CFD Analysis of Fluid Flow and Heat Transfer of a Triangle Object in a Rectangular Cavity

Anzaman Hossen, Anik Hira, Nazmus Sakib, Khandkar Aftab Hossain
Department of Mechanical Engineering
Khulna University of Engineering and Technology (KUET)
Khulna, Bangladesh

Abstract

This work is analyzed numerically on two dimensional, steady-state, forced convection heat transfer for Newtonian fluids in a rectangular enclosure having a heated equilateral triangular block (blockage of 20%) at center position which is maintained at constant wall temperature (T_w) and thermal conditions. The rectangular cavity is filled with fluid and the flow of fluid in the enclosure is initiated by the top wall moving in the positive x-direction when other walls remain stationary. The bottom wall and top wall both remain at the adiabatic condition and the other two walls remain at constant wall temperature (T_c) where ($T_w > T_c$). The dependence of average Nusselt number values on Reynolds number is to be developed and presented for its possible utilization in different engineering and design purpose. The dependence of various dimensionless parameters as Reynolds number, Prandtl number, and Nusselt number also has been analyzed. The result has been shown in form of Isotherm pattern and streamlines function for analyzing the heat transfer and fluid flow characteristics in between the cavity. A range of Reynolds number from 1 to 1000 at a difference of 100 has been analyzed for fixed Prandtl numbers 1, 50, and 100. It has been shown from the investigation that with the increase of Reynolds number average Nusselt number increase and then decrease after a certain Reynolds number for this kind of specific orientation.

Keywords

Forced Convection, Reynolds number, Fluid flow, CFD, Nusselt number

1. Introduction

The heat transfer phenomenon is one of the important things in most of the mechanical and electrical industries. Heat transfer rate has been examined in a different enclosures with different heated blocks in the past few years. Hydrodynamic and thermal characteristics are helpful in many fields such as electronic gadgets cooling systems. Micro-Electro-Mechanical Systems, furnace, building insulation, lubricating systems, food processing, solar heat collectors, etc. The enclosure of different shapes containing different types of blocks is used for controlling fluid flow due to convection. Thereafter enclosure containing a heated triangular block is not analyzed properly. The cooling of the mechanical instrument is an important topic. Without proper cooling, a device can be underworked or break down and the effective life can be decreased. So that proper cooling is needed for most of the devices.

In process industries such as heat exchanger problem mixed convection on the heated block has some importance. Cavity/enclosure with separate obstacles of varying shapes (circular, square, triangular, etc.) is used to control convection heat transfer. There are several applications where the fluid flow due to convection, such as crystal growth or solidification, should be limited. In such specific applications, the presence of an ideal form object (circular, square, or triangular) may be used to control the fluid flow. In addition, natural, forced or mixed convection heat transfer from heated triangular/cylindrical bodies has some particular application such as heat exchangers. A rectangular body is a bluff body that has more sharp edges than circular bodies, heat transfer and fluid flow characteristics are more significant in this type of enclosure. Compared to other cases triangular body has more impact on heat transfer characteristics. This report has been done by numerical analysis in forced convection characteristics with a heated triangular block at the center of the enclosure with constant wall temperature thermal conditions for a variety of Reynolds number and Prandtl number. The flow in a rectangular cavity is generally laminar because the flow velocity of the fluid is very low. Heat transfer rate in an enclosed rectangular cavity can be increased by increasing the velocity within the limit of laminar flow. Many internal arrangements can also increase the rate of heat transfer such as baffle or internal fin etc.

1.1 Flow Phenomena

Flow phenomena of the rectangular enclosure with a triangular object in the middle of the enclosure have of great importance in Heat transfer engineering problems as well as in Fluid mechanics. In practice, fluid flow is always affected to some extent by any solid boundaries. However, here the only laminar flow is considered.

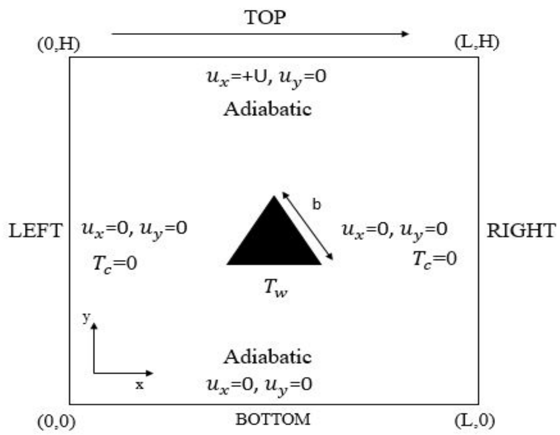


Fig 1: Rectangular cavity with a triangular object in the middle

In this case, the fluid is filled in the rectangular enclosure and the flow of the fluid is initiated by the moving of the upper wall translation in the positive x-direction ($U_x = +U$). While the other wall remains stationary as they don't move. The aspect ratio of the rectangular cavity is maintained at $A_R = \frac{L}{H} = 1$ (Where L and H are cross-sectional lengths of the cavity as in fig: 1.1.1). There is a triangular blockage at the middle of the rectangular enclosure. The blockage percentage is about 20% of the total rectangular enclosure. The working fluid is considered as Newtonian fluid. The viscous dissipation effects and radiation heat transfer are neglected.

1.2 Objectives

The main objectives of this analysis are:

- Investigating the effect of Reynolds number on Nusselt number
- Investigating Streamline patterns for different Reynolds number, Prandtl number.
- Investigating Isotherm patterns for different Reynolds number, Prandtl number.

2. Literature Review

Yua et al(2010). [1] performed an investigation of numerical analysis on a cylindrical enclosure containing an inner co-axial triangular block and showed that for low Prandtl number (about 0.03) heat transfer characteristics are unique and more than .07 they are almost independent. Braga et al (2005), [2] investigated natural convection in a square enclosure filled with several circular and square rods. Heat transfer was compared with two different types of geometry across a square cavity partially filled with a fixed amount of a conducting solid shape. They solved momentum and energy equations numerically for heat transfer problems in both the solid and the void space. Prasad et al.(1996) [3] considered both natural and forced convection heat transfer in a deep lid-driven cavity and found Gr/Re^2 ratio about .1 to 1000 variation. They also considered Reynolds number varied from 0 to 2000 and Grashof number varied from $10^7$to $5*10^9$. The corresponding range of these dimensionless parameters showed the relative strength of the two convection in a mixed environment. Lee et al. (2006) [4] performed an investigation on Numerical simulation of a heat-generating conducting body in a rectangular cavity. Here thermal conductivity k ratios vary from 0.1 to 50, Rayleigh numbers in the range of 10^3≤Re≤ 10^6. Oztopa et al.(2009) [5] observed a mixed convection phenomenon on rectangular enclosure having a circular body and showed that it is influenced by three different thermal boundary conditions for the circular body as isothermal, conductive, and adiabatic. Billah et al.(2011) [6] found that the circular hollow cylinder can be used as a control parameter for heat transfer, fluid flow, and temperature distribution in a lid-driven cavity having a heated circular hollow cylinder. Malleswaran etal,(2016) [7] performed a numerical investigation on mixed convection on the lid-driven cavity with a corner heater. They presented the effect of the average Nusselt number with the Ricardson number and found that the average Nusselt number increases with an increase of Ricardson number. It has also been observed that a cavity with corner heaters is completely different from the differentially heated cavity in which the thermal boundary layer occurs near both hot and cold walls whereas no such boundary layer exists in the cavity with corner heaters at forced convection. Nemati et al.(2010) [8] investigated thermal and hydrodynamic characteristics for different Reynolds number and different volume fractions of nanoparticles in terms of streamline and isothermal patterns in a lid-driven cavity. Sharif (2007), [9] considered the inclination of the cavity from 0° to 30° for thermal characteristics in shallow inclined drieven cavity and observed that average Nusselt number has been increased with the increase of inclination. Khanafer et al.(2013), [10] found a less variety of Richardson number and Reynolds number but a great variety of streamline pattern and isotherm patterns for the different thermal conditions in a lid-driven cavity with a circular cylinder. It is also shown that the average Nusselt number increase with the increase of the Richardson number for any kind of shape of the cylinder. But average Nusselt number didn't increase with the increase of the shape of the non-dimensional radius of the cylinder for a particular Richardson number. Saidi et al. (2012), [11] investigated the vortex formation characteristics around a rectangular block placed at different places in between the rectangular cavity and found that if the block is placed at the middle position of the rectangular cavity it shows a great impact of the vortex. Gangawane et al.(2017), [12] performed a numerical investigation on mixed convection characteristics in a lid-driven cavity containing heated triangular block and showed the result as isotherm and streamline pattern. Moreover, it is shown that for a given Prandtl number average Nusselt number has been increased before a certain range then it started to decrease sharply. It shows a range of Reynolds number from 1 to 1000 and Prandtl number 1, 50 and 100 and Grashof number 0 to 10^5. Sahu and Singh et al. (2013), [13] investigated transfer and flow due to natural convection in air around heated triangular cylinders of different sizes inside a square with Rayleigh number varied from 10^4 to 10^6 and the result has been presented as an isotherm pattern and streamline function. Furthermore, it has been shown that the fluid motion and circulation rate increase with an increase in enclosure size. The fluid motion is almost uniform for lower Ra (smaller enclosure size) and the fluid motion is prominent near the walls and the fluid is almost stagnant in the core region for higher Ra.

3.1 Methodology

The general momentum equation is also called the equation of motion or the Navier-Stokes equation; in addition, the equation of continuity is frequently used in conjunction with the momentum equation. The equation of continuity is developed simply by applying the law of conservation of mass to a small volume element within flowing fluid. The form of continuity equation or the conservation of mass equation is:

$$\frac{\partial U_x}{\partial x} + \frac{\partial U_y}{\partial y} = 0$$

The entire domain is solved with a single momentum equation and the resulting velocity field is shared between the phases. The following momentum equations are:

X-momentum equation:

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

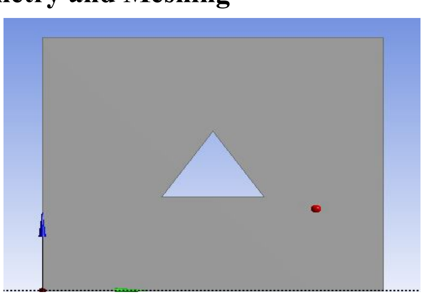
Y-momentum equation:

$$U_x \frac{\partial U_y}{\partial y} + U_y \frac{\partial U_y}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + \vartheta \left(\frac{\partial Ux^2}{\partial^2 y} + \frac{\partial Uy^2}{\partial^2 y}\right)$$

The energy conservation equation is written as:

$$U_x \frac{\partial T}{\partial x} + U_y \frac{\partial T}{\partial y} = \frac{k}{\rho C_p}\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right)$$

3.2 Geometry and Meshing



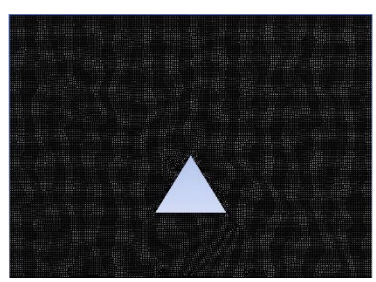


Fig 2: Geometry and Meshing

3.3 Boundary Condition

-At the top wall, the velocity component, v is zero and u has some magnitude in the positive x-direction so that velocity in terms of stream function can be written as, u=$\frac{\delta\Psi}{\delta y}$ and v=$-\frac{\delta\Psi}{\delta x}$. And the top wall is at adiabatic condition so, $\frac{\delta T}{\delta y}$=0

-At the bottom wall, both velocity component at the bottom wall is zero so that it can be written that, u=0 and v=0. The bottom wall is at adiabatic condition so that, $\frac{\delta T}{\delta y}$=0

-At the Left wall, both velocity component at the left wall is zero so that it can be written that, u=0 and v=0.

-At the right wall, both velocity component at the right wall is zero so that it can be written that, u=0 and v=0

-At the Triangular wall, the velocity component at the triangular wall is zero so that it can be written as u=0 and v=0. The triangular wall is at constant wall temperature condition.

3.4 Numerical Methodology:

Finite volume method is used to discretize the governing partial differential equations to yield a set of linear algebraic equations. The field governing equations are solved using the ANSYS FLUENT version 16.2 commercial CFD solver. First, the unstructured 'triangular' cells of uniform grid spacing were generated. Then for better mesh quality "Quadrilateral" mesh has been generated and test the result with the previous result with gives a better result. The well-known semi-implicit method for the pressure-linked equations (SIMPLE) scheme was used to solve the pressure-velocity decoupling. The system of algebraic equations is solved by using the Green-Gauss cell-based iterative method has been used to solve the iteration to meet desired convergence criteria. The momentum and energy terms are discretized using third-order accurate QUICK (Quadratic Upwind Interpolation for Convection Kinetics).

3.5 Validation

A validation has been done with the paper of reference [12], and the proposed result has been shown below fig: 3.5.1(pr=1) and fig:3.5.2 (pr=50). There is a slight difference between the two graphs of average Nusselt number vs Reynolds number

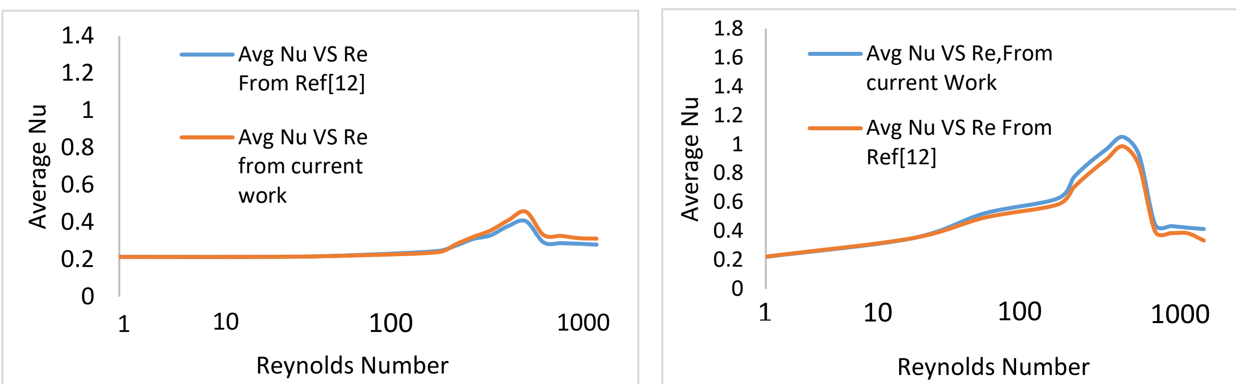


Fig 3: Validation with reference Paper

4. Result and Discussion

The forced convection in a square cavity with a moving upper wall has been investigated numerically. The effects of the heated triangular block for blockages about 20% on the convection characteristics have been analyzed limiting thermal conditions of CWT. Reynolds number is varied as Re=1 to 1000 at a difference of 100 corresponding to laminar range. The Prandtl number is varied as Pr=1, 50 and 100 (thus encompassing the fluids ranging from air to engine oil). Results encompassing the influences of mentioned parameters on the heat transfer characteristics (such as streamline, isotherm contours, average Nusselt numbers, etc.) are presented and discussed here in this section.

The governing non-dimensional partial differential equations along with the boundary conditions have been solved using numerically by using Ansys fluent 16.2 to get stream function and temperatures at every internal grid points in the computational domain. The length and height of the enclosure have been chosen 20mm and 20mm so the aspect ratio defined by L/H of the flow domain is 1.0.

Figures 4 and 5 show a different isotherm contour at different Reynolds number and Prandtl number. The change of heat transfer characteristics with the change of the Reynolds number and Prandtl number has been shown.

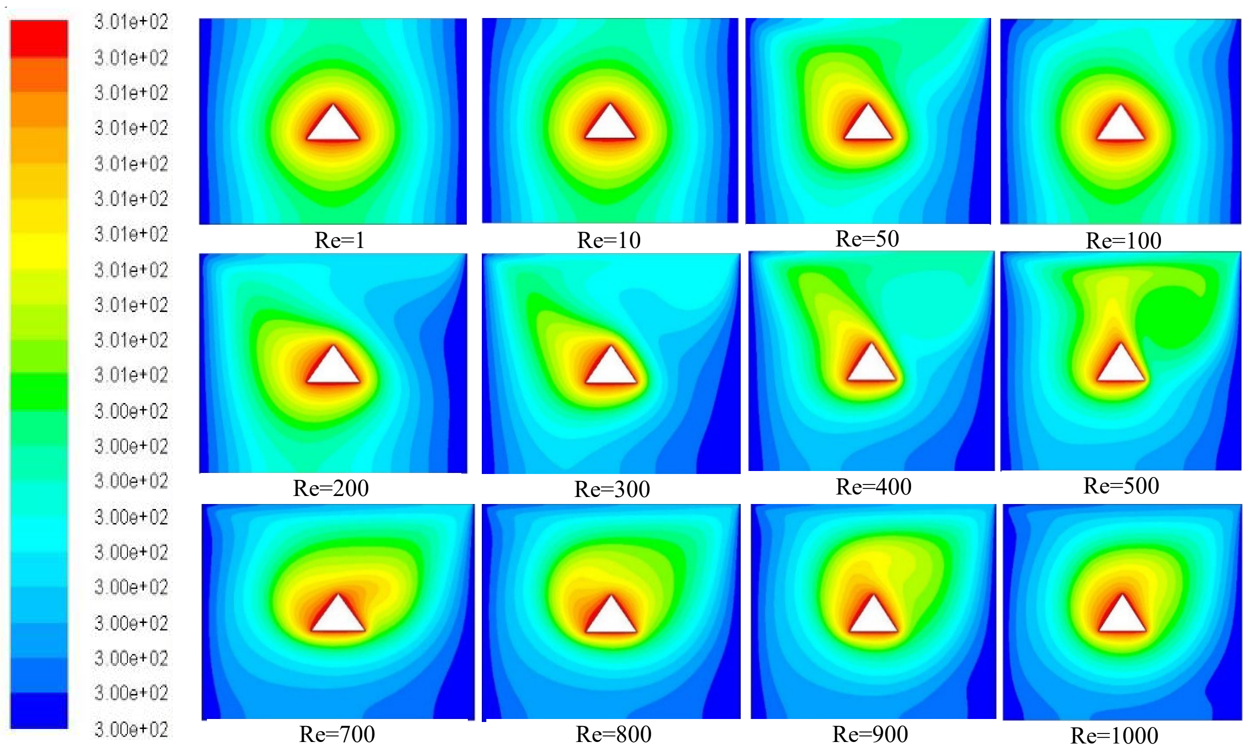


Fig 4: Isotherm Pattern for Prandtl number=1

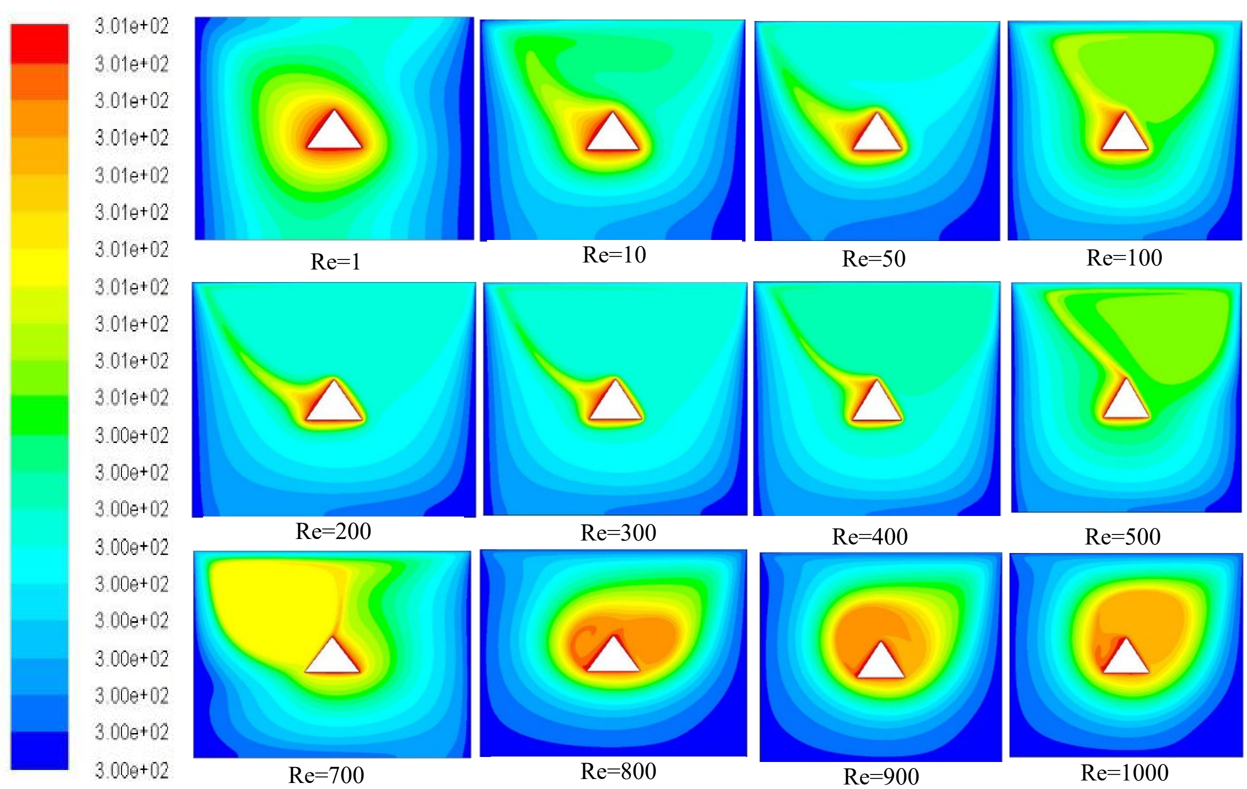


Fig 5: Isotherm Pattern for Prandtl number=50

From fig 4, at Re=1 and Pr=1, the isotherms are distributed in parallel to vertical walls. With increasing in Reynolds number in 10, the isotherms changed a little bit from the vertical position and inclined a little on the right side. When the Reynolds number increased to 100 isotherms started to incline slightly in the direction of the lid. As Reynolds number increased to 200 isotherms started changing rapidly in the direction of the lid. When Reynolds number increases to a value of 300, in this case, isotherm change more than the previous and now the isotherms are not exactly vertical to the walls. So that heat started dissipating from the hot triangular object. After that, when Reynolds number reached 500, the isotherm has shown a different nature. The cause is the decreased heat transfer sharply after Re=500. When the Reynolds number reached 700 isotherms started distributed circularly as the velocity of the lid is quite high. at Reynolds number 800 heat transfer rate has been decreased but the heat has been dissipated continuously and the isotherms are more circular than before. At Re=1000, as the fluid circulation strength increase, the isotherms are originated to distributed circularly along the block.

When fluid viscosity has been increased and the Prandtl number increases to a value of 50 then the inclined isotherms are observed even for lower Reynolds number values. In fig 5, for lower Reynolds number 1, the isotherms are not exactly parallel to vertical walls as observed for Reynolds number 1 and Prandtl number 1. This phenomenon is due to higher Pr fluids having higher viscosity and such fluids are easily displaced due to moving lid. Also, in pure forced convection example block works as an obstruction for the fluid circulation in the cavity. Consequently, as fluid rotates in a clockwise manner, a fluid plume has formed on the left side of the block. Because of the viscosity effect, the plume is reported to be slightly inclined. At Reynolds number 10 the isotherms changes a lot and start to form circulation which is observed for Prandtl number 1 at a higher Reynolds number. When the Reynolds number increases to 100 heat transfer rate has been increased to a great level and isotherms are now changed a lot. When the Reynolds number increases to 200 and 300, in this case, isotherm patterns are more stable than before and almost parallel to the vertical walls. When the Reynolds number increases to 500 heat transfer started decreasing sharply. But heat dissipation rate increased due to higher velocity. With the increasing velocity at Reynolds, at 600 isotherms are inclined to lid and formed a circulation around the triangular object. At Reynolds number 700, the isotherms made a circulation and the circulation became more intensified with the augmentation of Reynolds number until 1000.

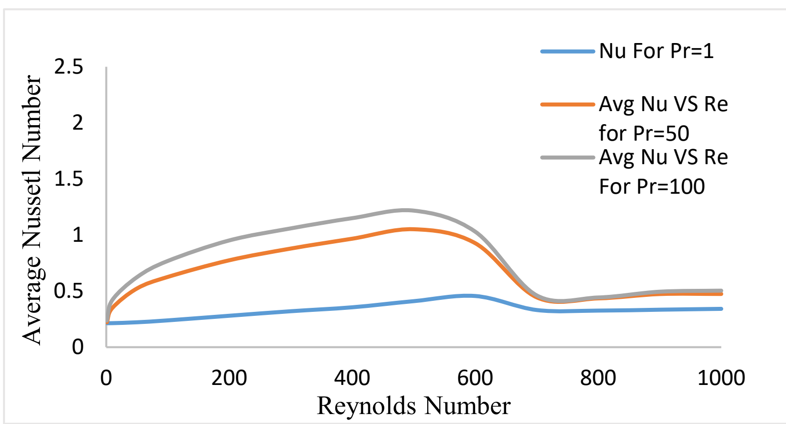


Fig 6: Average Nu vs Re for Different Pr

In fig 6, a graph has been plotted between average Nusselt number vs Reynolds number for three different Prandtl number. From this graph and the isotherm pattern, it has been clearly stated that with the increasing Prandtl number the average Nusselt number has been increased continuously. So Prandtl number has a great impact on Reynolds number and average Nusselt number. With the increase of the Prandtl number the average Nusselt number also has been increased.

From the above isotherm patterns and average Nusselt number versus Reynolds number curve, it is clearly stated that heat transfer rate is increasing with the increasing Prandtl number and the heat transfer rate has been increased until a certain Reynolds number then started to decrease dramatically when the triangular object is placed at the center of the rectangular cavity. The point after which the average Nusselt number started to decrease is termed as "Critical Reynolds Number" [12]. The heat transfer augmentation is observed before Re_{cr}, after which deterioration or no significant change is observed in heat transfer rate.

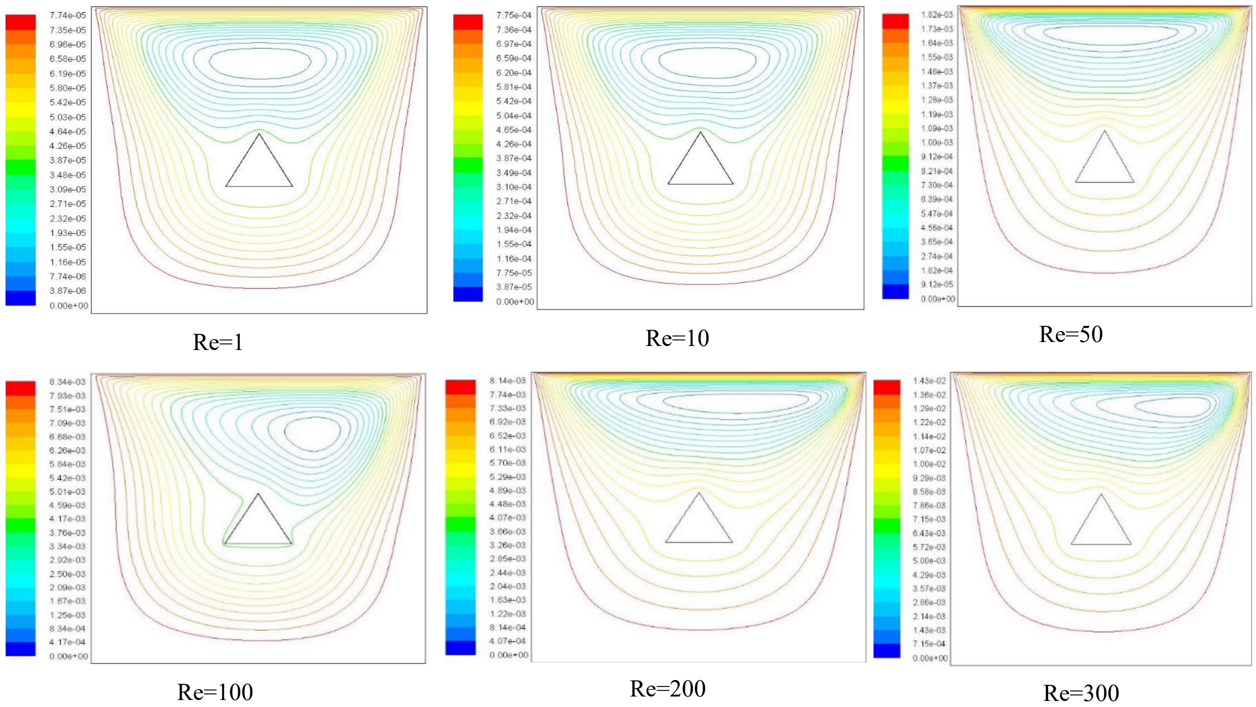


Fig 7: Streamline Patterns at Different Re and Pr=1

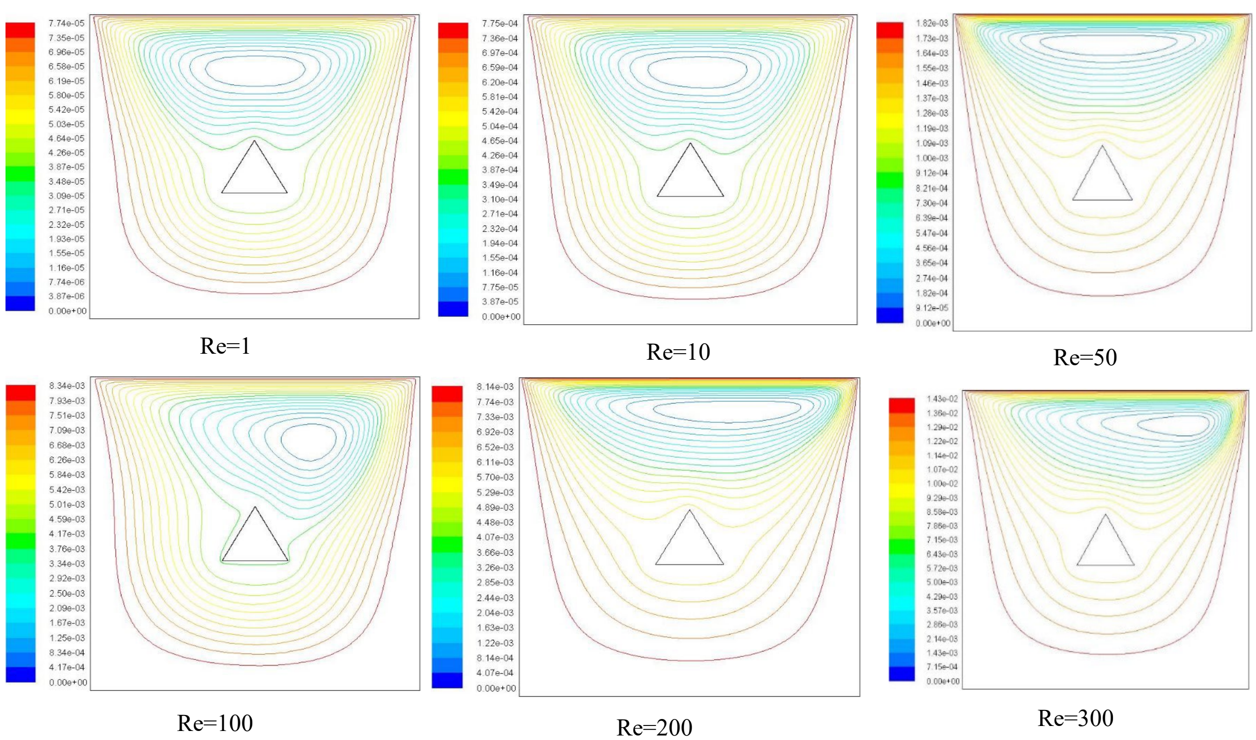


Fig 8: Streamline Patterns at Different Re and Pr=50

Fig 7 and 8 shown streamline pattern for different Reynolds number and Prandtl number. These figures represent the streamline changes for the pure forced convection problem and other significant conditions of Re and Pr and specify the impact of moving wall on fluid circulation in the cavity. For Reynolds number 1 (mostly lower inertial forces), high-velocity fluid gathers under moving wall and above block. This high-velocity fluid zone is denoted as a convection cell. As fluid velocity started increasing (Re=10) from figure fluid below the moving lid has not changed significantly. When the Reynolds number is raised to 50 from the convection cell has been pulled a little bit in the direction of the lid. At Reynolds number 100 fluid below the moving lid has been observed to be pulled towards the direction of the lid. It moves the convection cell towards the right top of the cavity in the direction of the moving lid. A small vortex has been formed because of the circulation of fluid around the rectangular cavity. Next due to increased velocity, the circulation has been increased and the vortex moved in the rightward direction more. With the increase of the viscosity of the fluid Prandtl number also has been increased. A very Limited influence (Fig: 8) is found only on the size of convection cell for Prandtl number 50. Even the observations are so unnoticeable.

In low Reynolds number, fluid circulation has not started but this barrier can be overcome by the higher velocity of the fluid. Because of the increasing Reynolds number, the fluid velocity increases and fluid started circulation around the triangular object and. This barrier also can be overcome by moving the triangular object at a different position in the y-direction and increasing the size of the triangular object.

5. Conclusion

A numerical investigation of laminar forced convection of triangular object in rectangular surface with top surface moving in the positive x-direction and constant wall temperature of the triangular object is performed. From the above discussion and graph, it also has been found that with the rising Reynolds number average Nusselt number also increases as so does the heat transfer. With the rising viscosity of the fluid ($1 \le Pr \le 50$), Reynolds number ($1 \le Re \le 1000$), and velocity isotherms patterns and streamlines change simultaneously. So, heat transfer and fluid flow phenomena are directly related to the Reynolds number and Prandtl number. As Re=80-500 is considered as critical Reynolds number range (Re_{cr}), this work can be enhanced by removing the critical Reynolds number barrier as well as by increasing the heat transfer rate by different methods. There will also be a possibility in the future to analyze heat transfer and fluid flow characteristics by changing the position of the triangular object and increasing the size of the triangular object and analyzing the change with current observation.

References

Zi-Tao Yua, Li-Wu Fanb, Ya-Cai Hua, Ke-Fa Cen, "Prandtl number dependence of laminar natural convection heat transfer in a horizontal cylindrical enclosure with an inner coaxial triangular cylinder", *International Journal of Heat and Mass Transfer* 53 (2010) 1333–1340.

Edimilson J. Braga, Marcelo J.S. de Lemos, "Laminar natural convection in cavities filled with circular and square rods" *International Communications in Heat and Mass Transfer* 32 (2005) 1289–1297.

Ajay K. Prasad, Jeffrey R. Koseff, "Combined forced and natural convection heat transfer in a deep lid-driven cavity flow" *Int. J. Heat and Fluid Flow*, Vol. 17, No. 5, October 1996.

Jae Ryong Lee, Man Yeong Ha, "Numerical simulation of natural convection in a horizontal enclosure with a heat -generating conducting body" *International Journal of Heat and Mass Transfer* 49 (2006) 2684–2702.

Hakan F. Oztopa, Zepu Zhaob, BoYu, "Fluid flow due to combined convection in lid-driven enclosure having a circular body" *International Journal of Heat and Fluid Flow* 30 (2009) 886–901.

M.M. Billah, M.M. Rahman, Uddin M. Sharif, N.A. Rahim, R. Saidur, M. Hasanuzzaman, "Numerical Simulation on Buoyancy-driven Heat Transfer Enhancement of Nanofluids in an Inclined Triangular Enclosure" *International Communications in Heat and Mass Transfer* 38 (2011) 1093–1103

A. Malleswaran and S. Sivasankaran, "A Numerical Simulation on MHD Mixed Convection in a Lid-driven Cavity with Corner Heaters" *JAFM*, Vol. 9, No. 1, pp. 311-319, 2016.

H. Nemati, M. Farhadi, K. Sedighi, E. Fattahi, A.A.R. Darzi, "Lattice Boltzmann simulation of nanofluid in lid-driven cavity" *International Communications in Heat and Mass Transfer* 37 (2010) 1528–1534.

M.A.R. Sharif, "Laminar mixed convection in shallow inclined driven cavities with hot moving lid on top and cooled from bottom" *Applied Thermal Engineering* 27 (2007) 1036–1042.

Khalil Khanafer, S.M. Aithal, "Laminar mixed convection flow and heat transfer characteristics in a lid driven cavity with a circular cylinder" *International Journal of Heat and Mass Transfer* 66 (2013) 200–209.

Maysam Saidi, Hassan Basirat Tabrizi, Reza Maddahian, "Vortex Formation in Lid-driven Cavity with Disturbance Block" *World Academy of Science, Engineering and Technology* 69 2012.

Krunal M. Gangawane, B. Manikandan, "Mixed convection characteristics in lid-driven cavity containing heated triangular block" *International Journal of Heat and Mass Transfer*, 2017, doi:10.1016/j.cjche.2017.03.009.

K. B. Sahu, Ravi Kumar Singh, "Analysis of Heat Transfer and Flow Due to Natural Convection in Air Around Heated Triangular Cylinders of Different Sizes Inside a Square Enclosure" *10th International Conference on Mechanical Engineering, ICME* 2013.

Biographies

Anzaman Hossen graduated in Mechanical Engineering from Khulna University of Engineering and Technology (KUET), Khulna, Bangladesh.

Anika Hira graduated in Mechanical Engineering from Khulna University of Engineering and Technology (KUET), Khulna, Bangladesh.

Nazmus Sakib graduated in Mechanical Engineering from Khulna University of Engineering and Technology (KUET), Khulna, Bangladesh.

Dr. Khandkar Aftab Hossain is a Professor in the Department of Mechanical Engineering at Khulna University of Engineering and Technology (KUET), Khulna, Bangladesh.