ALKU Journal of Science 2023, Sayı 5(3): 118-130
e-ISSN: 2667-7814

Original Article

A Numerical Study on Cross Flow Heat Exchanger with Different Reynolds (Re) Numbers

Gökhan CANBOLAT[1*]
[1]Alanya Alaaddin Keykubat University, Rafet Kayış Faculty of Engineering, Department of Mechanical Engineering, Antalya, Turkey
**gokhan.canbolat@alanya.edu.tr*

Abstract

Heat exchangers are highly popular in engineering and industrial applications. Numerical studies on heat exchangers to investigate the performance of heat transfer have been carried out widely by Computational Fluid Dynamics (CFD) in recent years. In this study, a circular pipe with hot water in cross flow is investigated in different Reynolds (Re) numbers. Flow is turbulent flow and the Re number varies from 3165 to 4643 in the circular pipe. The air is at a temperature of 303 K and the water is at 333 K. Variation of flow characteristics and thermal performance is observed according to an increase in Re numbers such as Wall Shear Stress (WSS), Skin Friction Coefficient (C_f), Nusselt Number (Nu), heat transfer coefficient (h) and surface temperature of the circular pipe. Results show that there are no significant changes for the WSS and C_f values in the specified range of the Re number. However, when the thermal performance is evaluated, the temperature of the surface of the circular pipe, heat transfer coefficient, and Nu number values are increased by an increase in the Re number. Here, the increase is approximately 2% for the specified range of Re number, and it is shown that it can be increased by the flow conditions. The maximum Nu number is 4482.37 at the Re number of 4643. As a result, the Re number is highly effective in controlling the heat transfer performance of a heat exchanger.

Keywords: Heat Exchanger, Cross-Flow, Computational Fluid Dynamics (CFD), Heat Transfer

Farklı Reynolds (Re) Sayılarında Çapraz Akışlı Bir Isı Değiştirici Üzerine Nümerik Bir Çalışma

Özet

Isı değiştiriciler mühendislik ve endüstriyel uygulamalarda oldukça popülerdir. Isı değiştiriciler üzerindeki ısı transferi performansının araştırılmasına yönelik sayısal çalışmalar son yıllarda Hesaplamalı Akışkanlar Dinamiği (HAD) yöntemiyle yaygın şekilde gerçekleştirilmektedir. Bu çalışmada farklı Reynolds (Re) sayıları ile çapraz akışlı sıcak su içeren dairesel bir boru incelenmiştir. Türbülanslı bir akışta, dairesel boru için Re sayısı 3165 ile 4643 arasında değişmektedir. Hava 303 K, su ise 333 K sıcaklıktadır. Akış karakteristikleri ve termal performans açısından Duvardaki kayma gerilmesi, Yüzey sürtünme katsayısı (Cf), Nusselt sayısı (Nu), Isı transferi kaysayısı (h) ve yüzey sıcaklığı Re sayısındaki artışa göre incelenmiştir. Sonuçlar, belirtilen Re sayısı aralığında kayma gerilmesi ve Cf değerleri için önemli bir değişiklik olmadığını göstermektedir. Ancak ısıl performans değerlendirildiğinde, Re sayısının artmasıyla dairesel borunun yüzey sıcaklığı, ısı transfer katsayısı ve Nu sayısı değerleri de artmaktadır. Burada, belirtilen Re sayısı aralığı için artış yaklaşık % 2 olup, akış koşullarıyla artırılabileceği gösterilmektedir. Re sayısı 4643'te maksimum Nu sayısı 4482.37 olarak hesaplamıştır. Sonuç olarak Re numarası, bir ısı değiştiricinin ısı transfer performansının kontrolünde oldukça etkilidir.

Anahtar Kelimeler: Isı Değiştirici, Çapraz Akış, Hesaplamalı Akışkanlar Dinamiği (HAD), Isı Transferi

Geliş/Received: 28 Eylül/September 2023
Kabul Ediliş/Accepted: 04 Ekim/October 2023

1. INTRODUCTION

Heat exchangers are popularly utilized in industrial and engineering applications. In the design of heat exchangers, cross-flow in tubes is widely used in engineering and industrial applications such as flow across over the tubes, cooling systems, electronic types of equipment, production process, boiler economizers, air preheater, air conditioning, and automotive radiator, etc. Therefore, various studies have been performed to observe the heat transfer in such flow [1]. Due to the increasing demands for energy saving in recent years, the need for efficient design of heat exchangers with low pressure drop and high heat transfer increase has increased [2].

The procedure of design for the heat exchanger is highly complicated. The analysis of heat transfer is needed to perform efficiently in terms of long-term performance and economy. While new technologies are integrated to increase the heat transfer by the improvement in the heat transfer rate, the pressure drop also increases, resulting the a higher pumping cost. Cross-flow exchangers are generally used by fins or without. Here, the high difference between the heat transfer coefficient for fluids such as gas and liquid, the heat transfer rate will be limited with lower heat transfer coefficient for gas. Here, fins are not mandatory for installation but fins increase the heat transfer area [3,4]. The heat transfer enhancement can be performed by various techniques for the heat exchangers. Here, these techniques are divided into two methods they are passive and active. In the active methods, an external method is needed such as electric, mechanical equipment, and vibration. On the other hand, in the passive methods, external power is not needed but surface geometry or additive fluid causes the heat transfer phenomenon. However, passive ones are commercially interested in heat transfer enhancement. Heat exchangers can be classified primarily according to flow types they are counter, parallel, and cross-flow. Cross-flow as in this study, the fluid flow is perpendicular to each other [5].

In a study, a tube exchanger is studied both numerically and experimentally. Large eddy simulation (LES) is also integrated to study to investigate the fluid flow and heat transfer. Small tubes with a diameter of 5.2 mm are used as flow channels. It is concluded that small tubes increase the heat transfer coefficient compared the large tubes and the LES method is highly effective in predicting fluid mechanics and heat transfer for the heat exchangers [6]. In a study, a steady, laminar, and two-dimensional incompressible flow is investigated for the tube bundles for a heat exchanger. The effects of Reynolds number, Prandtl number, and aspect ratio are studied to observe pressure drop and heat transfer performance. They reported that the Nu number increases with an increase in the Reynolds number. A higher length for the aspect ratio is more efficient in increasing the heat transfer ratio. Equivalent circular tubes perform better heat transfer according to flat tube banks. When the pressure drop is investigated, flat tube banks are better according to circular tubes [7]. In a study, the effect of the longitudinal pitch on heat transfer performance in a cross-flow is investigated over an inline tube heat exchanger. A geometry with symmetric and periodic boundary conditions is implemented for analysis. A single phase and turbulent flow are studied to examine the turbulent flow. In the study by Zukauskas, it was reported that the heat transfer coefficient can be reduced by correlation with a decrease in the longitudinal pitch [8]. The heat transfer performance can be estimated with the empirical correlations in the literature [9].

In this study, a cross-flow heat exchanger is studied with different Re numbers under turbulent flow conditions. The geometry is constructed as a one-way hot water to air that is perpendicular to water flow. Analysis is performed by the CFD and uniform mesh generation is aimed before the studies. Flow is three-dimensional, incompressible, and transitional flow is also taken into account during the flow. The Re numbers are 3165, 3587, 3904,4221, and 4643 for the turbulent flows. Air is at a temperature of 303 K and water is at 333 K. So the temperature difference is 30 K between water and air. The variation of WSS, C_f, Nu, h, and temperature on the pipe surface is investigated for comparison. In the given range of the Re numbers, the increase in heat transfer is approximately 2%. For the case of Re number is 4643, the maximum Nu number is 4482.37 obtained by CFD studies. CFD calculations are highly effective in guiding heat transfer enhancement for heat exchangers.

2. MATERIAL AND METHOD

In this section, the definition of the geometry for the heat exchanger, generation of mesh and mesh independency test, boundary conditions for the flows for air and water, governing equations for the fluid flow and heat transfer, and the numerical approach are shown.

2.1 Geometry, Generation of Mesh and Mesh Sensitivity Test

The geometry with mesh is shown in Figure 1. It is seen in the figure that L is the length of the pipe with 50 mm, H is the height with 50 mm, and W is the width with 50 mm. So, the heat exchanger is constructed as 50 mm x 50 mm x 50 mm. Here, developing flow effects are also considered for the flow, and pipe length is not provided ten times the pipe diameter. In a turbulent flow, the length needs to be a minimum ten times of the pipe diameter [10]. Here, while the air inlet as in the seen figure is through the Y direction, the water inlet is through the Z direction. The pipe is constructed in the center of the XY plane at the coordinate system. The blue regions are the inlets of air and water in the figure.

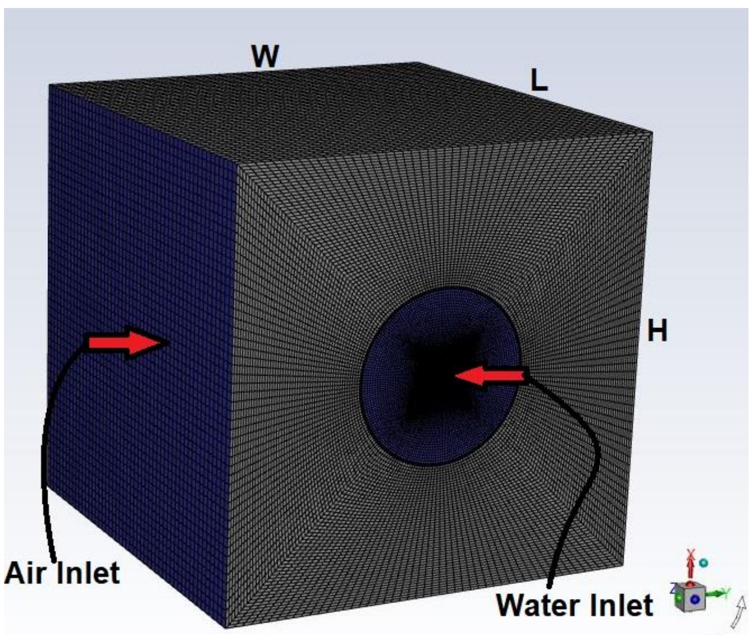

Figure 1. Meshed geometry for the heat exchanger

The hexahedral structured mesh is generated during meshing for the airflow and water flow separately by using the Ansys Fluent meshing tool. The meshing is an important stage during the numerical analysis and uniformity and adequate mesh number are needed to be supplied [11-17].

Detailed mesh views are shown in Figure 2. The general mesh is shown in Figure 2 (a) and a detailed view of the fluid domain is shown in Figure 2 (b). The hexahedral mesh is also used for the circular pipe. The mesh closed the pipe has small elements and it increases from the pipe wall to outward. Also, the mesh in the fluid domain is quite dense to solve the boundary layer. Here, y^+ value and WSS are used to obtain sufficient element numbers.

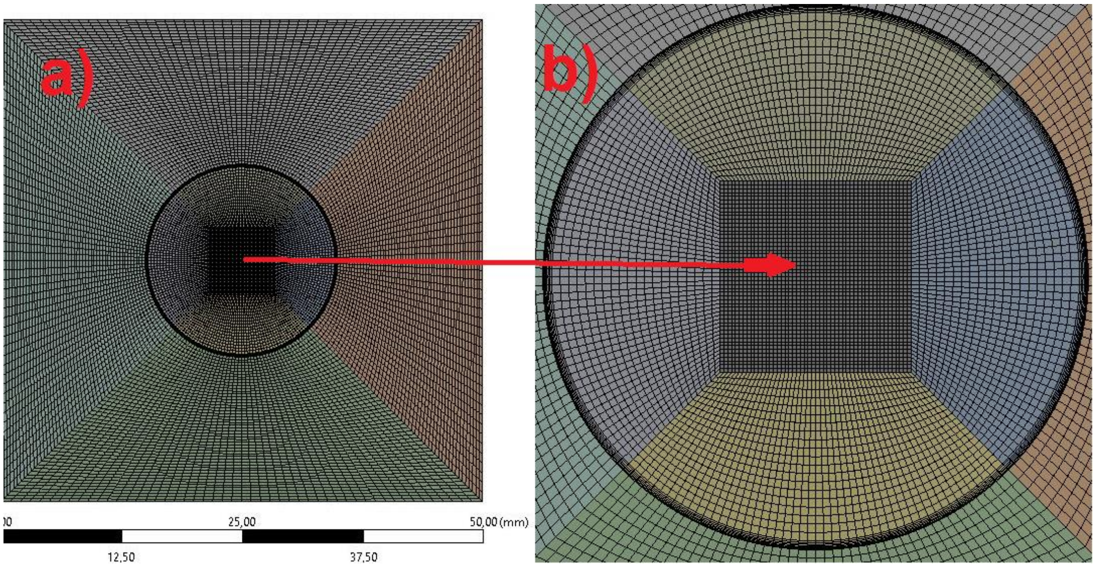

Figure 2. Detailed mesh view of the water inlet side of the heat exchanger

The mesh independency test is shown in Figure 3. This test is performed according to WSS and y^+ values. Blue bars indicate the y^+ values along the pipe surface in turbulent flow, and orange bars indicate the WSS values along the pipe surface. The mesh independency test studies are started by 103268 the element number to 8495087. It is increased within the element size and WSS and y^+ values are reported. In the first mesh, the y^+ value is 3.56 and the WSS value is 7.5 Pa. However, these values decrease to 1.183 and 10.37 Pa with the decrease in the element size. The 7-layer inflation method is constructed in the fluid domain to investigate the boundary layer in detail with the 1.2 growth rate. The dense mesh is generated by the inflation method. Here, the mesh with y^+ value for 1.2 and 10.35 Pa for the WSS is determined for the numerical analysis. The element number is 0.75 mm. Thus it is avoided to perform the numerical calculation with the insufficient mesh element number and it is also avoided the computational cost with large mesh element numbers. y^+ is a nondimensionalized variable with the same formulation of the Re number except for velocity, the velocity is friction velocity for the velocity y^+.

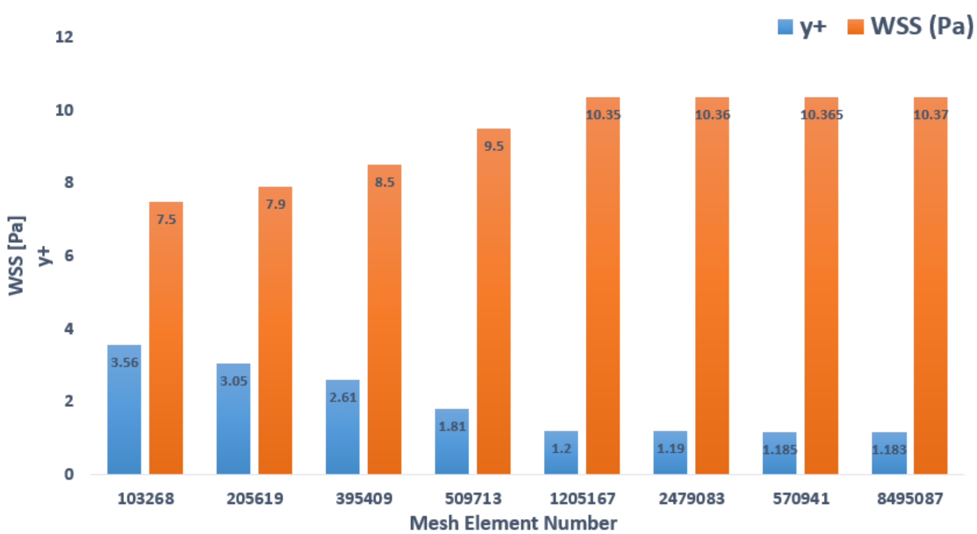

Figure 3. Mesh independency test

2.2 Boundary Conditions, Numerical Methods and Governing Equations

In this study, an incompressible, fully developed, and steady-state flow is studied. The no-slip boundary condition is applied at the pipe and heat exchanger walls. The pipe's inner wall is the heat transfer interface surface for the flows. So heat transfer occurs from hot water inlet to air by this surface.

Table 1. Thermophysical properties of the water

Density (ρ)	983.3	kg/m^3
Viscosity (μ)	0.467×10^{-3}	kg/m.s
Thermal Conductivity (*k*)	0.654	W/m.K
Specific Heat (c_p)	4185	J/kg.K

Table 1 presents the thermophysical properties of the water at the temperature of 333 K. The inlet temperature is 333 K for the hot water. Table 2 presents the thermophysical properties of the air at the temperature of 303 K. The inlet temperature is 303 K for the air in this study.

Table 2. Thermophysical properties of the air

Density (ρ)	1.164	kg/m^3
Viscosity (μ)	1.872×10^{-5}	kg/m.s
Thermal Conductivity (*k*)	0.02588	W/m.K
Specific Heat (c_p)	1007	J/kg.K

$$\mathrm{Re} = \frac{\rho U D_h}{\mu} \tag{1}$$

Reynolds Number (Re) was used to predict the fluid motion whether it is laminar or turbulent flow in fluid mechanics. Re is the ratio of inertial forces to viscous forces. It is an important dimensionless number. ρ, D_h, U and μ are density [kg/m3], characteristic length [m], velocity [m/s], and dynamic viscosity [Pas] in Equation 1.

The fluid motion was expressed by the governing equations for Newtonian, steady-state, and incompressible, flow. The continuity (conservation of mass) and momentum Equations (2) and (4), respectively [18].

$$\nabla . \vec{V} = 0 \tag{2}$$

$$\nabla = \vec{i}\frac{\partial}{\partial x} + \vec{j}\frac{\partial}{\partial y} + \vec{k}\frac{\partial}{\partial z} \tag{3}$$

The above Equation (2) is known as the Continuity Equation. $\vec{V}$ denotes the velocity vector. In this study, fluid is incompressible, hence, density was constant. Newtonian, incompressible, steady-state momentum equation is governed by Equation (4) for the flow.

$$\rho\left(\vec{V}.\nabla\right)\vec{V} = -\nabla P + \rho\vec{g} + \mu\nabla^2\vec{V} \tag{4}$$

p, μ and $\vec{g}$ are pressure [Pa], dynamic viscosity [Pas], and gravity vector above, respectively.

$$\rho\, c_p \left(\vec{V}.\nabla\right) T = k\left(\nabla^2 T\right) \tag{5}$$

$$\rho\, c_p \left(u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} + w\frac{\partial T}{\partial z}\right) = k\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}\right) \tag{6}$$

The energy equation is given in Equation (5).

$$\tau_w = \mu \frac{du}{dr} \tag{7}$$

In Equation (7); τ_w, μ and $\frac{du}{dr}$ are the shear stress, dynamic viscosity, and shear rate along the r direction perpendicular to the pipe surface respectively.

$$Nu = \frac{h\,D}{k} \tag{8}$$

where k is the thermal conductivity of fluid [W/m.K], D is the diameter of the tube [m], and h is the local heat transfer coefficient [W/m^{2}K] respectively, and the Nusselt number is defined by Equation (8).

The SST k-ω model was used to predict the turbulent flow in this study. Transport equations of the SST k-ω model are given by Equation (9) and Equation (10);

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\Gamma_k \frac{\partial k}{\partial x_j}\right] + G_k - Y_k + S_k \tag{9}$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho \omega u_j)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\Gamma_\omega \frac{\partial \omega}{\partial x_j}\right] + G_\omega - Y_\omega + D_\omega + S_\omega \tag{10}$$

k and ω are the kinetic energy and specific dissipation rate in the SST k- ω turbulent model. G_k is defined as the generation of turbulence kinetic energy. G_ω is the generation of ω. Γ_ω and Γ_k are the effective diffusivity of ω and k, respectively. Y_ω and Y_k and are the dissipation of ω and k. S_k and S_ω are user-defined source terms in Equation [19].

Numerical calculations are performed by all second-order upwind discretization to discretize the pressure, momentum, turbulent dissipation rate, and turbulent kinetic energy. The convergence criteria for residuals is chosen 10^{-5}. Flow is steady-state and the energy equation is enabled during calculations.

3. RESULTS AND DISCUSSION

In this study, the aim is to observe the effects of the Re number on the thermal performance of the heat exchanger. The hot inlet is supplied by water at the temperature of 333 K, and the cold inlet is supplied by the air at the temperature of 303 K. Wall Shear Stress (WSS), Skin Friction Coefficient (Cf), Nusselt Number (Nu), heat transfer coefficient (h), and surface temperature are compared according to increase in Re number range from 3165 to 4643. Results are presented for Wall Shear Stress (WSS), Skin Friction Coefficient (Cf), Nusselt Number (Nu), heat transfer coefficient (h), and surface temperature by Re numbers in this section.

Figure 4 presents the variation of velocity for the air passing through the pipe. Air enters the heat exchanger with a velocity of 10 m/s and a temperature of 303 K. As it is seen, the velocity of air is nearly zero at the point that contacts the pipe surface because of the stagnation point that occurs there. It increases on the upper side and the lower side by the narrowing in the airway.

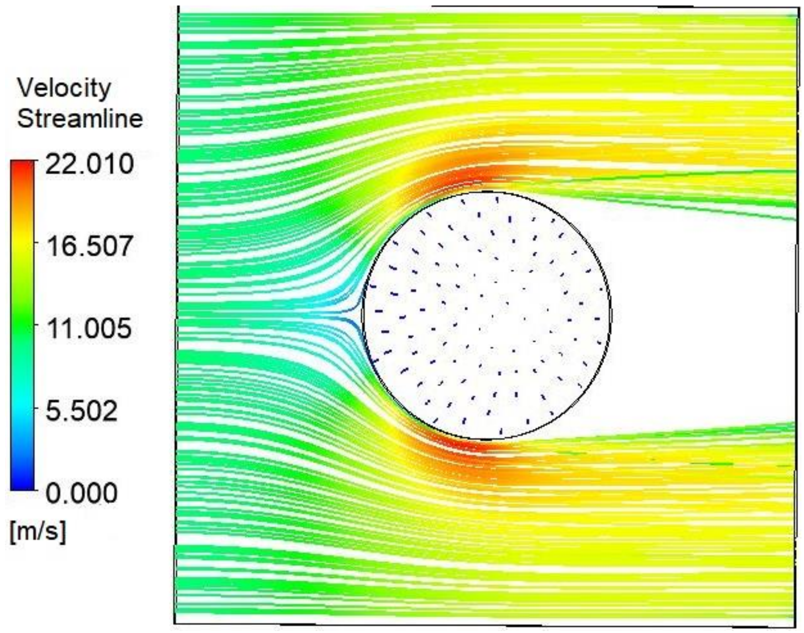

Figure 4. Velocity streamlines for the air passing through the pipe

Figure 5 shows the temperature distribution near the pipe surface for the air and water. The water enters the pipe with 333 K. As seen in the figure the temperature of the air is 303 K closed by the heat exchanger wall. But, the air temperature increases close to the pipe surface because of the heat transfer from the hot water to the air. The temperature of the water is seen as 333 K in the pipe but it decreases to the wall that contact with the air.

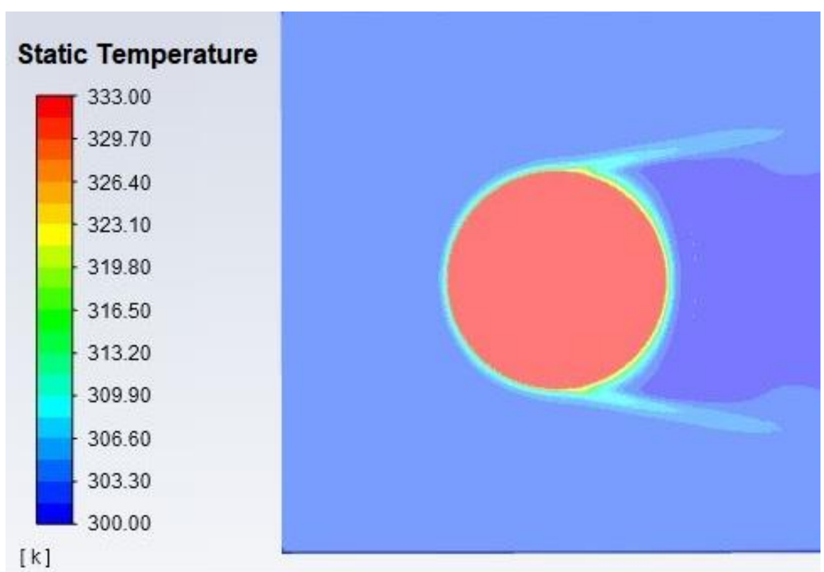

Figure 5. Temperature contour for the air passing through the pipe

Figure 6 shows the variation of the C_f values on the surface of the pipe along the flow direction. The C_f values are similar for the specified Re number for the flows. Due to the range of Re number is in a limited range. C_f values are presented to observe the transition from laminar flow to turbulent flow. C_f value is an indicator to investigate the transition from laminar to turbulent flow [20, 21].

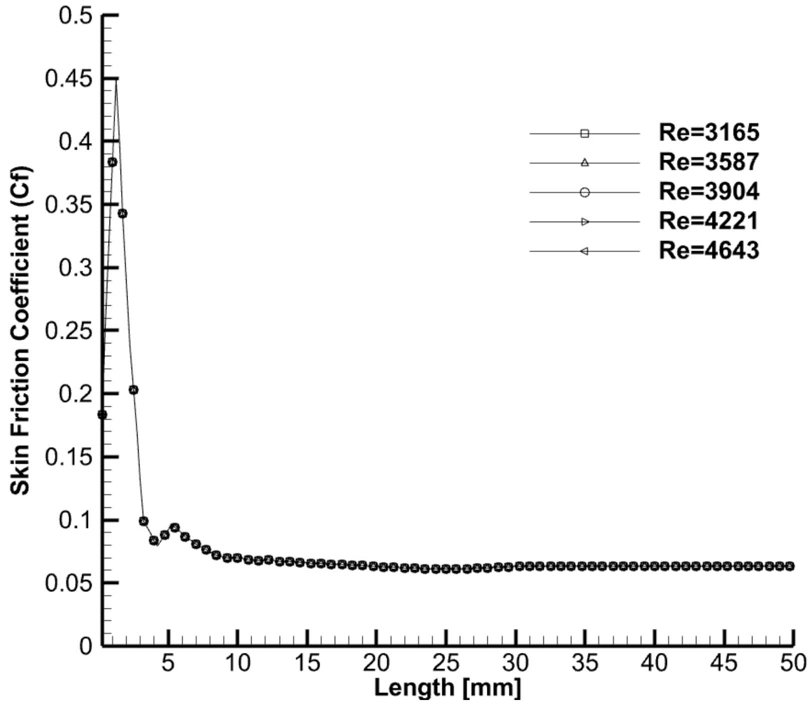

Figure 6. Variation of C_f along the pipe

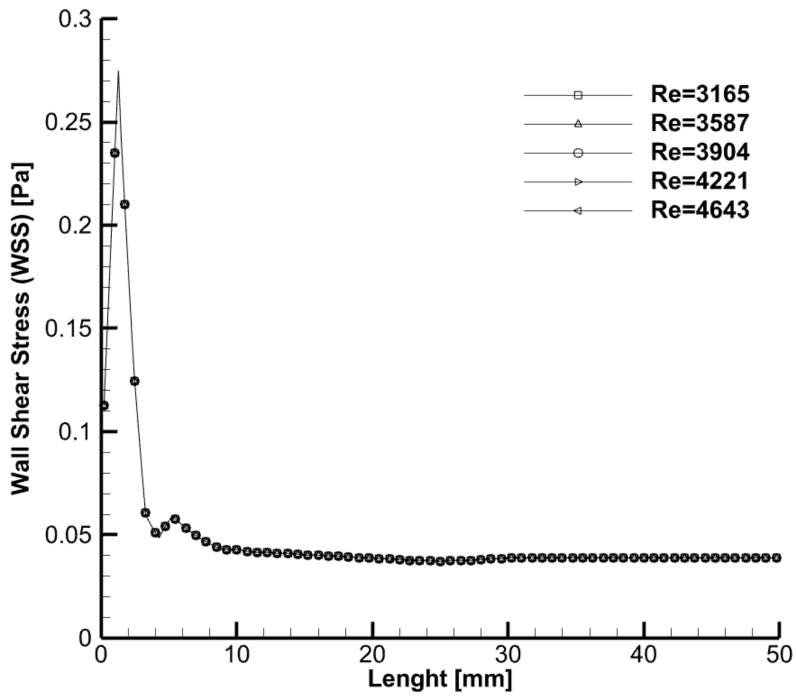

Figure 7. Variation of WSS along the pipe

Figure 7 shows the variation of the WSS values on the surface of the pipe along the flow direction. As seen in the C_f values, WSS values are also similar for the given Re numbers because of a limited range. The transition from laminar to turbulent flow is also seen in the WSS values along the pipe. The WSS is nearly 0.04 Pa during the turbulent flow. In the transition region, it is between 0.05- 0.06 Pa. It reaches its higher value when the flow is fully turbulent, and then it decreases [18].

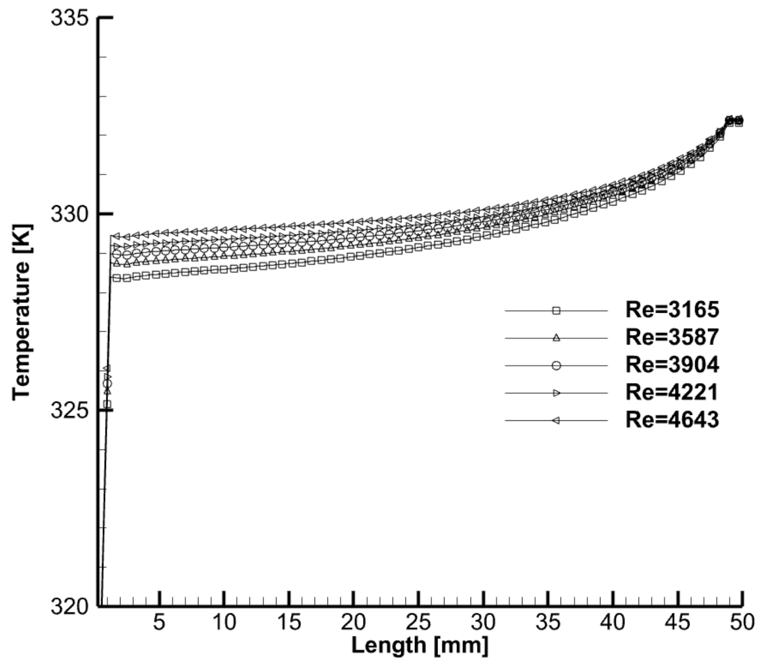

Figure 8. Variation of the surface temperature along the pipe

Figure 8 shows the variation of the surface temperature along the flow direction. Here the highest surface temperature occurs for Re number 4643. The lowest surface temperature occurs for the Re 3165. As expected the increase in Re number also increases the surface temperature along the pipe because of heat transfer from water to the air [22-25]. Here, the increase is approximately 2% for the specified range of Re number for the surface temperature for the specified pipe length. The difference decreases along the flow direction. As seen in the figure temperature difference is higher in the transition region. Even though surface temperature increases, the difference between the flow for specified Re numbers decreases.

Figure 9 shows the variation of the heat transfer coefficient along the flow direction. Here the highest heat transfer coefficient occurs for Re number 4643. The lowest heat transfer coefficient occurs for the Re number 3165. As expected the increase in Re number also increases the heat transfer coefficient because of heat transfer from water to the air. Here, the increase is approximately 2% for the specified range of Re number for the heat transfer coefficient for the specified pipe length. The difference decreases along the flow direction. As seen in the figure heat transfer coefficient difference is higher in the transition region. Even though surface temperature increases, the difference between the flow for specified Re numbers decreases.

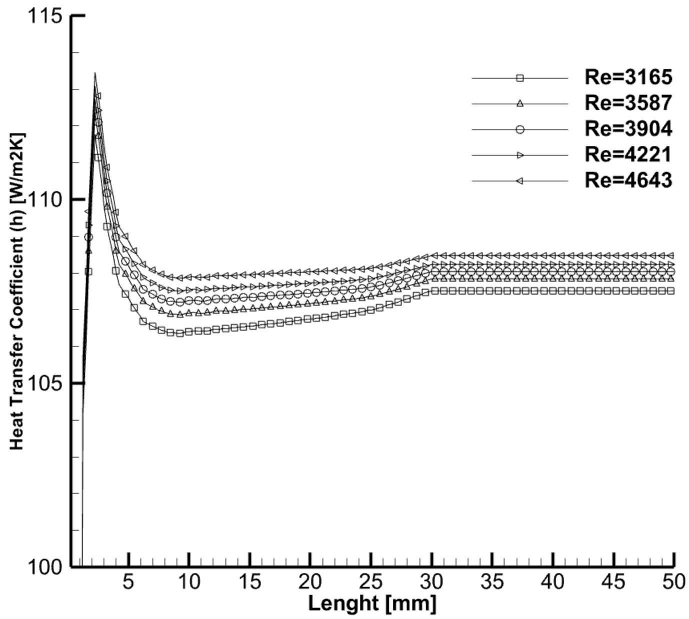

Figure 9. Variation of the heat transfer coefficient along the pipe

Figure 10 shows the variation of the Nu number along the flow direction. Here the highest Nu number occurs for Re number 4643. The lowest Nu number occurs for the Re number 3165. As expected the increase in Re number also increases the Nu number because of heat transfer from water to the air. Here, the increase is approximately 2% for the specified range of Re number for the Nu number for the specified pipe length. The difference decreases along the flow direction. As seen in the figure Nu number difference is higher in the transition region [26–29]. Even though the Nu number increases, the difference between the flow for specified Re numbers decreases.

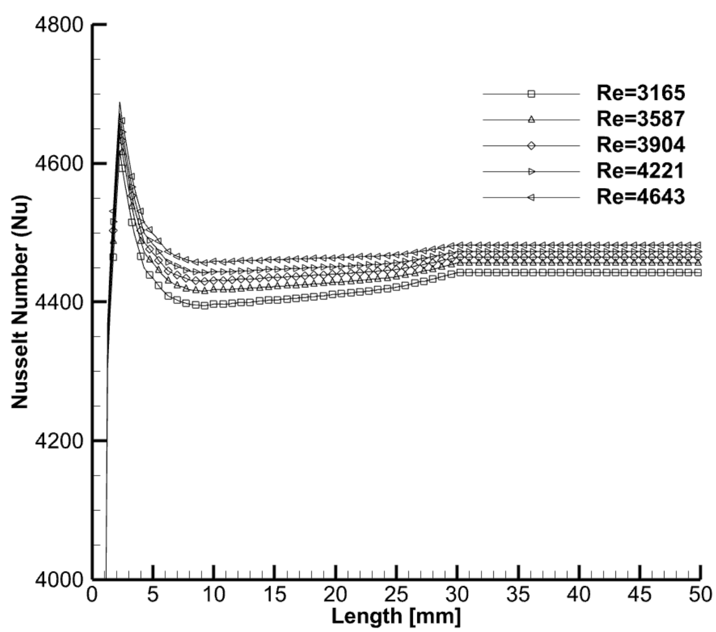

Figure 10. Variation of the Nu number along the pipe

4. CONCLUSION

In this study, a heat exchanger is studied numerically in terms of thermal performance according to the variation of the Re number. The Re number has changed the range from 3165 to 4643 for the five different values. The numerical calculations were repeated for all Re number values with the same boundary conditions except for Re number. The inlet velocity of the air is 10 m/s at the temperature of 303 K and the water temperature is 333 K so the temperature difference is 30 K.

According to the results, the following evaluations were reached:

- When the thermal performance of the heat exchanger is assessed for different Re numbers, the difference is highly low for the given limited ranger.
- The low differences for the investigated results that Wall Shear Stress (WSS), Skin Friction Coefficient (Cf), Nusselt Number (Nu), heat transfer coefficient (h), and surface temperature can be increased by an increase in the Re number. Because of a low limited range for Re numbers studied in the study, therefore differences are observed smaller than 2 %.
- The thermal indicators observed in this study are higher in the transition region according to the laminar and fully turbulent region.
- The increase in Re number increases the heat transfer coefficient and Nu number.

REFERENCES

[1] S. S. Paul, S. J. Ormiston, and M. F. Tachie, “Experimental and numerical investigation of turbulent cross-flow in a staggered tube bundle,” Int. J. Heat Fluid Flow, vol. 29, no. 2, pp. 387–414, 2008, doi: 10.1016/j.ijheatfluidflow.2007.10.001.

[2] J. M. Park, O. J. Kim, S. J. Kim, and Y. C. Shin, “Heat transfer characteristics of circular and elliptic cylinders in cross flow,” Adv. Mech. Eng., vol. 7, no. 11, pp. 1–8, 2015, doi: 10.1177/1687814015619553.

[3] S. Toolthaisong and N. Kasayapanand, “Effect of attack angles on air side thermal and pressure drop of the cross flow heat exchangers with staggered tube arrangement,” Energy Procedia, vol. 34, pp. 417–429, 2013, doi: 10.1016/j.egypro.2013.06.770.

[4] S. Liu and M. Sakr, “A comprehensive review on passive heat transfer enhancements in pipe exchangers,” Renew. Sustain. Energy Rev., vol. 19, pp. 64–81, 2013, doi: 10.1016/j.rser.2012.11.021.

[5] S. A. E. Sayed Ahmed, O. M. Mesalhy, and M. A. Abdelatief, “Flow and heat transfer enhancement in tube heat exchangers,” Heat Mass Transf. und Stoffuebertragung, vol. 51, no. 11, pp. 1607–1630, 2015, doi: 10.1007/s00231-015-1669-1.

[6] Y. Lai, M. Lu, and Q. Wang, “A large eddy simulation of plate-fin and tube heat exchangers with small diameter tubes,” Heat Transf. Eng., vol. 35, no. 11–12, pp. 1137–1143, 2014, doi: 10.1080/01457632.2013.863555.

[7] H. M. S. Bahaidarah, N. K. Anand, and H. C. Chen, “A numerical study of fluid flow and heat transfer over a bank of flat tubes,” Numer. Heat Transf. Part A Appl., vol. 48, no. 4, pp. 359–385, 2005, doi: 10.1080/10407780590957134.

[8] A. Žukauskas, “Heat Transfer from Tubes in Crossflow,” 1972, pp. 93–160.

[9] T. Kim, "Effect of longitudinal pitch on convective heat transfer in crossflow over in-line tube banks," Ann. Nucl. Energy, vol. 57, pp. 209–215, 2013, doi: 10.1016/j.anucene.2013.01.060.

[10] A. P. R. Bruce Roy Munson, T H Okiishi, Wade W Huebsch, Fundamentals of fluid mechanics. Hoboken, NJ: J. Wiley & Sons, 2013.

[11] M. Etli, G. Canbolat, O. Karahan, and M. Koru, "Numerical investigation of patient-specific thoracic aortic aneurysms and comparison with normal subject via computational fluid dynamics (CFD)," Med. Biol. Eng. Comput., vol. 59, no. 1, pp. 71–84, 2021, doi: 10.1007/s11517-020-02287-6.

[12] R. Rzehak and S. Kriebitzsch, "Multiphase CFD-simulation of bubbly pipe flow: A code comparison," Int. J. Multiph. Flow, vol. 68, pp. 135–152, 2015, doi: 10.1016/j.ijmultiphaseflow.2014.09.005.

[13] G. Canbolat, M. Etli, O. Karahan, M. Koru, and E. Korkmaz, "Investigation of vascular flow in a thoracic aorta in terms of flow models and blood rheology via Computational Fluid Dynamics (CFD)," J. Mech. Med. Biol., Aug. 2023, doi: 10.1142/S021951942350094X.

[14] F. Darıcık, G. Canbolat, and M. Koru, "Investigation of a fiber reinforced polymer composite tube by two way coupling fluid-structure interaction," Coupled Syst. Mech., vol. 11, no. 4, pp. 315–333, 2022, doi: 10.12989/csm.2022.11.4.315.

[15] G. Canbolat, A. Yıldızeli, H. A. Köse, and S. Çadırcı, "Düz Bir Plaka Üzerindeki Hidrodinamik ve Isıl Sınır Tabaka Akışının Sayısal Olarak İncelenmesi ve Geçiş Kontrolü," Int. J. Adv. Eng. Pure Sci., vol. 32, no. 4, pp. 390–397, 2020. doi: 10.7240/jeps.636786.

[16] M. Elkarii, R. Boukharfane, S. Benjelloun, and C. Bouallou, "A CFD-based surrogate model for predicting slurry pipe flow pressure drops," Part. Sci. Technol., vol. 41, no. 3, pp. 432–442, 2023. doi: 10.1080/02726351.2022.2110341.

[17] G. Canbolat, A. Yıldızeli, H. A. Köse, and S. Çadırcı, "Numerical Investigation of Transitional Flow over a Flat Plate under Constant Heat Fluxes," Acad. Perspect. Procedia, vol. 1, no. 1, pp. 187–195, Nov. 2018. doi: 10.33793/acperpro.01.01.39.

[18] J. Yunus A and M. Cimbala., Fluid Mechanics Fundamentals and Applications. Boston: HillHigher Education, 2006.

[19] ANSYS, "Ansys Fluent Theory Guide," PA 15317, 2013.

[20] W. H. GIEDT, "Effect of Turbulence Level of Incident Air Stream on Local Heat Transfer and Skin Friction on a Cylinder," J. Aeronaut. Sci., vol. 18, no. 11, pp. 725–730, Nov. 1951, doi: 10.2514/8.2092.

[21] S. T. McClain, B. K. Hodge, and J. P. Bons, "Predicting Skin Friction and Heat Transfer for Turbulent Flow Over Real Gas Turbine Surface Roughness Using the Discrete Element Method," J. Turbomach., vol. 126, no. 2, pp. 259–267, Apr. 2004, doi: 10.1115/1.1740779.

[22] A. Mirabdolah Lavasani, H. Bayat, and T. Maarefdoost, "Experimental study of convective heat transfer from in-line cam shaped tube bank in crossflow," Appl. Therm. Eng., vol. 65, no. 1–2, pp. 85–93, 2014, doi: 10.1016/j.applthermaleng.2013.12.078.

[23] C. K. Mangrulkar, A. S. Dhoble, S. G. Chakrabarty, and U. S. Wankhede, "Experimental and CFD prediction of heat transfer and friction factor characteristics in cross flow tube bank with integral splitter plate," Int. J.

Heat Mass Transf., vol. 104, pp. 964–978, 2017. doi: 10.1016/j.ijheatmasstransfer.2016.09.013.

[24] A. Sohankar, M. Khodadadi, and E. Rangraz, “Control of fluid flow and heat transfer around a square cylinder by uniform suction and blowing at low Reynolds numbers,” Comput. Fluids, vol. 109, pp. 155–167, 2015. doi: 10.1016/j.compfluid.2014.12.020.

[25] A. Lemouedda, A. Schmid, E. Franz, M. Breuer, and A. Delgado, “Numerical investigations for the optimization of serrated finned-tube heat exchangers,” Appl. Therm. Eng., vol. 31, no. 8–9, pp. 1393–1401, 2011. doi: 10.1016/j.applthermaleng.2010.12.035.

[26] Y. Wang, L. C. Wang, Z. M. Lin, Y. H. Yao, and L. B. Wang, “The condition requiring conjugate numerical method in study of heat transfer characteristics of tube bank fin heat exchanger,” Int. J. Heat Mass Transf., vol. 55, no. 9–10, pp. 2353–2364, 2012. doi: 10.1016/j.ijheatmasstransfer.2012.01.029.

[27] C. K. Mangrulkar, A. S. Dhoble, S. Chamoli, A. Gupta, and V. B. Gawande, “Recent advancement in heat transfer and fluid flow characteristics in cross flow heat exchangers,” Renew. Sustain. Energy Rev., vol. 113, no. November 2018, p. 109220, 2019. doi: 10.1016/j.rser.2019.06.027.

[28] L. Zhao, X. Gu, L. Gao, and Z. Yang, “Numerical study on airside thermal-hydraulic performance of rectangular finned elliptical tube heat exchanger with large row number in turbulent flow regime,” Int. J. Heat Mass Transf., vol. 114, pp. 1314–1330, 2017. doi: 10.1016/j.ijheatmasstransfer.2017.06.049.

[29] F. Duan, K. W. Song, H. R. Li, L. M. Chang, Y. H. Zhang, and L. B. Wang, “Numerical study of laminar flow and heat transfer characteristics in the fin side of the intermittent wavy finned flat tube heat exchanger,” Appl. Therm. Eng., vol. 103, pp. 112–127, 2016. doi: 10.1016/j.applthermaleng.2016.04.081.