Published by Avanti Publishers

Journal of Advanced Thermal Science Research

ISSN (online): 2409-5826

Investigation of Convective Heat Transfer and Friction Factor in Corrugated Channels with Different Inclination Angles Using Computational Fluid Dynamics

Erman Aslan[1,*] and Haydar Kepekçi[2]

[1]*Kocaeli University Engineering Faculty Mechanical Engineering Department, Izmit, Kocaeli, Turkey*

[2]*Nişantaşı University, Engineering Faculty, Mechatronics Engineering Department, Sarıyer, Istanbul, Turkey*

ARTICLE INFO

Article Type: Research Article

Keywords:
FVM
Nusselt number
Turbulence models
Corrugated channel
Convective heat transfer

Timeline:
Received: July 12, 2021
Accepted: August 29, 2021
Published: September 20, 2021

Citation: Aslan E, Kepekçi H. Investigation of Convective Heat Transfer and Friction Factor in Corrugated Channels with Different Inclination Angles Using Computational Fluid Dynamics. J Adv Therm Sci Res. 2021; 8: 21-29.

DOI: https://doi.org/10.15377/2409-5826.2021.08.3

ABSTRACT

Convective heat transfer and friction factor properties for periodic undulating channels are numerically investigated. The finite volume method (FVM) was used in the numerical study. Three different Reynolds averaged numerical simulation (RANS) based turbulence models, shear stress transport SST k-ω and transition SST model were used and compared with each other. Two different channels were used; the first geometry is a sharp corrugated channel with a 30° inclination angle, and the second geometry is a sharp corrugated channel with a 45° inclination angle. The Reynolds number varies in the range of 2000-7500. The Prandtl number was kept constant at 0.7. Nusselt number, friction factor, Reynolds number, and variations of the good factor were investigated. The effect of inclination angle and pitch are discussed.

*Corresponding Author
Email: erman.aslan@kocaeli.edu.tr
Tel: 0090 533 8151082

1. Introduction

With the increasing population and developing technology, the need for energy has increased even more. For this reason, scientists and researchers work for more efficient use of energy resources. Improving the performance of heat exchangers used in all areas of energy is one of the main objectives of engineers working in this field. One of the most important and common processes in engineering applications is heat exchange between two or more fluids at different temperatures. The devices in which this process is carried out are called heat exchangers. In practice, heat exchangers used in various places can be of different forms, capacities, sizes, and types according to their intended use [1].

In order to increase the performance of heat exchangers, it is aimed to increase the heat transfer rate. One of the steps taken for this purpose is to add corrugated channels to the heat exchangers. It has been observed that the geometric design revision made in this way has a positive effect on the heat transfer performance, especially in turbulent flows [2]. There are three different techniques to increase heat transfer in heat exchangers. These are the active flow control technique, passive flow control technique, and compound flow control technique. Flow vibration and surface vibration are examples of active flow control techniques. Using this method, extra power input to the system is required to increase heat transfer. Passive flow control technique can be done with geometric revision. Examples include machining rough surfaces or narrowing the canal diameter. One of the most common passive flow control techniques is corrugated ducts. Using this method, no extra power input is required to the system to increase heat transfer. The compound flow control technique is a combination of active and passive flow control methods. An example of this method is flow vibration with additives [3].

In this study, a sharp corrugated channel was used as a corrugated channel type. In the literature, there are many studies on sharp corrugated channels. Some of these studies were experimental, and some of them were done with CFD methods. Computational Fluid Dynamics (CFD) analysis is becoming increasingly common in both academic projects and the private sector. The main reason for this is that it is cheaper than experimental studies. It is also advantageous in terms of time-saving. There are models suitable for all kinds of flows in CFD codes. The most used turbulence models are RANS-based. RANS-based models do not have discrete flow features, so computational error rates are high. Large Vortex Simulation (LES), one of the CFD codes, is the best result in turbulent flow calculations. However, this model is not preferred in applications due to its long solution time [4]. Basic turbulence models based on RANS are known as k-ω and k-ε. The k-ω turbulence method does not seem to be an ideal model for applications in regions where the boundary layer is weak.

On the other hand, the k-ε turbulence model gives better results in the outer part of the boundary layer and in regions where it is weak. Therefore, the combination of both the k-ω turbulence method and the k-ε turbulence method has been sought. As a result of the studies, the ideal combination sought was determined as SST. For this reason, SST is the most preferred among RANS-based turbulence methods [5].

O'Brien and Sparrow [6] conducted an experimental study using sharp corrugated pipes to investigate the flow properties. As a result, they found that the EF value, which increases the heat transfer, varied between 2.14 and 2.71. Sparrow and Comb investigated the height of the corrugated walls and the effect of inlet conditions on the behavior of the flow. As a result of their studies, they found that the Nusselt number increased with the increase of the channel height [7]. Ali and Ramadhyani, in their experimental study using corrugated channels, found that the heat transfer and pressure drop increased as the Re number increased [8]. According to Sarangi *et al*., using the SST k-ω method, they analyzed the fin geometry they designed to improve the overall performance of the heat exchanger. They stated that they preferred this turbulence method because the RANS method gave the closest result to the experimental data [9]. Guardo *et al*. conducted a numerical study examining the effect of heat transfer from the wall to the fluid in beds where pipes of different diameters are placed. They used CFD analysis during this study. They calculated the pressure drop and thermal zones inside the bed [10]. Dağdekin and Öztop numerically investigated laminar flow and heat transfer in nested pipes. In this study, they used the SIMPLE algorithm. Velocity distribution, pressure distribution, and changes in local Nu number were obtained in the calculations made for 100, 500, and 1000 values of Re and 0.1, 0.7, 7, and 10 values of Pr number [11]. Zhang and Tian performed a numerical simulation of flow and heat transfer in cross-corrugated plate heat exchangers using a RANS-based k- ε RNG turbulence model. The effect of corrugation tilt angle on heat transfer and pressure drop was investigated. They observed that the numerical results matched well with the experimental results [12]. Eimsa-ard and Promvonge (2008) performed a numerical analysis of turbulent forced convection in a two-dimensional channel with transverse grooves using CFD software. In their numerical studies, four different turbulence models were used: Standard k- ε , RNG k- ε , k- ω, and SST. As a result of the calculations, they found that the RNG k-e and k- ε turbulence models were more compatible with the experimental results. They also concluded that corrugated channels increased 158% compared to the heat transfer provided by straight channels [13].

Talip and Hilo investigated convective heat transfer and flow field properties numerically and experimentally through a channel connected by a corrugated wall. They accepted the Reynolds number (Re) range during flow to be between 5000 and 20,000. They used the RNG k − ε turbulence model as the turbulence method. The effects of the wavelength and amplitude height of the corrugated wall on the friction factor and Nusselt number were investigated. The results showed that fluid flow significantly increases heat transfer with increased friction when the corrugated wall is used. The average increase in heat transfer rate and friction factor in the experiment was 40.7% and 46.2%, respectively [14]. Hoand *et al*. investigated the convective heat transfer and pressure loss properties of flow in a V-sharp corrugated channel using the LES numerical approach under vibrating inlet conditions. They aimed to examine the effect of pulsating parameters and Re for flow in the corrugated channels. As a result, they found that corrugated channels and pulsating flow significantly increased the overall thermal performance [15].

The study aims to determine the geometry that will increase the Nusselt number, which is one of the basic concepts of heat transfer. This study was carried out in order to explain in more detail the effect of channel height angles on the Nusselt number and friction coefficient of the widely used heat exchangers today. RANS-based turbulence models were used while performing numerical calculations. First, estimates are made for a corrugated channel with a 30° inclination angle. Different turbulence models are used in these calculations. The turbulence model that gave the closest result to the experimental data was determined. This model was used in the calculations for the corrugated channel with a 45° inclination angle. Finally, corrugated channels with a 30° inclination angle and corrugated channels with a 45° inclination angle were compared, and the effect of sharpening angle on heat transfer was investigated. In this study, k- ω , SST, and transitional SST were used as turbulence models.

2. Problem Definitions

In this study, sharp corrugated symmetrical channels with different inclination angles were used. The channel type used in the study is given in Figure **1** with its geometrical parameters. The list of parametric values used is shown in Table **1**. Inclination angle values of 30° and 45° were used. Flow development and temperature distributions along the channel geometry were investigated, and the effects of different inclination angles on heat transfer were investigated. In this numerical study, the Reynolds number (Re) varies between 2000 and 7500, and the Prandtl number (Pr) is taken as 0.7.

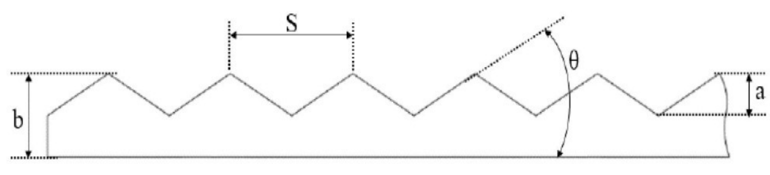

Figure 1: The channel including a sharp corrugation inclination with its geometric parameters.

Table 1: Geometric values of corrugated walls (mm).

Channel no.	θ (°)	S	a	b	$H_{min} = (b - a)x2$
1	30	17.32	5	2.5	5
2	45	10.231	5	2.5	5

Maximum channel height is given as,

$$H_{max} = 2b \quad (1)$$

The minimum channel height is given as,

$$H_{min} = 2(b - a) \quad (2)$$

3. Numerical Method

In this study, buoyancy forces and natural convection are neglected. In numerical solutions, turbulence models are solved in two dimensions. ANSYS Fluent was used in the analysis. The geometries used were drawn in the SolidWorks program, and meshing was done in Pointwise software. Three different models were used as turbulence modeling. These are k-ω model [16], SST model [17] and Transition SST Model [18]. The quadratic upwind procedure is used to discretize convective terms [19]. Low relaxation factors are used by default. (pressure: 0.3, momentum: 0.7, turbulence amounts: 0.8, energy: 1.0). While the convergence criterion in energy equations is 10-8, it is 10-6 in all other equations. The velocity and temperature profile at the entry point is taken as constant. It is assumed that the turbulence intensity at the inlet is 4%, and the mixing length equals 30% of the hydraulic diameter. These values were determined by experiment [20].

3.1. Computational Grids

The grid dependency calculation is required for CFD analysis. Mesh files are created at different grid numbers, and each is analyzed using the same parameters. After the analysis is completed, a lower grid number is based on mesh files that give similar results. No such procedure was performed in this study. This is because 192000 grid cells were decided as a result of the grid dependency process carried out in a similar survey [20]. To save time, the same grid number is adopted in this study. One of the most important things is that the y^+ value of the mesh used in this study was determined as 0.7, which is a fineness value of the mesh near the wall in all mesh used in this study. The small value of this value caused the sensitivity of the solution to increasing.

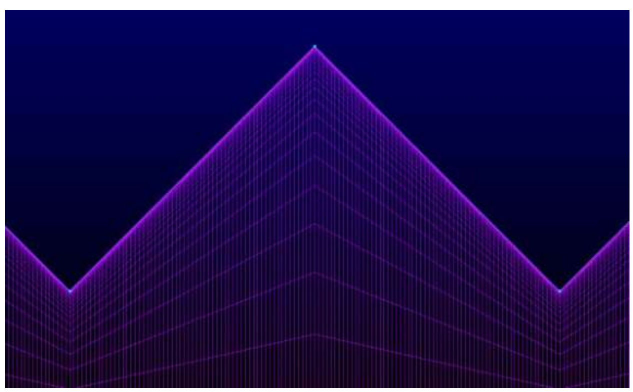

Figure 2: The detailed view of created grid cells.

3.2. Dimensionless Numbers

In this study, two different parameter calculations were made. There are Nusselt number and friction factors. The calculated parameters were compared with the experimental data. In the experiment, the flow became regular in the last three cycles of the channel. For this reason, the numerical values used in the calculations were taken from that region. The formulation of the Nusselt number is as follows:

$$\mathrm{Nu} = \frac{\mathrm{hD_h}}{\mathrm{k}} \quad (3)$$

where $D_h$is hydraulic diameter h is the cycle-average heat transfer coefficient, and k is the thermal conductivity. The hydraulic diameter is twice the average channel height. The average channel height is the average of the minimum channel height and the maximum channel height. The hydraulic diameter formula is given below:

$$D_h = H_{min} + H_{max} \tag{4}$$

The cycle averaged heat transfer coefficient (h) is calculated by integrating the local heat transfer coefficient for the last three cycles. The formula of the cycle averaged heat transfer coefficient is given below:

$$h = \frac{1}{3S}\int_0^S h_x dx \tag{5}$$

The h value given above is used to calculate the average Nusselt number. The last three cycles are used to calculate the friction factor. The friction factor formula is given below:

$$f = \frac{-\frac{dP}{dX}D_h}{\frac{1}{2}\rho V^2} \tag{6}$$

In the above equation $\rho$is density, V is velocity.

4. Results

In Figure **3**, the axial velocity distributions of the analyzes performed with the SST turbulence model in two corrugated channel geometry are channels with (a) 30° and (b) 45° inclination angles. The input velocity used in these figures is 6.79212 m/s. For this velocity value, the Reynolds number in both channels is 7380. The flow in the channel appears to have fully developed in recent cycles. The flow in the corrugated channel with an inclination angle of 45° appears at a higher velocity than the flow in the channel with an inclination angle of 30°. Also, near the walls, dense recirculation zones are observed in the channel with an angle of inclination of 45°; therefore, a duct with a 45° inclination angle produces more Nusselt numbers than a duct with a 30° inclination angle.

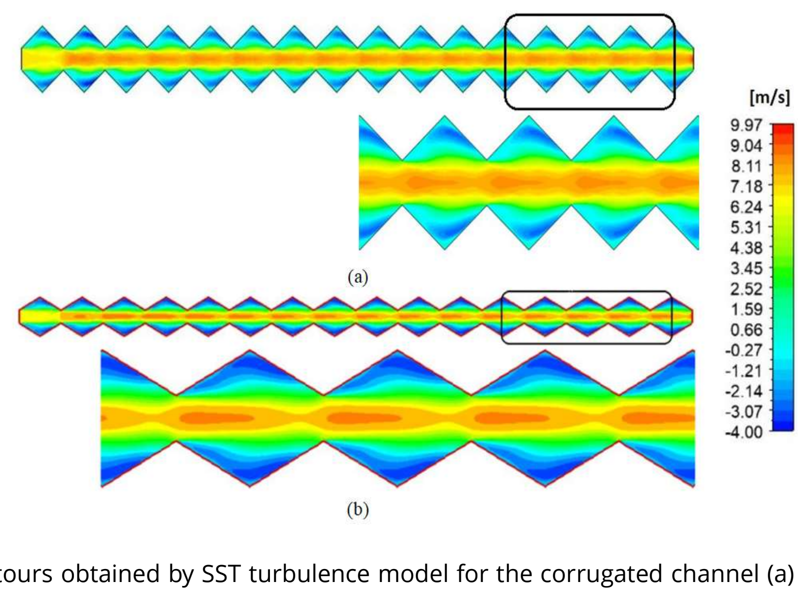

Figure 3: Axial velocity contours obtained by SST turbulence model for the corrugated channel (a) with a 30° inclination angle and (b) with a 45° inclination angle.

The temperature distributions obtained from the analysis with the SST turbulence model for two different channel geometries are given in Figure **4a** (channel with 30° inclination angle) and Figure **4b** (channel with 45° inclination angle). In the figures, it can be seen that the initiation and growth of the recirculation zones favor the mixing of the cold liquid from the core with the hot liquid near the boundary layer. This causes high-temperature variations at the wall edges of the corrugated channels. The wall temperature in the corrugated channel with an inclination angle of 45° appears to be higher than the wall temperature in the channel with an inclination angle of 30°.

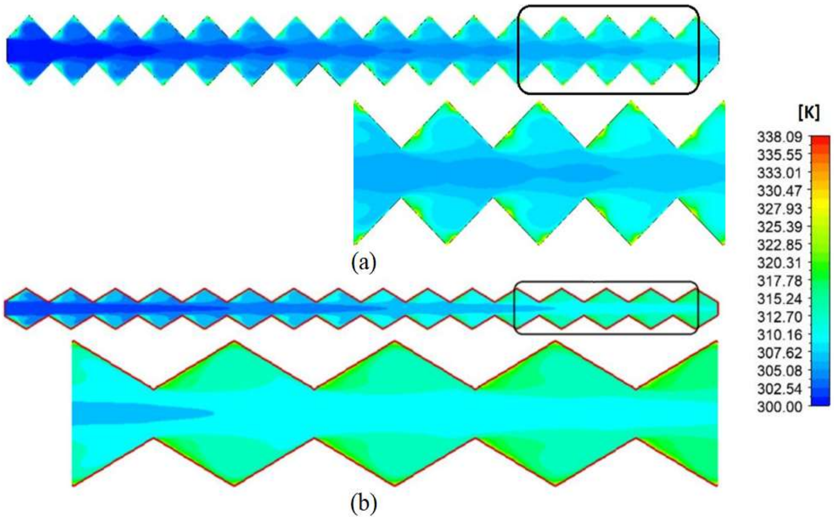

Figure 4: Axial temperature contours obtained by SST turbulence model for the corrugated channel (a) with a 30° inclination angle and (b) with a 45° inclination angle.

In Figure **5**, Nusselt numbers obtained from different turbulence models and Nusselt numbers obtained from the experimental study [20] for the corrugated channel with a 30° inclination angle are shown in the graph. The relationship between them can be interpreted from this graph. As the Reynolds number increased, so did the Nusselt number. In addition, it is seen that the analysis that gives the closest result to the experimental data is SST [21]. Therefore, it was decided to use the SST turbulence model to analyze the corrugated channel with a 45° inclination angle.

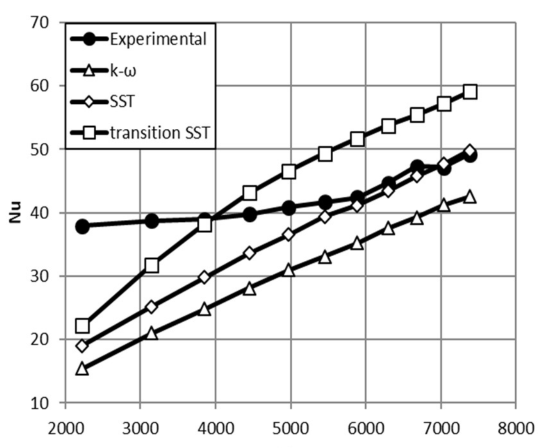

Figure 5: Comparison of Nusselt numbers from different turbulence methods.

In Figure **6**, the Nusselt numbers obtained as a result of the calculations using the SST turbulence method of the corrugated channel with 30° inclination angle and the corrugated channel with 45° inclination angle are shown in the graph. According to the curves in this graph, the Nusselt number is higher in corrugated channels with high inclination angles. It is seen that the Nusselt number increases as the inclination angle increases in the corrugated channels. It can be said that the higher the inclination angle, the higher the heat transfer in corrugated channels.

Figure **7**. shows the variation of the friction factor with the Reynolds number obtained as a result of the analysis made with different turbulence methods in a corrugated channel with a 30° inclination angle. When compared with the experimental data [20], it is seen that the most likely result is obtained from the SST turbulence method. It has also been determined that the analyses for high Reynolds number flow give more accurate results than low Reynolds number analyses.

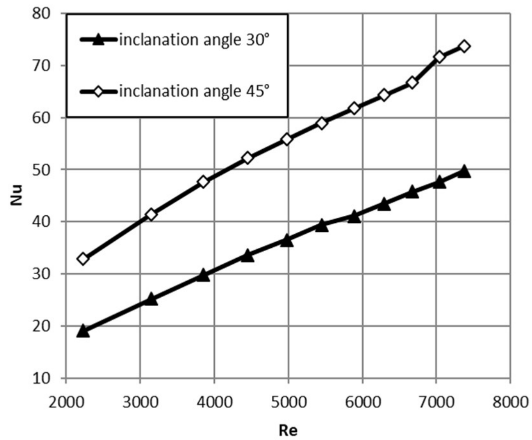

Figure 6: Comparison of Nusselt numbers from different inclination angle analysis.

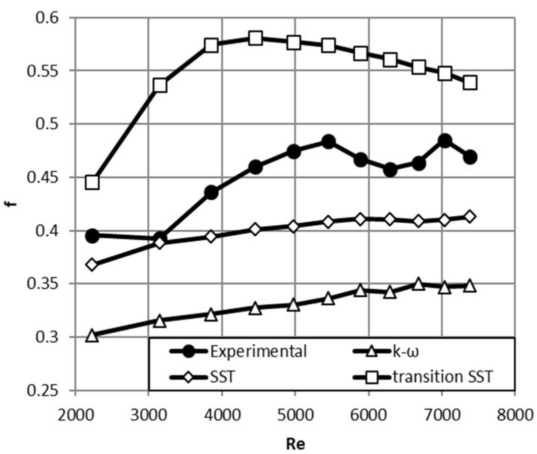

Figure 7: Comparison of friction factor from different turbulence methods

In Figure **8**, the friction factor values obtained from the calculations made using the SST turbulence method of the corrugated channel with 30° inclination angle and the corrugated channel with 45° inclination angle are shown on the graph. According to the curves in this graph, the friction factor is lower in corrugated channels with high inclination angles. It is seen that the friction factor decreases as the inclination angle increases in corrugated channels.

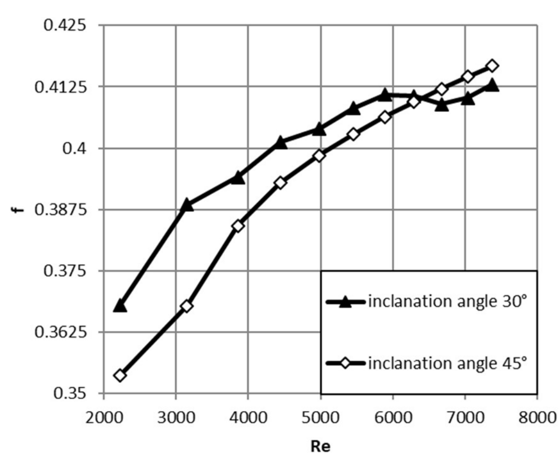

Figure 8: Comparison of friction factor from different inclination angle analysis.

5. Conclusions

In this study, the effect of different inclination angles on forced convection in corrugated channels was investigated. During this study, the first geometry, a corrugated channel with a 30° inclination angle, was created and analyzed with different RANS turbulence models. These models are selected as k-ω, SST, and transitional SST. All obtained results have been compared with experimental data. Among the preferred turbulence models, it was seen that the SST model gave the most accurate result. Then the second geometry, the 45° inclined grooved channel, has been drawn. The SST turbulence method has been chosen because it gives the closest result to the experimental data when analyzing the geometry prepared by CFD. Data from this analysis are compared with data from analysis performed with the same turbulence method of grooved channel geometry with a 30° inclination angle. As a result, it was seen that the Nusselt number increased with the increase of the inclination angle. This means that the heat transfer performance is increased. In addition, it has been determined that the friction factor decreases with the increase of the inclination angle in the corrugated channels.

References

[1] Bayram G, Şahin AŞ.Plakalı Isı Eşanjöründe Farklı Soğutkanlar Kullanılarak İki Farklı Soğutma Sisteminin Deneysel Analizi, J. Yekarum 2014; 2(2): 20-26.

[2] Kurtulmuş N, Şahin B. A review of hydrodynamics and heat transfer through corrugated channels, Int. Commun. Heat Mass Transfer 2019; 108, 104-307. https://doi.org/10.1016/j.icheatmasstransfer.2019.104307

[3] Wang G, Vanka SP. Convective heat transfer in periodic wavy passages, Int. J. Heat Mass Transfer 1995; 38(17): 3219-3230. https://doi.org/10.1016/0017-9310(95)00051-A

[4] Kepekci H, Zafer B, Guven HR. Aeroacostics Investigations of Unsteady 3D Airfoil For Different Turbulence Models Using Computational Fluid Dynamics Software, Fresenius Env. Bulletin 2019; 28(10): 7564-7573.

[5] Kepekci H, Zafer B, Güven HR. Aeroacoustics Investigations of Unsteady 3D Airfoil for Different Angle Using Computational Fluid Dynamics Software, Int. Scholarly Scientific Research Innovation 2018; 12(8): 813-820.

[6] O'Brien JE, Sparrow EM. Corrugated-duct heat transfer pressure drop and flow visualization, J. Heat Transfer 1982; 104(3): 410-416. https://doi.org/10.1115/1.3245108

[7] Sparrow EM, Comb JW.Effect of Interwall spacing and fluid flow inlet conditions on a corrugated-wall heat exchanger, Int. J. Heat Mass Transfer 1983; 26 (7): 993-1005. https://doi.org/10.1016/S0017-9310(83)80124-0

[8] Ali MM, Ramadhyani S. Experiments on convective heat transfer in corrugated channels, Exp. Heat Transfer 1992; 5(3): 175-193. https://doi.org/10.1080/08916159208946440

[9] Sarangi SK, Mishra DP, Ramachandran H, Anand N, Masih V, Brar LS. Analysis and optimization of the curved trapezoidal winglet geometry in a compact heat exchanger, Appl. Thermal Eng. 2021; 182, 1-14. https://doi.org/10.1016/j.applthermaleng.2020.116088

[10] Guardo A, Coussirat M, Larrayoz MA, Recasensa F, Egusquiza E. Influence of the turbulence model in CFD modeling of wall-to-fluid heat transfer in packed beds, Chem. Eng. Science 2005; 60(1): 1733-1742. https://doi.org/10.1016/j.ces.2004.10.034

[11] Dagtekin İ, Oztop HF. İç içe Yerleştirilmiş Borularda Laminer Akışta Isı Transferinin İncelenmesi, Turkish J. Eng. Env. Sci. 2000; 24, 193-202.

[12] Zhang G, Tian M. Simulation and Analysis of flow pattern in cross-corrugated plate heat exchangers, J. Hydr. 2006; 18(5): 547-551. https://doi.org/10.1016/S1001-6058(06)60133-9

[13] Eimsa-ard S, Promvonge P. Numerical study on heat transfer of turbulent channel flow over rare grooves, Int. Commun. Heat Mass Transfer 2008; 35(7): 844-852. https://doi.org/10.1016/j.icheatmasstransfer.2008.03.008

[14] Talib A.R.A, Hilo A.K. (2021) Fluid flow and heat transfer over corrugated backward facing step channel, Case Studies in Thermal Engineering 2021; 24(100): 1-15. https://doi.org/10.1016/j.csite.2021.100862

[15] Hoang V.Q, Hoang T.T., Dinh C.T., Plourde F. Large eddy simulation of the turbulence heat and mass transfer of pulsating flow in a V-sharp corrugated channel, International Journal of Heat and Mass Transfer 2021; 166(120): 1-12. https://doi.org/10.1016/j.ijheatmasstransfer.2020.120720

[16] Wilcox DC. Turbulence Modeling for CFD, DCW Industries, California, USA. 1998

[17] Menter FR. Two-equation eddy-viscosity turbulence models for engineering applications, AIAA Journal 1994; 32(8): 1598-1605. https://doi.org/10.2514/3.12149

[18] Menter FR, Langtry R, Volker S. Transition modeling for general-purpose CFD codes, Flow Turb. Comb. 2006; 77(1): 277-303. https://doi.org/10.1007/s10494-006-9047-1

[19] ANSYS-Fluent, Ansys-Fluent 14.5 User's Guide, Canonsburg, USA. 2012.

[20] Aslan E, Taymaz I, Islamoglu Y. Finite volume simulation for convective heat transfer in wavy channels, Heat Mass Transfer 2016; 52(3): 483-497. https://doi.org/10.1007/s00231-015-1571-x

[21] Aslan E, Taymaz I, Islamoglu Y, Engin M, Colpan I, Karabas G, Ozcelik G. Computational investigation of the velocity and temperature fields in corrugated heat exchanger channels using RANS based turbulence models with experimental validation, Prog. Compt. Fluid Dyn. 2018; 18(1):33-45. https://doi.org/10.1504/PCFD.2018.089526