HEAT TRANSFER ANALYSIS OF DOUBLE TUBE HEAT EXCHANGER WITH WAVY INNER TUBE

by

Ceren HASGUL and Gulsah CAKMAK*

Mechanical Engineering Department, Firat University, Elazıg, Turkey

Original scientific paper
https://doi.org/10.2298/TSCI210628266H

The effect of the design on the heat transfer is numerically investigated by using the wavy inner tube in a double-pipe heat exchanger. A wavy inner tube was used in the design to give a turbulent effect to the fluid along the inner tube of a double tube heat exchanger. In numerical study, ANSYS 12.0 Fluent code program was used, and the basic protection equations were solved for steady-state, 3-D and turbulent flow conditions. The study was examined at Reynolds numbers ranging from 2700-5300. The obtained results were compared with the experimental data performed under the same conditions. As a result of this comparison, after it was seen that the results obtained from the numerical analysis and the experimental results were compatible with each other, the wave number of the inner tube was increased and analyzed with the ANSYS fluent code program. When the data obtained as a result of the analyzes were evaluated, it was seen that the highest heat transfer was obtained from the 16 wave tube heat exchanger, which has the highest number of waves and under counter flow conditions. The increase in heat transfer increased by 270% compared to the straight tube.

Key words: *double tube heat exchanger, wavy inner tube, finite volume method*

Introduction

Energy is at the highest level among the most important needs of the world in terms of ranking. Increasing human population day by day, air pollution and the gradual depletion of energy resources together with many other factors cause this need to increase gradually. Therefore, the efficient and economical use of existing energy resources is even more important today. Many systems have been designed and researches have been carried out in order to meet the energy need and to use energy resources more efficiently and economically at the same time. All organizations around the world are turning to energy-saving energy-recycling systems. One of the systems that provide energy recycling is heat exchangers [1]. A large part of the energy produced today is used in heating and cooling systems in industry, commercial and home. In such places, heat transfer is provided by heat exchangers. The usage area of heat exchangers is very wide. The importance of heat exchangers is increasing day by day in terms of energy saving and the successful application of energy sources. In heat exchangers, energy savings and efficient use of energy are achieved by increasing the heat transfer [2]. In order to increase the performance of the heat exchangers, the heat transfer rate should be increased. The heat transfer rate can be increased by decreasing the temperature difference between the hot fluid and the cold fluid

* Corresponding author, e-mail: gulcakmak@firat.edu.tr

at the exit of the heat exchanger. There are two methods, active and passive, for increasing the heat convection coefficients. In industrial applications, passive methods are generally preferred. Active methods are costlier than passive methods.

The CFD can be called a flexible experimental laboratory located on the computer. In this way, it provides the opportunity to create virtual prototypes during analysis and to experiment on these prototypes. The most important factor in this study is to save long time and cost in the experimental study by using CFD.

Many researchers have worked on the analysis of heat exchangers using CFD method [3-7]. Dogan *et al.* [8] compared the thermal and hydraulic performances of circular and elliptical tube heat exchangers using CFD method. First, the obtained data of finned and circular pipe design were verified and compared with the performance results of finned and circular pipe design in the literature. Numerical studies were carried out with ANSYS FLUENT software in the range of Re = 750-2850 depending on the wingspan. As a result of numerical validation studies, all analyzes were solved using the SST k-ω turbulence model, which gives closer values to similar studies in the literature. As a result of this study, they showed that the flow entering between the fins in the circular tube model cooled the fastest, and the circular tube model created a higher pressure drop than the other models.

Abeykoon [9] aimed to investigate the design stages of a heat exchanger theoretically and analyzed and optimized the performance using CFD. A computational model of the same heat exchanger was implemented with ANSYS and then this model was expanded to six different models by changing the basic design parameters for optimization purposes. Eventually, these models were used to analyze the heat transfer behavior, mass-flow rates, pressure drops, flow rates, and eddies of shell and tube flows inside the heat exchanger. In general, the results are: it confirms that CFD modelling can be promising for the design and optimization of heat exchangers, making it possible to test numerous design options without producing physical prototypes. Maakoul *et al.* [10] numerically investigated the design and thermo-hydraulic performance of a double tube heat exchanger with helical fins on the ring side. A 3-D CFD model using FLUENT software was made to investigate the ring side fluid-flow, heat transfer coefficient and pressure drop for different configurations. The results obtained for a helically bent annular side showed higher heat transfer performance and higher pressure drop compared to simple twin tube heat exchangers.

In the study, it is aimed to numerically investigate the increase of heat transfer by giving the effect of turbulence to the fluid along the pipe in a double-pipe heat exchanger by using one of the passive methods of increasing the heat transfer. For this purpose, four different types of wavy inner tubes were mounted inside the tube in order to produce swirling flow in the heat exchanger. Investigation was made for wavy pipes with 10, 12, 14, and 16 wave numbers. The effect of the wavy pipes placed in the heat exchanger on the heat transfer was observed by increasing the wave numbers. The study was performed between 2700 and 5300 Reynolds numbers.

Computational fluid dynamics

The CFD method is basically based on three main equations (continuity, momentum, and energy equations). By solving these equations using boundary conditions suitable for the problems, the temperature distribution in the boundary-layer and the heat transfer coefficient are calculated from there:

– Continuity equation

The velocity components of the fluid in the r, θ, and z direction are in the form of u_r, u_θ, and u_z are expressed:

$$\frac{1}{r}\frac{\partial(ru_r)}{\partial r}+\frac{1}{r}\frac{\partial(u_\theta)}{\partial\theta}+\frac{\partial(u_z)}{\partial z} \tag{1}$$

– Energy equation

The energy equation for the temperature distribution in a cylindrical geometry is given in eq. (2). Where ρ is the fluid density, k – the transmission coefficient, T – the temperature, μ – the dynamic viscosity, c_p – the specific heat, F – the heat generation rate, and u_r, u_θ, and u_z are the fluid velocity components:

$$\rho c_p\left(\frac{\partial T}{\partial t}+u_r\frac{\partial T}{\partial_r}+\frac{u_\theta}{\partial\theta}+u_z\frac{\partial T}{\partial_z}\right)=k\left[\frac{1}{r}\frac{\partial}{\partial_r}\left(r\frac{\partial T}{\partial_r}\right)\frac{1}{r^2}\frac{\partial^2 T}{\partial\theta^2}+\frac{\partial^2 T}{\partial_{z^2}}\right]+\mu\Phi \tag{2}$$

– Momentum equations:

The r, θ, and z components of the conservation of momentum equations are given in eqs. (3)-(5).The r component of the incompressible Navier-Stokes equation:

$$\rho\left(\frac{\partial u_r}{\partial_t}+u_r\frac{\partial u_r}{\partial_r}+\frac{u_\theta}{r}\frac{\partial u_r}{\partial\theta}-\frac{u\theta^2}{r}+u_z\frac{\partial u_r}{\partial_z}\right)=$$

$$=-\frac{\partial P}{\partial_r}+\rho g_r+\mu\left[\frac{1}{r}\frac{\partial}{\partial_r}\left(r\frac{\partial u_r}{\partial_r}\right)-\frac{u_r}{r^2}-\frac{1}{r^2}\frac{\partial^2 u_r}{\partial\theta^2}-\frac{2}{r^2}\frac{\partial u_\theta}{\partial\theta}+\frac{\partial^2 u_r}{\partial z^2}\right] \tag{3}$$

The θ component of the incompressible Navier-Stokes equation:

$$\rho\left(\frac{\partial u_r}{\partial_t}+u_r\frac{\partial u_r}{\partial_r}+\frac{u_\theta}{r}\frac{\partial u_r}{\partial\theta}-\frac{u\theta^2}{r}+u_z\frac{\partial u_r}{\partial_z}\right)=$$

$$=-\frac{\partial P}{\partial\theta}+\rho g_\theta+\mu\left[\frac{1}{r}\frac{\partial}{\partial_r}\left(r\frac{\partial u_\theta}{\partial_r}\right)-\frac{u_\theta}{r^2}-\frac{1}{r^2}\frac{\partial^2 u_\theta}{\partial\theta^2}-\frac{2}{r^2}\frac{\partial u_r}{\partial\theta}+\frac{\partial^2 u_\theta}{\partial z^2}\right] \tag{4}$$

The z component of the incompressible Navier-Stokes equation:

$$\rho\left(\frac{\partial u_z}{\partial_t}+u_r\frac{\partial u_r}{\partial_r}+\frac{u_\theta}{r}\frac{\partial u_z}{\partial\theta}+u_z\frac{\partial u_z}{\partial_z}\right)=$$

$$=-\frac{\partial P}{\partial z}+\rho g_z+\mu\left[\frac{1}{r}\frac{\partial}{\partial_r}\left(r\frac{\partial u_z}{\partial_r}\right)-\frac{1}{r^2}\frac{\partial^2 u_z}{\partial\theta^2}+\frac{\partial^2 u_z}{\partial z^2}\right] \tag{5}$$

The geometry of double tube heat exchanger

The outer tube of the double-pipe heat exchanger used in the design has a length of L = 1100 mm, an inner diameter of 82 mm, and an outer diameter of 88 mm (De). The appearance of the 10 wave tube heat exchanger in a 3-D Solidworks environment is shown in fig. 1.

The length of the wavy inner tube is the same as the outer tube and is 1100 mm long. The cross-section of the wavy pipe is given in fig. 2 and the sample representation of the open plate state of the waves is given in fig. 3. The wave number was chosen according to the inner tube length. The wave number of the wavy pipes used in the design was changed to be 10, 12, 14, and 16, and the effect of this change on the heat transfer was investigated by numerical analysis method (ANSYS FLUENT code program).

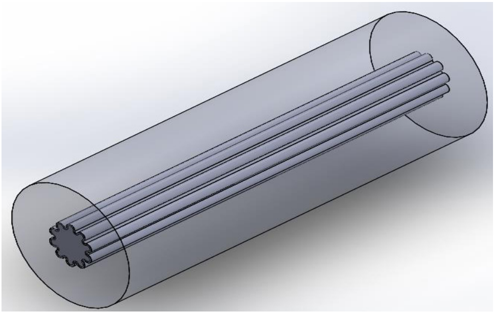

Figure 1. View of 10 wave tube heat exchanger in 3-D solidworks environment

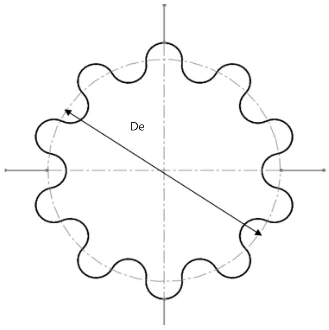

Figure 2. Cross-section view of wavy pipe

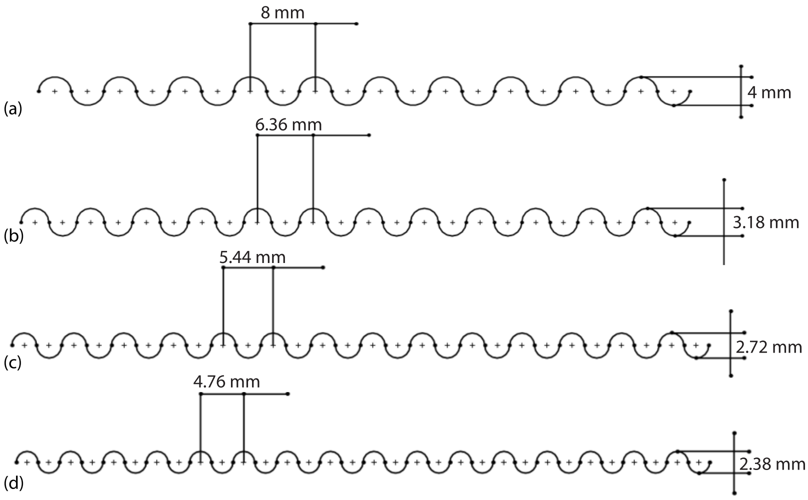

Figure 3. Flat plate representation of waves; (a) 10 waves, (b) 12 waves, (c) 14 waves, and (d) 16 waves

The material of the wavy pipes is 1 mm thick galvanized iron plate. While hot air is passed through the inner pipe, cold water flows between the corrugated pipe and the outer pipe. The outer tube of the heat exchanger is insulated to prevent heat losses.

In this study, the effect of the design on the heat transfer has been numerically investigated by using a *wavy inner tube* in a double-pipe heat exchanger. By increasing the wave numbers of the wavy pipes, separate analyzes were made for counter flow and parallel flow. The results obtained by the numerical study were compared with the results of the experimental study [11] and the accuracy of the analysis was checked. The numerical study was carried out in steady regime, turbulent flow conditions and in 3-D. Two fluids, air and water, were used in the heat exchanger. Water was used as the cold fluid and air was used as the hot fluid. In the study, the flow rate of cold water flowing through the outer pipe was kept constant and different flow rates of the hot fluid air passing through the inner pipe were used. The Reynolds number was studied in the range of 2700 < Re < 5300 for the hot fluid air.

The initial conditions for air and water fluids are:

– Air (inner fluid) inlet temperature: 45°C
– Water (outer fluid) inlet temperature: 20°C

– Air inlet velocity: 3.013877 m/s (variable value) (worked for six different air velocity values).
– Water inlet velocity:0.00259 m/s (fixed value).

Results and discussion

Comparison of numerical and experimental results

Counter and parallel flow analysis was performed for each wave tube and straight tube heat exchanger using the ANSYS FLUENT code program. In controlling the accuracy of the analysis results, comparison of the results of the experimental study and numerical study performed under the same conditions is important for the interpretation of the accuracy of the study. Our analyzes were compared with the experimental study [11] performed under the same conditions. Comparison was made for straight tube and 10 wave tube heat exchangers. The variation of the obtained Nusselt number with respect to the Reynolds number is shown in fig. 4.

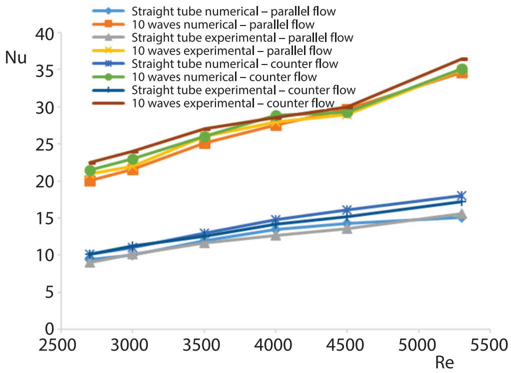

Figure 4. Variation of Nu-Re numbers of experimental and numerical study

In fig. 4, it is understood from the graphs that the experimental and numerical Nusselt number results are generally compatible with each other. It is seen that the difference between the experimental results and numerical results for the parallel flow and counter flow case in straight tube and 10 wave tube heat exchangers is 2.66%, 3.09%, 1.52%, and 2.80%, respectively. Budak *et al*. [12] analyzed, experimentally and numerically, the effect of turbulators with different geometries at the inner tube inlet in a concentric type heat exchanger. In this study, it is seen that there is a difference of 1.85%, 3.8%, 2.25%, and 2.1% for the four turbulators, respectively, in the agreement of the experimental results with the numerical results.

Evaluation of numerical results

Figures 5 and 6 show the variation of Nusselt numbers with Reynolds numbers in the analysis performed under parallel flow and counter flow conditions for straight tube, 10, 12, 14, and 16 wave tube heat exchangers. As can be seen from the figures, the Nusselt number increases

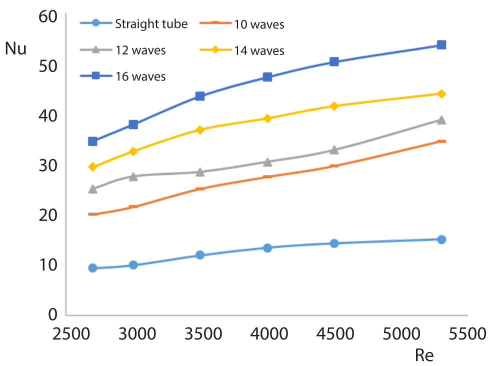

Figure 5. Variation of Nusselt number with Reynolds number in parallel flow heat exchangers

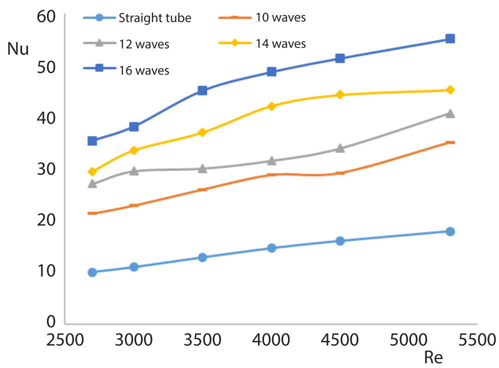

Figure 6. Variation of Nusselt number with Reynolds number in counterflow heat exchangers

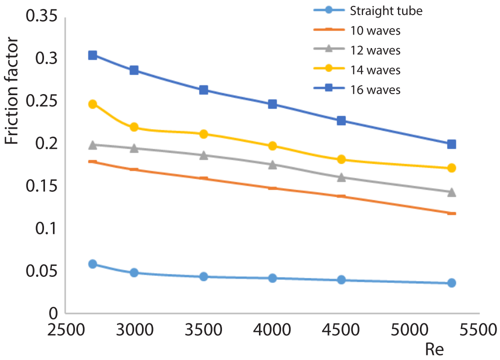

Figure 7. Variation of friction factor with Reynolds number

as the Reynolds number increases. The increase in Nusselt number in wave tubes is higher than in straight pipe. In addition, as the wave number increases, the Nusselt number increases.

The Nusselt number increased by an average of 113%, 147%, 200%, and 260% for parallel flow in 10, 12, 14, and 16 wave tube, respectively, compared to straight pipe. For counterflow, it increased by an average of 116%, 160%, 213%, and 270%, respectively.

In fig. 7, the graph of the variation of the friction factor with Reynolds numbers is given. The friction factor decreases as the Reynolds number increases. As the number of waves increases, the friction factor gradually increases. This is due to the fact that the wave pipes placed in the heat exchanger increase the pressure drop.

Temperature distribution results

In fig. 8, the temperature distribution results performed under counter flow conditions for straight tube, 10, 12, 14, and 16 wave tube heat exchangers with Re = 5300 are given.

As can be seen from the temperature distributions as a result of the analyses, the air temperature at the outlet of the inner pipe decreased gradually due to the turbulence effect in the flow with the increase in the number of waves used in the tube. When the examination is made between the straight tube and the wave tube, it is seen that the temperature drop in the wave tube is higher than the straight tube. With the wave tube used in the heat exchangers, the cooling took place faster and thus the wave tube increased the heat transfer from the air to the water. The temperature distributions in the inner tube change with increasing the number of waves. Wave tube added to the heat exchangers create a turbulent effect in the flow. An increase in heat transfer is observed with the turbulence effect in the flow. It is understood from the temperature distributions that the cooling in the straight tube is slower and the cooling obtained in the wave tube is faster. When the number of waves in the heat exchanger is increased, the cooling gradually increased as the air moved away from the inlet area, and thus an increase in heat transfer occurred.

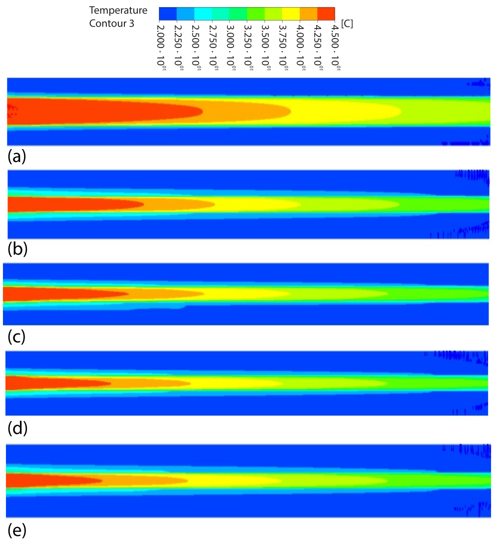

Figure 8. Temperature distributions for counterflow heat exchangers with Re = 5300; (a) straight tube, (b) 10 waves, (c) 12 waves, (d) 14 waves, and (e) 16 waves

The velocity vectors in the *r-q* plane of the straight tube and wave tube heat exchanger are shown in figs. 9 and 10. As can be seen, the straight tube heat exchanger is more uniformly distributed than the wave tube heat exchanger's appearance in the *r-q* plane. The eddy effect created in the flow by the wave tube is reflected in the appearance of the velocity vector in the

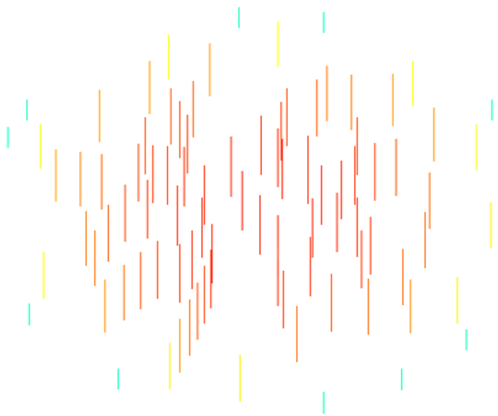

Figure 9. The view of velocity vectors in the *r-q* plane in a straight tube heat exchanger

Figure 10. The view of velocity vectors in the *r-q* plane in a wavy tube heat exchanger

r-q plane. Thus, the turbulence effect has increased compared to the straight tube and it has been observed that heat transfer has been improved.

Conclusion

Economic use of energy resources has become a part of life all over the world in our age. Therefore, the increase in heat transfer that will occur in heat exchangers, which is the most important device of the industry, is of great importance in terms of energy saving. In this study, wave tubes were placed in a double tube heat exchanger and the effect of these placed wave tubes on heat transfer was numerically investigated. Analysis was made for straight tube heat exchanger and 10, 12, 14, and 16 wave tube heat exchangers. The increase in heat transfer was observed by increasing the wave numbers of the wave pipes used in the design. First, the numerical study was compared with the experimental study. As a result of the comparison, a difference of 2.66% in parallel flow conditions and 3.09% in counter flow conditions was observed for the straight tube heat exchanger. It was observed that there was a difference of 1.52% in parallel flow conditions and 2.80% in counter flow conditions for 10 wave pipes. These differences are small and acceptable values. After this verification, the number of waves was increased and analyzes were made using the ANSYS FLUENT package program based on the finite volume method from CFD code programs in both parallel and counter flow conditions. When all analyzes were evaluated, it was observed that there was an increase in heat transfer in wave tube heat exchangers compared to straight tube. The highest Nusselt value was obtained from 16 wave tube heat exchangers performed under counter flow conditions. In the counter flow and 16 wave tube heat exchanger, the increase in heat transfer is 270% compared to the straight tube. In future studies, the optimum wave number in the wave pipes placed in the heat exchanger can be determined and its effects on heat transfer can be examined.

Acknowledgment

This study is derived from the master thesis of *Numerical investigation of heat transfer and flow characteristics in a heat exchanger with wavy inner pipe* by Ceren HASGUL in the department of Mechanical Engineering of Fırat University.

References

[1] Heyhat, M. M., *et al*., Performance Evaluation and Exergy Analysis of a Double Pipe Heat Exchanger Under Air Bubble Injection, *Applied Thermal Engineering*, *143* (2018), Oct., pp. 582-593
[2] Kumar, R., Chandra, P., Thermal Analysis of Compact Shell and Wire Coil-Inserted Helical Coil Tube Heat Exchanger, *International Journal Ambient Energy*, *43* (2019), 1, pp. 1068-1073
[3] Serageldin, A. A*., et al*., Thermo-Hydraulic Performance of the U-Tube Borehole Heat Exchanger with a Novel Oval Cross-Section: Numerical Approach, *Energy Conversion and Management*, *177* (2018), Dec., pp. 405-406
[4] Padmanabhan, S., *et al.*, Heat Transfer Analysis of Double Tube Heat Exchanger with Helical Inserts, *Materials Today: Proceedings*, *49* (2021), 9, pp. 3588-3595
[5] Xiong, O., *et al*., The 3-D Numerical Study of Conical and Fusiform Turbulators for Heat Transfer Improvement in a Double-Pipe Heat Exchanger, *International Journal of Heat and Mass Transfer*, *170* (2021), May, pp. 120-995
[6] Vivekanandan, M., *et al*., Experimental and CFD Investigation of Spiral Tube Heat Exchanger, *Materials Today: Proceedings*, *37* (2021), 2, pp. 3689-3696
[7] Altwieb, M., *et al*., A New 3-D CFD Model for Efficiency Optimisation of Fluid-Toair Multi-Fin Heat Exchanger, *Thermal Science and Engineering Progress*, *19* (2020), Oct., pp. 100-658
[8] Dogan, S., *et al*., Numerical Comparison of Thermal and Hydraulic Performances for Heat Exchangers Having Circular and Elliptic Cross-Section, *International Journal of Heat and Mass Transfer*, *145* (2019), 118731
[9] Abeykoon, C., Compact Heat Exchangers – Design and Optimization with CFD, *International Journal of Heat and Mass Transfer*, *146* (2020), 118766
[10] Maakoul, A., *et al*., Numerical Design and Investigation of Heat Transfer Enhancement and Performance for an Annulus with Continuous Helical Baffles In a Double-Pipe Heat Exchanger, *Energy Conversion and Managemen*t, *133* (2017), Feb., pp. 76-86
[11] Cakmak, G., *et al*., Experimental Investigation of Thermal Performance in a Concentric-Tube Heat Exchanger with Wavy Inner Pipe, *International Journal Thermophys*, *33* (2012), 6, pp. 1055-1067
[12] Budak, N., *et al*., Experimental and Numerical Investigation of the Effect of Turbulator on Heat Transfer in a Concentric-Type Heat Exchanger, *Experimental Heat Transfer*, *29* (2016), 3, pp. 322-336

Paper submitted: June 28, 2021
Paper revised: July 17, 2021
Paper accepted: July 23, 2021