Journal Home page : http://dergipark.gov.tr/ejens

EJENS

European Journal of Engineering and Natural Sciences

DergiPark AKADEMİK

CODEN: EJENS

Investigation of Heat Transfer from Heated Square Patterned Surfaces in a Rectangular Channel with an Air Jet Impingement

Dogan Engin Alnak[1], Koray Karabulut[2]*, Ferhat Koca[1]

[1]*Sivas Cumhuriyet University, Department of Automotive Engineering, Technology Faculty, 58140, Sivas, Turkey.*

[2]*Sivas Cumhuriyet University, Department of Electric and Energ, Sivas Vocational High School, 58140, Sivas, Turkey.*

**Corresponding Author email: kkarabulut@cumhuriyet.edu.tr*

Abstract

Heat transfer is a very important precaution for proper design and safe operation of electronic packages and systems. Impinging jets are usually used to solve thermal problems of electronic components in industry due to providing a good heat transfer performance. In this study, cooling of copper plate with five square patterned surfaces inside a rectangular channel comprising of one open and three blocked sides was numerically investigated by using a single air jet flow. The numerical computations were performed by solving a steady, three-dimensional Navier-Stokes equation and an energy equation by using Ansys-Fluent 17.0 software program with k-ε turbulance model. Air was taken as working fluid with inlet temperature of 300 K. A constant heat flux with 1000 W/m^2 was applied to square patterned surfaces while the top and side surfaces were adiabatic. The study was carried out for different Reynolds numbers (Re) of 4000, 6000, 8000 and 10000 and different jet-to-plate distances (H/D$_h$) of 4, 6, 10 and 12. The numerical results agreed well with the numerical and experimental datas of study existed in literature. The results were presented as the variations of the mean Nu numbers and temperatures for each square patterned indentation surface. The temperature and velocity distributions of jet fluid flow and mean temperature and Nu values of whole five square patterned surfaces and value of air jet outlet temperature were also researched for different Re numbers and H/D$_h$ ratios. It was seen that increasing the Re number increases the Nusselt number for all cases. Average Nusselt number increases of 59.28% from Re=4000 to Re=10000 for H/D$_h$=4. However, Nu number was less sensitive to H/D$_h$ ratio in the range of H/D$_h$=4-12. Average Nusselt number decreases of 9.11% from H/D$_h$=4 to H/D$_h$=12 for Re=6000. The highest average Nusselt number was attained for Re=10000 and H/D$_h$ =6.

Key words

Impingement Air Jet, Patterned Surface, Square Channel, Numerical Analysis

1. INTRODUCTION

Rapid improvement of technology leads electronic devices to have both more compact and higher processing power. The reliability of the electronic parts of a system is a primarily factor in the overall reliability of the system. Electronic components depend on transition of the electric current to implement their duties and they become potential regions for excessive heating, since the current flow through a resistance brings along heat generation. Unless properly designed and controlled, high rates of heat generation cause to high operating temperatures for electronic device, which endangers its reliability and safety. Besides, the high thermal stresses in the solder joints of the electronic equipments mounted on circuit boards resulting from temperature variations are major reasons of defects. Therefore, thermal control has become increasingly important in the design and operation of the electronic devices. For this purpose, it is needed to develop new cooling techniques instead of conventional technology. Impinging liquid and gaseous jets are used widely because of their easy application and high heat transfer coefficient. Jet impingement is employed for heating, cooling and drying in cases where coefficients of high heat transfer are aimed. Thus, it is considered that jet impingement can be used for cooling of the electronic components generating high heat and having high performance. Because high heat producing electronics evolve, it becomes proof that using air cooling alone will not provide adequate performance. Jet impingement has the ability to take away large amounts of heat from these environments with high heat flux. In the one of the earliest investigations into jet impingement flow was carried out a review of literature that included several examples [1]. In an another review study, Carlomagno and Ianiro [2] did a detailed work of the effects of Reynolds number and jet to plate distance on the heat transfer and flow physics of the impinging jets. Argus et al. [3] researched jet flow and heat transfer by using only one jet, numerically. Popovac and Hanjalic [4] investigated cooling of a heated cubic plate using one impingement jet. Yang and Hwang [5] exhibited numerical simulations of flow properties of a turbulent slot jet impinging on convex surface with a semi-cylindrical. Mushatat [6] performed a numerical study in order to study heat transfer and turbulent flow characteristics of impinging slot jets. Zuckerman and Lior [7] numerically analyzed the effect of nozzle type on initial turbulance, pressure drop, shearing force of free jet and jet velocity profile. They also described various experimental and computational techniques from other authors. Before submitting your final paper, check that the format conforms to this template. Specifically, check the appearance of the title and author block, the appearance of section headings, document margins, column width and other features. Please make sure that the use of other languages in figures and tables is avoided.

Different from the literature, it was mainly carried out numerical investigation of heat transfer from heated square patterned surfaces, inside a rectangular channel having three closed and one open side by using an air jet with single slot in the present work. The slot jet was mounted next to closed side. Air jet geometry and channel were designed of similar to an implementation of cooling of electronic components inside various devices. The numerical computations were performed by solving a steady, three-dimensional Navier-Stokes equation and an energy equation by using Ansys-Fluent 17.0 software program with k-ε turbulance model. Air was used as working fluid with inlet temperature of 300 K. It was exerted to a constant heat flux ($\ddot{q}$) with 1000 W/m^2 on square patterned surfaces. The numerical results agreed well with the numerical and experimental datas of study existed in literature. The results were presented as the variations of the mean Nu numbers and temperatures for each square patterned indentation surface. The temperature and velocity distributions of jet fluid flow and mean temperature and Nu values of whole five square patterned surfaces and values of air jet outlet temperature were also investigated for different Re numbers and H/D_h ratios.

2. NUMERICAL METHOD

An easy way to comply with the symposium paper formatting requirements is to use this document as a template and simply type your text into it.

The aim of finite volume method is to break down the domain of the problem into a finite number of elements to be solved to find a solution for each of these sections and then by uniting these solutions to find a general solution to the problem. This method uses a technique which is based on the control volume for transforming heat flow equations into algebraic equations which can be solved numerically. In other words, this technique is based on the principle of taking the heat flow equations integration in each control volume. This integration result provides equations which characterize each control volume which occurs. For preparing the most appropriate grid model, a fine grid should be formed in regions where the change in variables such as velocity, pressure and temperature is bigger. Therefore, the finest grid was especially used for the channel surfaces with the indentation and in other zones a sparser grid was preferred. Convergence of the computations was stopped for the continuity and the momentum equations when residues were less than 10^{-6} and for the energy equation when residues were less than 10^{-7}. A grid structure which consisted of tetrahedral was used for simulation. Also, a standard k-ε turbulance model was performed for the selected model with square patterned surfaces in the numerical investigations.

The flow and heat transfer through the geometry are governed by the partial differential equation derived from the laws of conservation of mass, momentum and energy with steady state conditions without a body force, which are expressed as follows [8].

Continuity equation

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{1}$$

Momentum equation

x momentum equation

$$\rho\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{dx}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right) \tag{2.1}$$

y momentum equation

$$\rho\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right) \tag{2.2}$$

z momentum equation

$$\rho\left(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right) \tag{2.3}$$

Energy equation

$$u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}=\left(\frac{k}{\rho c_p}\right)\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}\right) \tag{3}$$

In the equations, ρ is density, μ dynamic viscosity, p pressure, k thermal conductivity, T temperature, c_p specific heat and u, v, w are velocities of the x, y and z direction, respectively.

In the used standard k-ε turbulance model, the turbulence kinetic energy and its rate of dissipation and the viscous dissipation term are used.

Steady flow turbulence kinetic energy equation

$$\frac{\partial(\rho u k')}{\partial x}+\frac{\partial(\rho v k')}{\partial y}+\frac{\partial(\rho w k')}{\partial z}=\frac{\partial}{\partial x}\left(\frac{\mu_t}{\sigma_k}\frac{\partial k'}{\partial x}\right)+\frac{\partial}{\partial y}\left(\frac{\mu_t}{\sigma_k}\frac{\partial k'}{\partial y}\right)+\frac{\partial}{\partial z}\left(\frac{\mu_t}{\sigma_k}\frac{\partial k'}{\partial z}\right)+\mu_t\phi-\rho\varepsilon \tag{4}$$

Turbulent viscosity

$$\mu_t=C_\mu.\rho.\frac{k'^2}{\varepsilon} \tag{5}$$

Turbulence kinetic energy

$$k'=\frac{1}{2}\left(\overline{u'^2}+\overline{v'^2}+\overline{w'^2}\right) \tag{6}$$

Viscous dissipation term

$$\phi=2\mu\left[\left(\frac{\partial u}{\partial x}\right)^2+\left(\frac{\partial v}{\partial y}\right)^2\right]+\mu\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right)^2 \tag{7}$$

Turbulence kinetic energy disappearance equation

$$\frac{\partial(\rho u\varepsilon)}{\partial x}+\frac{\partial(\rho v\varepsilon)}{\partial y}+\frac{\partial(\rho w\varepsilon)}{\partial z}=\frac{\partial}{\partial x}\left(\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial\varepsilon}{\partial x}\right)+\frac{\partial}{\partial y}\left(\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial\varepsilon}{\partial y}\right)+\frac{\partial}{\partial z}\left(\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial\varepsilon}{\partial z}\right)+C_{1\varepsilon}\mu_t\frac{\varepsilon}{k'}\phi-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k'} \tag{8}$$

The model constants C_μ, $C_{1\varepsilon}$, $C_{2\varepsilon}$, σ_k and σ_ε have typically default values for used standard k-ε turbulance model [8]. The values of these constants have been arrived at by numerous iterations of data fitting for a wide range of turbulent flows. These are as follows;

C_μ =0.09, $C_{1\varepsilon}$ =1.44, $C_{2\varepsilon}$ =1.92, σ_k=1 and σ_ε=1.3.

Reynolds number is calculated by the equation given below;

$$Re=\frac{V_\infty.D_h}{\nu} \tag{9}$$

Here, D_h is the hydraulic diameter of the jet inlet

$$D_h = \frac{4A_c}{P} = \frac{4(a.W)}{2(a+W)} \tag{10}$$

where A_c is the cross-section area of the jet inlet, P is the perimeter of the jet inlet.

The Nusselt number is evaluated as the conductive heat transfer rate of the fluid over the solid boundary equal to the convective heat rate as:

$$-k\left(\frac{dT}{dn}\right)_{surface} = h\left(T_\infty - T_s\right) \text{ and } Nu = \frac{h.D_h}{k} \tag{11}$$

Where h is the local heat transfer coefficient on the surface, n is the vertical direction to isotherm and the local Nusselt number is obtained as above.

3. GEOMETRIC MODEL

Perspective view of the channel having sizes was shown in Fig. 1. The boundary conditions can also be seen in this figure. The jet nozzle hydraulic diameter (D_h) was kept at 9.9 mm, the length (L) and width (W) of the channel were taken as 200 mm and 50 mm, respectively. 5.5x50 mm rectangular nozzle with an inlet velocity ranging between 6.23 and 15.58 m/s was used in the study. Uniform velocity profile for jet inlet was existed in the inlet of the rectangular nozzle. The channel height was determined at different measurements as $4xD_h$, $6xD_h$, $10xD_h$ and $12xD_h$. While space between two indentations was taken as D_h, indentation width and height were $2xD_h$ and D_h, respectively. There are five surfaces with square indentation and the constant heat flux ($\ddot{q}$) with 1000 W/m^2 was applied to only these surfaces for all simulations when the top and side surfaces were adiabatic. However, the Reynolds numbers (Re) of the jet ranged from 4000 to 10000.

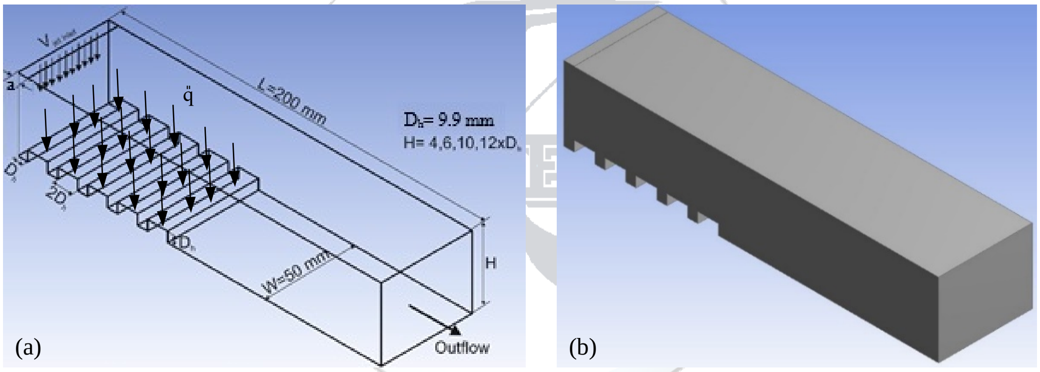


Figure. 1. Perspective view of the channel (a) Domain with boundary conditions and sizes (b) CFD simulation domain

The study was conducted under the following assumptions:

i) The flow field was assumed to be three-dimensional, steady-state and turbulent for the channel;
ii) Calculations were carried out for incompressible fluid;
iii) Air was used as working fluid for the cooling of indentation surface;
iv) Constant heat flux of 1000 W/m^2 was applied to indentation surfaces;
v) Thermal properties of the fluid were constant;
vi) There was no heat generation for both the jet fluid and the solid surfaces.

4. RESULTS AND DISCUSSION

Figure 2 exhibits a comparison of the effects of Re number with 6000 and 8000 on the Nu number as experimental and numerical for jet-to-plate distance (H/D_h) value of 6 between Kilic et al. [9] and the present study numerically. The comparison was performed for smooth copper plate surface that was used in [9] at Re numbers of 6000 and 8000. While the deviation of the Nu number between the experimental results of Kilic et al. [9] and the present numerical study is 3.99% at Re=6000, it is found as 1% for numerical results at the impingement region. However, difference between the experimental and numerical results increases for Re=8000 at the impingement region due to higher turbulance intensity. When the deviation of numerical results with experimental is 9.15% due to higher turbulance intensity in this region, the difference is 1.02% for the numerical results when compared the present study with Kilic et al. [9]. Therefore, it can be said that the results of the present study are well comparable with experimental and numerical study results of the Kilic et al. [9] and the numerical study is reasonable and appropriate.

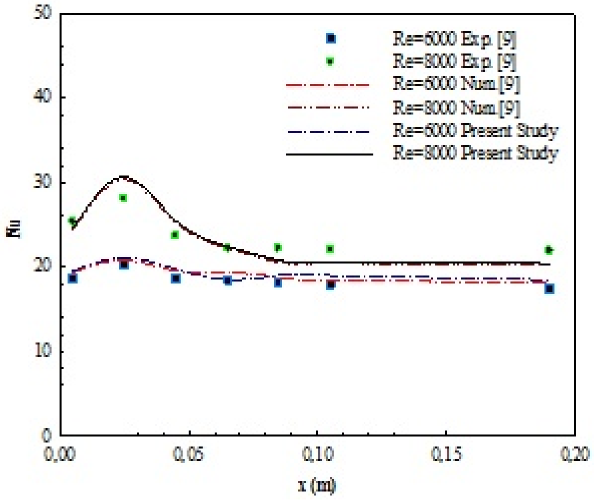


Figure 2. Comparison of the present results and those of Kilic et al. [9]

To determine the effect of the grid size on the mean Nu number and outlet temperature of jet air ($T_{out\text{-}jet}$), a grid independence test (as shown in Table 1) was carried out for H/Dh=4 and Re=4000. The test showed that 1714584 grids on a duct cross-section are adequate (< 0.01% difference compared with 1954741 grids). Mesh structure of the channel having square patterned surfaces was also shown in Fig. 3 with zoomed image in order to clearly see the mesh shape of the square pattern.

Table 1. Grid independence test results for Nu_m and $T_{out\text{-}jet}$

Mesh Numbers	**Nu_m**	**$T_{out\text{-}jet}$ (K)**
1525412	9.7189	324.648
1714584	9.7298	324.663
1954741	9.7297	324.665

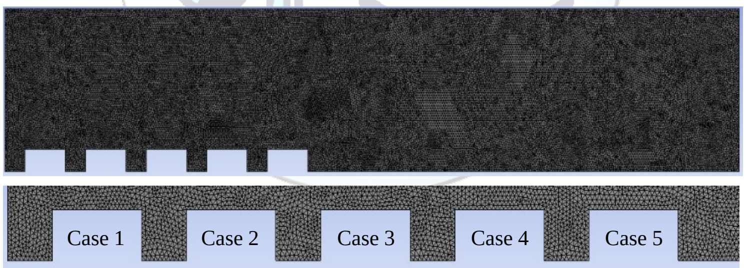


Figure 3. Mesh structure of the channel with circle patterned surface used in calculations

The effects of Reynolds number and H/D_h ratios on the Nusselt number variation for different location cases of rectangular patterned surfaces are shown in Fig. 4. Cases in the graphs indicates the location order of the rectangular surfaces beginning from the left side of the channel. Enhancement in Re number increases the Nu number on the surfaces of the rectangular patterned. Recirculations come into existence at the bottom of the left wall for all Reynolds numbers. These recirculations cause a change of direction of the jet flow. Therefore, values of the highest local Nu number were determined for the case 2 for all Re numbers. However, because the effect of recirculation increases on the location place of the highest local Nu number with increase of H/D_h ratio, the local Nu number values change according to ratio of H/D_h at the different cases from 1 to 5. Average Nusselt number increases of 59.28% from Re=4000 to Re=10000 for H/D_h=4. Besides, Nu number was less sensitive to H/D_h ratio in the range of H/D_h=4-12. Average Nusselt number decreases of 9.11 % from H/D_h=4 to H/D_h=12 for Re=6000 because of a decline of turbulance intensity. Also, the highest average Nusselt number was attained for Re=10000 and H/D_h=6.

Fig. 5 exhibits the mean surface temperatures of the square patterned for different jet-to-plate distances (H/D_h) of 4, 6, 10 and 12 and Re numbers. Increasing of the Re number provides to reducing of the mean surface temperatures by causing to enhance the turbulence intensity and so heat transfer. However, H/D_h increment from 4 to 12 sharply affects the surface temperature of the case 1 depending on directed of the jet flow thanks to recirculations. While the lowest surface temperature value was obtained for the case of 2, the case 5 was found to have the highest surface temperature due to losing the jet effect and reduction of flow velocity toward exit of the channel for all Re numbers (Figs. 5a-d). A surface temperature increase of 1.83% was observed from the case 2 to case 5 for H/D_h=4 and Re=10000.

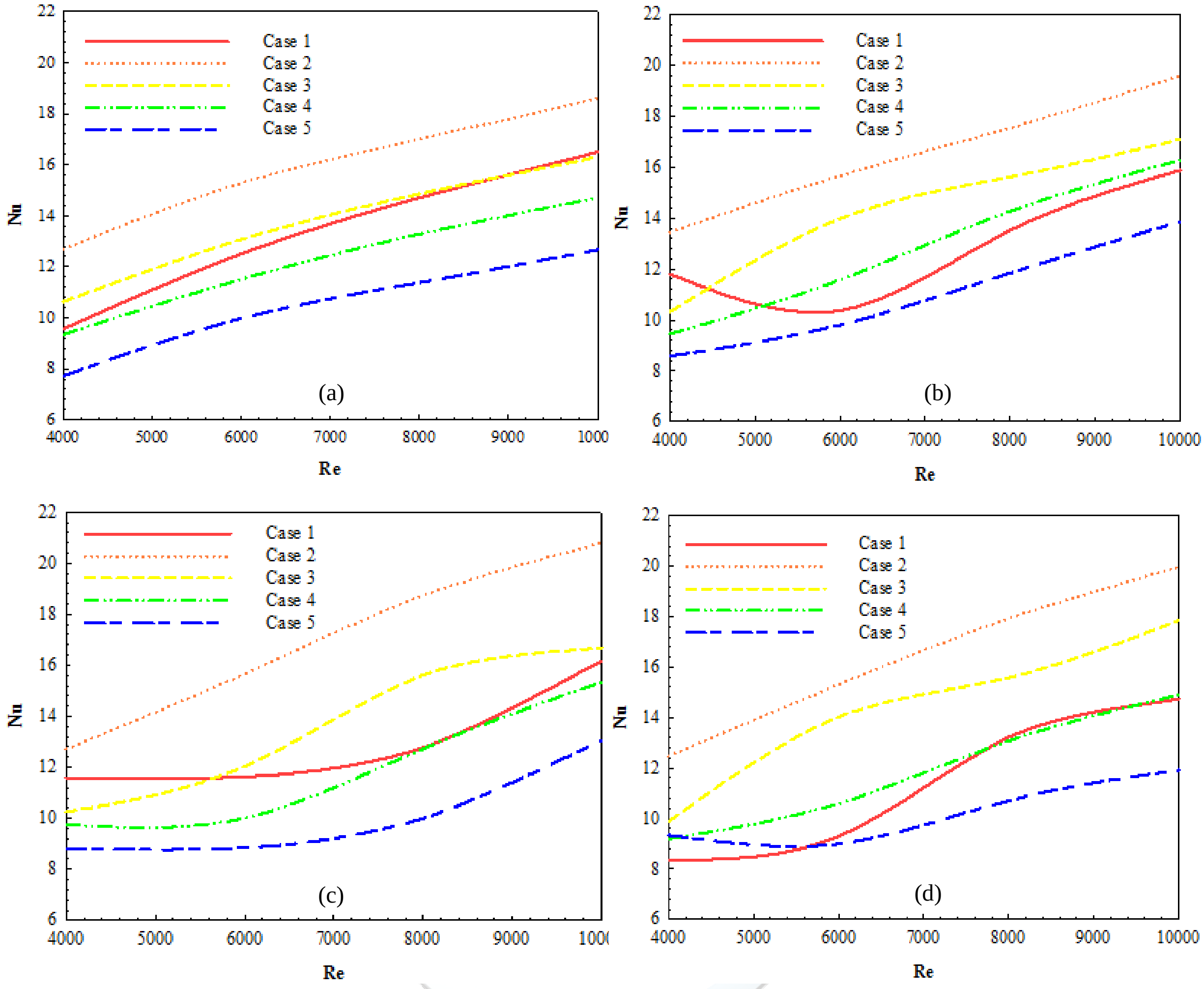


Figure. 4. Variations of mean Nu number versus different Re number for different H/D_h and cases (a) H/D_h=4 (b) H/D_h=6 (c) H/D_h=10 and (d) H/D_h=12

It can be seen in Figs. 6A and B, recirculations happen at the bottom of the left wall for analyzed Reynolds numbers. However, these occurred recirculations affect main jet flow and cause to change location of maximum heat transfer point. Besides, recirculation sizes decreases with increasing Re number from 4000 to 10000 due to prevention of expansion of recirculations by increasing jet flow velocity. When the velocity of jet flow is high at the impingement region, it reduces toward channel outlet. Therefore, surface temperatures of the square indentations increase. Because of the separation of the jet flow from the last square surface, recirculations occur. One can see that recirculations increase with increasing jet-to-plate distance H/D_h. Thus, thickness of thermal boundary layer enhances with enhancing recirculations, which causes to increase the temperature. Increasing channel height causes to decrease length of wall jet region. The reason for this is reducing of flow velocity at a longer channel height. Decreasing flow velocity, on the surface of copper indentation plate, causes an increase in temperature.

Mean Nu numbers and surface temperatures of square indentations and air jet temperature at the outlet of the channel for different Re numbers and H/D_h ratios are given in Table 2. When Nu_m increases with increasing of Re, it decreases with increasing H/D_h ratio from 4 to 12 due to reducing air jet velocity in the impingement region. However, mean surface temperatures of the square indentations enhance with enhancing jet-to-plate distance H/D_h but they decrease with increasing Re number. Besides, outlet temperature of the air jet decreases with enhancing Re number and H/D_h ratio.

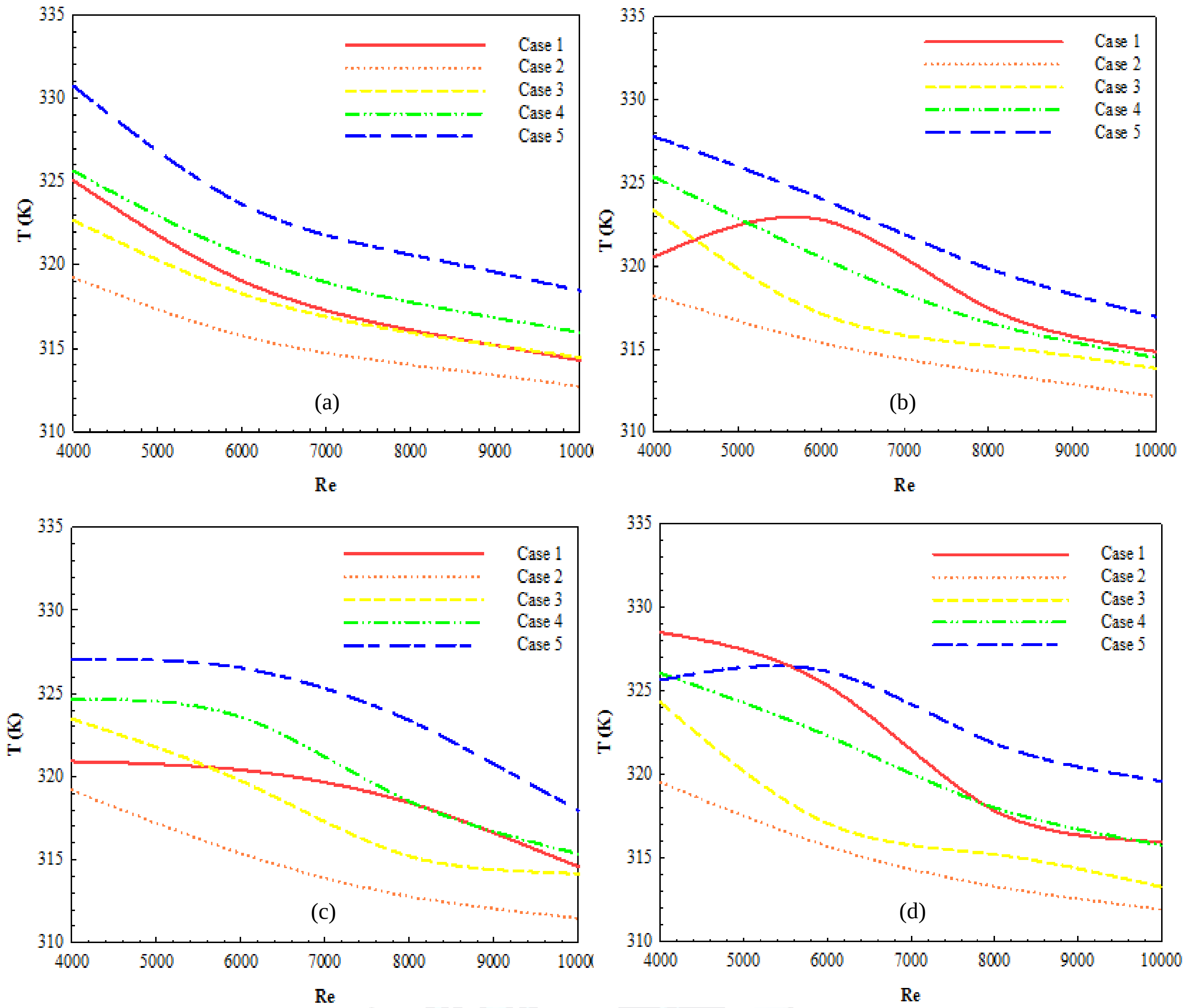
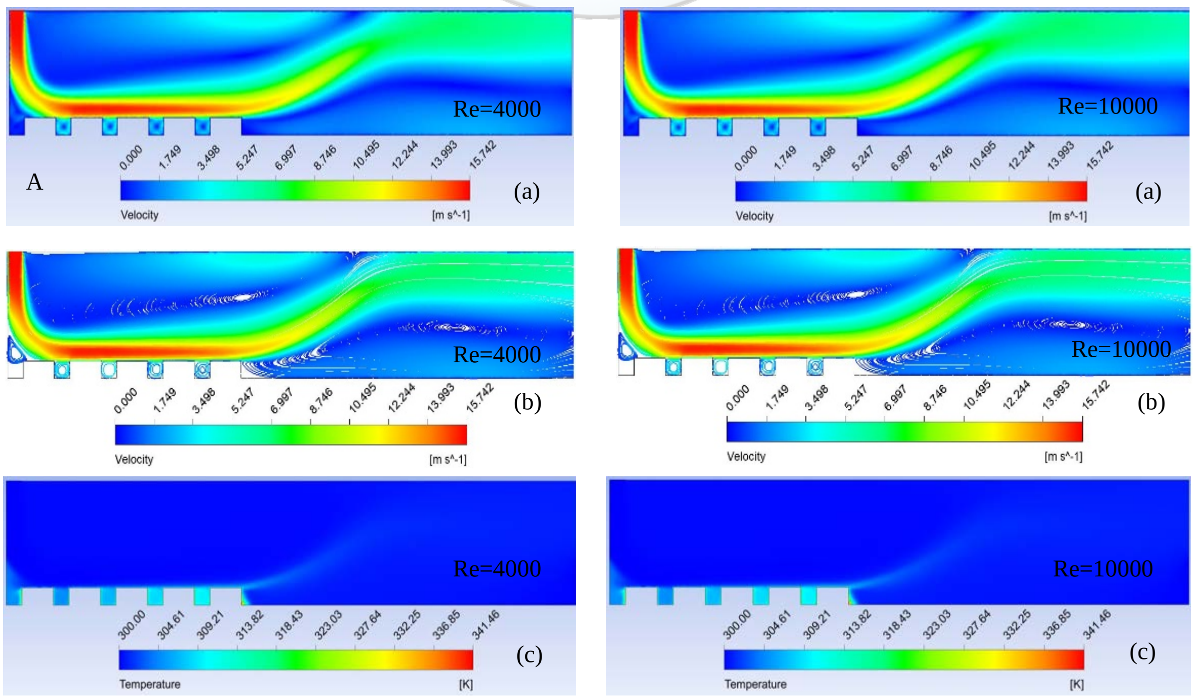


Figure. 5. Variations of mean surface temperature versus different Re number for different H/D_h and cases (a) H/D_h=4 (b) H/D_h=6 (c) H/D_h=10 and (d) H/D_h=12



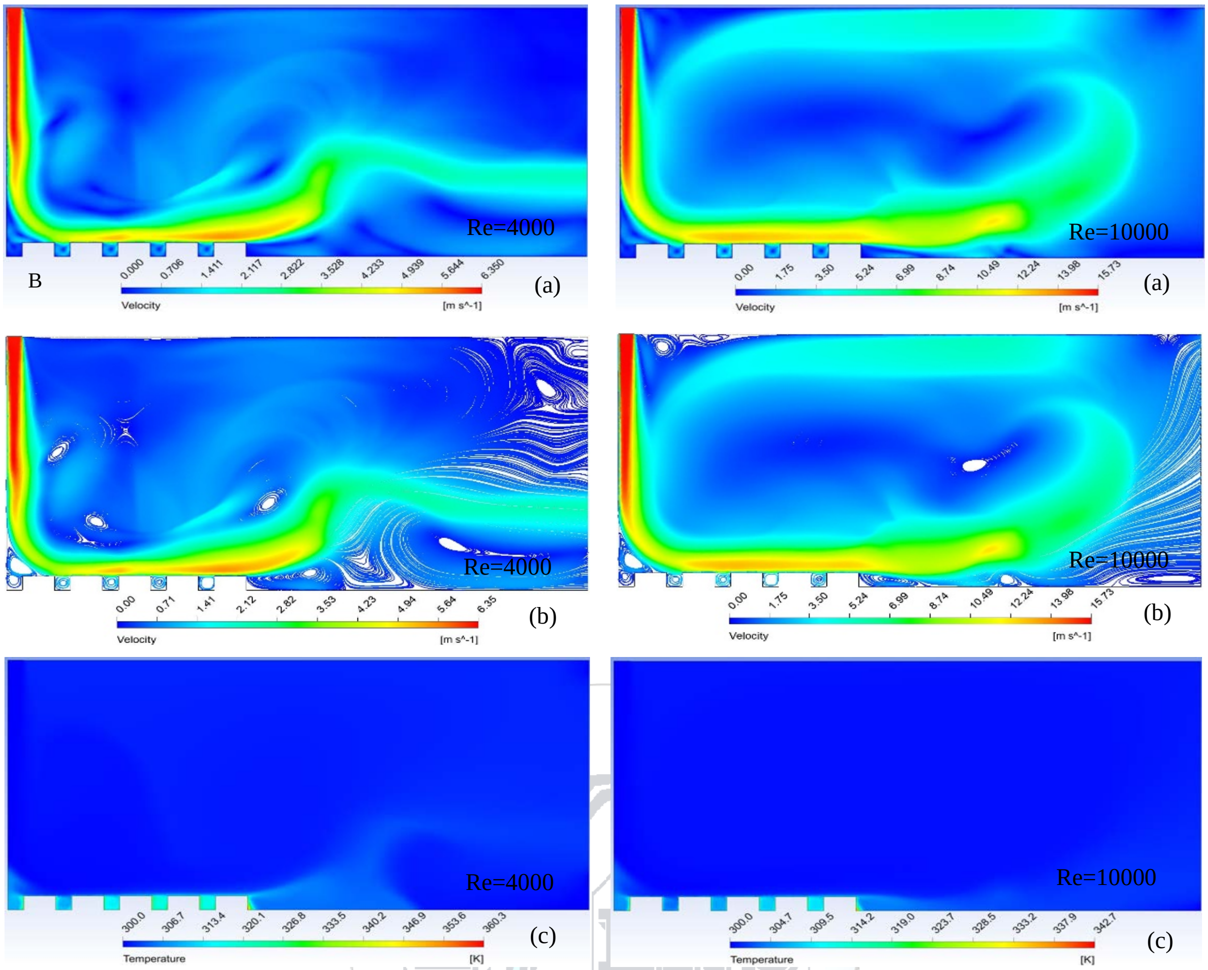


Figure. 6. A-H/D$_h$=4, B- H/D$_h$=10 a) Velocity b) Streamline c) Temperature distributions for different Re numbers

Table 2. Results of Nu$_m$, T$_{sm}$ and T$_{out\text{-}jet}$ air for different Re and H/D$_h$

Square Surface	Nu$_m$		T$_{sm}$		T$_{out\text{-}air\ jet}$	
	H/D$_h$=4	**H/D$_h$==12**	**H/D$_h$==4**	**H/D$_h$==12**	**H/D$_h$==4**	**H/D$_h$==12**
Re=4000	9.7298	9.6675	324.663	324.802	302.564	302.541
Re=6000	12.2246	11.1108	319.454	321.312	301.698	301.691
Re=10000	15.4981	15.3918	315.171	315.293	301.001	301.043

5. CONCLUSION

The present study was conducted into the numerical research of heat transfer from a heated square patterned copper surface, inside a rectangular channel by using a single jet flow. The numerical results were compared with the numerical and experimental datas of study existed in literature and found they were in well agreement. The results were presented as the variations of the mean Nu numbers and temperatures for each square patterned indentation surface. The temperature and velocity distributions and streamlines of jet fluid flow and mean surface temperature and Nu values of whole square surfaces and values of air jet outlet temperature were also analyzed for different Re numbers and jet-to-plate distance H/D$_h$ ratios. The following conclusions can be drawn from the numerical results;

-Increase in Reynolds number increases the heat transfer at all jet-to-plate distances. Average Nusselt number increases of 59.28% from Re=4000 to Re=10000 for H/D$_h$=4. Besides, Nu number was less sensitive to H/D$_h$ ratio in the range of H/D$_h$=4-12. Average Nusselt number decreases of 9.11% from H/D$_h$=4 to H/D$_h$=12 for Re=6000 because of a decline of turbulance intensity. Also, the highest average Nusselt number was attained for Re=10000 and H/D$_h$=6.

-Recirculations occur at the bottom of the left wall for all Reynolds numbers. These recirculations cause a change of direction of the jet flow. Thus, values of the highest local Nu number were determined for the case 2 for all Re numbers.

- Occurred recirculation sizes decrease with increasing Re number from 4000 to 10000 due to prevention of expansion of recirculations by increasing jet flow velocity.

-When the velocity of jet flow is high at the impingement region, it reduces toward channel outlet. Therefore, surface temperatures of the square indentations increase.

- Increasing the channel height causes to decrease length of wall jet region. The reason for this is reducing of flow velocity at a longer channel height. Decreasing flow velocity, on the surface of copper indentation plate, causes an increase in temperature.

-In a conclusion, the heat transfer, including local and average Nusselt numbers, are significantly affected by jet Reynolds numbers; while it is less sensitive to jet-to-plate distance. Also, it is considered that geometry of air jet and channel used in this study can be employed to cool electronic components due to resembling various electronic equipment application.

REFERENCES

[1]. B. W. Webb and C. Ma, “Single-phase liquid jet impingement heat transfer”, Adv. Heat Transf., vol. 26 pp. 105–117, 1995.
[2]. G. M. Carlomagno and A. Ianiro, “Thermo-fluid-dynamics of submerged jets impinging at short nozzle-to-plate distance: a review”. Exp Thermal Fluid Sci., vol. 58, pp. 15–35, 2014.
[3]. E. Argus, M. A. Rady and S. A. Nada, “A numerical investigation and parametric study of cooling an array of multiple protruding heat sources by a laminar slot air jet”. Int J Heat Mass Transf., vol. 28, pp. 787–805, 2006.
[4]. M. Popovac and K. Hanjalic K, “Large-eddy simulation of flow over a jet-impinged wall-mounted cube in a cross stream”, Int J Heat Fluid Flow, vol. 28, pp. 1360–1378, 2007.
[5]. Y. T. Yang, C. H. Hwang, “Numerical simulations on the hydrodynamics of a turbulent slot jet on a semi-cylindrical convex surface”, Num Heat Transfer, vol. 46, pp. 995–1008, 2004.
[6]. K. S. Mushatat, “Analysis of the turbulent flow and heat transfer of the impingement cooling in a channel with cross flow”, Eng Sci, vol. 18, pp. 101–122, 2007.
[7]. N. Zuckerman and N. Lior, “Jet impingement heat transfer: physics, correlations, and numerical modeling”, Adv. Heat Transf., vol. 39, pp. 565–631, 2006.
[8]. FLUENT User's Guide, Fluent Inc. Lebanon, NH, 2003.
[9]. M. Kilic, T. Calisir, S. Baskaya, “Experimental and numerical study of heat transfer from a heated flat plate in a rectangular channel with an impinging air jet”, J Braz. Soc. Mech. Sci. Eng., vol. 39, pp. 329-344, 2017.

ACKNOWLEGMENT

The authors would like to thank Scientific Research Projects Coordination Unit of Sivas Cumhuriyet University (Sivas/Turkey) for their financial support to this research under project number TEKNO-021.