International Journal of Engineering Science Invention (IJESI)
ISSN (Online): 2319-6734, ISSN (Print): 2319-6726
www.ijesi.org ||Volume 9 Issue 11 Series I || November 2020 || PP 25-40

A Study on Free Convection Heat Transfer From a Heated Finned Object in a Channel

M. M. Alam[1], M. A. Jafor[1]
[1]*Department of Mechanical Engineering, Bangladesh Army University of Science and Technology, Bangladesh*

ABSTRACT: *In this work, the fluid flow and heat transfer characteristics are investigated numerically with a rectangular object with fin placed in a channel. A heated object is placed at the bottom of the channel and side walls are assumed adiabatic and top wall is at a temperature in a laminar flow condition. The object with different sizes (where the object sizes are* $40\,mm \times 40\,mm$, $46\,mm \times 46\,mm$, $52\,mm \times 52\,mm$, $58\,mm \times 58\,mm$, $64\,mm \times 64\,mm$, $70\,mm \times 70\,mm$*) are used for this investigation. Numerical simulations with Rayleigh Numbers (Ra) in the ranges* $20 \leq Ra \leq 4.0x10^4$ *investigated the flow field. The two-dimensional (2D) mathematical models include the systems of partial differential equations such as Continuity equation, Navier-Stokes equation, and Energy equation which are solved by the finite volume method. Uniform grids are used in the entire computational domain. Numerical results are obtained for different values of Ra, Prandtl Numbers (Pr) and different object sizes in the form of velocity vectors, velocity contour, temperature contour, isothermal line, and streamlines. Heat transfer rate at the heated objects are presented in terms of average Nusselt number (Nu). This work shows that the heat transfer increase with the increase of Ra and heat transfer found the maximum for small object size. It also illustrates that the average Nu remains the same for small Grashof number (Gr) but it increases rapidly for various Pr at high Gr.*

KEYWORDS: *Free Convection heat transfer; 2D finned object, Rayleigh number, Prandtl number*

Date of Submission: 31-10-2020 Date of Acceptance: 12-11-2020

I. INTRODUCTION

Free convection heat transfer from a heated body attracts extra thoughtfulness regarding the analysts, since it doesn't requires any outside vitality for exchanging heat. Only when the fluid motion is set up by buoyancy effects resulting from the variation of density caused by temperature difference in the fluid, the free convection takes place. Yet, in the event that Fins are connecting in the heated object, it builds heat exchange from the surfaces of the heated object. Fin is one sort of flimsy portions of metal. Fin are utilized at first glance where the heat transfer coefficient is low. Typically, the fin material has a high thermal conductivity. The fin is exposed to a flowing fluid, which cools or heats it, with the high thermal conductivity allowing increased heat being conducted from the wall thru the fin. Heat transfer is the region that arrangements with the instrument in charge of exchanging vitality starting with one place then onto the next when a temperature contrast exists. Natural convection is one of the most economical and practical methods of cooling and heating. Natural convection is caused by temperature or concentration induced density gradient within the fluid. Natural convection stream happens as a consequence of impact of gravity strengths on liquids in which thickness slopes have been thermally settled. In a channel complex stream structures is frequently presented. Hence it is critical to comprehend the heat transfer attributes of free convection over a channel. The analysis of heat transfer characteristics of a heated object in fluid, for example, human body in atmospheric air in room, crops in Green House, electronic parts in gadgets and so forth are turning into a prominent subject of exploration in present time. Likewise, it has application in nuclear engineering regarding reactors cooling.

Heat exchangers are used in a wide range of applications such as power generation, auto and aerospace industry, electronics, and HVAC. The main purpose of a heat exchanger is to efficiently transfer the heat from one fluid to another which in most of the cases is separated by a solid wall. The cooling of mechanical or electronic gadgets has always developing significance because of the across the board presentation of microelectronics together with expanding requests up on practically and dependability. Heat is trapped in thermal devices for various reasons. Because of absence of proficient cooling of heat entrapped in the thermal devices, they can fail to meet expectations or even breakdown under certain condition. The ever increasing miniaturization, packaging density, quality demands and reduction in life cost will put ever increasing pressure on the solution of these problems. The solution of such heat transfer depends on successfully solving Navier-Stokes equation along with the energy equation.

In this project work, a heated object with fin is placed at the bottom in a rectangular channel. Here the flow is considered to be steady state and laminar. Free convection takes place between the heated object and air due to the buoyancy effect which is the upward force. The investigation is carried out for different values of *Ra*, *Pr* and different object sizes. The current investigations were simulated by using the commercial finite-volume code ANSYS FLUENT (version 16.1). Figure 1 shows the detailed geometry of the channel. Natural convection flows are largely dominated by buoyancy forces. The buoyancy forces are generated by density gradients, which vary primarily with temperature since pressure gradients are relatively small in these flows. When fluid flows inside a channel and if there remains temperature difference between fluid and solid surface, heat transfer takes place by convection heat transfer mechanism. After taking heat from the heated object fluid density decrease and it lift up due to buoyancy force. Thus there create a vacancy which is filled up by relatively cooler fluid with relatively larger density.

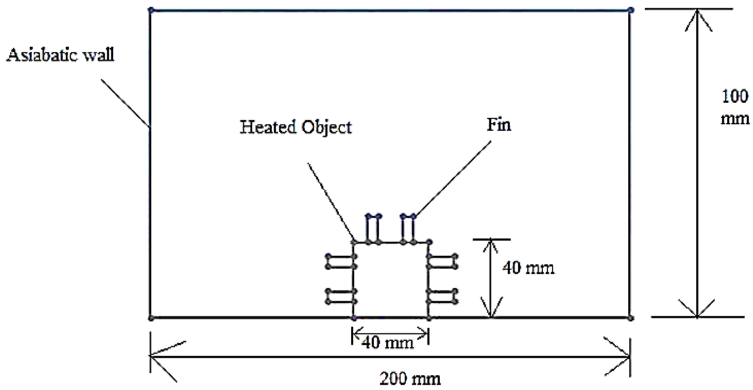


Figure 1. Arrangement of a heated object with fin in a channel

Flow thru a channel with a rectangular object with fin has a great impact on engineering applications. Typical applications of natural convection include electronic systems such as transformers, rectifiers, telecommunications module, Chips in laptop or other portable devices, heated wire, heating elements on electric furnace etc. These devices generally do not have fans or blowers. Instead, they are cooled by buoyancy-driven flow that carries heat from heated components and conducts thru the outer casing. Natural air ventilation is encountered in many practical applications such as energy conservation in the heating or cooling of environmental systems. Free convection heat transfer has great importance in shell-tube heat exchanger, air-cooled solar collectors, internally cooled turbine blades, chemical reactors, filtration, cooling of IC chips and other thermal devices. Recently this process has been proposed in nuclear power plant to cool the surface of the bodies in which heat is generated by nuclear fusion.

II. LITERATURE REVIEW

A number of scientists have worked on incompressible flows of free convection. Lee JR et al. [1], analysis numerically the natural convection in a horizontal enclosure heated from the bottom wall, cooled at the top wall and having a square adiabatic body in the center. The discretized equations with proper boundary conditions are solved based on stream function and vorticity formulation of the 2D Navier-Stokes equation and continuity equation and then the temperature distribution at all nodes are calculated by solving Energy equation. For the investigation of the effect of *Pr* on natural convection, a range of *Pr* are considered. Numerical results are obtained for different values of *Pr*. Sarris IE et al. [2] studied natural convection in a two dimensional, rectangular enclosure with sinusoidal temperature profile on the upper wall and adiabatic conditions on the bottom and sidewalls numerically. The applied sinusoidal temperature is symmetric with respect to the mid-plane of the enclosure. Numerical calculations are produced for *Ra* in the range 10^2 to 10^8, and results are presented in the form of streamlines, isotherm contours, and distributions of local *Nu*.

Nimkar MP et al. [3], the study of natural convection in a horizontal plate with vertical channels. The parameters varied during the experiment are heat input, aspect ratio and horizontal plate with and without V-slot. The study was aimed to determine the heat transfer characteristics and temperature distribution. Bakkas M et al. [4], studied numerically *2D* steady natural convection in a horizontal channel with the upper wall maintained cold at constant temperature and the lower one provided with rectangular heating blocks, periodically distributed. The blocks are connected with adiabatic segments and their surfaces are assumed to release a uniform heat flux. The study is performed using air as the working fluid. The spacing between the blocks is maintained constant while the *Ra* and the relative height of the blocks are varied. Zeitoun O et al. [5], investigate *2D* laminar natural convection heat transfer in the air around horizontal ducts with rectangular and square cross-section numerically. Different aspect ratios are used for a wide range of *Ra*. Temperature and velocity profiles are obtained near each surface of the duct. For different aspect ratios, heat transfer data are generated and presented in terms of Nusselt number and Rayleigh number.

Alami ME et al. [6] performed a numerical study of natural convection from a *2D* horizontal channel with the rectangular heated block. The studied domain reduced to a "*T*" form cavity that presents asymmetry with respect to a vertical axis passing thru the middle of the openings. The results are obtained for the varied parameters as the *Ra* and blocks gap. Boyalakuntla DS et al. [7] numerically analyze the possibility of using buoyant flow in the display panel of a laptop for electronics cooling. Three-dimensional (*3D*) channels with embedded pin fin arrays are analyzed using an unstructured finite volume method. Singh S et al. [8] performed almost similar investigations. They considered six placement configurations of the inlet and outlet openings. They observed that the maximum cooling effect is achieved if the inlet is kept near the bottom wall while the outlet is placed near the top of the hot wall. Han CY et al. [9] perform an investigation to study the radiation-affected steady-laminar natural convection induced by a hot inner cylinder under a large temperature difference in the cylindrical annuli filled with a gray gas for better visualization.

Based on the result of this study, when there exists a large temperature difference between two cylinders, the existence of a radiative participating medium is found to incur a distinct difference in fluid dynamics as well as thermal behavior. The fin creates a great impact on heat transfer characteristics of the object in free convection Numerical results are expressed for different values of *Ra*, *Pr* and different object sizes in the form of velocity vectors, velocity contour, temperature contour, isothermal line, and streamlines.

III. MATHEMATICAL MODELLING AND BOUNDARY CONDITION

Computational Fluid Dynamics (CFD) is a numerical approach for simulating fluid flow. It allows the practitioners and researchers to predict characteristics of a system, including flow velocity, pressure, temperature and heat transfer. The use of CFD is a numerical methodology for solving the governing equations of fluid flow. The governing equations of fluid flow are partial differential equations; when discretized on a mesh, they transform into algebraic equations, which can be solved by a finite-difference/finite-volume algorithm. In this work, the flow is considered *2D*, steady state, incompressible, laminar and all physical properties are assumed constant through the channel with a heated object with fin at the bottom. In CFD the fundamental governing equations of fluid dynamics are the continuity, momentum and energy equations. The physics of any fluid flow and heat transfer is governed by these equations.

(i) Navier-Stokes equation for 2D is given below:

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \vartheta\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right)(1)$$

Whereas,

$X_{momentum}$: $$\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial w}{\partial y}\right) = -\frac{\partial P}{\partial x} + \square\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) + \rho g_x \quad (2)$$

$Y_{momentum}$: $$\rho\left(\frac{\partial v}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial w}{\partial y}\right) = -\frac{\partial P}{\partial x} + \square\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) + \rho g_y \quad (3)$$

(ii) Continuity equation- $$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (4)$$

(iii) Energy equation- $$u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} = \frac{k}{\rho c_p}\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right)(5)$$

To determine the dimensionless parameters that govern the heat transfer in free convection, we need to non-dimensionalized the above governing equations. The following dimensionless parameters are defined as follows.

$$X = \frac{x}{W} \qquad Y = \frac{y}{H} \qquad U = \frac{u}{U_0} \qquad V = \frac{v}{U_0} \qquad \theta = \frac{T - T_c}{T_h - T_c} \quad (6)$$

Where $U_0$is the reference velocity.

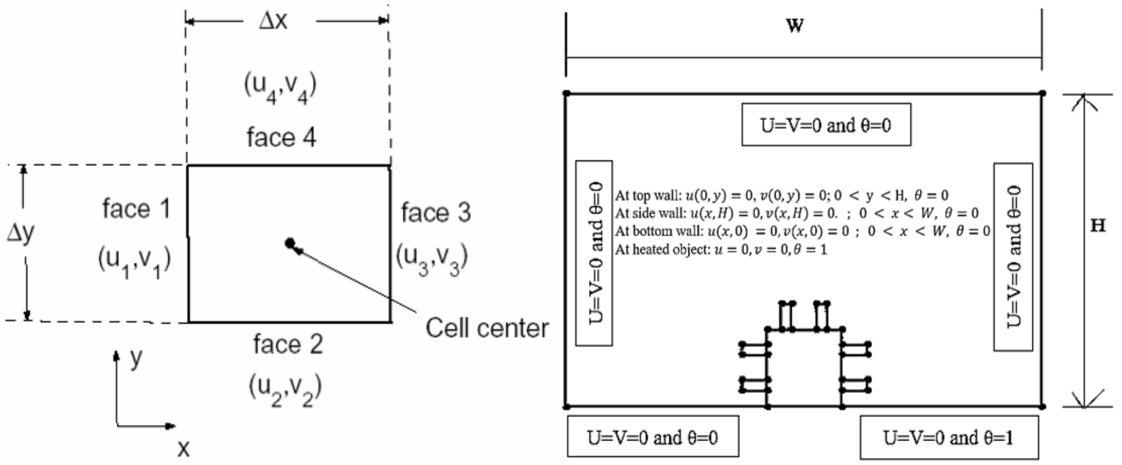
For discretization using the finite-volumemethod considering steady, incompressible flow, this equation reduces to

$$\int_S V \bullet \hat{n}\, dS = 0$$

The integration is over the surface *S* of the control volume and $\hat{n}$ is the outward normal at the surface. Consider the rectangular cell shown below in Figure 2(a). The velocity at face *i*is taken to be$V_i = u_i\hat{i} + v_i\hat{j}$.Applying the mass conservation equation to the control volume defined by the cell gives

$$-u_1\Delta y - v_2\Delta x + u_3\Delta y + v_4\Delta x = 0$$

The face values $u1$, $v2$, etc. are obtained by suitably interpolating the cell-centre values for adjacent cells. Similarly, one can obtain discrete equations for the conservation of momentum and energy for the cell. One can readily extend these ideas to any general cell shape in 2D or 3D and any conservation equation. For our forgoing simulation of 2D convection heat, transfer from a heated finned object in a closed channel the following boundary condition is considered as shown in Figure 2(b). In the present investigation, a uniform grid system with sizes 10^{-1} mm is used. Five combinations of different sizes 10^{-1} mm, 10^{-2} mm, 10-^{3} mm, 10^{-4} mm and 10^{-5} mm are used to test the effect of grid size on the accuracy of predicted results. The deviation between the various distribution along the channel length is very little. The result of grid size 10^{-1} mm is presented throughout this paper.



(a) (b)

Figure 2. (a) Rectangular cell discretization, (b) Typical sketch of problem geometry and coordinates

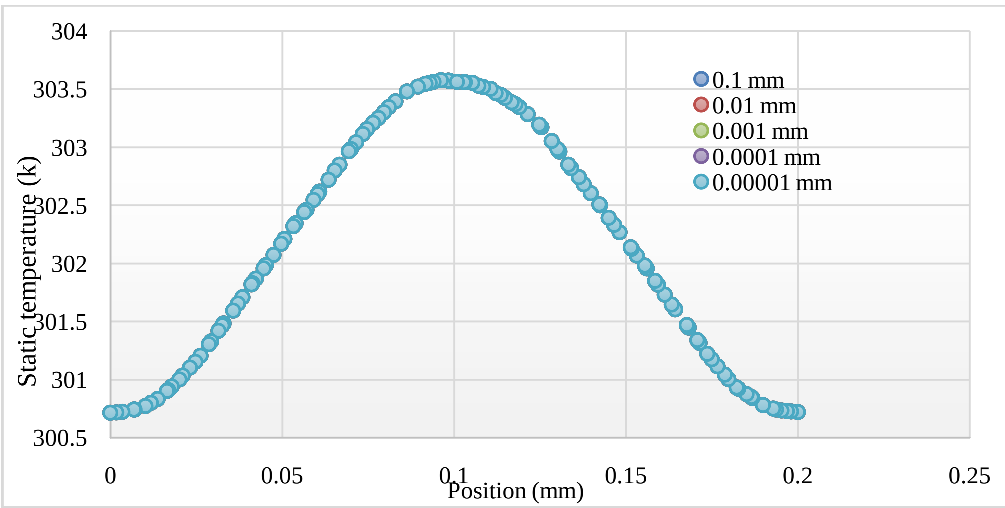


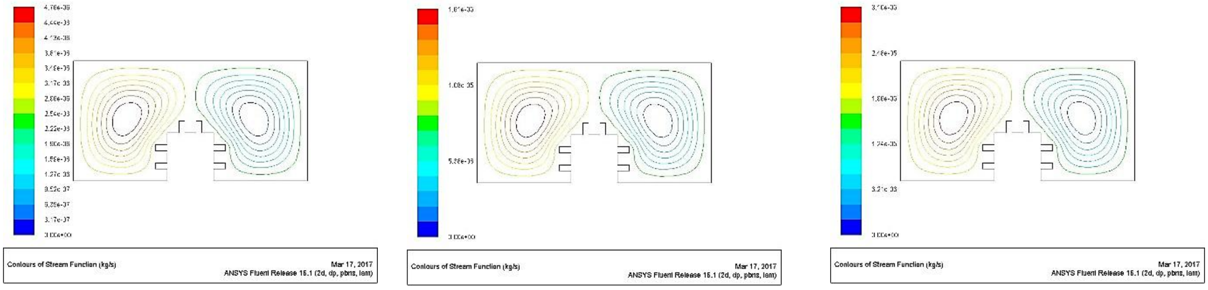
Figure 3. Grid sensitivity test for grid sizes 10^{-1} mm, 10^{-2} mm, 10^{-3} mm, 10^{-4} mm, 10^{-5} mm.

The governing non-dimensional partial differential equations were solved using ANSYS Fluent to get stream function, velocity vector, velocity magnitude, isothermal line and temperature at every internal grid points in the computational domain. The mesh size 10^{-1} mm was used on the analysis field. The length of the channel is chosen $200\,mm\ \times 100\,mm$ and the sizes of the heated object are $40\,mm \times 40\,mm$, $46\,mm \times 46\,mm$, $52\,mm \times 52\,mm$, $58\,mm \times 58\,mm$, $64\,mm \times 64\,mm$, $70\,mm \times 70\,mm$. The governing physical parameters in the problem considered are *Ra* and *Pr*, where *Ra* is the product of *Gr* and *Pr*. The flow domain is solved for *Ra* in the range $20 \le Ra \le 4.0 \times 10^4$.

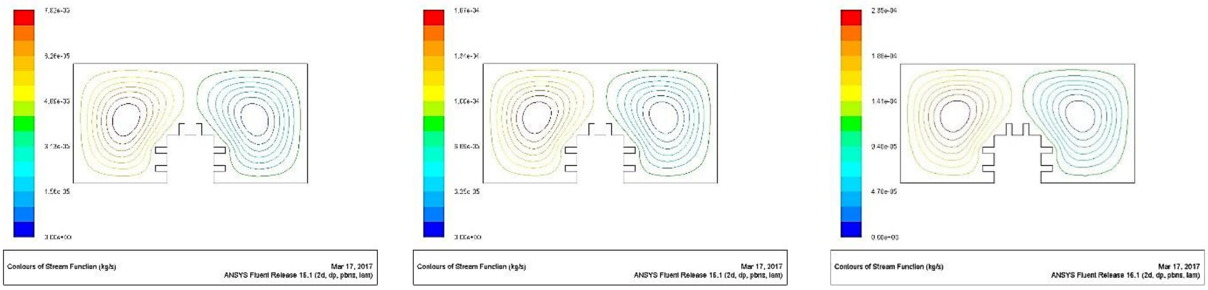
IV. RESULTS AND DISCUSSION

Effect of Rayleigh Numbers (Ra)

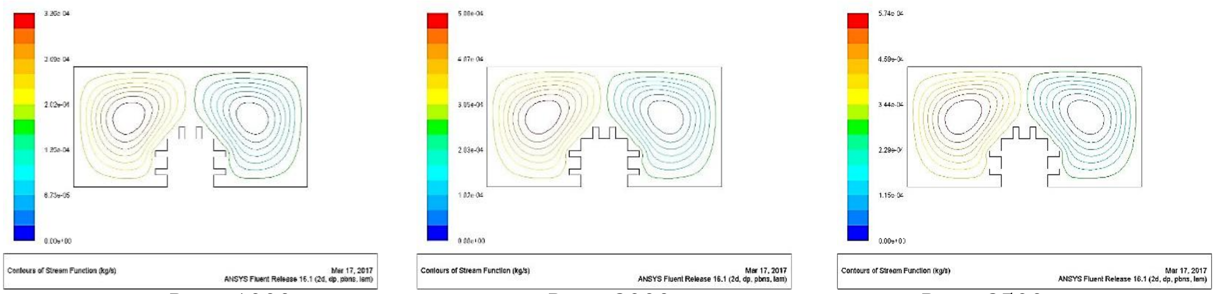
From the figure, the effect of *Ra* is finely observed. The main flow is symmetric from hot surface zone of the object to the upper zone near to the top wall. The flow from the left side of the object swept by rotating anti-clockwise direction and flow from the right side swept with clockwise rotation. The flow is laminar and there is no external force except buoyancy force is acted on fluid, which drives the hot fluids, by a greater circulation at the two sides of the object. At first from the surrounding, the cold fluid, which is heavier, comes closer to the heated object and takes the heat as because of the temperature gradient between them. When the air gets heated, its density reduced and becomes lighter in comparison to the cold fluid. Because of this density gradient there creates a buoyancy force, which tries to drive the hot and lighter fluid in upper direction, but as in the upper region, there remain cold and heavier fluid formerly this hot and lighter fluid makes their way from the two lower corners to get out of the channel. When the hot fluid gets out of the channel then the cold fluid from the upper region comes to that place to fill up the vacancy, gets heated, and thus repeat the similar occurrence. This is why there is found a circulation of flow.



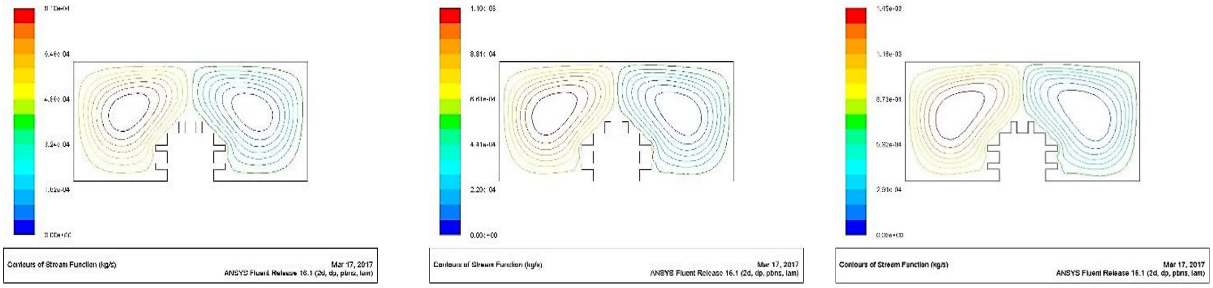
$Ra = 20$ $Ra = 60$ $Ra = 100$



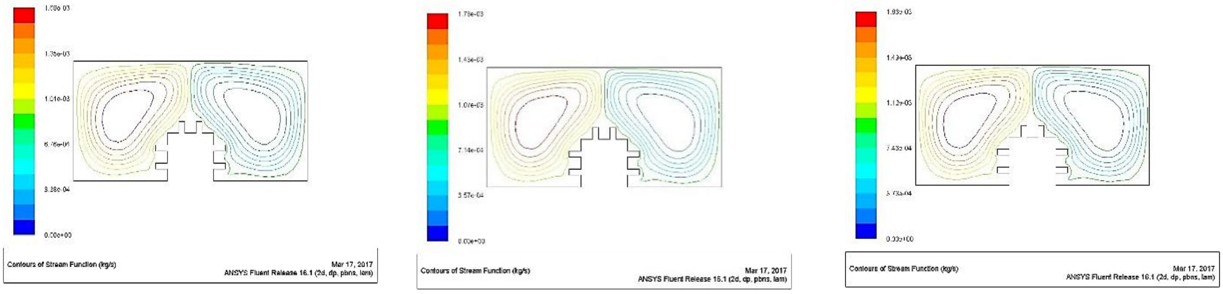
$Ra = 200$ $Ra = 400$ $Ra = 600$



$Ra = 1000$ $Ra = 2000$ $Ra = 2500$



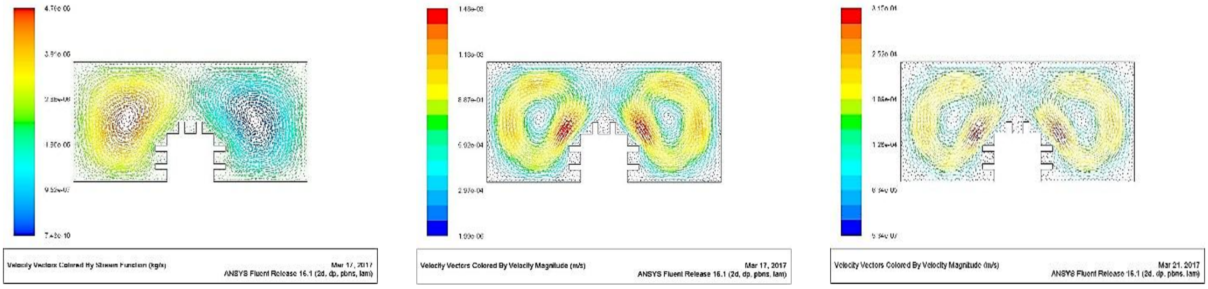
$Ra = 5000$ $Ra = 10000$ $Ra = 20000$



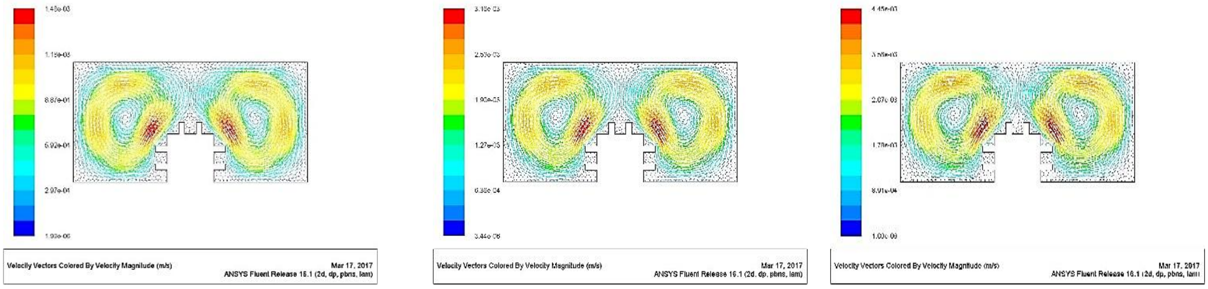
$Ra = 30000$ $Ra = 35000$ $Ra = 40000$

Figure 4. Steam function at fixed $Pr = 0.71$ and different *Ra*

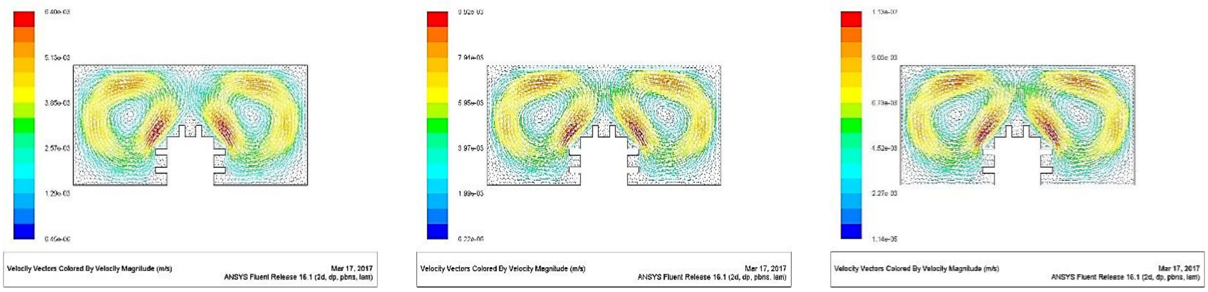
At low *Ra*, the low thermal driving force induces weak airflow and having low velocity and less dense streamline, which is shown from Stream function (Figure 4), and Velocity vector (Figure 5). In addition, conduction is seen to dominate at low *Ra*. With the increase of *Ra* it is found that streamlines get denser and towards from each side in the middle of the channel which physically indicates the higher velocity and higher dense flow. That results in higher heat transfer from the heated object to the fluid. These characteristics observed from the figures of stream functions and velocity vectors from Figure 5.



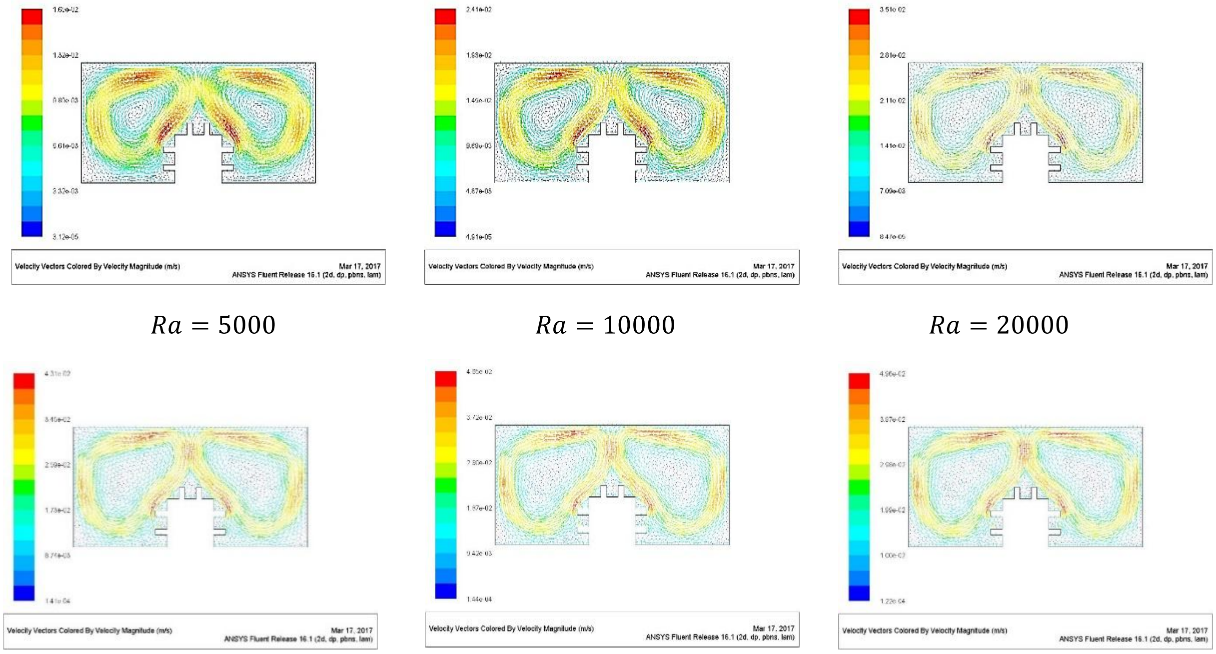
$Ra = 20$ $Ra = 60$ $Ra = 100$



$Ra = 200$ $Ra = 400$ $Ra = 600$



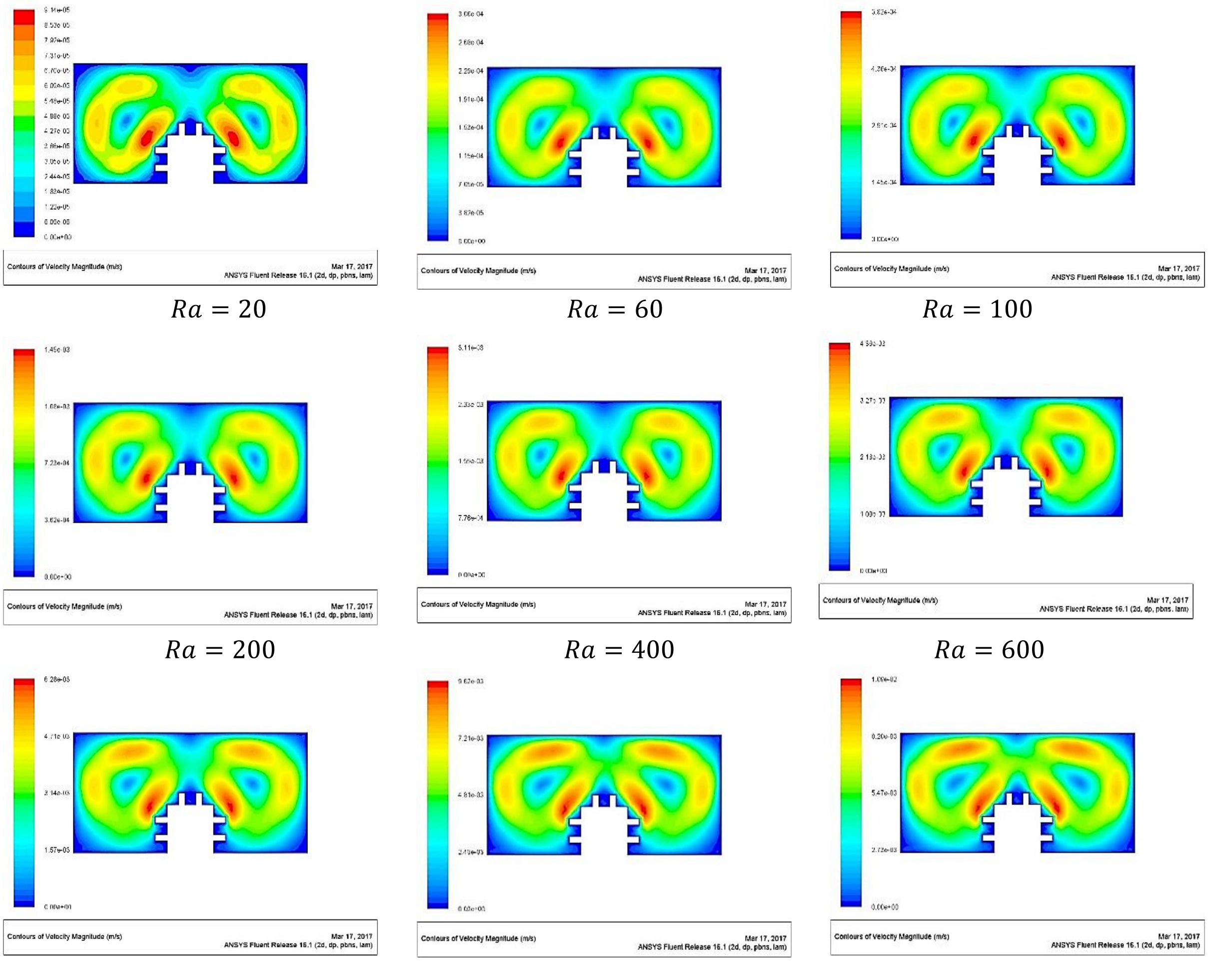
$Ra = 1000$ $Ra = 2000$ $Ra = 2500$



$Ra = 5000$ $Ra = 10000$ $Ra = 20000$

$Ra = 30000$ $Ra = 35000$ $Ra = 40000$

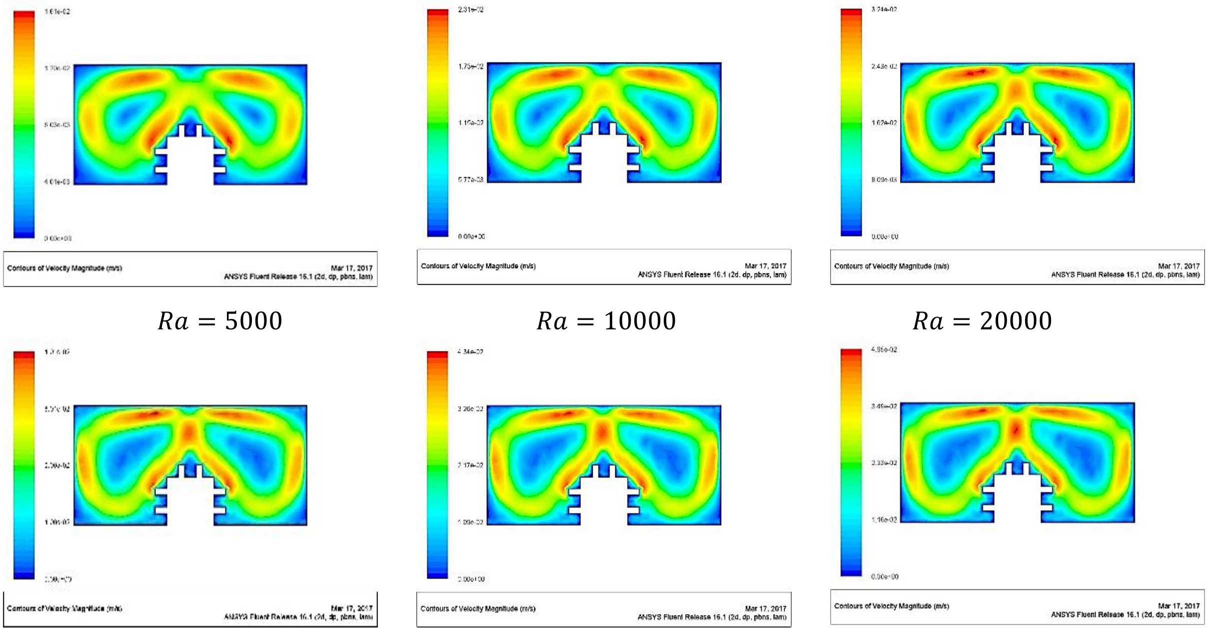
Figure 5. Velocity vector at fixed $Pr = 0.71$ and different Ra.



$Ra = 20$ $Ra = 60$ $Ra = 100$

$Ra = 200$ $Ra = 400$ $Ra = 600$

$Ra = 1000$ $Ra = 2000$ $Ra = 2500$

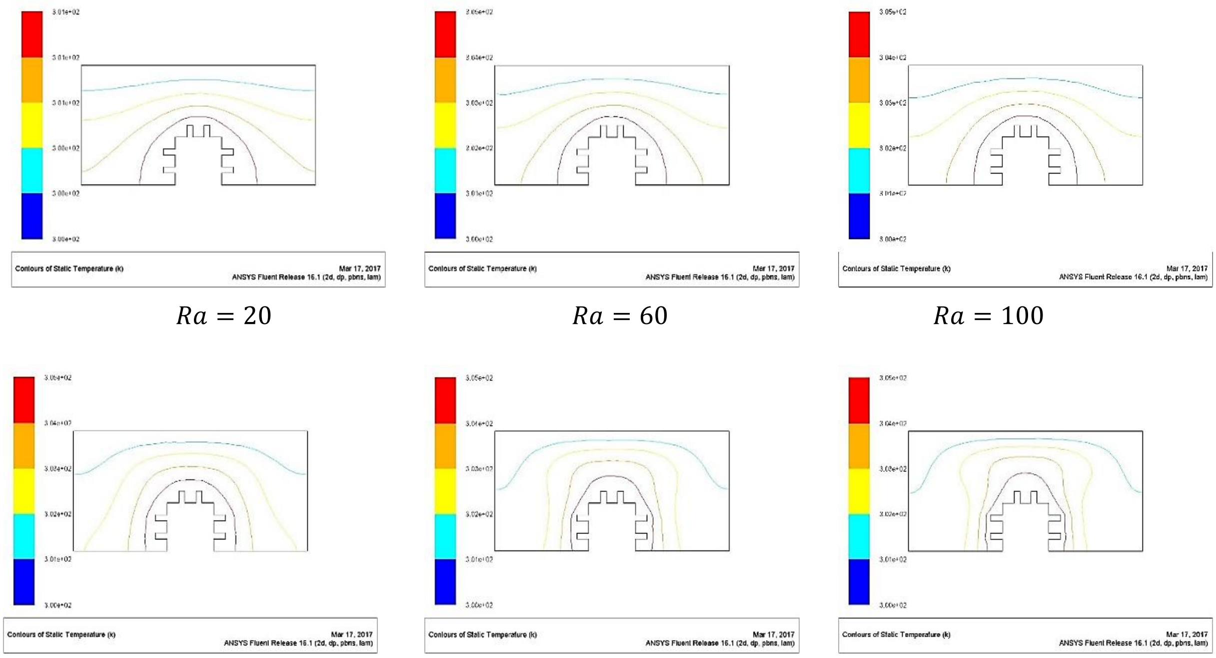


$Ra = 5000$ $Ra = 10000$ $Ra = 20000$

$Ra = 30000$ $Ra = 35000$ $Ra = 40000$

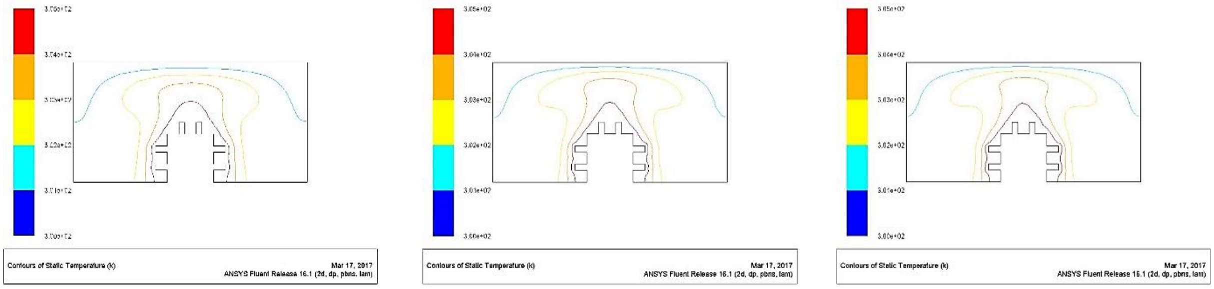
Figure 6. Velocity contour at fixed $Pr = 0.71$ and different *Ra*.

The temperature distribution is seen for *Ra* from Figure 7 and Figure 8. Velocity increases with the increase of *Ra*. Increase of *Ra* means increase of temperature distribution so as buoyancy effect or decrease of viscous effect which helps to increase velocity naturally. The higher velocity occurs at lower left corner near the heated object and as well as lower right corner near the heated object of the channel. From the figure of color contour of temperature field it is found that the highest temperature remains around the heated object. Temperature distribution is smooth. From the figure it is also found that the upper both corner regions remain cold compared to the vicinity of the object.

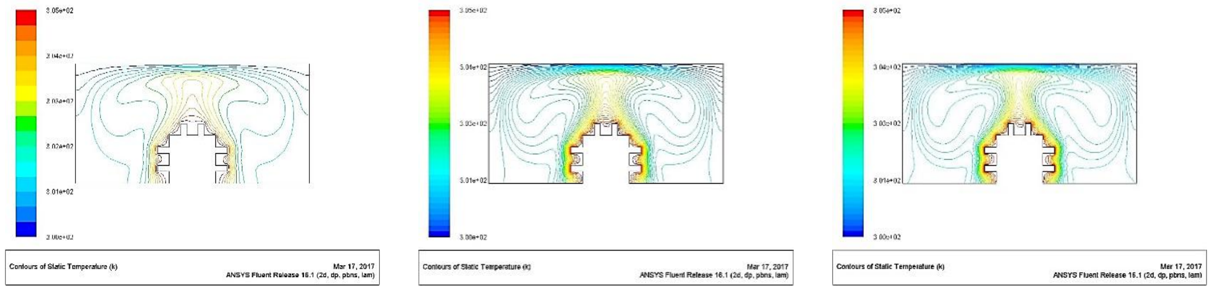


$Ra = 20$ $Ra = 60$ $Ra = 100$

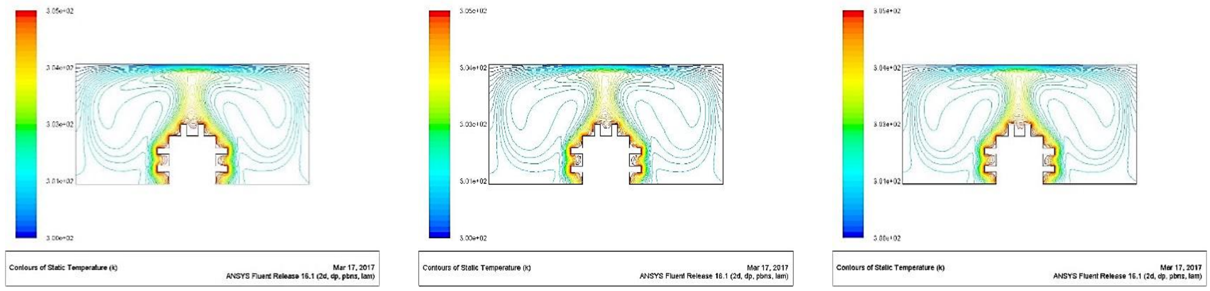
$Ra = 200$ $Ra = 400$ $Ra = 600$



$Ra = 1000$ $Ra = 2000$ $Ra = 2500$



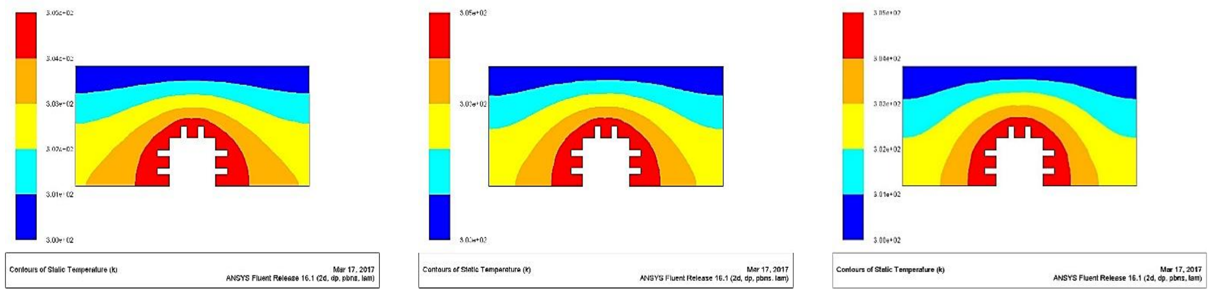
$Ra = 5000$ $Ra = 10000$ $Ra = 20000$



$Ra = 30000$ $Ra = 35000$ $Ra = 40000$

Figure 7. Isothermal line at fixed $Pr = 0.71$ and different Ra.

For Ra less than 5000, there is a rapid change of temperature distribution, which is observed from figure. At higher Ra higher than 5000, the rate of heat transfer increases such that the temperature distribution increases.



$Ra = 20$ $Ra = 60$ $Ra = 100$

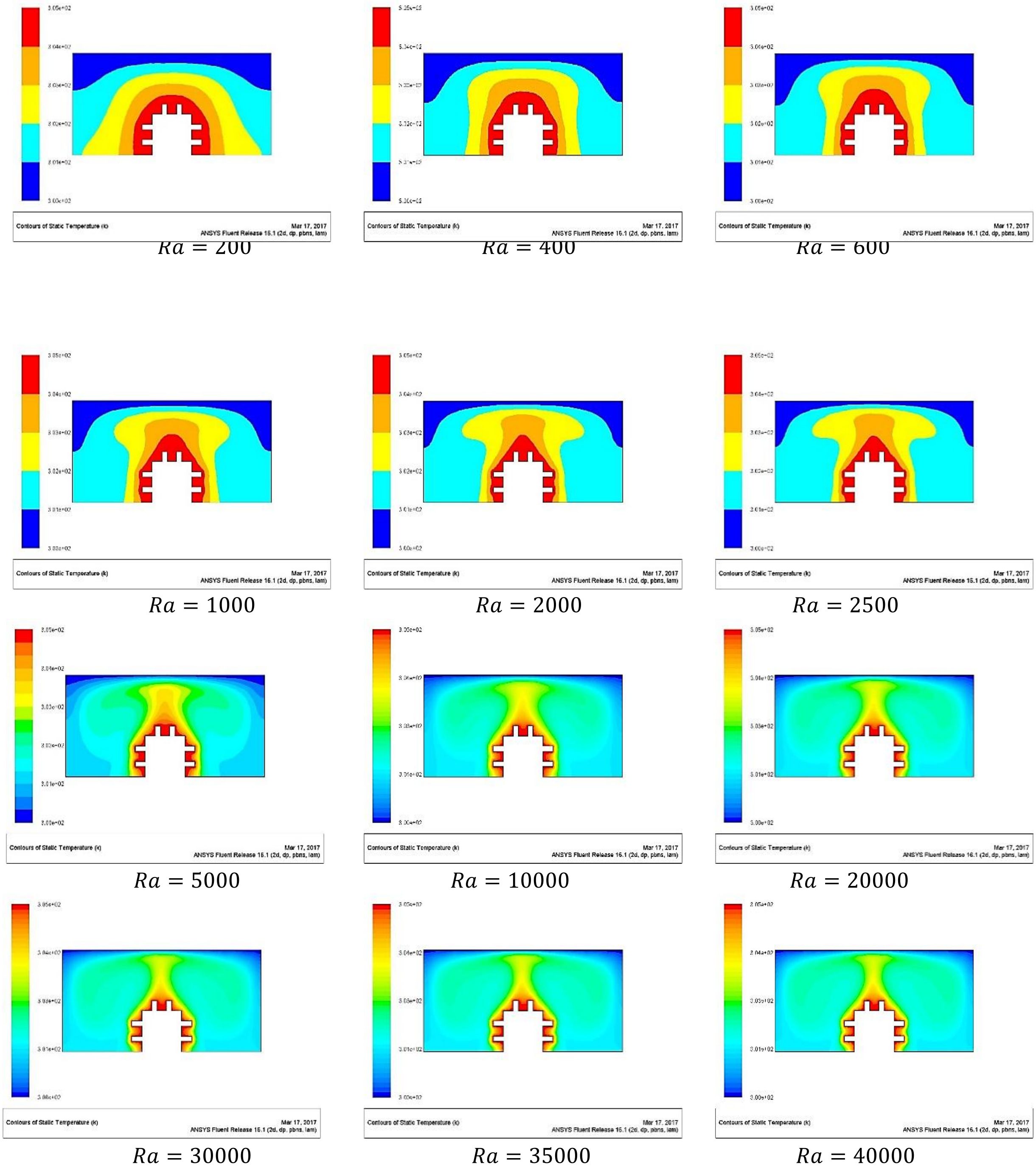
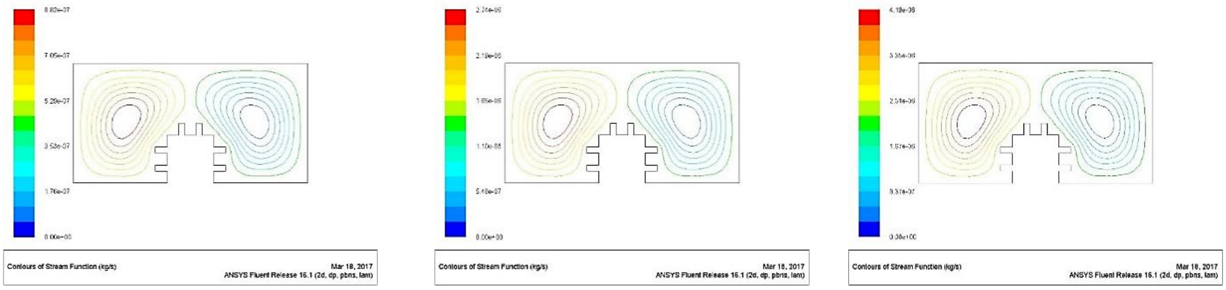


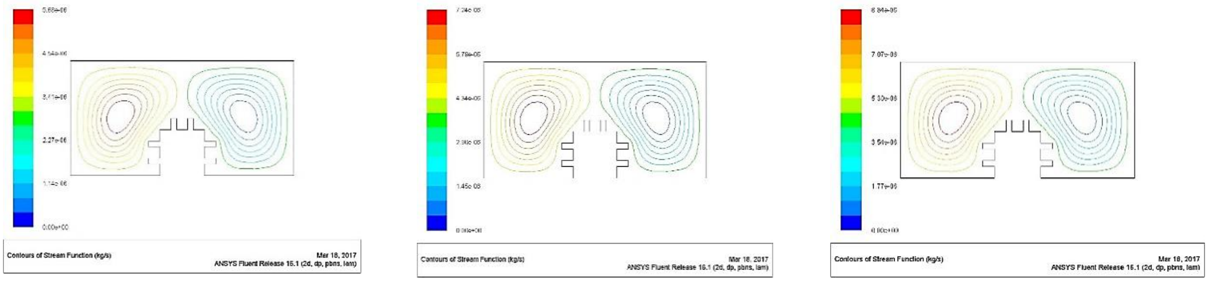
Figure 8. Temperature contour at fixed $Pr = 0.71$and different *Ra*.

Effect of Prandtl Numbers (*Pr*)

Figure 9 to Figure 12 exhibits the flow field under a fixed *Ra* 500 and sizes 40 mm by 40 mm while *Pr* is allowed to vary. In the last of the paper Figure 19(a) illustrates the effect of *Gr* and *Pr* on the average *Nu*. The average *Nu* exhibits a positive dependence on both *Gr* and *Pr*. It shows at low *Gr* or *Ra*, heat transfer occurs principally by conduction and therefore the average *Nu* is expected to approach constant value independent of *Gr* and *Pr*. For the higher value of *Gr* the conduction effect decrease and buoyancy effect increase so that the average *Nu* increase rapidly with the *Pr*.

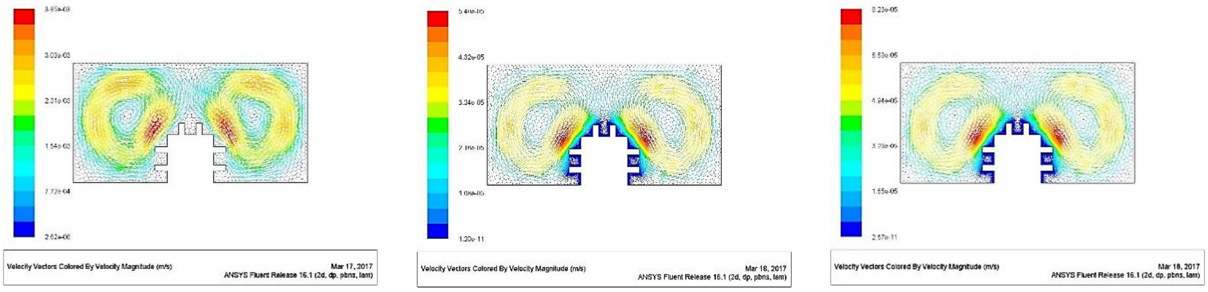


$Pr = 1 Pr = 2 Pr = 3$

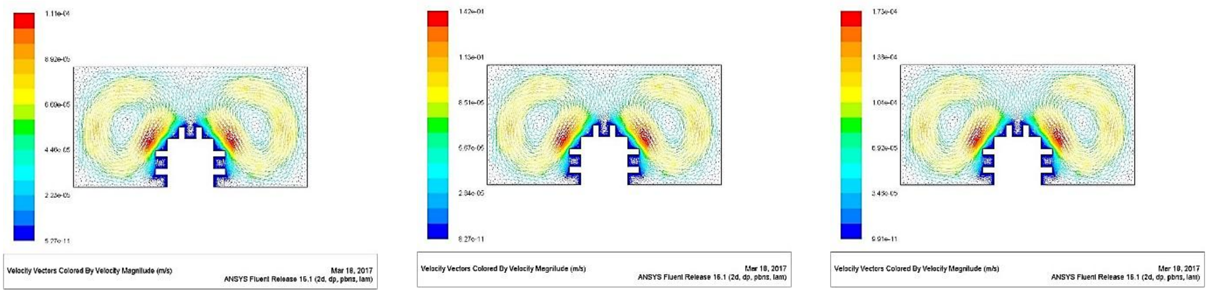


$Pr = 4 Pr = 5 Pr = 6$

Figure 9. Stream function at $Ra = 500$ and $1 \leq Pr \leq 6$

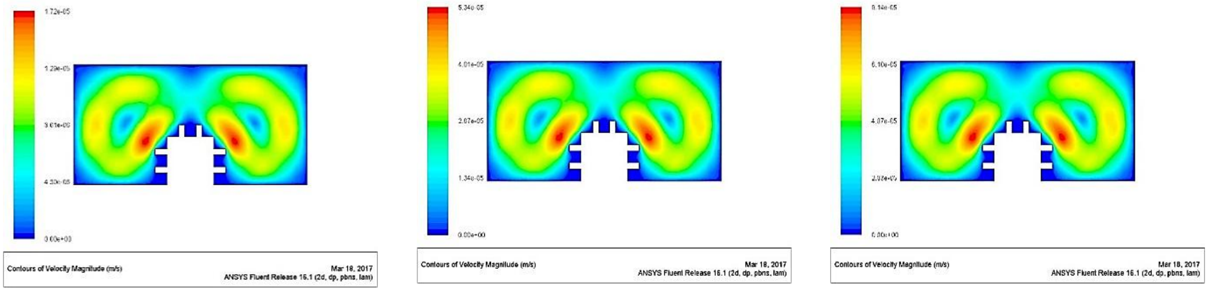


$Pr = 1 Pr = 2 Pr = 3$

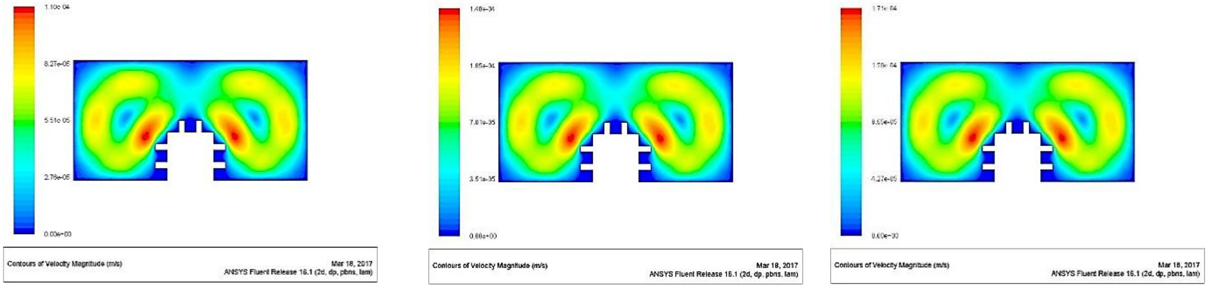


$Pr = 4 Pr = 5 Pr = 6$

Figure 10. Velocity vector at $Ra = 500$ and $1 \leq Pr \leq 6$.

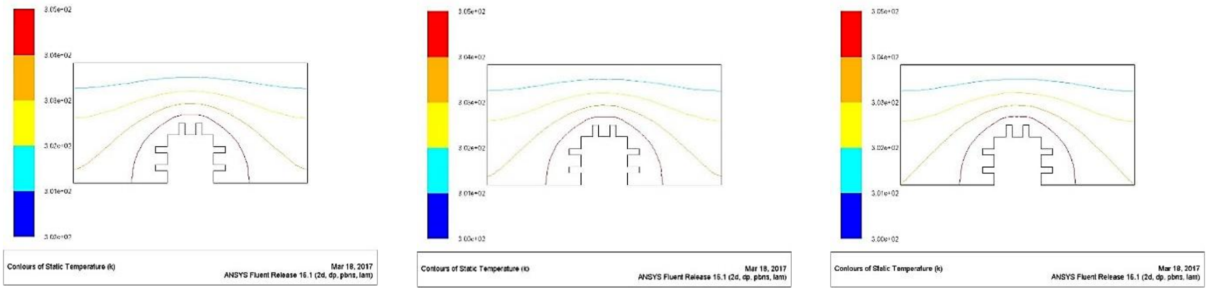


$Pr = 1 Pr = 2 Pr = 3$

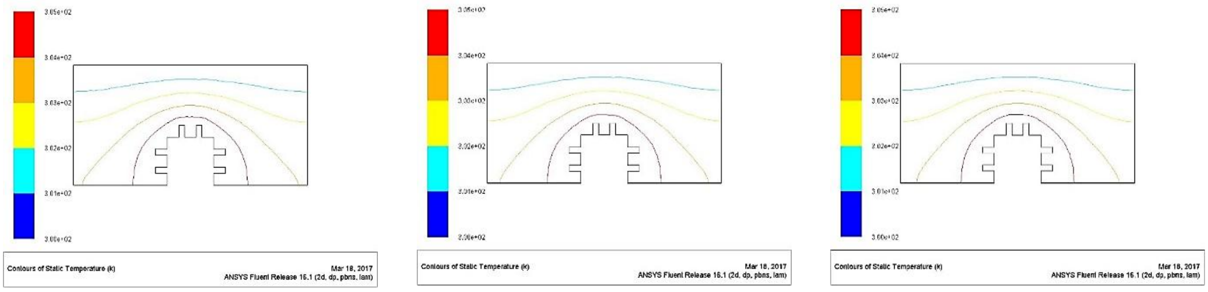


$Pr = 4 Pr = 5 Pr = 6$

Figure 11. Velocity contour at $Ra = 500$ and $1 \leq Pr \leq 6$



$Pr = 1 Pr = 2 Pr = 3$

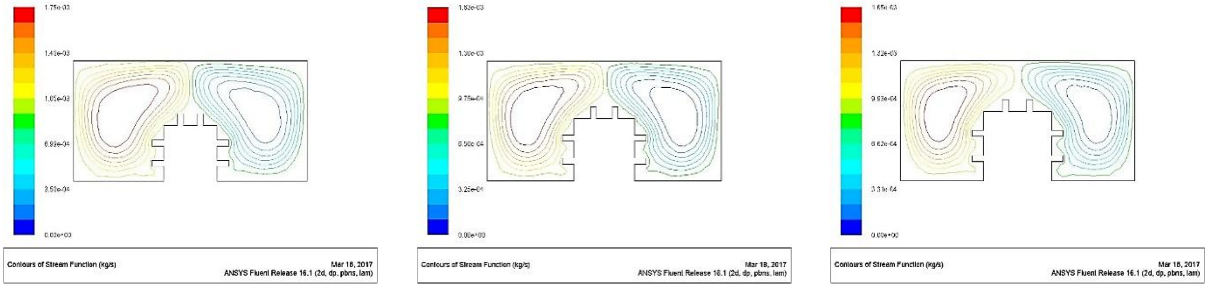


$Pr = 4 Pr = 5 Pr = 6$

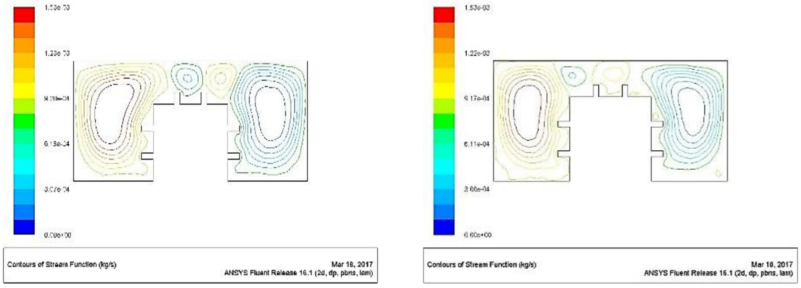
Figure 12. Isothermal line at $Ra = 500$ and $1 \leq Pr \leq 6$

Effect of Object Sizes

The effect of various object sizes are well depicted in Figure 13 to Figure 17.When the object sizes are increasing then the temperature of the enclosure is increasing so that the enclosure will be heated rapidly. The increasing of temperature causes decreasing in heat transfer rate. From the Figure 18 and Figure 19 it is shown that the average *Nu* decreases with theincreasing of heated object sizes. The maximum *Nu* also obtained in the smaller object size. From Figure 18 and Figure 19 it shows that the average *Nu* increases with the increase of Rayleigh number.

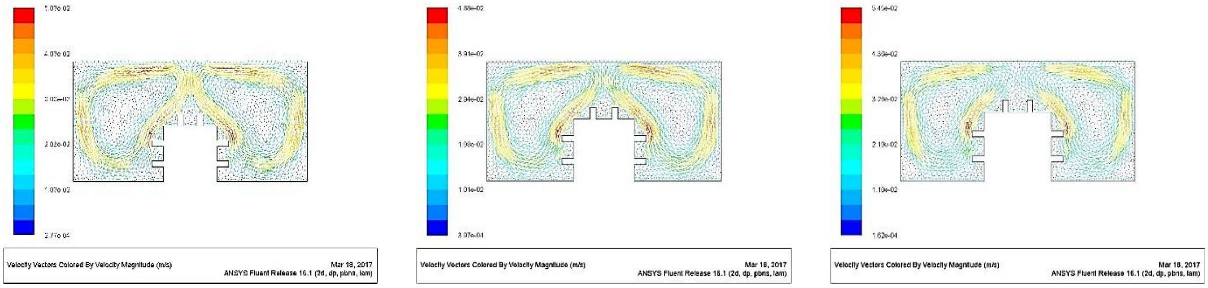


$46\ mm \times 46\ mm 52\ mm \times 52\ mm 58\ mm \times 58\ mm$

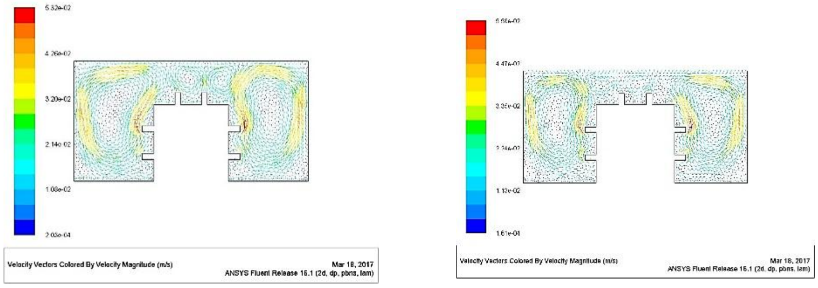


$64\ mm \times 64\ mm 70\ mm \times 70\ mm$

Figure 13. Stream function at $Ra = 40000$ and $Pr = 0.71$ with different object size.

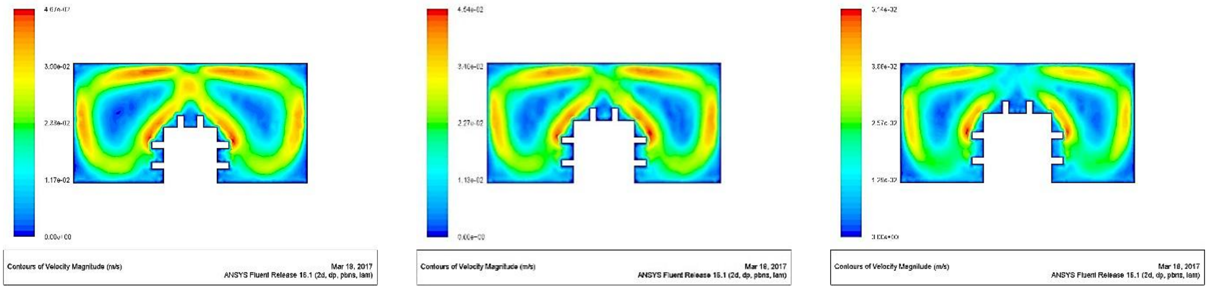


$46\ mm \times 46\ mm 52\ mm \times 52\ mm 58\ mm \times 58\ mm$

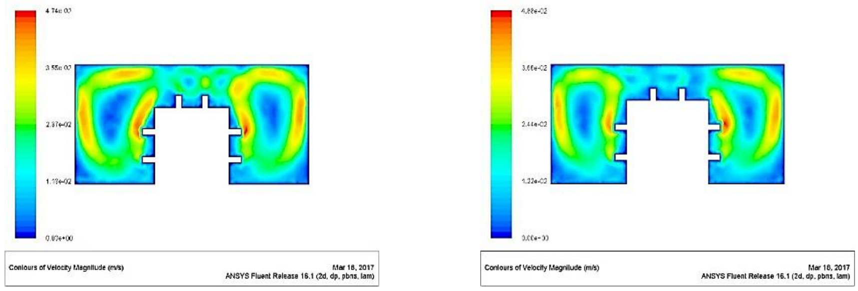


$64\ mm \times 64\ mm 70\ mm \times 70\ mm$

Figure 14. Velocity vector at $Ra = 40000$ and $Pr = 0.71$ with different object size

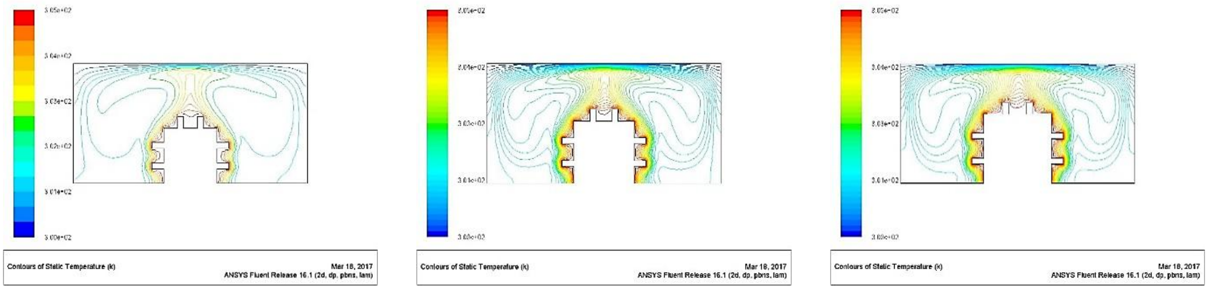


$46\ mm \times 46\ mm 52\ mm \times 52\ mm 58\ mm \times 58\ mm$

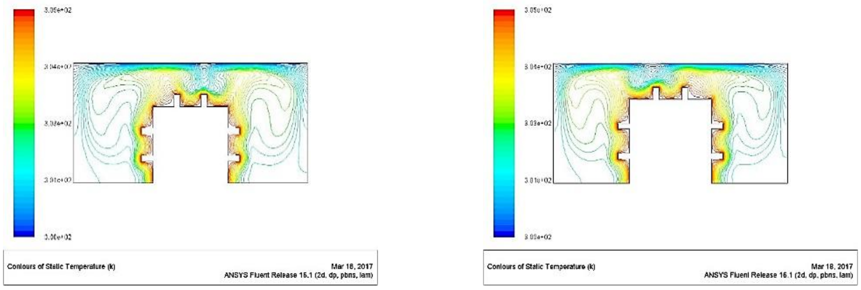


$64\ mm \times 64\ mm 70\ mm \times 70\ mm$

Figure 15. Velocity contour at$Ra = 40000$ and $Pr = 0.71$ with different object size.

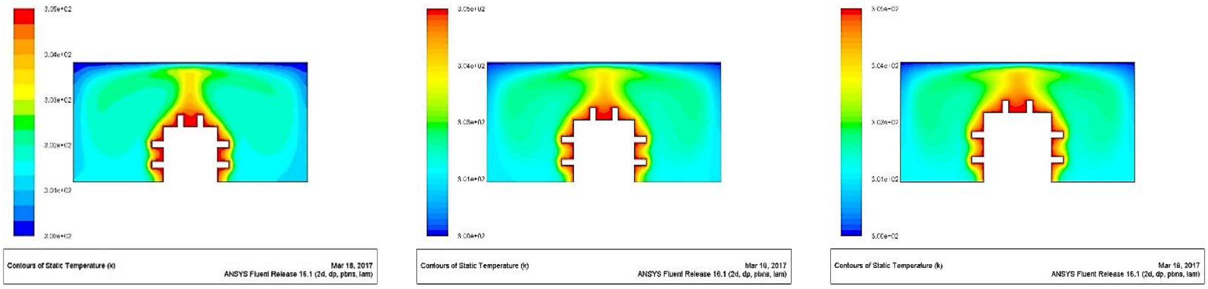


$46\ mm \times 46\ mm 52\ mm \times 52\ mm 58\ mm \times 58\ mm$

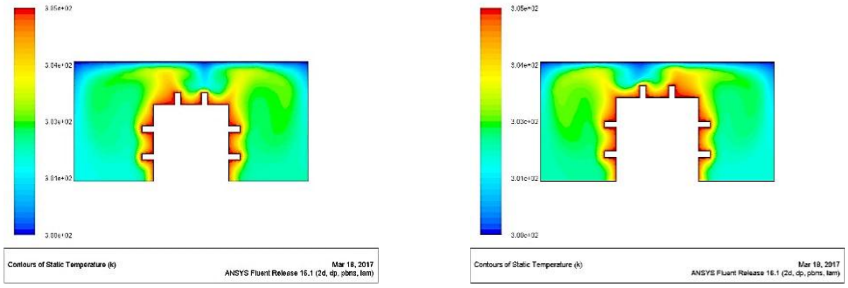


$64\ mm \times 64\ mm 70\ mm \times 70\ mm$

Figure 16. Isothermal line at $Ra = 40000$ and $Pr = 0.71$ with different object size.



$46\ mm \times 46\ mm 52\ mm \times 52\ mm 58\ mm \times 58\ mm$



$64\ mm \times 64\ mm 70\ mm \times 70\ mm$

Figure 17. Temperature contour at $Ra = 40000$ and $Pr = 0.71$ with different object size.

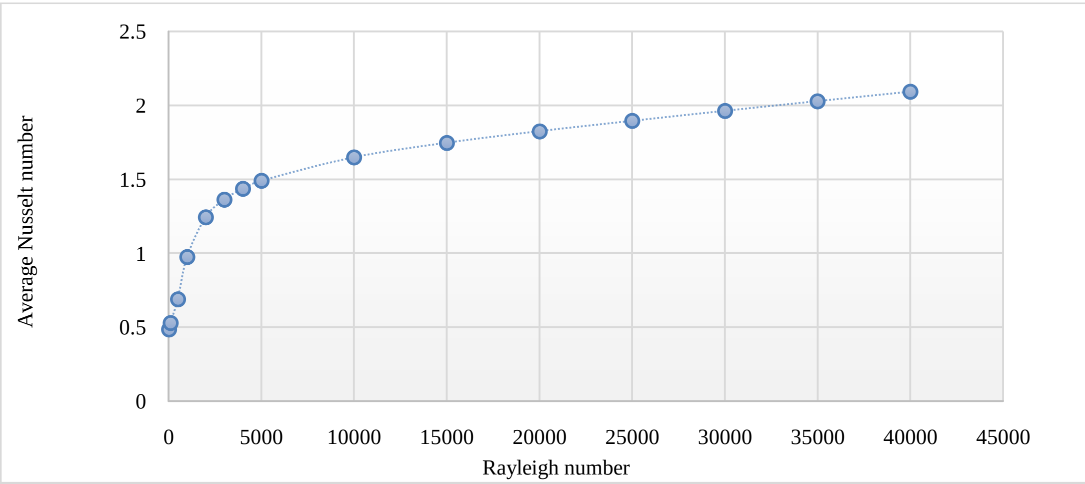
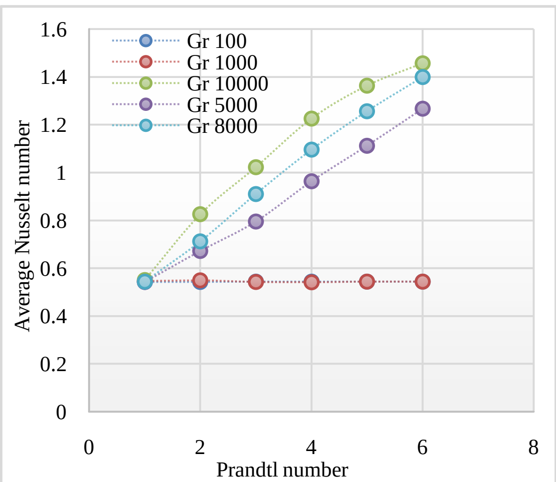


Figure 18. Average *Nu* vs. *Ra*



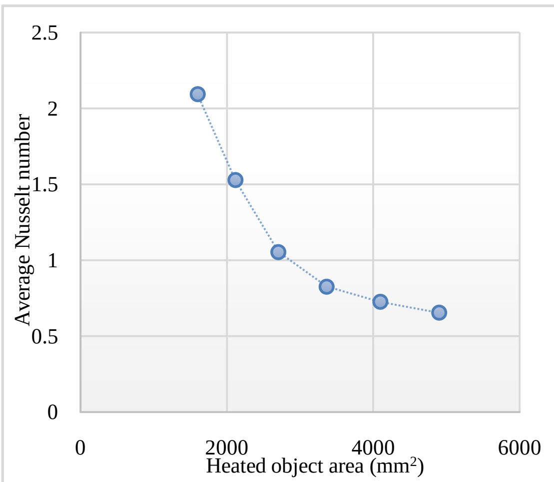


Figure 19. (a) Average *Nu* vs. *Pr* for different *Gr*, (b) Average *Nu* vs. Heated object area.

V. CONCLUSION

Laminar free convection in a two-dimensional channel with a heated object with fin placed on the bottom adiabatic wall is simulated numerically in this work. The investigation led to several conclusions. The flow of heat transfer reduced with the increase of object size and maximum heat transfer takes place in the lower left and right corner of the small object size. However, heat transfer increase with the increasing of *Ra* but remain same for the variation of *Pr* at lower *Gr*. On the contrary, heat transfer increase rapidly with the variation of *Pr* at higher *Gr*.

REFERENCES

[1]. Lee JRand Park IS. Numerical analysis for Prandtl number dependency on natural convection in an enclosure having a vertical thermal gradient with a square insulator inside. Nuclear Engineering and Technology 2012;44(3):283-296.

[2]. Sarris IE, Lekakis I, Valchos NS. Natural convection in a 2D enclosure with sinusoidal upper wall temperature. Numerical Heat Transfer-An International Journal of Computation and Methodology 2002;42(5):513-530.

[3]. Nimkar MP, Prayagi SV. Heat transfer by natural convection in two vertical one horizontal plate-an overview. International Journal of Engineering Science and Technology 2011;3(2):1008-1013.

[4]. Bakkas M, Amahmid A, Hasnaoui M. Numerical study of natural convection heat transfer in a horizontal channel provided with rectangular blocks releasing uniform heat flux and mounted on its lower wall. Energy Conversion and Management 2008; 49(10):2757-2766.

[5]. Zeitoun O, Ali M. Numerical investigation of natural convection around isothermal horizontal rectangular ducts. Numerical Heat Transfer-An International Journal of Computation and Methodology 2006;50(2):189-204.

[6]. Alami1 ME, Najam M, Semma E, Oubarra A, Penot F. Electronic components cooling by natural convection in horizontal channel with slots. Energy Conversion and Management 2005;46(17):2762-2772.

[7]. Boyalakuntla DS, Murthy JY, Amon CH. Computation of natural convection in channels with pin fins. IEEE Transaction on Components and Packaging Technologies 2004;27(1):138-146.

[8]. Singh S, Sharif MAR. Mixed convective cooling of a rectangular cavity with inlet and exit openings on differentially heated side walls. Numerical Heat Transfer-An International Journal of Computation and Methodology 2003;44(3):233-253.

[9]. Han CY, Baek SW. Natural convection phenomena affected by radiation in concentric and eccentric horizontal cylindrical annuli. Department of Aerospace Engineering, Korea Advanced Institute of Science and Technology, Numerical Heat Transfer, Part A 1999; 36:473-488.

M. M. Alam, et. al. "A Study on Free Convection Heat Transfer From a Heated Finned Object in a Channel." *International Journal of Engineering Science Invention (IJESI)*, Vol. 09(11), 2020, PP 25-40. Journal DOI- 10.35629/6734