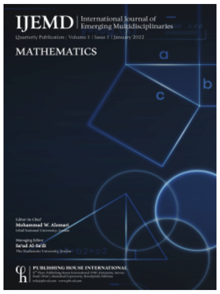
IJEMD-M, 1 (2) (2022) 49- 57 https://doi.org/10.54938/ijemdm.2022.01.2.28

International Journal of Emerging Multidiciplinaries: Mathematics

Research Paper
Journal Homepage: www.ijemd.com
ISSN: 2790-1998 (print), 2790-3257 (online)



Numerical Investigation of Viscous Fluid Flow and Heat Transfer in the Closed Configuration Installed with Baffles

Afraz Hussain Majeed[1*], Aqsa Afzal[1] and Rashid Mahmood[1]

[1]*Department of Mathematics, Air University, PAF Complex E-9, Islamabad 44000, Pakistan*
**Corresponding author*

Abstract

In this study, the flow and heat transfer of viscous fluid features inside the closed configuration with a heated baffles are investigated. Due to the non-linearity of the model, the numerical approach is adopted to get the solution. Initially, the governing equations were discretized in the 2D domain using the Finite Element Method (FEM). We employ a stable finite element pair $\mathbb{P}_2 - \mathbb{P}_1$ for the velocity and pressure approximations while temperature approximation is done using the space of linear polynomial as $\mathbb{P}_1$. To improve accuracy, a hybrid mesh is built at a coarse level, then the grid refinement level is increased. The baffle gap (B.g) is varied from 0.2 to 0.6 and three Reynolds numbers are chosen for this investigation: $Re = 5,10,20$. The Grashof number is fixed in all the cases to $Gr = 1000$. The major findings of the study are shown using velocity profile, streamlines, and isotherms. Moreover, the kinetic energy and average Nusselt number (Nu_{avg}) for various values of parameters are also calculated. Furthermore, by increasing the baffle gap, the thermal flow regime raises showing more fluid gets heated inside the cavity.

Keywords: Flow and heat transfer; Square cavity; Hybrid mesh; FEM computation; Heated baffles;
2010 Mathematics Subject Classification: 80A20; 37N10; 78M10.

1. Introduction

Studying the behavior of the fluid inside some closed domain is an important phenomenon and has various industrial applications. In a fluid-filled container, due to the investigation of heat and transfer of mass by

Email addresses: chafrazhussain@gmail.com **(A.H.Majeed),** aqsa96afzal@gmail.com **(A.Afzal),** rashid.mahmood@mail.au.edu.pk **(R.Mahmood)**

natural convection, it has a wide scope in material assembling frameworks, cryogenic businesses, cooling problems, crystal developing industries, and designing fields like chemical, mechanical, aeronautical, and so on. In this chapter, the driven cavity flow problem is discussed by considering heated baffles inside the domain. The flow is generated by moving both the vertical walls with some velocity. The first numerical investigation for the flow in the driven cavity is presented by Kawaguti [1] who performed simulations for creeping as well as inertial flows in the cavity with different aspect ratios. In view of the aforementioned highly practical utilization researchers have shown persuasive focus towards the analysis of convective heat transfer phenomenon like Eckert et al. [2] examined heat transfer through convection in a rectangular enclosure. Gibanov and Sheremet [3] also observed natural convection in a closed domain of various geometries consisting of a heat source to provide temperature difference in the non-Newtonian fluid. Mahalakshmi et al. [4] numerically investigated the behavior of a shear-dependent fluid contained in a closed region with a heat source at the center of the cavity. J. Buongiorno [5] produced an analytical model for nanofluids by incorporating Brownian diffusion and established an argument in unusual convective heat transfer. Li et al [6] made an experimental study to identify the key mechanisms affecting the transient boiling heat transfer coefficient. Their results showed that the heat transfer coefficient was improved due to the increased thermal conductivity of the nano-suspension. Yadav et al. [7] investigated the onset of magnetic field on heat transfer enhancement of nanofluid flow over non-linear surface. Poulikatos et al. [8] derived actual source for generation of natural convection in flow domain i.e. gravity aspects and also determined different ranges about Rayleigh number which controls convection process.

Flow analysis is both fascinating and instigating. There are many technological applications and scientific challenges, but more numerical and experimental research is needed in a wide range of fields. The viscous fluid flows over solid surfaces or within ducts and channels are the focus of frictional effects and heat exchange. To account for these effects, the fluid properties must be evaluated. The problem can be solved experimentally or theoretically. Various proposals have been made over the years to solve incompressible Newtonian flow problems using numerical methods such as finite differences, finite element, and finite volume schemes. The most well-known method is the classic PARADISO algorithm, which introduces the stalled grid concept when combined with the pressure correction equation. Various 2-D flow configurations were successfully resolved using this method. With the invention of fast, multi-processor personal computers, including simple computer cluster management, three-dimensional numerical simulation of a variety of physical issues seems to have become widespread in recent decades. It was noted that information on non-Newtonian fluid flows, particularly polymer melts in the molding process, is still scarce in the literature [9-13].

In this study, the viscous fluid in a square cavity having heated baffles is studied by considering the different gaps values. The paper is organized as follows: In section 2 problem statement and its mathematical formulation is defined. In section 3 and 4 mathematical modelling and numerical procedure are proposed. In section 5 outcomes of the problem are summed up. In the last section conclusion based on the results and discussion, a section is drawn.

2. Problem Statement and Mathematical Formulation

Considers a square cavity with an insulated baffle connected to every one of its horizontal walls at a symmetric position. The baffles with constant temperature, are connected to horizontal walls at positions in the center as shown in the Fig. 1. The horizontal walls of the cavity are at no-slip condition while the vertical walls are moving with velocity v = 1. The baffles are fixed with horizontal walls with equal distance from the center. All the walls are at zero temperature while the baffles are heated with temperature T = 1. The non-dimensional baffles length taken in this research is 0.75, while the width is 0.02. It was tracked down that the baffles fundamentally impact the heat transfer rate and the average Nusselt numbers decreased to roughly 15% compared with the nonpartitioned cavity.

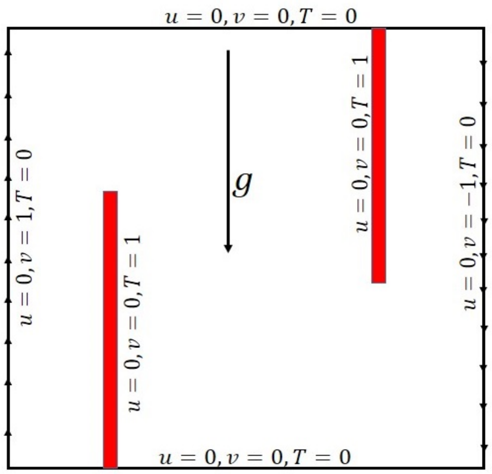


Fig. 1: Schematic diagram of the problem

3. Mathematical modeling

For a two-dimensional, steady and incompressible non-dimensional governing equations Eqs.(2.1)-(2.4) defined below (see ref. [9])

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{2.1}$$

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + \frac{\partial p}{\partial x} = \frac{1}{Re}\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right), \tag{2.2}$$

$$u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + \frac{\partial p}{\partial y} = \frac{1}{Re}\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) + \frac{Gr}{Re^2}\theta, \tag{2.3}$$

$$u\frac{\partial \theta}{\partial x} + v\frac{\partial \theta}{\partial y} = \frac{1}{PrRe}\left(\frac{\partial^2 \theta}{\partial x^2} + \frac{\partial^2 \theta}{\partial y^2}\right), \tag{2.4}$$

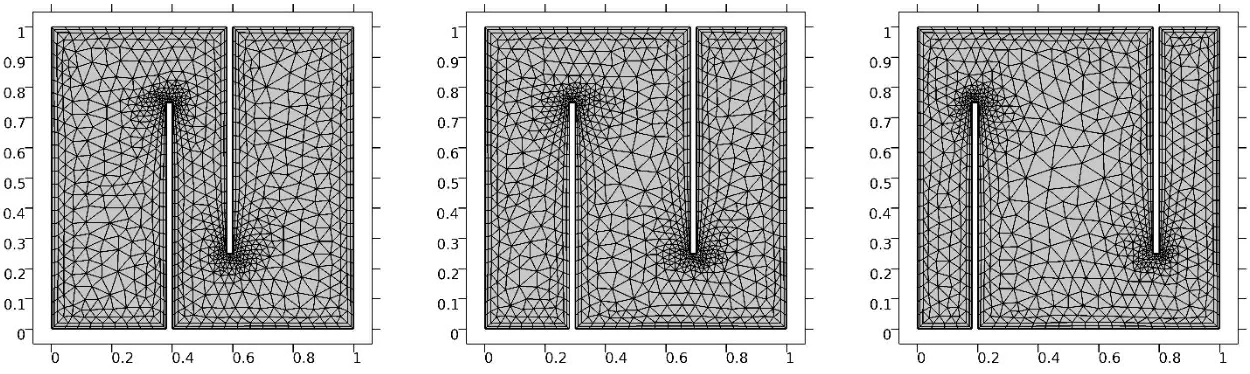
The appropriate boundary conditions for the given problem are described as

- Horizontal walls are kept cold with no-slip
- Vertical walls are cold with $u = 0, v = 1$ (left) while $u = 0, v = -1$.
- On baffles $T = 1$ with $u = v = 0$

This research is conducted by taking three different baffles gaps. For baffle gap 0.2 the x coordinate of center of left baffle is x = 0.38 and for right baffle it is x = 0.58. For baffle gap 0.4 the x coordinate of center of left baffle is x = 0.28 and for right baffle it is x = 0.68 and for baffle gap 0.6 the x coordinate of center of left baffle is x = 0.18 and for right baffle it is x = 0.78

4. Numerical procedure

It is obvious that by increasing the refinement levels the number of elements inside the domain will increase and computations will require more time. The total number of elements and degree of freedoms involved in different refinement levels for three different baffles gaps are listed in Table 1. Fig. 2 is the demonstration of domain discretization at the Coarse level for three different baffle gaps



B.g = 0.2 B.g = 0.4 B.g = 0.6

Fig. 2: Coarse refinement level mesh for various baffle gap ratio.

Table 1: Degree of freedom and number of elements for different refinement levels.

Refinement Level	$B.g = 0.2$		$B.g = 0.4$		$B.g = 0.6$	
	$\#EL$	$\#DOF$	$\#EL$	$\#DOF$	$\#EL$	$\#DOF$
Extremely Coarse	590	3609	622	3,769	628	3,835
Extra Coarse	820	4957	812	4,908	788	4,788
Coarser	1,182	6,983	1,160	6,855	1,148	6,813
Coarse	2,064	11,886	2,050	11,798	1,946	11,278
Normal	2,938	16,670	3,004	17,000	2,918	16,570
Fine	4,454	24,763	4,504	24,986	4,424	24,595
Finer	12,022	65,285	12,228	66,315	11,974	65,045
Extra Fine	31,490	167,431	31,906	169,511	30,924	164,601
Extremely Fine	38,252	201,241	38,424	202,101	38,230	201,131

5. Results and Discussions

In Fig. 3, the influence of different baffle gaps and different Reynolds numbers on velocity profiles can be seen. From left to right, there are three different Reynolds numbers i.e. $Re = 5,10,20$ in increasing order while from top to bottom there are three increasing baffle gaps i.e. B.g$= 0.2, 0.4$and0.6. The Gr number is kept constant at $Gr = 1000$ while performing these simulations. It can be observed that by increasing baffle gap, boundary layer thickness decreases in velocity profiles, the velocity is starting shifting between the baffles.

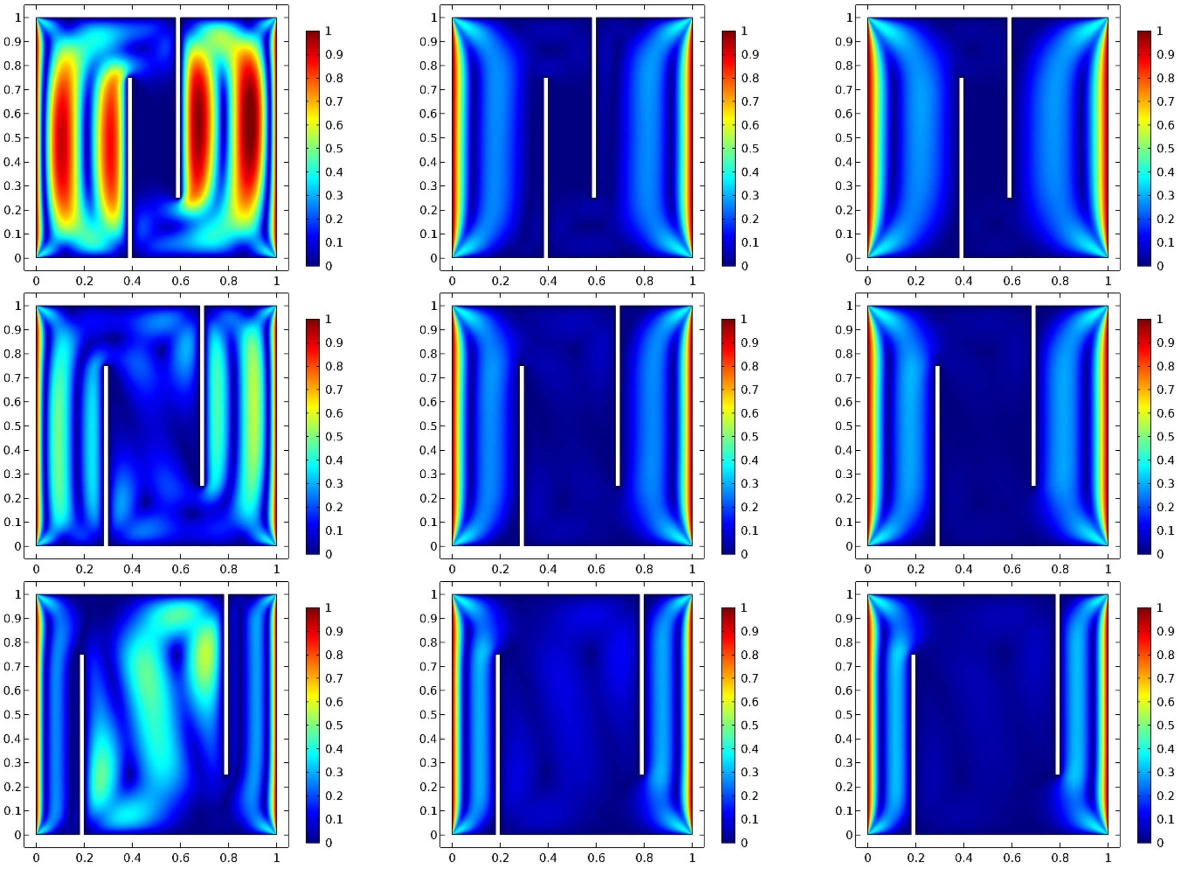
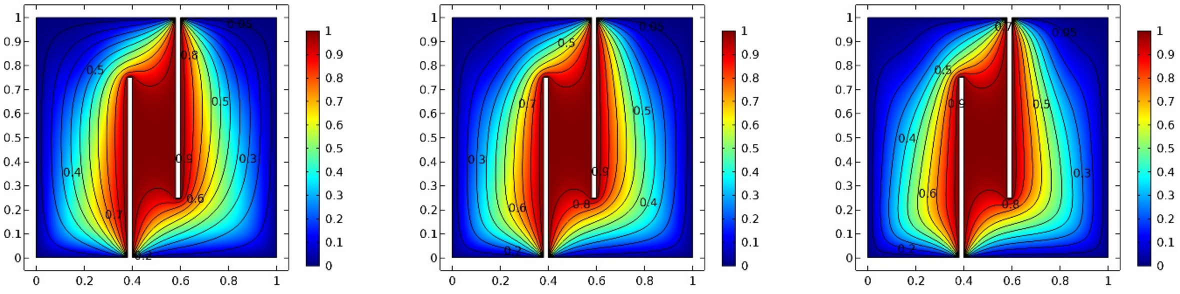


Fig. 3: Influence on velocity streamline with baffle gap ratio $B.g = 0.2, 0.4, 0.6$ from top to bottom for Re=1,5,10 from left to right with constant $Gr = 1000$.

In Fin. 4, the influence of different baffle gaps and different Reynolds numbers on isotherm contours of temperature distribution inside the cavity can be seen. From left to right, there are three different Reynolds numbers i.e. $Re = 5, 10, 20$ in increasing order while from top to bottom there are three increasing baffle gaps i.e. B.g = 0.2,0.4 and 0.6. The Prandtl number is kept constant at $Pr = 5$ while performing these simulations. It can be observed that by increasing baffle gap, the heat transfer region increases showing more fluid gets heated inside the cavity



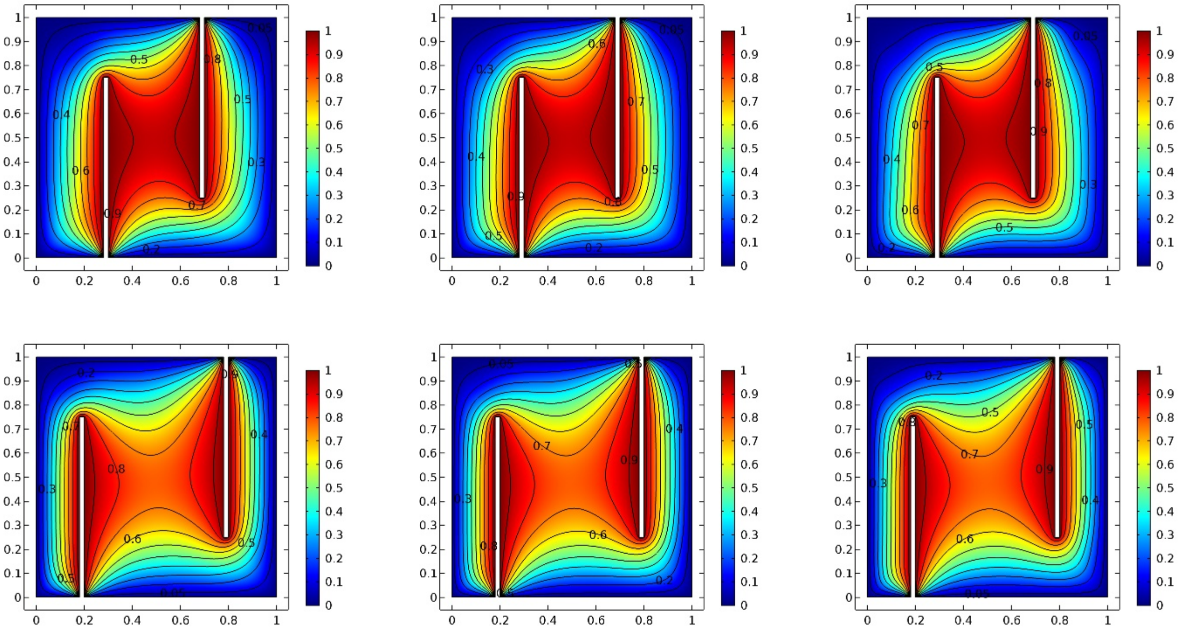


Fig. 4: Influence on isotherm contours for $(Pr = 5)$ with baffle gap ratio B.g$= 0.2$,0.4,0.6 from top to bottom for Re=1,5,10 from left to right with constant $Gr = 1000$.

Observing the behavior of velocity along a straight line inside the 2D domain is a very interesting phenomenon and is done by many researchers. This straight line can be horizontal or vertical. Due to the attachment of baffles with the horizontal wall, there will be a jump in the velocity if we draw any horizontal cutline. So, the velocity is observed along the vertical cutline at x = 0.5. In Figs. 5 (a-c), u and v components of velocity are observed along the y-axis at x=0.5 while the baffle gap inside the cavity is B.g = 0.2, 0.4, 0.6 and three different curves are representing three different Reynolds Numbers i.e. Re = 5,10,20, respectively.

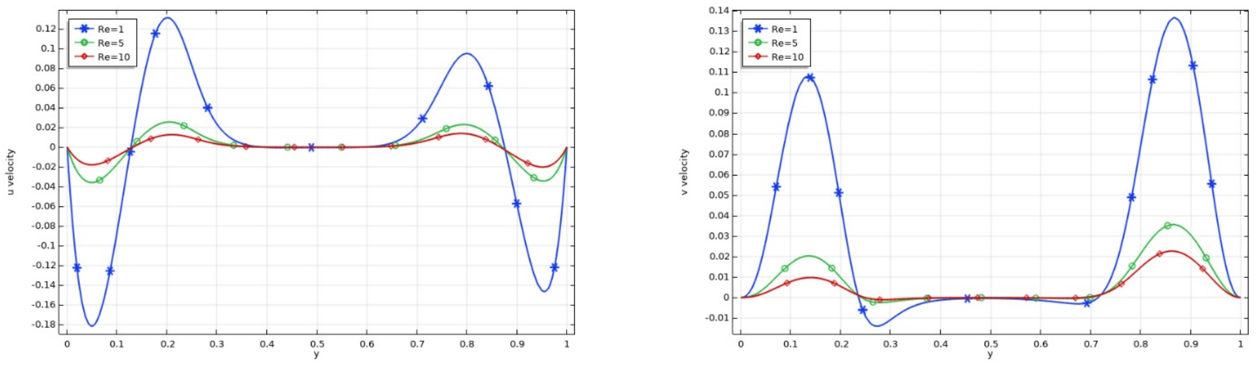


Fig. 5(a): Vertical cutlines at x=0.5 to observe u and v component of velocity along y axis for baffle gap=0.2

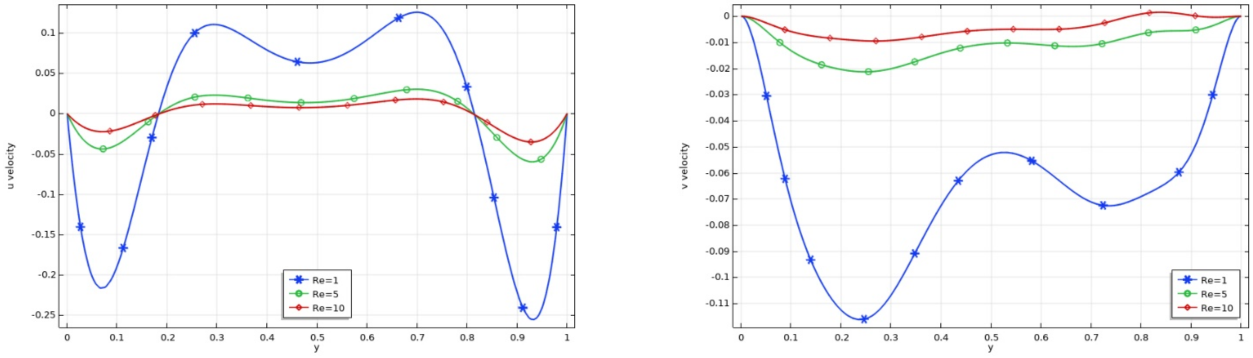


Fig. 5(b): Vertical cutlines at x=0.5 to observe u and v component of velocity along y axis for baffle gap=0.4

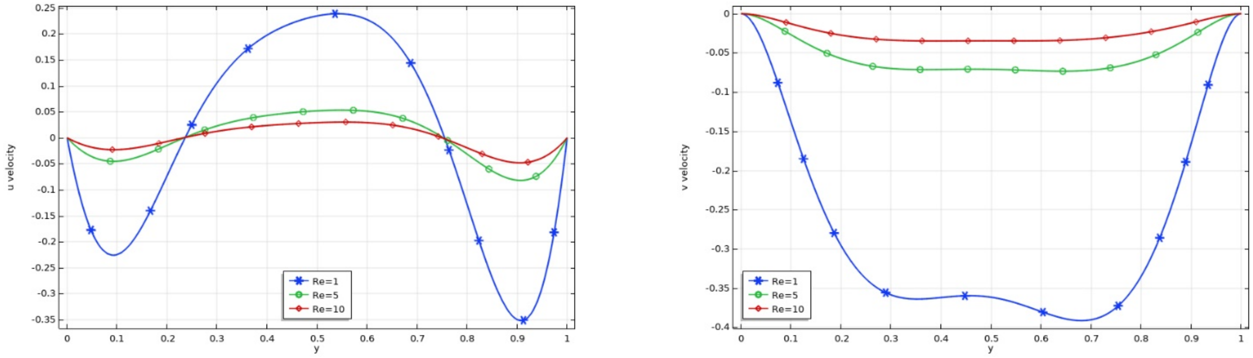


Fig. 5(c): Vertical cutlines at x=0.5 to observe u and v component of velocity along y axis for baffle gap=0.6

Inside the cavity, two major physical phenomena are happening, fluid motion and heat exchange. Reynolds number tells the behavior of fluid flow while Nusselt number is the rate of heat exchange within the cavity. So, the graph of Reynolds vs Nusselt number is plotted for three different baffle gaps. It can be observed from Fig. 6(a) that with the increase in Reynolds number, the Nusselt number is also increasing for all three cases. But for baffle gap 0.6 there is the highest amount of heat exchange within the cavity. Kinetic Energy is a form of energy that a particle has by the reason of motion. The graph of Reynolds number vs overall kinetic energy inside the domain is plotted and presented in Fig. 6(b). It can be observed from the graph that by increasing the Reynolds number kinetic energy is increasing inside the domain while there is a big jump in kinetic energy for the case of baffle gap B.g=0.2.

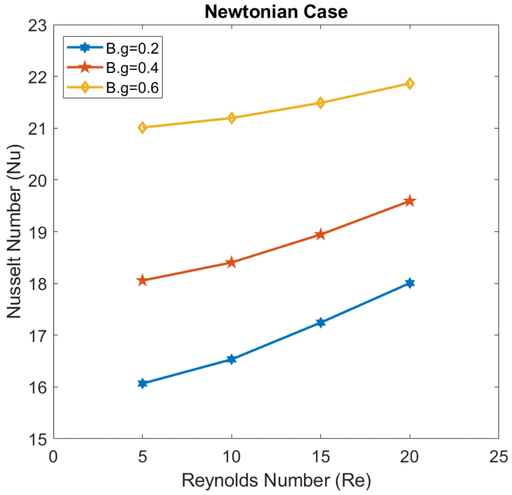


Fig. 6(a): Influence of average Nusselt number on baffles for various Reynolds Number for B.g=0.2,0.4,0.6 from top to bottom

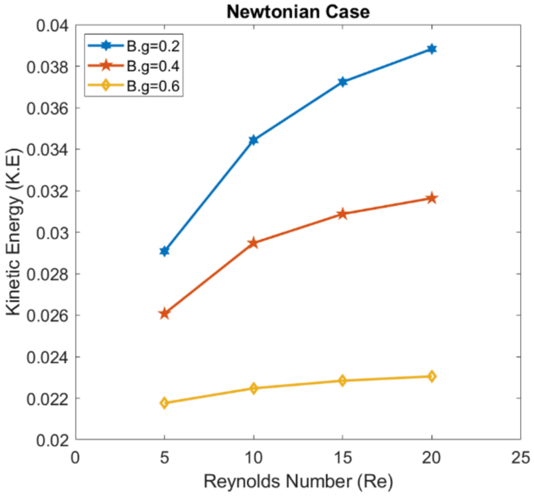


Fig. 6(b): Impact of Reynolds number on kinetic energy for baffle gap 0.2.

6. Conclusion

This research concerns the analysis of viscous fluid flow and heat transfer in a closed configuration. Mathematical modeling of the Newtonian fluid is practiced by constructing partial differential equations from generalized Navier Stokes equations. Due to the non-linearity of the model, the finite element scheme is applied to compute the solution of the attained partial differential system along with implemented boundary constraints. Discretization of the domain consists of the arrangement of rectangular and triangular elements. Graphical illustrations in the form of flow circulations, isothermal contours are depicted against involved parameters. The quantities of engineering interest like kinetic energy and Nusselt number are also computed. By increasing baffle gap for the Newtonian fluid, boundary layer thickness decreases in velocity profiles with increase in Reynolds number. With the increase in Reynold number, the number of eddies is shows that higher mixing will be obtained. By increasing the baffle gap, the heat transfer region increases showing more fluid gets heated inside the cavity.

Author Contributions

Mr. Majeed has done modeling and computing data.
Ms. Aqsa has done written a complete manuscript
Dr. Mahmood has supervised us.

References

[1] Kawaguti, M. Numerical solution of the Navier-Stokes equations for the flow in a two-dimensional cavity. *J. Phys. Soc. Japan*. **16**, 2307-2315 (1961).

[2] Eckert, E. R. G., Carlson, W. O. Natural convection in an air layer enclosed between two vertical plates with different temperatures, *Int. J. Heat Mass Transfer,* **2** (1961).

[3] Gibanov, N. S., Sheremet, M. A., Natural convection in a cubical cavity with different heat source configurations, *TSEP*, **7**, 138-145(2008).

[4] Mahalakshmi, T., Nithyadevi, N., Oztop, H. F., Abu-Hamdeh, N. Natural convective heat transfer of Ag-water nanofluid flow inside enclosure with center heater and bottom heat source, *Chin. J. Phys.***56**, 1497-1507 (2018).

[5] Buongiorno, J. Convective Transport in Nanofluids, *Journal of Heat Transfer, **128**, 240-250* (2006).

[6] Li, et al., Transient pool boiling and particulate deposition of copper oxide nano-suspensions, *Int. J. of Heat and Mass Trans.*, **156** (2020).

[7] Yadav, D., Lee, D., Cho, H. H. The onset of double-diffusive nanofluid convection in a rotating porous medium layer with thermal conductivity and viscosity variation: a revised model, *J. Porous Media,* **19**, 31-46 (2016).

[8] Poulikakos, D., Bejan, A., Selimos, B., Blake, K., High Rayleigh number convection in a fluid overlaying a porous bed, *Int. J. Heat Fluid Flow, 7, 109-116* (1986).

[9] Bilal, S., Mahmood, R., Majeed, A. H., Khan, I., Nisar, K. S. Finite element method visualization about heat transfer analysis of Newtonian material in triangular cavity with square cylinder," *J. Mater. Res. Technol.*, **9**, 4904–4918 (2020).

[10] Mahmood, R., Bilal, S., Majeed, A. H., Khan, I., Sherif, E. S. M. A comparative analysis of flow features of Newtonian and power law material: A New configuration, *J. Mater. Res. Technol.*, (2019).

[11] Mahmood, R., Bilal, S., Majeed, A. H., Khan, I., Nisar, K. S. CFD analysis for characterization of non-linear power law material in a channel driven cavity with a square cylinder by measuring variation in drag and lift forces, *J. Mater. Res. Technol.*, (2020).

[12] Mahmood, R., Bilal, S., Majeed, A. H., Khan, I., Nisar, K. S. Assessment of pseudo-plastic and dilatant materials flow in channel driven cavity: Application of metallurgical processes, *J. Mater. Res. Technol.*, **9**, 3829–3837, (2020).

[13] Mehmood, A., Mahmood, R., Majeed, A. H., Awan, F. J. Flow of the Bingham-Papanastasiou Regularized Material in a Channel in the Presence of Obstacles: Correlation between Hydrodynamic Forces and Spacing of Obstacles, *Model. Simul. Eng.*, (2021).