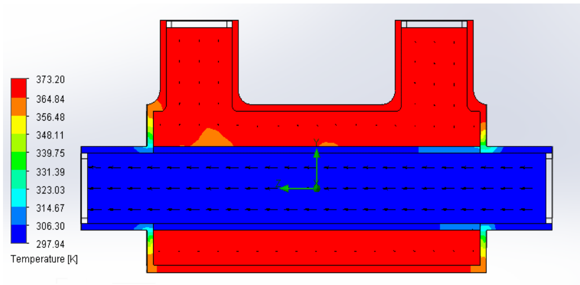

Figure 4:CFD temperature contour plot of the heat exchanger at initial condition.

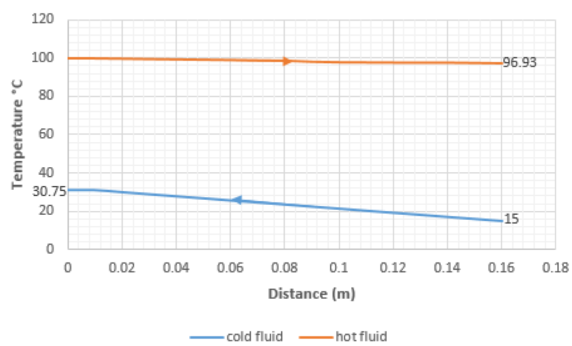

Figure 5: The variation of the fluid temperatures in the heat exchanger at initial conditions.

From the figures, it is seen that at boundary conditions, the efficiency of the unit is 4.43%. This is indicative of a poor performance, hence the effect each of the primary variable has on the efficiency was investigated.

Effect of mass inflow rate of cold water ($\dot{m}_t$) on efficiency (η).

Figure 6 reveals that increase in the mass inflow of cold water will cause an increase in the efficiency of the system. The trend line suggests that a gentle/very slight increase in efficiency is experienced with increase in the mass inflow rate of the cold water. The relationship between the efficiency and the cold water inlet mass flow rate is expressed thus:

$$\eta = 4.384 + 0.2404\dot{m}_t \quad (12)$$

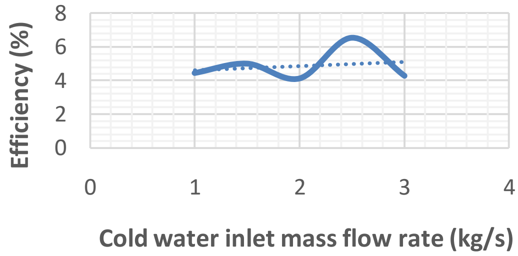

Figure 6: Effect of mass inflow rate of cold water ($\dot{m}_t$) on efficiency (η).

Effect of mass inflow rate of hot water ($\dot{m}_s$) on efficiency (η).

Figure 7 shows that increase in the mass flow rate of the hot fluid would lead to a reduction in the efficiency of the system. This is so because a lower flow rate is accompanied with lower values of hot fluid mass, and subsequently, a lower capacity. At a steady heat transfer rate, the hot fluid will have a larger quantity of heat removed from it, leading to an increase in overall system efficiency. Also at mass inflow rates above 0.4, a monotonic relationship would exist between the efficiency and the hot fluid flow rate. The relationship between both variables is expressed as:

$$\eta = 196.55 - 403.8\dot{m}_s \quad (13)$$

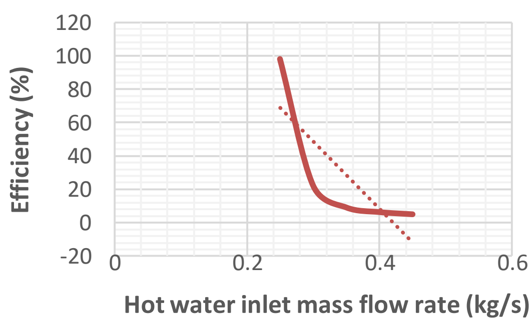

Figure 7: Effect of mass inflow rate of hot water ($\dot{m}_s$) on efficiency (η).

Effect of the inlet velocity of cold water (v_t) on efficiency (η).

From figure 8, the trendline of the graph shows that increasing the inlet velocity of the cold water will lead to a reduction in the overall efficiency of the system. The relationship between both variables is expressed as:

$$\eta = 5.54 - 0.863v_t \quad (14)$$

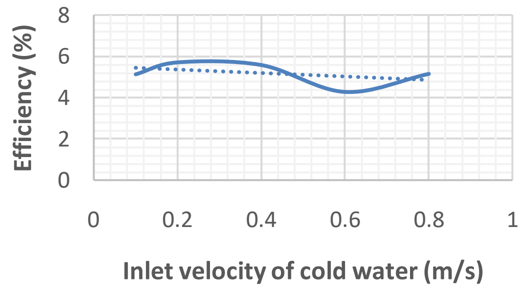

Figure 8: Effect of the inlet velocity of cold water (v_t) on efficiency (η).

Effect of the inlet velocity of hot water (v_s) on efficiency (η).

The inlet velocity of the hot fluid was seen to be proportional to the overall efficiency of the system. This is revealed by figure 9. It is due to the lower value of the hot fluid capacity when compared to the cold fluid capacity. i.e.$\dot{m}_s C < \dot{m}_t C$.The relationship between both variables is expressed as:

$$\eta = 74.15 + 10.1v_s \quad (15)$$

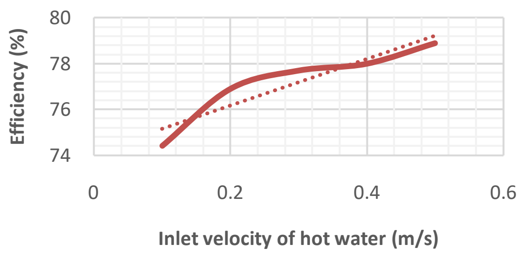

Figure 9: Effect of the inlet velocity of cold water (v_t) on efficiency (η).

From the results, it is seen that the inlet conditions of the hot fluid contributes the most to the performance of the system. This is largely due to the lower fluid capacity it possesses. This implies that the objective function would be affected mostly by the primary variables of the hot fluid.

Objective function

The result of the optimization analysis yielded the objective function and this is expressed in equation 16.

$$\eta = 1.30 + 7.09\dot{m}_s + 16.58\dot{m}_s^2 - 16.79v_s^2 + 119.2\dot{m}_s v_s \quad (16)$$

Result of the optimal conditions for maximizing the efficiency of the heat exchanger is shown in table 6.

Table 6: Optimal values for inlet conditionsof the heat exchanger.

Parameter	Value
Cold water inlet mass flow rate	2.5 kg/s
Hot water inlet mass flow rate	0.5 kg/s
Cold water linear velocity	0.2 m/s
Hot water linear velocity	0.34 m/s

The CFD temperature contour plot of the heat exchanger and the variation of the fluid temperatures in the heat exchanger at optimal condition are given in figures 10 and 11 respectively.

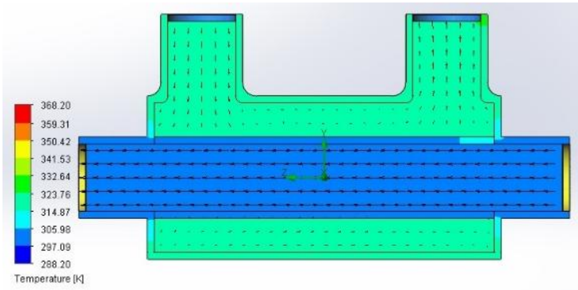

Figure 10: CFD temperature contour plot of the heat exchanger at optimal condition.

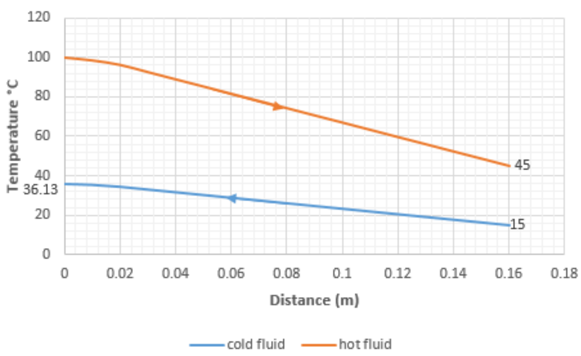

Figure 11: The variation of the fluid temperatures in the heat exchanger at optimal conditions.

The figures reveal that at optimal values, the efficiency of the system is maximized to a value of 86.11%. Also, the pressure drop across the inlet and outlet ends of both shell and tube are within the acceptable region. Figure 12 shows the variations in pressure across both shell and tube of the heat exchanger.

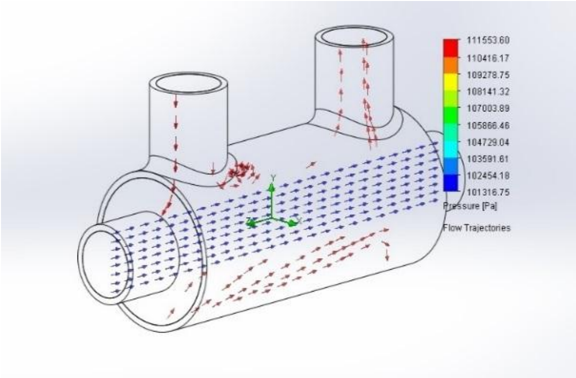

Figure 12: CFD pressure contour plot of the heat exchanger at optimal condition.

CONCLUSION

This paper presents a research carried out to optimize the input conditions of a shell and tube heat exchanger. The heat exchanger is modelled to exchange heat between hot water and cold water. CFD analysis was run on the model using Solid Works flow simulation tool in order to generate a series of responses for 20 input variables. The result obtained was used to generate an objective function which is an optimization model for maximizing the efficiency of the unit. The result of the CFD operation revealed that when exchange of heat in a shell and tube heat exchanger, is between same fluid type, the efficiency of the unit would largely be influenced by the input variables associated with a lower fluid capacity. For the model presented in the study, the optimized input conditions are 2.5 kg/s, 0.5 kg/s, 0.2 m/s and 0.34 m/s for the cold water inlet mass flow rate, hot water inlet mass flow rate, cold water linear velocity, and the hot water linear velocity respectively. The efficiency of the unit as a result of the optimal values of the inlet condition rose to 86.11% as against 4.43% at initial conditions.

REFERENCES

1. Chukwudi, B.C. and Ogunedo M.B. 2018. “Design and Construction of a Shell and Tube Heat Exchanger.”*Elixir Mech. Engg.*118 (2018): 50687-50691.
2. Chukwudi BC, Ogunedo MB. 2018. “Thermal design of an oil fired crucible furnace using CFD technique.” *International Journal of Research and Review.* 5(2):75-81.
3. Ogunedo, M.B. and Okoro, V.I. 2017. “CFD Simulation of a Candle Flame Propagation.” *Elixir Mech. Engg*. 108 (2017): 47813-47817.
4. Mohammed I., K. Mohammed, G. Rajmohan. 2017. “CFD Analysis of Shell and Tube Heat Exchanger.” *International Journal of Engineering Science and Computing.* 7(4): 6453-6457.
5. Jomer, C.S., S. Thomas, D. Rakesh, P. Nidheesh. 2015. “Optimization of Shell and Tube Heat Exchanger by Baffle Inclination and Baffle Cut.” *International Journal of Innovative Research in Science, Engineering and Technology.* 4(12): 69-73.

How to cite this article: Chukwudi BC, Ogunedo MB. Optimization of the inlet conditions of a shell and tube heat exchanger. International Journal of Research and Review. 2018; 5(6):145-150.

******