COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF A DOUBLE PIPE HEAT EXCHANGER: MODELLING AND EXPERIMENTAL VALIDATION

Trinh Hoai Thanh[1*], Tran Thi To Nga[2]

[1]*Ho Chi Minh City University of Food Industry*

[2]*Ho Chi Minh City University of Education*

[*]Email*: onalone2000@gmail.com*

Received: 10 June 2022; Accepted: 1 September 2022

ABSTRACT

Concentric tubes (dual pipe), shell and tube, and plate heat exchangers are the most popular heat transfer devices. It has been used in various food processes such as sterilization, pasteurization, cooling, etc. In this paper, a Computational Fluid Dynamics (CFD) model of a double pipe heat exchanger was set up using the finite element method (FEM) to analyze the fluid characteristic inside the heat exchange. The model of the heat exchanger was built by coupling two major transport equations for fluid flow and energy at a steady-state, i.e., a k-epsilon turbulence model for the turbulence flow and energy transport equation for heat transfer. According to the results, the experimental data and prediction from the simulation are similar for the tube side. However, the experimental data was different from the prediction for outlet temperature at the shell side due to the heat loss from the shell to the outer environment which did not take into account in the model.

Keyword: CFD modelling, double pipe heat exchanger, finite element method.

1. INTRODUCTION

A heat exchanger (HX) is equipment that transfers or exchanges heat energy for various purposes. Heat exchangers are extremely significant in the manufacturing industry and play a vital role to recover heat energy from fluids [1]. Concentric tubes (dual pipe), shell and tube, and plate heat exchangers are the most popular heat transfer devices. The basic and simplest heat exchanger equipment is the double pipe heat exchanger, which is used to exchange heat energy in either a parallel or counter-flow arrangement [2]. The double-pipe heat exchanger is used when the desired heat transfer area is small (up to 50 m^2) [3]. Double pipe heat exchangers have various applications in industries, especially in the food industry because of their flexibility, low design, maintenance and installation costs. It is frequently used in many food industrial processes such as convenience food processing, sauces processing, vegetable creams and infant food processing, industrial vegetable processing, dairy processing, broths, industrial fruit processing, and soft drinks processing. In these processes, it involves the sterilizer process, aseptic process with direct cooling, tubular pasteurizer, and tubular cooler. Therefore, a well-understanding of the heat exchanger's behavior will help the rating or choosing a proper configuration and maintain a high heat transfer efficiency [4].

The numerical approach to solving mass, momentum, energy, and species conservation equations and associated phenomena on computers using programming languages is known as computational fluid dynamics (CFD) [5]. The basics of CFD are partial differential equations; and thus, knowledge of numerical methods is essential to find approximate solutions using the appropriate numerical technique. The computational branch of fluid dynamics has expanded beyond simple modeling and simulation of flow. The emergence of powerful and cost-effective desktop computers has prompted this development. They can model real-world physical events that are extremely complicated in nature and include a variety of physical principles, such as mass, momentum, heat, species, and electric or magnetic charge conservation. With the availability of desktop computing, the term "computational fluid dynamics" or "CFD modeling and simulation" are often now referred to as Multiphysics Modeling and simulation.

CFD analysis has been utilized to study the characteristic of a double pipe heat exchanger by many researchers such as Barzegar [6], Bejena [1], Dhrubajyoti [7], Rubén Cabello [8]. In this manuscript, a CFD model of a double-pipe heat exchanger was set up using the finite element method (FEM) to analyze the fluid characteristic inside the heat exchanger. In addition, the experimental data was conducted based on a lab-scale heat exchanger to verify the model.

2. MODEL DEVELOPMENT

The model of the heat exchanger was built by coupling two major transport modules for the transport of fluid flow and energy at a steady state. For the momentum transport, a k-epsilon turbulence model was included and solved together with Naviestock equations to describe the characteristic of the turbulence flow for both the shell side and tube side [9]. The model introduces two additional transport equations as below:

$$\frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right] + 2\mu_t E_{ij} E_{ij} - \rho\varepsilon \tag{1}$$

$$\frac{\partial(\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k} 2\mu_t E_{ij} E_{ij} - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{2}$$

where

k represents the turbulent kinetic energy

ε represents the turbulent dissipation rate

u_i represents the velocity component in the corresponding direction (and applied for 03 directions in the Cartesian coordinate)

E_{ij} represents a component of the rate of deformation

μ_t represents eddy viscosity, $\mu_t = \rho C_p \frac{k^2}{\varepsilon}$

The energy transport equation is

$$\rho C_p u \cdot \nabla T = \nabla \cdot (k \nabla T) + Q \tag{3}$$

The heat transfer between the shell side and the tube side is described as

$$-n \cdot (-k\nabla T) = d_s Q_s - \nabla_T \cdot (-d_s k_s \nabla_T T) \tag{4}$$

where

d_s is layer thickness, m

k_s layer thermal conductivity, W/(m.K)

C_p heat capacity at constant pressure, J/(kg·K

Q represents the heat sources in the fluid domain (W/m)

Q_s heat source in the thin film layer (W/m)

T is the temperature, *K*

ρ is the fluid density, kg/m^3

The model was then simulated using COMSOL Multiphysics 5.0 (licensed). To simplify this model, the 3-section heat exchanger was assumed to be a 1-section heat exchanger with a length of 3 meters and there was no heat loss in the model. In addition, the model geometry is in 3D and represented half of the equipment only. This model geometry is illustrated in Figure 1a. The mesh grid was calibrated for fluid dynamics applications with a total of 352220 elements after conducting the mesh convergence analysis as described in section 4.1. Figure 1b demonstrates the shape at one end of the double-pipe heat exchanger. The parameters for the model are in Table 1.

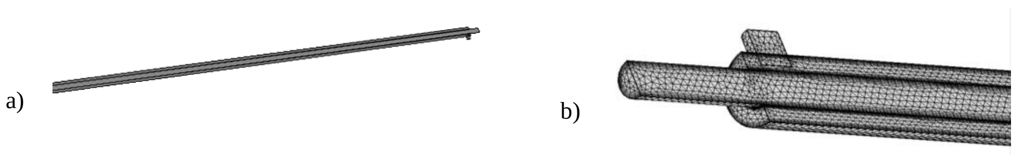

Figure 1. Double pipe heat exchanger model: a) Geometry configuration, b) mesh generated

Table 1. Double pipe model parameters

Parameter	
Shell side Diameter, mm	38
Tube side Diameter, mm	19
Tube layer thickness, mm	1
Tube length, m	3
Fluid properties	From the material library of water (temperature dependent)
Wall properties	From the material library of steel with a conductivity of 25 W/m.K (SS304, information from supplier)

The model boundary conditions were set up as in the following table.

Table 2. Boundary conditions set up for the model

Boundary conditions	k-ε module	Heat module
Inlet, Shell side	Hot stream velocity in	Inlet hot temperature
Inlet, Tube side	Cold stream velocity in	Inlet cold temperature
Oulet, Shell side	Pressure = 0	Outflow condition
Oulet, Tube side	Pressure = 0	Outflow condition
Symmetry	Symmetry faces	Symmetry faces

3. EXPERIMENTS FOR MODEL VERIFICATION

The experiments were conducted on a lab-scale double pipe heat exchanger located at the Processes and Units Operation lab, Ho Chi Minh city University of Food Industry, Vietnam. The experimental setup is shown in Figure 2 where the hot stream and cold stream are at the shell side and tube side, respectively. Both fluids were water during the experiments. The inner tube diameter is ϕ19/21 mm, and the shell diameter is ϕ38/42 mm. The length of the heat exchanger is 1 meter. The experiments were conducted to verify the CFD model, and the parameters are described in the below table.

In the experiments, the hot stream and cold stream were adjusted according to the specified flow rate mentioned in Table 3. Temperatures were measured at the inlet and outlet of equipment, the results are in

Table 5..

Table 3. Experiments parameters

Exp No.	Hot stream flow rate (*l*/min)	Cold stream flow rate (*l*/min)
1	3	3
2	6	3
3	3	9

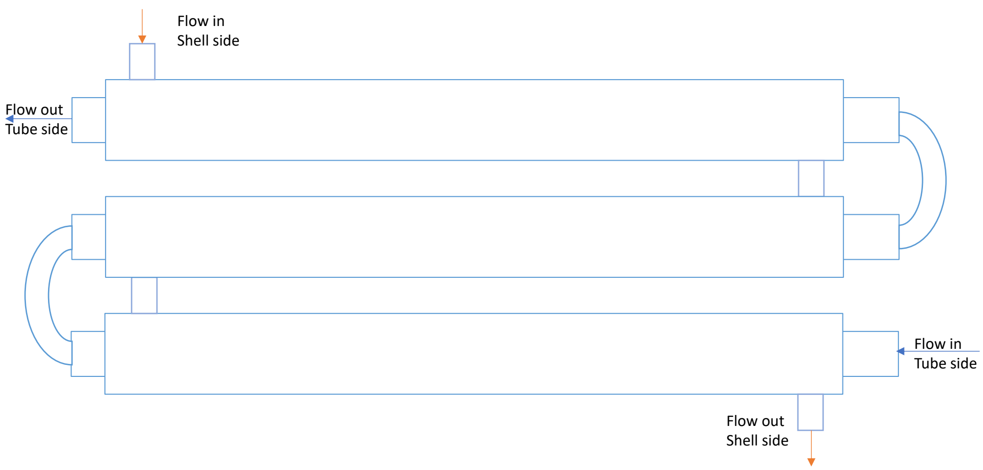

Figure 2. Experimental setup for model verification

4. RESULTS AND DISCUSSIONS

4.1. Mesh convergence analysis

Mesh convergence analysis was performed to estimate the accuracy of the simulation as in Table 3 and plotted in Figure 3. The accuracy of run no. 5 is about 1.34% and 0.93% for tube side and shell side temperature, respectively.

Table 4. The different number of elements and the predicted outlet temperature at the shell side and tube side

Run No.	Number of elements (domain + boundary + edge)	Outlet temperature – shell side	Outlet temperature – tube side
1	55123 elements	71.835	48.150
2	96295 elements	68.674	51.284
3	184865 elements	70.901	49.085
4	195689 elements	70.470	49.509
5	352220 elements	70.243	49.743

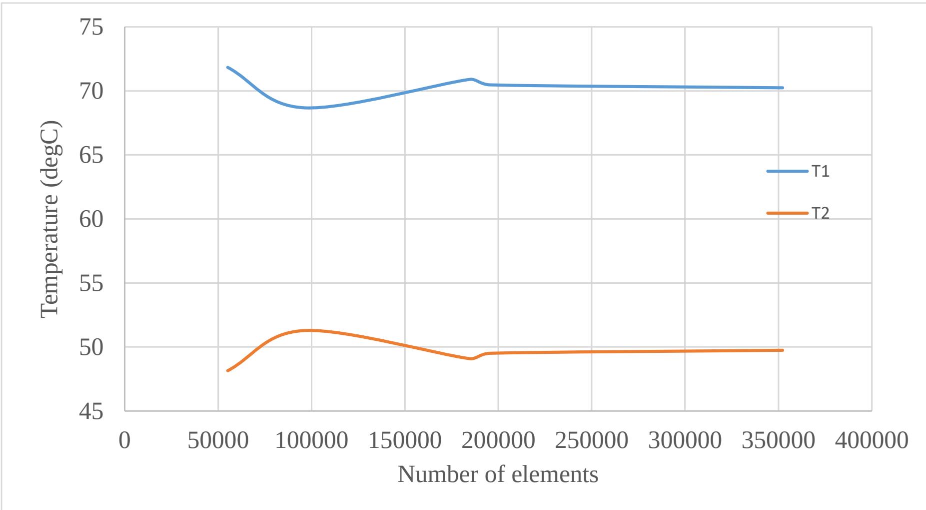

Figure 3. Mesh convergence analysis based on outlet temperatures (T1 - shell side, T2 - tube side)

As illustrated in Figure 3, the predicted temperature accuracy increased as the number of elements increased. It means that the predicted temperature stability is increased with the increase of total elements. From this analysis, the mesh configuration of run 5 was chosen as the model for laboratory experiment verification and also CFD analysis later on.

4.2. Lab-scale experiment verification and model analysis

Our experiments to validate the model were conducted in the Processes and Units Operation Lab, and the results are illustrated in

Table 5.. During the experiments, the temperatures of the inlet and outlet for the hot stream (shell side stream) and cold stream (tube side stream).

The simulations were run based on the inlet condition and predicted the outlet results and displayed in Table 6.. When comparing the predicted and experimental values, it can be seen that the tube side temperature is very close to each other while the shell side temperature is far from the experimental values. It can be explained by the heat loss from the shell to the outside environment, especially when the shell is not insulated (a simple calculation was made and found that heat losses in the experiments were about 25%). The current model does not take into account the heat loss to the environment hence it gave a difference in the prediction of the shell side of the heat exchanger when the heat loss is high. The future model should include the environmental conditions and takes into account the heat loss to the environment.

Table 5. Experimental results for double pipe heat exchanger with two water fluids, counterflow

Exp no.	Hot stream flow rate (*l*/min)	Cold stream flow rate (*l*/min)	Tube side inlet temperature (ºC)	Tube side outlet temperature (ºC)	Shell side inlet temperature (ºC)	Shell side outlet temperature (ºC)
1	3	3	45	54	75	64
2	6	3	51	61	76	71
3	3	9	50	53	77	63

Table 6. Simulation results for double pipe heat exchanger with two water fluids, counterflow

Simulation no.	Tube side inlet temperature (ºC)	Tube side outlet temperature (ºC)	Shell side inlet temperature (ºC)	Shell side outlet temperature (ºC)
1	45	52.963	75	67.087
2	51	60.385	76	71.354
3	50	53.091	77	67.807

Based on the model, the fluid flow characteristics can be predicted and shown in Figure 4 and Figure 5. Figure 6 represents the change in velocity magnitude and temperature of a hot stream and a cold stream along the length of the heat exchanger. Figure 4 used the "Slices plot" to illustrate the distribution of velocity magnitude across the heat exchanger. Ten slices were used across the model, and velocities distribution on each slide was presented. The surface plot in Figur 5 demonstrates the temperature distribution on the surface of the tube and shell of the double-pipe heat exchanger. In the software, the user can see and rotate the model to see the temperature distribution of the whole heat exchanger. However, only two ends of the heat exchangers were portrayed in this manuscript. Another picture usually gets interesting is the relationship between the temperature of the hot stream and cold stream along the length of the heat exchanger as depicted in Figure 6. It shows the behavior or the interaction between the hot and cold streams. There is much more information that can be extracted from the model, however, it will not be discussed here. In a nutshell, this model can be used to characterize the fluid flow inside the double-pipe heat exchanger, however, it is outside the scope of this manuscript.

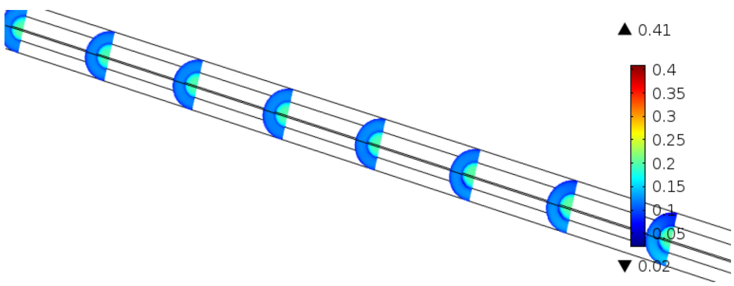

Figure 4. Velocity magnitude across the heat exchanger (unit of the legend is m/s).

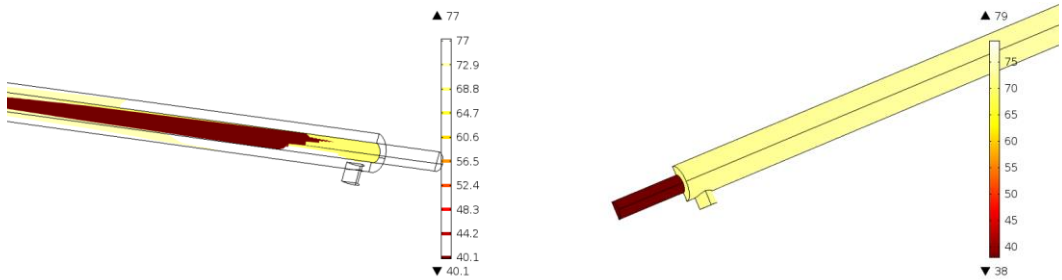

Figure 5. Temperature distribution at one end of the heat exchanger (tube side inlet, shell side outlet)

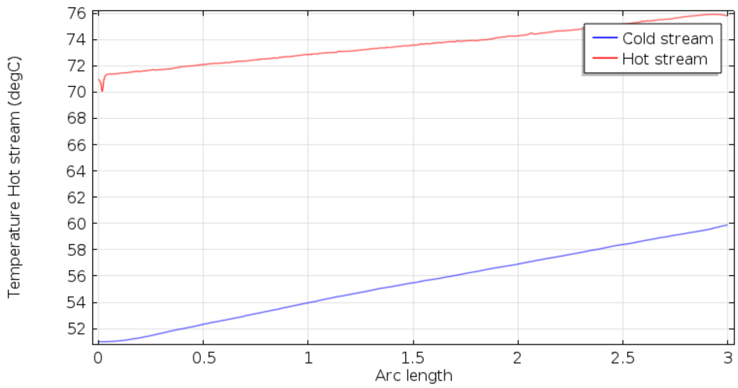

Figure 6. Temperature distribution across the length of the heat exchanger (shell side is the hot stream and the tube side is the cold stream).

5. CONCLUSION

The momentum and heat transport equations were applied and coupled to the double pipe heat exchanger to approximate the fluid flow behavior, and the results are well consistent with experiments for the tube side. However, the characteristic of the shell side is not fully simulated due to the extreme heat loss to the environment of the lab-scale equipment. It should include the environment factor in the future model and get a more accurate model for the shell side, especially when the heat loss is high. In a nutshell, CFD modeling helps to reduce the cost, time consumption, and effort in predicting the characteristic of a double-pipe heat exchanger.

REFERENCES

1. Baru Debtera and Ibsa Neme - CFD simulation of a double pipe heat exchanges: Analysis conduction and convection heat transfer, Int. J. Sci. Res. Rev. **7** (12) (2018) 329-338.
2. Holman J. - Heat transfer, 10th edition. Boston: McGraw-Hill Education (2009).
3. Vollaro A. D. L., Galli G., and Vallati A. - CFD analysis of convective heat transfer coefficient on external surfaces of buildings, Sustainability **7** (7) (2015) 9088-9099. https://doi.org/10.3390/su7079088.
4. George Saravacos and Athanasios E. Kostaropoulos - Handbook of food processing Equipment, 2nd ed. London: Springer (2016).
5. Martín M.M. - Introduction to Software for Chemical Engineers. Taylor & Francis, (2014).

6. Barzegar A. and Jalali Vahid D. - Numerical study on heat transfer enhancement and flow characteristics of double pipe heat exchanger fitted with rectangular cut twisted tape, Heat Mass Transf. **55** (12) (2019) 3455-3472. https://doi.org/10.1007/s00231-019-02667-1.
7. Dhrubajyoti Bhattacherjee - CFD analysis of double pipe counter flow heat exchanger, Int. J. Eng. Res. Technol. **9** (10) (2020) 506-518.
8. Rubén Cabello, Plesu-Popescu A.E., Bonet-Ruiz J., Cantarell D.C., Llorens J. - Validation of CFD models for double-pipe heat exchangers with empirical correlations, Chem. Eng. Trans. **88** (2021) 1243-1248. https://doi.org/10.3303/CET2188207.
9. K-epsilon turbulence model, Wikipedia (2022).

TÓM TẮT

MÔ HÌNH ĐỘNG HỌC LƯU CHẤT TRONG THIẾT BỊ TRUYỀN NHIỆT ỐNG LỒNG ỐNG: MÔ HÌNH HÓA VÀ KIỂM CHỨNG THỰC NGHIỆM

Trịnh Hoài Thanh[1]*, Trần Thị Tố Nga[2]
[1]*Trường Đại học Công nghiệp Thực phẩm TP.HCM*
[2]*Trường Đại học Sư phạm TP.Hồ Chí Minh*
*Email: *onalone2000@gmail.com*

Các loại thiết bị trao đổi nhiệt ống lồng ống, thiết bị vỏ ống, và tấm bản là những thiết bị trao đổi nhiệt phổ biến. Nó được sử dụng trong nhiều quá trình trong thực phẩm như là thanh trùng, tiệt trùng, làm nguội, v.v. Bài viết này trình bày mô hình động học lưu chất (Computational Fluid Dynamics - CFD) của một thiết bị trao đổi nhiệt dạng ống lồng ống. Phương pháp phần tử hữu hạn (FEM) được sử dụng để phân tích tính chất của các lưu chất trong thiết bị truyền nhiệt. Mô hình này kết hợp giải đồng thời hai phương trình vi phân chính của quá trình truyền vận cho dòng chảy và năng lượng ở trạng thái ổn định. Phương trình k-epsilon cho dòng chảy rối và phương trình truyền năng lượng cho quá trình truyền nhiệt. Kết quả mô phỏng và thực nghiệm cho thấy rằng giá trị tính toán được từ mô hình và thực nghiệm khá giống nhau đối với dòng chảy phía ống. Tuy nhiên, đối với dòng chảy phía vỏ, số liệu thực nghiệm thấp hơn nhiều so với số liệu tính toán từ mô hình do sự mất mát nhiệt từ ra ngoài thành thiết bị, một yếu tố chưa được đưa vào trong mô hình hiện tại.

Từ khóa: Mô phỏng động học dòng chảy, thiết bị truyền nhiệt ống lồng ống, phương pháp phần tử hữu hạn.