Simulation of shell and tube heat exchanger using COMSOL software

Trung Kim Nguyen¹², Tuan Nguyen Ba¹², Pha Bui Ngoc¹², Abdul Mutalib Embong¹, Ngoc Nguyen Thi Nhu¹², An Nguyen Si Xuan¹², Ngon Hoang Trung¹², Nurul Ashikin Abdul Kadir³, Tuan-Anh Nguyen¹², Loi Hoang Huy Phuoc Pham¹²*¹

¹ Faculty of Chemical Engineering, Ho Chi Minh City University of Technology (HCMUT), 268 Ly Thuong Kiet Street, District 10, Ho Chi Minh City, Vietnam
² Viet Nam National University Ho Chi Minh City (VNU-HCM), Linh Trung Ward, Thu Duc District, Ho Chi Minh, Vietnam
³ Centre for Foundation and Continuing Education, Universiti Malaysia Terengganu, 21030 Kuala Terengganu, Terengganu, Malaysia
⁴ School of Electrical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor, Malaysia

*Corresponding author’s e-mail: phhloi@hcmut.edu.vn

Abstract. The aim of this paper is to propose a model to simulate the behaviour of water flows in shell and tube heat exchanger. Particularly, the continuity equation, the general heat transfer equations and the energy equation in COMSOL Multiphysics software were implemented in the numerical modelling. Besides, the experiment was also conducted to validate the proposed COMSOL model. The water temperature at locations close to the inlet and outlet of the shell side was respectively predicted at 31.5°C and 34.6°C in the simulation, and it was respectively measured at 31.5°C and 35°C in the experiment. These findings showed that the simulation results had a good agreement with the experiment. Next, this model was extended to simulate the overall heat coefficient and the pressure drops of the water flows in such heat exchanger. The overall heat coefficient was at 736.62 W/m²K. The pressure drops at the inlet/outlet areas of the shell and tubes were at 849.93 Pa and 6255.50 Pa, respectively. Conclusive evidence showed that the proposed model is a reliable method for studying the heat transfer behaviour of the shell and heat exchanger.

1. Introduction

A heat exchanger is a typical device for transferring energy from one medium to another. There are several heat exchanger types available today, including shell and tube heat exchangers (STHE), plate heat exchangers (PHE), plate fin heat exchangers (PFHE), pillow plate heat exchangers (PPHE), and plate and shell heat exchangers (PAHE) [1]. They are widely applied in power plants, air conditioning, chemical plants, oil refineries, sewage treatment plants, petrochemical plants, and other chemical processes. Furthermore, the majority of conventional heat exchangers are traditional shell and tube heat exchangers which were generated in the 1930s. A STHE consists of a bundle of parallel tubes inside a shell [2], [3]. A huge number of tubes are implemented in this device due to enhancing the
heat transfer area and efficiently transferring heat. On the other hand, it is an effective method of using waste heat in order to secure energy [4].

Powerful simulation software has grown in popularity in line with the usage of computers [5]. Computer simulation, for example, gives a clear image of the complex physical phenomena that occurs in chemical engineering operations. This is possible because these simulations can provide visual representations of concepts that are difficult to visualize, such as concentration gradients, velocity profiles, and temperature gradients [6]. Although there is no alternative for laboratory conducted experiments, computer simulations can serve as a motivation to a strong understanding of fundamental chemical engineering concepts.

COMSOL Multiphysics is a software application. It is intended to integrate or connect several processes (such as heat and momentum transfer in the case of a shell and tube heat exchanger) into a single model [7]. As a result, the COMSOL software can solve several nonlinear PDE’s at the same time, and models may be created and solved in one, two, or even three dimensions. Furthermore, the COMSOL models are interactive and user-friendly, making them as an excellent tool for supplementing theoretical knowledge [5].

In one study [5], a 2-D COMSOL model of the STHE was established to simulate sea water cooling plant operation. Thus, sea water with temperatures ranging from 2 to 5°C is used to chill the cooling medium. The cooling medium is supplied at 12°C and must be cooled to 6°C. The heat transfer in the heat exchanger should result in a temperature drop of 6°C. This research aimed to plot and investigate the temperature and velocity profiles for various inlet flow velocity, pipe diameter, and pipe material. The simulation results reveal that aluminum is the preferable material of choice since it is consistent with all of the design criteria for the required heat transfer.

Currently, a numerical analysis was conducted to estimate the efficiency of a STHE as the temperature varied [8]. In this study, a 3-D model of shell and tube heat exchanger in COMSOL was implemented. Water and air were kept at 70-80°C and 5-10°C, respectively, in order to set up various combinations of inlet temperatures in the simulation. The effectiveness of the heat exchanger was predicted to range between 0.41 and 0.56 depending on the temperature. This observation demonstrated that raising the inlet temperature of the water and reducing the inlet temperature of the air can result in great efficiency.

In general, the above COMSOL models have demonstrated the heat transfer behavior of fluids in a STHE. These models, however, were not tested against the experimental data to verify their capability. This can lead to uncertainties in applying these models. Therefore, this paper aims to experimentally and numerically investigate the temperature of water flows in the STHE. The observed data from the conducted experiment are used to validate a 3-D COMSOL model including the continuity equation, the general heat transfer equation and the energy equation. Finally, the validated model is implemented to study the behavior of water flows in the STHE. It is expected the finding can develop a COMSOL model to simulate the behavior of the hot/cold water flows in the STHE.

2. Methodology to develop the model

COMSOL Multiphysics software allows for the modelling of fluid behavior in a STHE using a continuity equation, general heat transfer equations and an energy equation [9]. These models are used in this study to simulate the temperature of cold/hot water in the STHE. Details of the equations are described in the following sections. Some assumptions are used in order to simplify the numerical computations, as follows:

- Heat transfer and flow are both stable.
- There is no heat transfer to the surroundings.

2.1. Continuity equation

As mass is maintained within the control volume or infinitesimal fluid element, the rate of mass increase within a volume equals the net rate at which mass exceeds its bounding surface.
A scalar equation can be used to define mass conservation. The components of the velocity vector are space - time functions. The continuity equation can be seen as follows:

$$\nabla \cdot (\rho u) = 0$$  \hspace{1cm} (1)

where $\rho$ is the density in kg/m and $u$ is the velocity in m/s.

The second term in equation (1) is velocity divergence which is referred to as the convective term. It indicates the variation between mass flows into and mass flows out of boundaries. It must be balanced with the first term which depicts accumulation. If a fluid is incompressible, its density remains constant in both space and time.

2.2. General heat transfer equations

The equation for heat transfer in water phase is defined by equation (2).

$$\nabla \cdot (-K \nabla T) = Q - \rho C_p u \cdot \nabla T$$ \hspace{1cm} (2)

in which $K$ is the thermal conductivity in W/m.K, $T$ is the temperature in K, $Q$ is the sinks or the source term and $C_p$ is the specific heat capacity in J/kg.K.

The properties for water and metals were chosen from the built-in material library. The COMSOL software includes the effect of the turbulent fluctuations on the temperature profile by employing the Kays-Crawford model for the turbulent Prandtl number.

The equation for heat transfer in solid phase is presented as follows:

$$\nabla \cdot (K_s \nabla T_s) = 0$$ \hspace{1cm} (3)

where $K_s$ is the thermal conductivity of the solid phase in W/m and $T_s$ is the temperature of the solid phase in K.

2.3. Energy equation

$$\frac{\partial [u_i (\rho E + p)]}{\partial x_i} = \frac{\partial}{\partial x_i} \left\{ K_{\text{eff}} \frac{\partial T}{\partial x_i} + u_i (\tau_{ij})_{\text{eff}} \right\} + S_h$$ \hspace{1cm} (4)

in which $E$ is the total energy in J, $p$ is the pressure in Pa; $(\tau_{ij})_{\text{eff}}$ is the deviatory stress tensor, $S_h$ is the source, $x$ is the displacement in m, $i$ and $j$ are the vector direction and $K_{\text{eff}}$ is the effective thermal conductivity in W/m.K.

The turbulence produces an effective thermal conductivity, $K_{\text{eff}}$, as shown in equation (5).

$$K_{\text{eff}} = K + K_T$$ \hspace{1cm} (5)

$$K_T = \frac{C_p \mu_T}{Pr_T}$$ \hspace{1cm} (6)

where $\mu_T$ is the turbulent viscosity in Pa.s, $Pr_{T}$ is the turbulent Prandtl number and $K_T$ is the turbulent thermal conductivity in W/m.K.

3. Experimental setup

Figure 1 depicts the experimental setup layout which includes a STHE. In this study, the STHE type is a single-pass shell with four segmental baffles and four passes of tubes. On the tube and shell
sides, hot and cold water flows are employed, respectively. The STHE is horizontal with a carbon steel shell diameter of 80 mm. Table 1 shows the geometrical dimensions of the tubes and baffles arrangement. A heat stream pump and a heat stream tank are used to circulate the hot side of the system. Flowmeters and flow control valves are used to set a 8-LPM cold-water flow rate and an 4-LPM hot-water flow rate. The inlet hot temperature is maintained at around 55°C, while the inlet cold temperature is maintained at around 31.5°C.

![Figure 1. Sketch of experimental facility used in this study: 1) shell and tube heat exchanger, 2) reflux control valve, 3) hot stream tank, 4) heat stream pump, 5) flow control valve for hot water, 6) flowmeter for hot water, 7) flowmeter for cold water, 8) flow control valve for cold water.](image)

| Items                | Value  |
|----------------------|--------|
| Length of tube       | 340 mm |
| Number of tubes      | 20     |
| Tube Inner diameter  | 6.5 mm |
| Tube outer diameter  | 8 mm   |
| Number of baffles    | 4      |
| Baffle spacing       | 80 mm  |

Table 1. Description of tubes and baffles.

4. **Simulation setup**

In this study, the STHE in section 3 was simulated by five steps: firstly, designing a computational domain based on the geometry of the STHE and creating meshes for this domain; secondly, defining the conditions at boundaries of the domain; thirdly, setting up the above equations in COMSOL Multiphysics software to predict the inlet/outlet temperatures of hot/cold water; fourthly, validating the developed model through comparisons between simulation results and experimental data; lastly, numerically analysing behavior of hot/cold water flows in the STHE.
4.1. **Computational domain and mesh generation**

A 3-D domain was created based on the geometrical dimension of the STHE in section 3. Figure 2a displays the STHE computational domain and boundary conditions used in the simulation set-up. The computational domain was divided into a number of discrete volume elements. Tetrahedral meshes were used for this grid generation. Fine mesh has been created near the surfaces of the shell and tubes. The mesh becomes coarser near other domain boundaries. Figure 2b illustrates the generated mesh of computational domain. Because of the expensive computation and accuracy, the mesh size of 116,851 elements is used for the rest of the studies.

![Figure 2](image_url)

**Figure 2.** Computational domain and mesh used in the simulation: a) A 3-D domain of the STHE: 1) Inlet hot water, 2) Outlet hot water, 3) Inlet cold water, 4) Outlet cold water; b) Meshing of the domain.

4.2. **Boundary conditions**

Table 2 presents the boundary conditions as illustrated in Figure 2a.

| Location                  | Boundary condition                  |
|---------------------------|-------------------------------------|
| Inlet hot water (1)       | Inlet mass flow rate: 4LPM          |
|                           | Inlet temperature: 55°C             |
| Outlet hot water (2)      | Outflow                             |
| Inlet cold water (3)      | Inlet mass flow rate: 8LPM          |
|                           | Inlet temperature: 31.5°C           |
| Outlet cold water (4)     | Outflow                             |

4.3. **Solution method**

There are a wide range of solvers to choose for solving the equations in COMSOL. Thus, stationery segregated solver was selected to solve the above equations. The relative tolerance of the solver was set to $10^{-4}$.

5. **Results and discussion**

5.1. **Model validation study**

Figure 3 shows the predicted temperature profile of hot/cold water flows at various locations. From the figure 3, it can be seen from the figure 3 that temperature of hot water flow was at 55°C and 49.2°C at the locations close to the inlet and outlet of tube side, respectively. It was also found in the figure 3 that the water temperature was respectively at 31.5°C and 34.6°C at the locations near the inlet and outlet of shell side.
Figure 3. Temperature profiles for model validation.

The simulation results from the present COMSOL model were compared with the experimental data to verify its validity. Table 3 illustrates the comparisons between the predicted temperatures of water flows and the experiential data. As can be seen in the table 3 that there was a good agreement between the simulation results and the experimental data.

Table 3. Comparisons between the simulation and the experiment.

| Location          | Temperature (°C) | Deviation (%) |
|-------------------|------------------|---------------|
|                   | Simulation       | Experiment    |
| Inlet hot water   | 55               | 54.6          | 0.8            |
| Outlet hot water  | 49.2             | 47            | 4.7            |
| Inlet cold water  | 31.5             | 31.6          | 0.2            |
| Outlet cold water | 34.6             | 34.8          | 0.4            |

5.2. Case studies

In this study, an 8-LPM cold water flow was allowed to enter an 80-mm shell diameter. The tube sides were injected by hot water with a flow rate of 4 LPM. The temperature of cold water was kept constant at 55°C and the hot water temperature was maintained at 31.5°C. The following sections present the simulation results of the behavior of the hot/cold water flows in the STHE.

5.2.1. Prediction of temperature distribution in the STHE

Figure 4 shows the predicted temperature distribution of the hot/cold water flows in the STHE. From the figure 4, it can be found that there were two water flows in counter current. The temperature of the hot water was decreased and the cold water temperature was increased. This can be explained that the heat transfer process occurred in the STHE.
It can also be seen in the figure 4 that a larger heat transfer coefficient can be obtained after the baffles because of the greater velocities. This finding is similar to the simulation results in the previous studies [8], [10]. In the present study, overall heat transfer coefficient was predicted at 736.62 W/m²K.

![Figure 4. Predicted temperature profile in the STHE.](image)

5.2.2. Prediction of pressure drop in the STHE

A pressure profile in the STHE is presented in figure 5. As can be seen in the pressure profile that the pressure was decreased when the hot/cold water flows go through the STHE. This finding is in corresponding with the actual scenario. The pressure drops were 849.93 Pa and 6255.5 Pa at the inlet/outlet areas of the shell and tubes, respectively.

![Figure 5. Predicted pressure profile in the STHE.](image)

6. Conclusions
In this study, a COMSOL model was developed to simulate the behavior of the hot/cold water flows in the STHE. The experiment was also conducted to test the capacity of the developed model. The validation study showed that the developed model had a good agreement with the conducted experiment. Studied parameters such as the overall heat transfer coefficient and the pressure drop were considered in the simulations. Overall, the conclusions of these simulations are as follow:

- The overall heat transfer coefficient was 736.62 W/m$^2$K.
- The inlet/outlet areas of the shell and tubes had the pressure drops of 6255.5 Pa and 849.93 Pa, respectively.

Acknowledgement
We acknowledge the support of time and facilities from Ho Chi Minh City University of Technology (HCMUT), VNU-HCM for this study.

References
[1] Zapata B L, Escobar R F, Medina M A, and Zaragoza C M A 2009 State Variables Estimation for a Counter-Flow Double-Pipe Heat Exchanger Using Multi-linear Model 2009 Electronics, Robotics and Automotive Mechanics Conf. (CERMA 2009) (Cuernavaca) pp 349–354
[2] Jogwar S S and Daoutidis P 2009 Multi-scale dynamics in counter-current heat exchangers 2009 17th Mediterranean Conf. on Control and Automation (MED 2009) (Thessaloniki) pp 169–174
[3] Xuemei L, Jin-hu L and Lixing D 2009 The Heat Transfer Characteristics of Varied-Section Tube Heat Exchanger 2009 IITA Int. Conf. on Control, Automation and Systems Engineering (case 2009) (Zhangjiajie) pp 613–615
[4] Shi S, Niu G, Yan C and Lei W 2011 Research and Comparison of Heat Transfer and Pressure Drop in Lubricant Cooler 2011 Asia-Pacific Power and Energy Engineering Conf. (Wuhan) pp. 1–4
[5] Reddy S S, Dey T, Haribabu K, Krishnakumar H and Vishal G 2012 Optimization of Shell and Tube Heat Exchangers for Sea Water Cooling by COMSOL Multiphysics Int. J. Emerging Technol. Adv. Eng. 2 264–270
[6] Herrera M A 2010 COMSOL Assisted Modeling of a Climbing Film Evaporator Worcester Polytechnic Institute
[7] COMSOL 2018 COMSOL Multiphysics® v. 5.5 p 1742
[8] Aujla G S, Pandey S, Trivedi Y, Ghosh D, Sharma P, Tyagi A, Jain A and Kumar P 2017 Effectiveness analysis and design of shell and tube heat exchanger using comsrl multiphysics Int. J. Mech. Eng. Technol. 8 777–785
[9] William T V 2017 CFD Module User’s Guide pp 1–710
[10] Petrik M and Szepesi G L 2018 Shell side CFD analysis of a model shell-and-tube heat exchanger Chem. Eng. Trans. 70 313–318