EUROPEAN MECHANICAL SCIENCE
e-ISSN: 2587-1110
Research Paper

# Numerical Analysis of Flow and Heat Transfer for Semi Spheres Placed As Multiple Serials In a Channel

**Koray Karabulut[1*], Dogan Engin Alnak[2], Ferhat Koca[3], Coskun Ozalp[4]**

[1*]Cumhuriyet University, Engineering Faculty, Energy System Eng. Dept., Sivas, Turkey
[2,3]Cumhuriyet University, Technology Faculty, Automotive Eng. Dept., Sivas, Turkey
[4]Osmaniye Korkut Ata University, Engineering Faculty, Energy System Eng. Dept., Osmaniye, Turkey

## Abstract

Excessive heating of electronic equipments may cause both decreasing thermal performance of system and undesirable consequences such as increased thermal stresses and mechanical defects in the system. In the literature, there are many studies including applications of thermal analysis and technologies in order to make cooling more effective. In this study, flow and heat transfer were investigated numerically in the case of three dimensional, incompressible, fully developed and laminar flow for semi spheres placed as multiple serials in a channel. The continuity, Navier-Stokes and energy equations were solved numerically by using Ansys Fluent-17.0 software program. Air was taken as working fluid. Inlet temperature of the air and sphere surface temperatures were 300 K and 350 K, respectively. The effects of location of semi spheres as multiple serials, placement angle and positions of sphres according to each other on the pressure drop and heat transfer enhancement for case of different Re numbers (Re=100, 200, 400 ve 800) were researched. The obtained results showed that the positions of spheres had a great importance in the case of placement as multiple serials of semi spheres.

**Keywords:** Semi sphere, converging and diverging channels, heat transfer, laminar flow.

## 1. INTRODUCTION

Heat transfer has a great important role in the design of various heat exchangers, nuclear reactors, solar collectors, heaters, coolers, internal-combustion engines, combustion chambers, electric machines etc. It is required that the heat transfer is taken the highest values in order to design of compact devices. Nowadays, various type heat exchanger designs have been improved. Heat transfer coefficient belonging to surface geometry of these heat exchangers and flow properties with pressure loss coefficient should be determined. One of the heat transfer increment methods is continuous renewed of boundary layer. For this aim, shifted plate serials have been investigated in literature and it has been shown that the heat transfer coefficients can be increased.

One of the developed methods for enhancement of heat transfer surface area and the most important for improving of heat transfer coefficient by forming flow mixture section is communicating converging diverging channels. By means of these channels, enhancement in heat transfer can be carried out by increasing surface area in unit volume and especially by composing mixture section. Pressure drop also increases with increasing heat transfer at flow on these surfaces. Therefore, main goal for using of these surfaces is that when it is provided maximum enhancement in heat transfer, increment in pressure drop is obtained by minimizing of flow rate. The fundamental purpose should be become to determine optimum values which increase the system performance.

In this study, variations of Nusselt number, pressure loss coefficient, outlet temperature of fluid and amount of heat transfer according to three different multiple semi-sphere serials were numerically investigated at the communicating converging diverging channels composed of semi-sphere with 32 numbers and 4 arrays located in the channel as multiple serials. Continuity, Navier-Stokes and energy equations were solved together to calculate heat transfer at channels. These basic conservation equations describing the problem were solved with computer program of Ansys FLUENT-17.0 finite element method based. FLUENT provides to solve sets of nonlinear partial differential equations by using numerical methods. There are five main criterias to control validity of the obtained results with numerical methods. These are ensuring solution convergence, determination of the independence of the solution from iteration, providing

*Corresponding authour
Email: kkarabulut@cumhuriyet.edu.tr
Tel: 00 90 346 2191010/2270, Fax: 00 90 346 91110

European Mechanical Science, September 2018; 2(3): 76-82
doi: https://doi.org/10.26701/ems.377446
Received: January 11, 2018
Accepted: January 23, 2018

of conservation equations, determination that the solution is independent of cell structure and finally comparing results with experimental or accepted studies in literature

## 2. LITERATURE SURVEY

Converging-diverging channels as periodic has become subject of interest for many researchers. Sparrow and Prata [1] investigated flow and heat transfer with Reynolds number between 100 and 1000 at converging-diverging conic channels with unconnected as periodic both experimental and numerical. It was shown in study that pressure loss was little more according to flat tube and Nusselt number as depending on Prandtl number. It was seen that Nusselt number was less than flat tube for Pr<1, it was little more according to flat tube for Pr>1 in study. Patankar [2] studied flow and heat transfer under fully developed flow conditions in rectangular cross-section channels that were periodically variable cross section in flow direction. It was found that the uniform wall temperature was similar due to the fact that the velocity and temperature profiles changed periodically in the modules. It was determined that the temperature area in a given wall heat flux periodically repeated itself. With the concept of periodically fully developed flow and solution method, Patankar developed the first theorem of such heat transfer mechanisms. Here, he determined that the Nusselt number in the periodically fully developed flow was very high relative to the smooth channel flow and was a function of the Reynolds number. Kelkar and Patankar [3] investigated flow and heat transfer in channels with fins by studying constant fluid properties and considering flow as two-dimensional. In this study, two parallel plates were used and the flow was simulated by placing fins on the surfaces of the parallel plates. The calculations were examined by changing the variables such as Reynolds number, Prandtl number and fin conductivity in different geometric parameters. An increase in heat transfer was observed when the fins were placed on the plate surfaces as the fluid contacted more surfaces and moved with mixing. Also, it indicated that the increase in pressure drop helped to increase the heat transfer.

Wang and Bank [4] numerically examined flow and heat transfer in periodic sinusoidal channels. Their studies showed that heat transfer was increased in such channels but the increase in pressure loss was less than the increase in Nusselt number. Chunhua et al. [5] and Pankaj et al. [6] experimentally examined periodically interrelated axial vortex generators. They made comparison in their work for the appropriate fin types in the vortex generators. Zhu et al. [7], Chunhua et al. [8] calculated the effects of vortex generators on heat transfer by using a three-dimensional numerical method. They came to the conclusion that by using the vortex generators, the Nusselt number can be increased significantly. For the renewal of the boundary layer, general equations were extracted and analyzed by using analytical and numerical results for heat and mass transfer coefficients in shifted plate arrays channel and plate and it was shown that heat transfer coefficients can be increased (Sohankar [9], Patankar [10], Yılmaz [11]). Yılmaz and Ayhan [12] who were the first scientists theoretically and experimentally examined the heat transfer in the converging-diverging channels connected to each other. Here, it was stated that heat transfer can be increased significantly according to normal channels due to a good mixing as perpendicular to the flow in the channels and it was concluded that heat transfer can be increased further at the high Reynolds numbers.

It was indicated that heat transfer in communicating elements was be very high relative to unconnected elements by Fuji et al. [13] and the results of similar studies related to these were given. As a result of the experiments carried out, it was shown that this design was an effective method for increasing heat transfer. Kotcioğlu and Bölükbaşı [14] performed experiments by placing three different fins on a vertical channel with a rectangular section. The fins belonging to the test elements were shaped of converging and diverging fins according to flat surface, cylindrical and air flow direction by doing angle. It was deduced that the heat transfer coefficient improved in periodically placed fins as groups in especially converging and diverging fins due to periodic regeneration of the boundary layer.

## 3. MATERIALS AND METHODS

### 3.1 Basic Equations

Viscous, heat transfer involving, incompressible, three-dimensional and non-steady flow equations; conservation of mass Equation (1), Newton's second law conservation of momentum (Navier-Stokes) Equation (2) and conservation of energy, the first law of thermodynamics Equation (4) [10, 11,12 and 15].

Continuity equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\partial u_k)}{\partial x_k} = 0 \tag{1}$$

Navier-Stokes equation

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\partial u_i u_k)}{\partial x_k} = -\frac{\partial \rho}{\partial x_i} + \frac{1}{\mathrm{Re}}\frac{\partial \tau_{ij}}{\partial x_j} \tag{2}$$

Here, $\tau_{ij}$ is the viscous tensile tensor and is like Equation (3).

$$\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_j}\right) - \frac{2}{3}\mu\frac{\partial u_k}{\partial x_k}\delta_{ij} \tag{3}$$

This expression indicates the conservation of the three components of momentum. The expression on the left side shows the momentum change in the unit volume and the right side shows the viscous and pressure forces acting on fluid. In addition to these equations, the energy conservation equation is also used since the heat transfer in the flow is also examined in equation (4).


doi: https://doi.org/10.26701/ems.377446

$$c_v\frac{(\rho T)}{\partial t}+c_v\frac{(\partial u_k T)}{\partial x_k}$$
$$=\frac{\gamma}{\mathrm{Re}\,\mathrm{Pr}}\frac{\partial}{\partial x_k}\left(k\frac{\partial T}{\partial x_k}\right)-(\gamma-1)\rho\frac{\partial u_k}{\partial x_k}+\frac{\gamma-1}{\mathrm{Re}}\phi \tag{4}$$

The expressions on the right side of the equation show the change in fluid temperature due to conduction, pressure work and viscous heat, respectively, and $\phi$ represents the viscous loss function as seen from Eq. (5).

$$\phi=\tau_{ij}\frac{\partial u_i}{\partial x_j} \tag{5}$$

Friction coefficient (pressure loss coefficient) between parallel plates can be analytically calculated by Equation (6) [15].

$$f=\frac{96}{\mathrm{Re}} \tag{6}$$

### 3.2 Problem Geometry and Boundary Conditions

Three different models consisting of a rectangular channel and 32 semi-cylinders in various cases were used in order to numerically solve the flow and temperature fields of the semi-spheres in a converging-diverging channel as shown in Fig. 1. Here, the flow is toward the inside of the channel. Figure 2 shows the geometric boundary conditions. The bottom, top, and side parts are solid surfaces defined as walls when the left side of the channel is given as velocity input. The cross-sectional area on the right side of the duct is defined as the pressure outlet. There is no need to define the pressure at the exit because the velocity is defined in the input according to Patankar [10]. At the inlet of the jet, velocity, turbulence kinetic energy, turbulence kinetic energy loss rate and temperature are uniformly defined.

### 3.3 Convergence Criteria

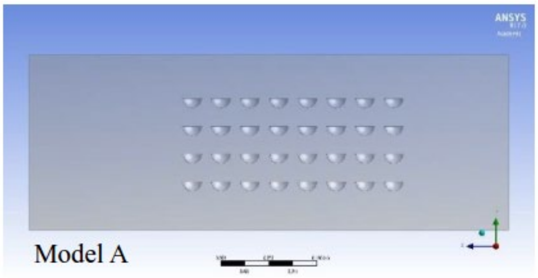
Convergence in a numerical study is very important in reaching the correct solution. In CFD, the distance between convergence mesh elements is either faster or diffuculty in convergent, depending on the dimensions of the control volume or the sizes of the element. Sometimes, it can lead to unwanted false results in analysis. When talking about convergence, you need to talk about residuals. Because, it is possible to see from the graph that the equations of the flow is converge or not. Convergence is the iterative analysis until the difference between the value of a variable at a point and the previous value and however the previous value of the variable is equal to a predetermined convergence criterion.

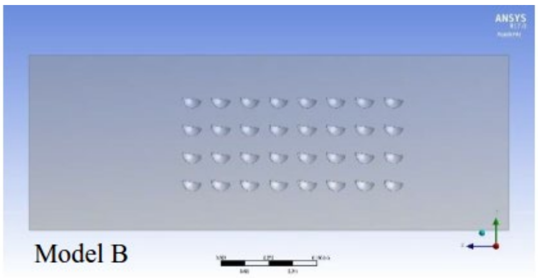
### 3.4 Working Area and Applied Mesh Structure

The working area consists of a rectangular channel. The surfaces of the semi-spheres are heated to 350 K. A fluid (air) with an average temperature of 300 K enters the inlet of the duct and leaves the work area from the outlet of the duct. The Reynolds numbers for inlet velocities are Re = 100, 200, 400 and 800, respectively.

Numerical solution of work-spaces having high-quality mesh has a great importance in getting the right results. Therefore, when CFD users who compose in meshes increase the number of elements used and they carry out analysis with this new mesh number, they have study opportunities with the most exact and enough mesh element. This is the independence of the analysis from mesh element number. In this study, the analyzes were carried out with mesh structures involving four different numbers of elements, and it was indicated that how the results changed depending on them. The results were shown by using 238524 (sparse mesh), 403547 (normal mesh), 752026 (dense mesh) and 925405 (very dense mesh), respectively.

x, y components of the velocity were dimensionless with the channel inlet velocity when it was provided to become dimensionless of the turbulence kinetic energy by dividing the square of the channel inlet velocity. Both velocity components and turbulence kinetic energy were completely independent of the number of elements and only varied a little in sparse form (Figure 3).



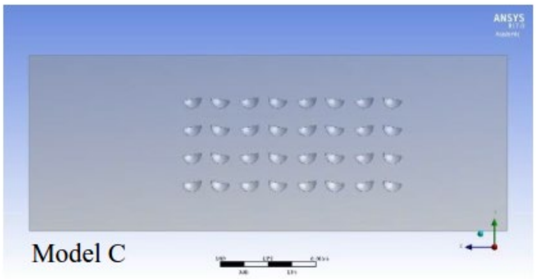




Figure 1: Models used in numerical calculations

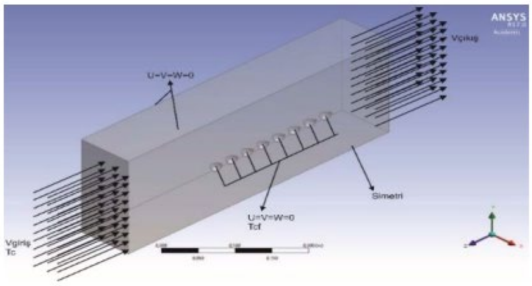



Figure 2: Boundary conditions used in numerical calculations


doi: https://doi.org/10.26701/ems.377446

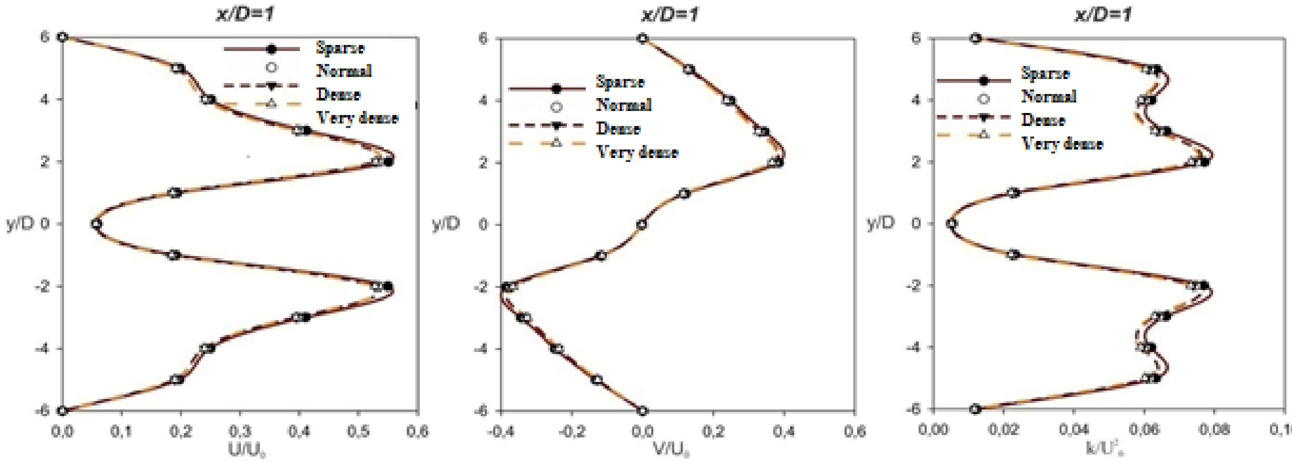



Figure 3: Variation of x, y velocity component and turbulence kinetic energy according to different mesh numbers respectively, Re = 100

However, this change was high between the sparse mesh structure and very little among normal, dense and very dense mesh structures. Especially, it was seen that the analysis with very dense mesh structure was not different from the results obtained using the dense mesh structure. Since the difference between the normal and the dense mesh was small, it was decided that all analyzes should be carried out with a dense mesh of about 752026 elements (Figure 4).

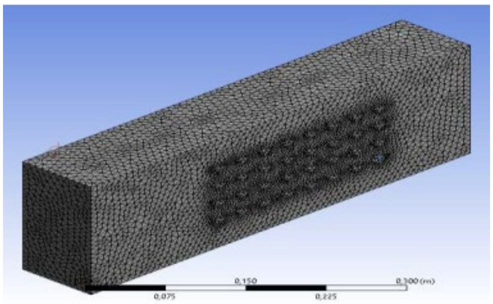


Figure 4: Mesh structure applied to model

## 4. FINDINGS AND EVALUATION

The results of the f-friction factor variation of the presented numerical study with the Re number for parallel plate are shown in Figure 5 by comparing analytical (Eq. 6) and numerical results obtained from Erdinc [15]. It is seen that the results are very compatible with each other and therefore it is considered that the numerical study is correct and acceptable.

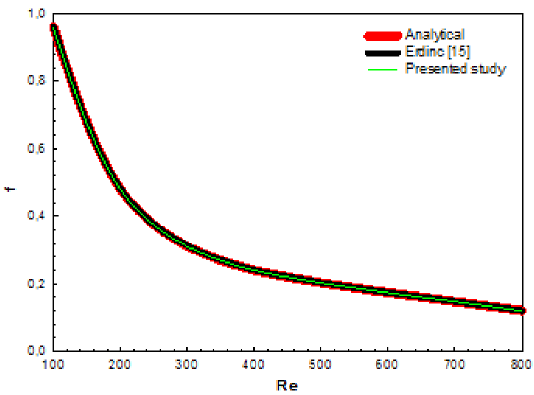



Figure 5: Comparing the results obtained for f-friction factor

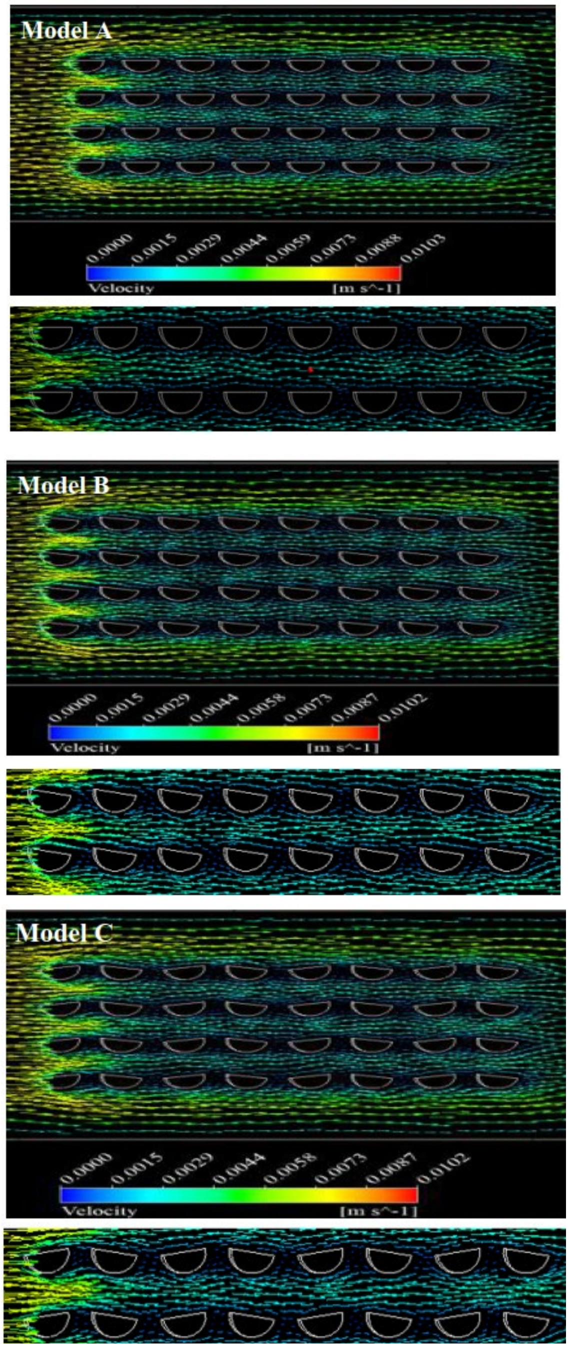



Figure 6: Presentation of velocity vectors for Models A, B and C, Re=100


doi: https://doi.org/10.26701/ems.377446

In Figure 6, the motion of the fluid between the semi-spheres in the channel at Re = 100 for Model A, B, and C is visualized using velocity vectors. In addition, semi-spheres at the second and third array are indicated by zooming in order to better show the flow between the semi-spheres by starting from the top array for also the three models. For model A, the flow around the semi-spheres is very small as the flow passes through the channels between the spheres. On the other hand, for model B, due to the fact that the semi-spheres are positioned horizontally at an angle of 20°, the fluid is easily directed between the spheres and provided to mix with the fluid coming from the bottom of the sphere (Fig. 6). In Model C, due to the position of the semi-spheres in the channel although the fluid is contact with the spheres, the motion of the fluid both between the spheres and mixing with each other is more difficult than in Model B. In addition, as can be seen from the velocity distribution vectors for all three models in Fig. 6 the lowest velocity values along the channel are reached between the spheres because the spheres obstruct the flow.

Figure 7 shows the variation of the pressure loss coefficient (f-friction factor) with the number of Re for three different converging-diverging channels (Models A, B, and C) formed by differently arranging a total of 32 semi-spheres in the case of 8 serials with 4 rows. The loss coefficients obtained for models B and C are very close to each other but more loss coefficients are obtained in Model C where the passage of fluid between spheres is more difficult. In addition, the friction factor in the laminar flow is a function of the Re number and the friction factor decreases with the increase in the number of Re.

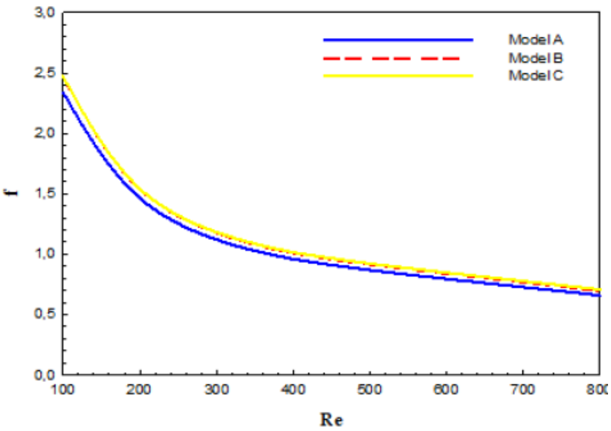



Figure 7: Variation of pressure loss coefficient with Reynolds number

Figure 8 shows the variation of the outlet temperature of the fluid from the channel with the Re number for Models A, B and C. Although the lowest pressure loss coefficient is obtained for Model A, the fluid temperature at the outlet is lower. Because the fluid is less circulated among the spheres than the other models. In Model B, the fluid contacts with more sphere surface because fluid passing is increased between the channels according to Models A and C. Thus the heat transfer surface area increases and at the same time it is reached to the higher fluid outlet temperature value for this model since existing fluid moves by mixing with the fluid coming from under the sphere.

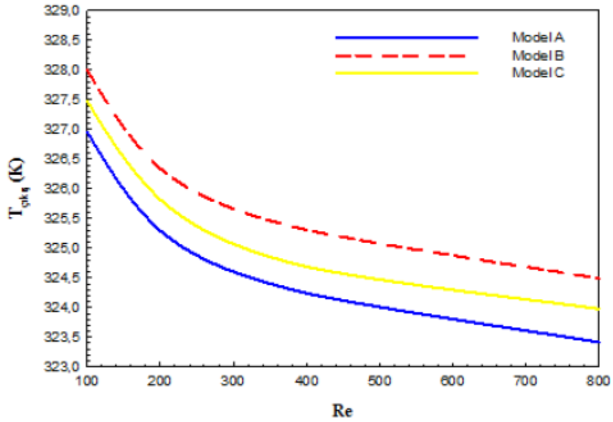



Figure 8: Variation of fluid outlet temperature with Reynolds number

Figure 9 shows variation of the mean Nusselt number (Nu) with the Re number for spheres. The temperature variation of the fluid results in a variation at the Nusselt number. However, as can be seen in Fig. 9 the Nu number is also increased because increasing of the fluid velocity (increase of Re) increases the heat transfer by convection. The highest Nu number value is also reached in Model B where the passing and mixing of the fluid are better.

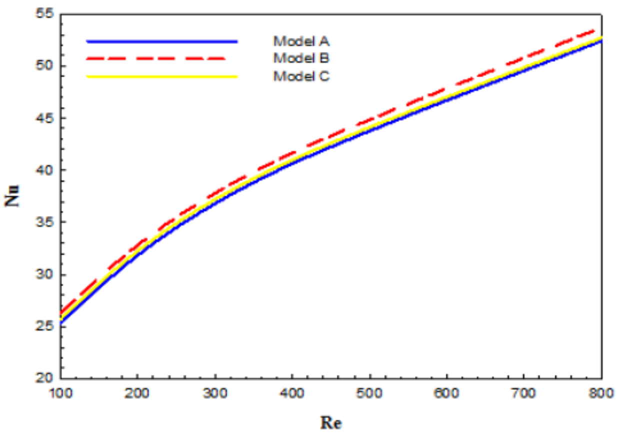



Figure 9: Variation of Nusselt number with Reynolds number

Figure 10 shows variation of heat transfer value to the fluid at the outlet of the channel according to Re number. When the increase in the Re increases the amount of heat transfer to the fluid, the highest heat transfer value is achieved for Model B.

Figures 11, 12 and 13 show the temperature and velocity contours in a converging-diverging channel consisting of 32 semi-spheres in a total with 8 semi-spheres in each row and with 4 rows for Models A, B and C at Re = 100, respectively. For all three models, because the spheres with the 8 serials make diffucult the fluid passing between the channels as can be seen from the temperature contours, according to other parts of the channel in especially in the second array after the spheres, they cause to obtain a hotter fluid region (Figures 11a, 12a and 13a). Thus, velocity values between the channels considerably reduce  due to the difficult fluid pas-


doi: https://doi.org/10.26701/ems.377446

sing according to other parts of the channel (Figs. 11b, 12b ve 13b). However, because the motion of the fluid between the spheres is more efficient than that of the Models A and C for Model B, the mixing of the fluid is improved and as a result of this, heat transfer also increases. For this reason, the temperature contour variation in Model B is higher than in the other two models especially at higher Re values. In addition, higher velocity values can be achieved due to improvement of fluid motion between the spheres.

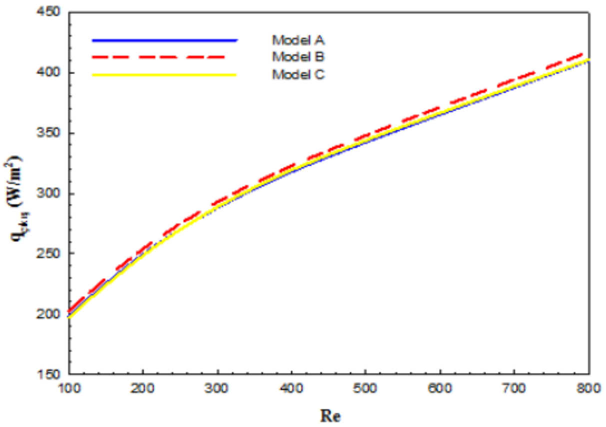



Figure 10: Variation of heat flux with Reynolds number in the outlet

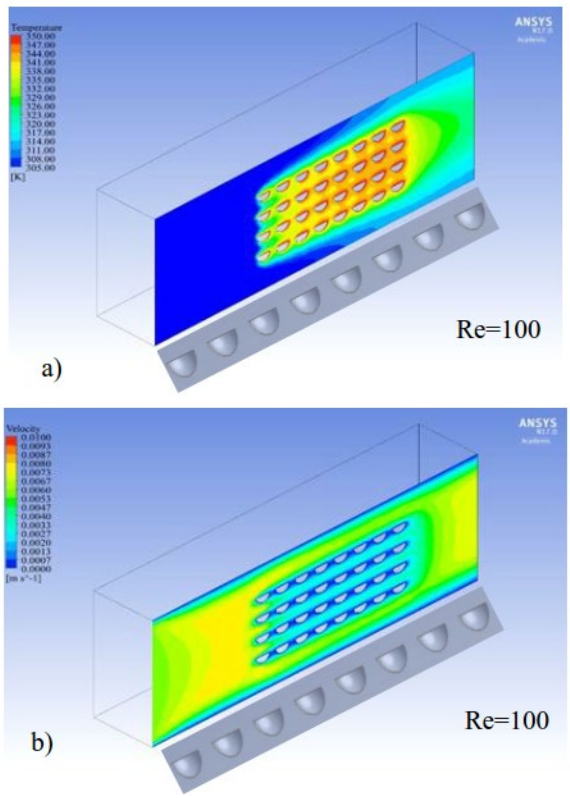



Figure 11: Temperature and velocity contours for Model A

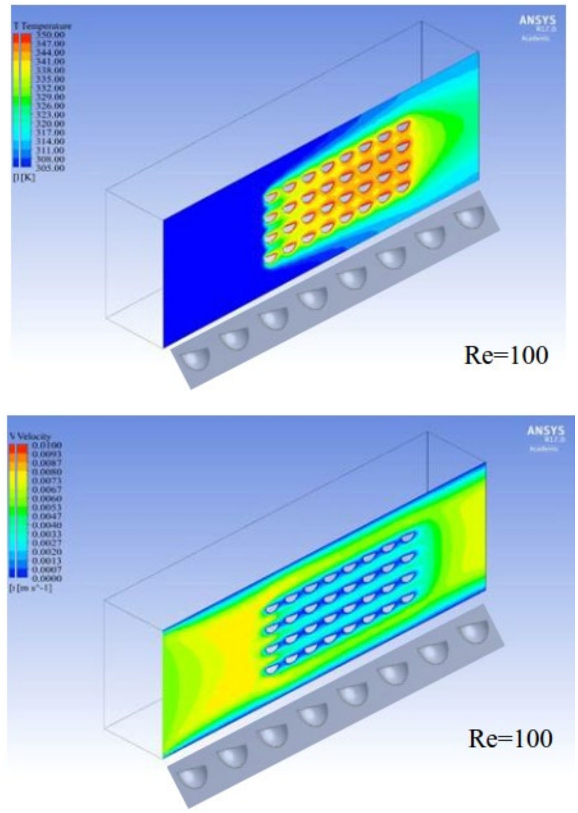



Figure 12: Temperature and velocity contours for Model B

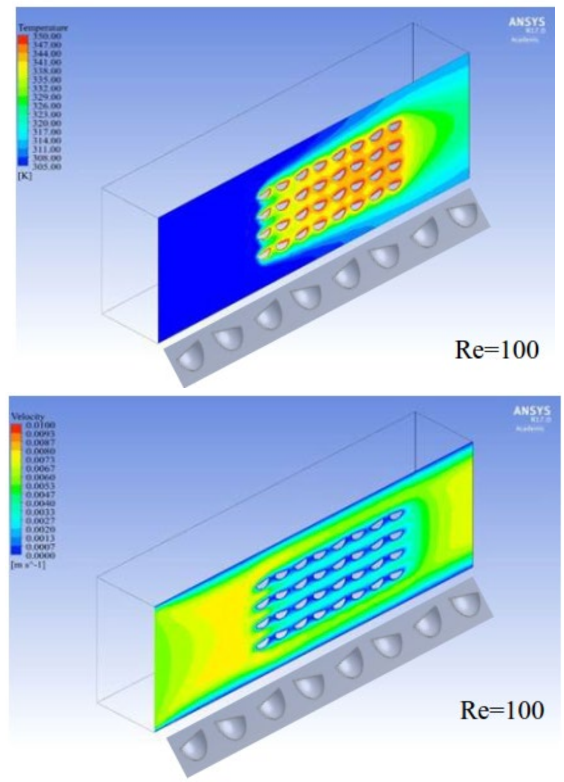



Figure 13: Temperature and velocity contours for Model C

## 5. RESULTS

In this study, the variation of the pressure loss coefficient, outlet temperature of the fluid from the channel, Nusselt number and the amount of heat transfer according to three different multi semi-sphere arrays were numerically investigated in the converging-diverging channels composed of


doi: https://doi.org/10.26701/ems.377446

32 semi-spheres with 4 rows placed as serials. Continuity, Navier-Stokes and energy equations for the calculation of the heat transfer in the channels were solved with the Ansys FLUENT-17.0 computer program based on the finite volume method.

For Model A, B and C, when the variation of the outlet temperature of the fluid with the Re number is analyzed, it is reached to the higher fluid outlet temperature value in Model B compared with the other models due to the fact that the motion of fluid between the channels is improved. However, the number of Nu also increases due to the increased heat transfer by convection because the fluid velocity (increase of Re) is increased. Also, the highest Nu number value is reached in Model B where the motion of fluid and the mixing between channels are better.

As can be seen from the velocity and temperature contours, as the result of the fluid movement between the channels composed of the hot spheres, it is attained a hotter fluid region especially towards the end of the channel. However, the flow velocity between the spheres decreases due to the flow obstruction. In addition, the heat transfer from the spheres is increasingly transferred to the flow in the channel by the continuous renewal of the boundary layer which is one of the reasons of the increasing the heat transfer of the converging-diverging channels. This case is achieved by directing the fluid between the converging and diverging channels formed by semi-spheres. The best example is especially in Model B, where the highest heat transfer is obtained according to other models.

Therefore, the best sphere placement position, sphere spacing and numbers should be chosen to ensure the best fluid circulation between the channels with low pressure loss in order to achieve the highest heat transfer rates.

## NOMENCLATURE

$c_p$ : specific heat, [$Jkg^{-1} K^{-1}$]
$f$ : friction factor
$k$ : thermal conductivity, [Wm-1 K-1]
**ρ** : density [$kgm^3$]
$\lambda_{ij}$ : viscous tensile tensor [$Nm^{-2}$]
$\phi$ : viscous loss function [$m^2s^{-3}$]
μ : dynamic viscosity [$kgs^{-1}m^{-1}$]
Nu : Nusselt number (=hd/k), [-]
$Pr$ : Prandtl number (=$k/\rho c_p$), [-]
$Re$ : Reynolds number (=$V_\infty d/v$), [-]
$T$ : temperature [K]
$t$ : time [s]
$u$ : velocity, [$ms^{-1}$]

## REFERENCES


[1] Sparrow, E.M. and Prata, A.T. (1983). Numerical solutions for laminar flow and heat transfer in a periodically converging-diverging tube with experimental confirmation. Numerical Heat Transfer, vol. 6, no. 4, p. 3219-3230.

[2] Patankar, S.V., Liu, C.H., Sparrow, E.M. (1977). The periodic thermally developed in ducts with streamwise periodic wall temperature or heat flux. International Journal of Heat and Mass Transfer. vol. 21, no. 5, p. 557-566.

[3] Kelkar, K.M. and Patankar, S.V. (1987). Numerical prediction of flow and heat transfer in a parallel plate channel with staggered fins. Journal of Heat Transfer. vol. 109, no. 1, p. 25-30.

[4] Wang, G. and Vanka, S.P. (1995). Convective heat transfer in periodic wavy passages. International Journal of Heat and Mass Transfer. vol. 38, no. 17, p. 3219-3230.

[5] Chunhua, M., Chengying, Q., Xiangfei, K., Jiangfeng, D. (2010). Experimental study of rectangular channel with modified rectangular longitudinal vortex generators. International Journal of Heat and Mass Transfer. vol. 53, no. 15-16, p. 3023-3029.

[6] Pankaj, S., Gautm, B., Subrata, S. (2014). Comparision of winglet-type vortex generators periodically deployed in a plate-fin heat exchanger-a synergy based analysis. International Journal of Heat and Mass Transfer. vol. 74, p. 292-305.

[7] Zhu, J.X., Fiebig, M., Mitra, N.K. (1995). Numerical ınvestigation of turbulent flows and heat transfer in a rib-roughed channel with longitudinal vortex generators. International Journal of Heat and Mass Transfer. vol. 38, no. 3, p. 495-501.

[8] Chunhua, M., Chengying, Q., Enyu, W., Liting, T., Yaju, Q. (2012). Numerical ınvestigation of turbulent flow and heat transfer in a channel with novel longitudinal vortex generators. International Journal of Heat and Mass Transfer. vol. 55, no. 23-24, p. 7268-7277.

[9] Sohankar, A. (2007). Heat transfer augmentation in a rectangular channel with vee-shaped vortex generator. International Journal of Heat and Fluid Flow. vol. 28, no. 2, p. 306-317.

[10] Patankar, S.V. and Prakash, C. (1981). Analysis of the effect of plate thickness on laminar flow and heat transfer in ınterrupted plate passages. International Journal of Heat and Mass Transfer. vol. 24, no. 11, p. 1801-1810.

[11] Yılmaz, T. (1982). Numerical solutions of navier-stokes equations for laminar flow in rows of plates in staggered arrangement. International Journal of Heat and Fluid Flow, vol. 3, no. 4, p. 201-206.

[12] Yılmaz, T. and Ayhan, T. (1983). Birbirleriyle bağlantılı daralan ve genişleyen kanallarda ısı transferi. 4. Ulusal Isı Bilimi ve Tekniği Kongresi, p.133-149.

[13] Fuji, M., Sensimo, Y., Yamanaka, G. (1988). Heat transfer and pressure drop of perforated surface heat excganger with pasage enlargement and contraction. International Journal of Heat and Mass Transfer. vol. 31, no. 1, p. 135-142.

[14] Kotcioğlu, I., Bölükbaşı, A. (2003). Düşey dikdörtgen kesitli bir kanalda farklı kanatçıklı yüzeylerde ısı transferinin incelenmesi. Dokuz Eylül Üniversitesi Mühendislik Fakültesi Fen ve Mühendislik Dergisi. cilt. 5, sayı 2, p. 89-102.

[15] Erdinc, M.T. (2014). Birbirleriyle bağlantılı daralan ve genişleyen kanallarda akış ve ısı geçişinin sayısal olarak incelenmesi. Osmaniye Korkut Ata Üniversitesi Fen Bilimleri Enstitüsü. Yüksek Lisans Tezi. p. 100.



doi: https://doi.org/10.26701/ems.377446