Application of Improved Blocked-Off Method to Simulate the Interacting Influences of Obstacle Shape and Wall Velocity on the Turbulent Mixed Convection Flow in a Trapezoidal Cavity

Meysam Atashafrooz*
Department of Mechanical Engineering
Sirjan University of Technology, Sirjan, Iran
E-mail: m.atashafrooz@sirjantech.ac.ir
Meysam.atashafrooz@yahoo.com
*Corresponding author

Received: 18 February 2022, Revised: 30 March 2022, Accepted: 31 March 2022

Abstract: In the current research, interaction influences of obstacle shape and top wall velocity on the hydrothermal behaviours of the turbulent mixed convection flow in a trapezoidal cavity are numerically simulated. To achieve this goal, three different shapes of the obstacles including semicircular, triangular, rectangular are considered. Dimensions of these obstacles are chosen so that the environment around all three of them is same. The RNG $k-\varepsilon$ model is chosen to simulate the turbulent flow. To model the inclined or curved walls of trapezoidal cavity and obstacles, the improved blocked-off method is applied. Results show that the obstacle shape and top wall velocity have a significant influence on the thermal and hydrodynamic behaviours. In fact, the highest magnitude of heat transfer rate along the bottom wall occurs in the cavity with the rectangular obstacle and for the highest magnitude of top wall velocity; whilst its lowest magnitude is related to the pure free convection and for the cavity with the semicircular obstacle. Besides, the lowest and highest magnitudes of temperatures fields occur for the cavities with rectangular and triangular obstacles, respectively.

Keywords: Improved Blocked-Off Method, Rectangular Obstacle, RNG $k-\varepsilon$ MethodTurbulent Flow, Semicircular Obstacle, Triangular Obstacle

How to cite this paper: Meysam Atashafrooz, "Application of Improved Blocked-Off Method to Simulate the Interacting Influences of Obstacle Shape and Wall Velocity on the Turbulent Mixed Convection Flow in a Trapezoidal Cavity", Int J of Advanced Design and Manufacturing Technology, Vol. 15/No. 2, 2022, pp. 57–67. DOI: 10.30495/admt.2022.1952976.1341.

Biographical notes: Meysam Atashafrooz is an associate professor in the Sirjan University of Technology, Sirjan, Iran. He received his MSc and PhD degrees from Shahid Bahonar University of Kerman in 2011 and 2015, respectively. He has several published papers about the fluid mechanics, thermodynamics, heat transfer and CFD research area.

1 INTRODUCTION

The study of laminar and turbulent forced, free and mixed convection fluid flow in cavities and enclosures has long been considered by a large number of thermal science researchers [1-8]. Koufi et al. [9] numerically investigated the influences of inlet and outlet openings on the turbulent mixed convection flow in an open cavity. Miroshnichenko et al. [10] simulated the coupling between the turbulent convection flow and surface thermal radiation in a square enclosure having local heater.

Huang et al. [11] analyzed the influence of buoyancy force on the turbulent mixed convection flow inside a cavity using the large eddy simulation (LES) approach. It can be concluded that the results of LES method are in good agreement with experimental data. Olazo-Gómez et al. [12] studied the impact of a vertical glazed wall on the conjugate laminar and turbulent convection flow in a cavity in the presence of surface thermal radiation. Rodrigues and de Lemos [13] applied the $k-\varepsilon$ and local thermal non-equilibrium approaches to analysis the turbulent convection flow in a cavity filled with a porous medium. This attention has been due to the importance of these types of flows in design of many engineering and industrial applications such as chemical catalytic reactors, chemical vapor deposition equipment, furnace engineering, solar collectors, nuclear reactors, and electronics cooling systems [14-19].

In addition, it should be noted that the laminar and turbulent convection fluid flow inside the cavities and enclosures is known as a benchmark problem and it is used to validate various issues [20-26]. Finding suitable methods to control the thermal and hydrodynamic behaviours in different systems is one of the main goals of thermal science researchers. So far, several different approaches have been proposed in this regard. One of the effective and useful methods proposed is to use an obstacle in the flow domain [27-34]. As a result, several studies have been conducted to investigate the effects of different obstacles on the heat transfer rates and friction coefficients in the various geometers such as ducts, cavities and enclosures for laminar and turbulent fluid flow regimes [35-42].

Among these studies, Kareem and Gao [43] numerically investigated the effects of a rotating cylinder on the hydrothermal behaviours of turbulent mixed convection flow in a lid-driven cavity. Motlagh and Sarvari [44] used the large eddy simulation and proper orthogonal decomposition (POD) methods to study the turbulent mixed convection flow in the ventilated cavity with an obstacle. Barman and Dash [45] investigated the effects of obstacle positions on the turbulent forced convection fluid flow in a duct with two forward facing steps. Menni et al. [46-48] analyzed the characteristics of various baffles on the trends of turbulent convection heat transfer inside channels under different conditions.

Siba and Jehhef [49] numerically studied the impacts of a triangular obstacle on the turbulent forced convection heat transfer in a channel with a sudden expansion. Although so far, several studies have been done to investigate the influences of obstacles on the hydrothermal behaviours of laminar or turbulent convection fluid flow in different geometries; but, to the best of the authors' knowledge, analysis of the interacting effects of obstacle shape and top wall velocity on the turbulent mixed convection flow in a trapezoidal cavity have not been investigated by other researchers. It should be mentioned that the obstacles shape and wall velocity can reinforce or weaken each other's effects on the hydrodynamic and thermal behaviours. Also, it is important to note that in this study, calculations are performed for three shapes of obstacle (rectangular, triangular and semicircular) at different values of top wall velocity. However, the popular and efficient RNG $k-\varepsilon$ approach is used to model the turbulent fluid flow, whereas, the improved blocked-off method is applied to simulate the inclined or curved walls of trapezoidal cavity and obstacles.

2 PROBLEM DESCRIPTION

As it is mentioned earlier, this paper attempts to analysis the interaction impacts of the top wall velocity and shape of various obstacles (rectangular, triangular and semi-circular) on the hydrodynamic and thermal behaviours of mixed turbulent convection flow in trapezoidal cavities. The geometries of these trapezoidal cavities are shown in "Figs. 1(a to c)".

It should be noted that the size of the cavities walls (except the size of obstacles walls) is the same and equal to:

$a = 4cm, b = 2cm, c = 4.7cm$ and $d = 14cm$

The dimensions of the obstacles are considered so that the environment around all three of them is equal to each other. Therefore, the radius of semicircular obstacle, the width of rectangular obstacle and the height of triangular obstacle are equal to:

$$R = \frac{b}{2} = 1cm, W = R\left(\frac{\pi}{2} - 1\right) = 0.57cm \text{ and}$$

$$H = R\sqrt{\frac{\pi^2}{4} - 1} = 1.21cm$$

Also, the flow and thermal boundary conditions for all three cavities are presented in "Table 1".

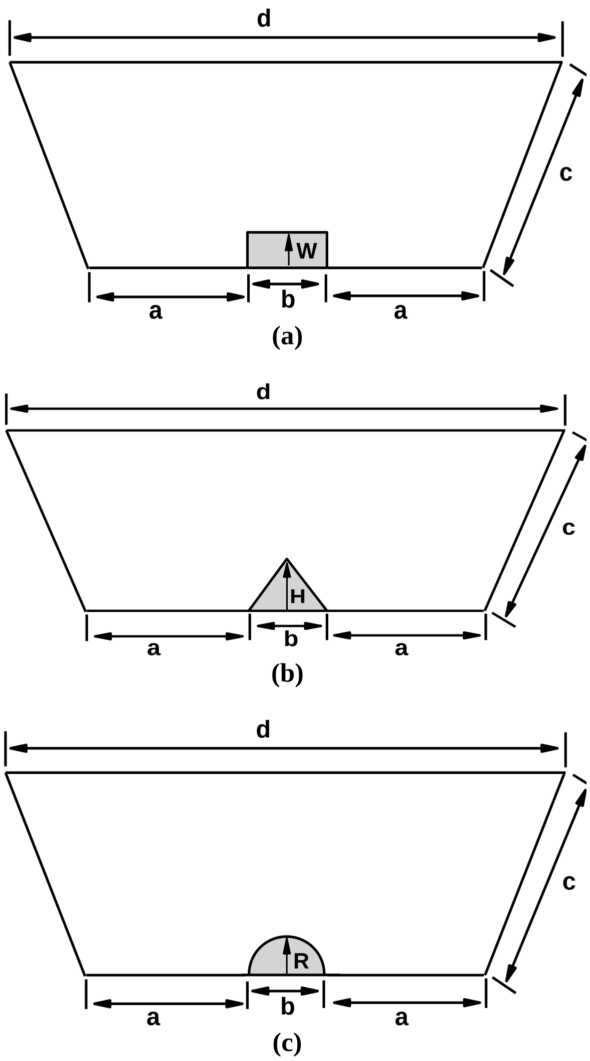


Fig. 1 Cavity with the semicircular obstacle: Geometry of the studied trapezoidal cavities.

Table 1 Flow and thermal boundary conditions for all studied cavities

	Flow boundary conditions	Thermal boundary conditions
Top walls	$u = U_0, v = 0$	$T = 300K$
Inclined side walls	$u = v = 0$	$\frac{\partial T}{\partial n} = 0$
Bottom walls including the obstacles surfaces	$u = v = 0$	$T = 400K$

3 GOVERNING EQUATIONS

The basic equations for turbulent mixed convection flow in cavity under study can be written in vector forms as follows:

$$\frac{\partial U_j}{\partial x_j} = 0 \tag{1}$$

$$\frac{\partial (U_j U_i)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\upsilon\left(\frac{\partial U_i}{\partial x_j}\right) - \overline{u_i u_j}\right] + B_i \tag{2}$$

$$\frac{\partial (U_j T)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\frac{\upsilon}{Pr} - \frac{\upsilon_t}{Pr_t}\right)\frac{\partial T}{\partial x_j}\right] \tag{3}$$

In these equations, the $\overline{u_i u_j}, \upsilon_t$ and Pr_t terms are respectively the Reynolds stress, turbulence viscosity and turbulence Prandtl number. Also, the B_i term is related to the contribution of buoyancy force. In this research, the turbulence terms are calculated using the RNG $k - \varepsilon$ turbulence model. According to this model:

$$\overline{u_i u_j} = -\upsilon_t\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_i}{\partial x_j}\right) + \frac{2}{3}\delta_{ij}k \tag{4}$$

$$d\left(\frac{\rho^2}{\sqrt{\varepsilon\mu}}\right) = 1.72\frac{\upsilon_t}{\sqrt{{\upsilon_t}^3 - 1 + C_v}}d\upsilon_t \tag{5}$$

In the above equations, the k and ε parameters are respectively the turbulent kinetic energy and the turbulent energy dissipation rate. The following equations are used to calculate these terms:

$$\rho\frac{\partial}{\partial x_j}(kU_i) = \frac{\partial}{\partial x_j}\left(a_k\mu_{eff}\frac{\partial k}{\partial x_j}\right) + G_k + G_b - \rho\varepsilon + S_k \tag{6}$$

$$\rho\frac{\partial}{\partial x_i}(\varepsilon U_i) = \frac{\partial}{\partial x_j}\left(a_\varepsilon\mu_{eff}\frac{\partial \varepsilon}{\partial x_j}\right) + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\,\rho\frac{\varepsilon^2}{k} - R_\varepsilon + S_\varepsilon \tag{7}$$

In these equations:

$$G_k = -\rho\overline{u_i u_j}\frac{\partial u_j}{\partial x_i} \tag{8}$$

$$R_\varepsilon = \frac{C_\mu\rho\eta^3(1 - \frac{\eta}{\eta_0})}{1 + \beta\eta^3}\frac{\varepsilon^2}{k} \tag{9}$$

$$\eta = \frac{S_k}{\varepsilon} \quad (10)$$

$$\mu_{eff} = \rho C_\mu \frac{k^2}{\varepsilon} \quad (11)$$

The constants shown in the above equations are presented in "Table 2".

Table 2 The constants used in the $k-\varepsilon$ equations [50-52]

β	c_v	η_0	c_μ	$C_{2\varepsilon}$	$C_{1\varepsilon}$	Pr_t
0.012	100	4.38	0.0845	1.68	1.42	0.85

However, more details of the RNG $k-\varepsilon$ turbulence model are presented in Refs. [50-52].

4 NUMERICAL SOLUTIONS AND CODE VALIDATION

To obtain the hydrothermal behaviours of turbulent mixed convection flow in the studied cavity, the basic equations (continuity, Navier-Stokes, energy, turbulent kinetic energy and turbulent energy dissipation) are discretized by integration on the volume of each element (FVM). Then, the obtained discrete equations are solved by using the line-by-line iterative approach and three-diagonal matrix method. It should be mentioned that the Simple algorithm is applied to couple the velocity and pressure fields.
The optimum meshes used to numerically solve the governing equations in all three cavities under study are presented in "Table 3". It is important to note that the considered meshes are concentrated near the cavities and obstacles walls to achieve more accurate results.

Table 3 The optimum meshes used to solve the governing equations in the studied cavities

Cavity with rectangular obstacle	Cavity with triangular obstacle	Cavity with semicircular obstacle
280×140	300×160	280×150

Also, the computational times for solving the governing equations by using the provided meshes are presented in "Table 4". As can be seen from this table, the cavities with triangular and rectangular obstacles have the highest and lowest computational times, respectively.
In the present study, the improved blocked-off method [53-54] is applied to model the inclined side walls of the trapezoidal cavity and obstacles surfaces. The difference between this method and the blocked-off method [55-58] is related to model the curved or inclined surfaces. In fact, the improved blocked-off method simulates exactly the inclined or curved walls same as the real irregular walls. A schematic of the simulation of curved or inclined surfaces using the improved blocked-off method is shown in "Fig. 2". More details of this approach were fully explained in Refs. [53-54]. Therefore, these explanations are not presented here to avoid repetition.

Table 4 The computational times for solving the governing equations in the studied cavities

Cavity with rectangular obstacle	Cavity with triangular obstacle	Cavity with semicircular obstacle
$24\ min$	$31\ min$	$27\ min$

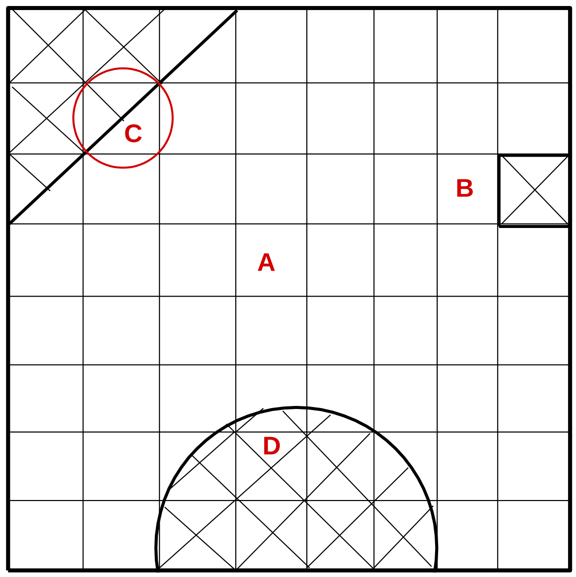


Fig. 2 A schematic of the simulation of curved or inclined surfaces using the improved blocked-off method.

Also, the convergence criterions considered to numerically solve the governing equations are as follows:

$$Max\left|\frac{\Lambda^{\omega}(m,n) - \Lambda^{\omega-1}(m,n)}{\Lambda^{\omega}(m,n)}\right| \le 10^{-6} \quad (12)$$

$$\sum_{m=1}^{m=M}\sum_{n=1}^{n=N}\left|\Lambda^{\omega}(m,n) - \Lambda^{\omega-1}(m,n)\right| \le 10^{-5} \quad (13)$$

In these relations, the Λ parameter denotes the velocity, pressure, temperature, turbulent kinetic energy and turbulent energy dissipation fields. Also, the ω symbol is the iteration step.

Furthermore, it should be mentioned that all computations in this research are done by using a homemade code written in Fortran 90 and a personal computer Intel (R), Core (TM) i5-7400, CPU 3.0 GHz and 8.00 GB of RAM.
To ensure the accuracy of the numerical methods and algorithms considered in this research, the results of this study are compared with the results presented by Promvonge [27] and Fawaz et al. [29]. These comparisons are shown in “Fig. 3”. As can be seen from this figure, the numerical approach considered in this research has a high accuracy.

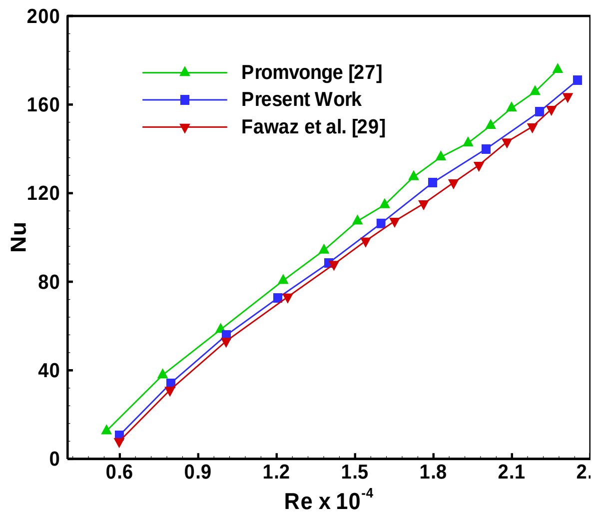
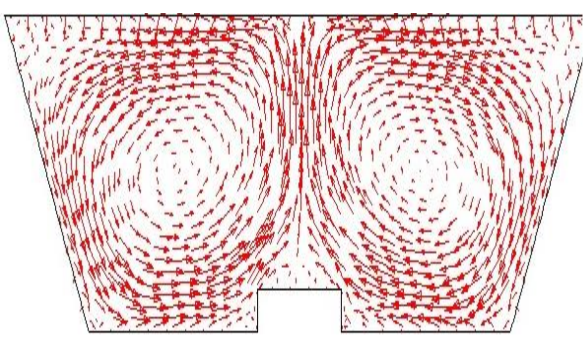


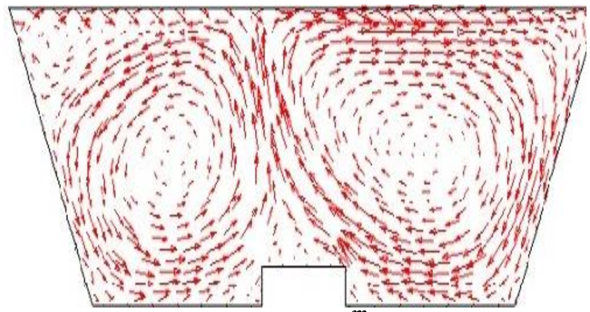
Fig. 3 Comparison of the magnitudes of average Nusselt number at various values of Reynolds numbers with the findings of Promvonge [27] and Fawaz et al. [29].

5 RESULTS AND DISCUSSIONS

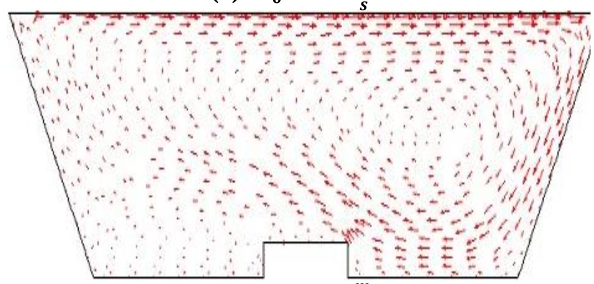
First, to analyze the interaction impacts of obstacle shape and top wall velocity on the flow features, distributions of the velocity vectors in the trapezoidal cavity are shown in “Figs. 4 to 6” for three shapes of obstacle (rectangular, triangular and semicircular) at different values of U_0.



(a): $U_0 = 0$

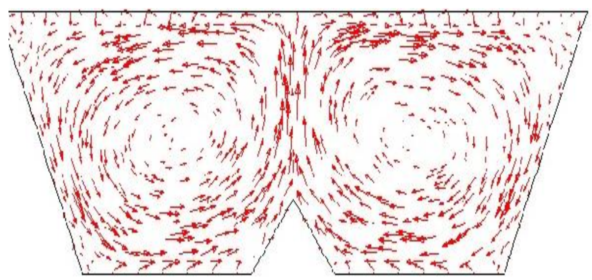


(b): $U_0 = 0.1\,\frac{m}{s}$

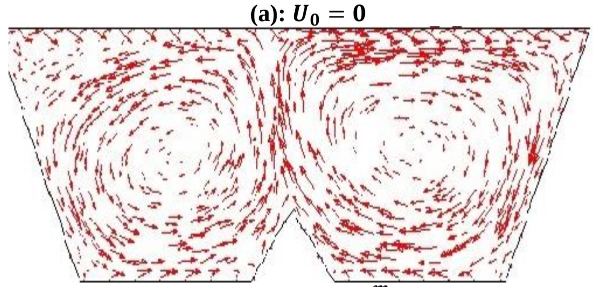


(c): $U_0 = 0.5\,\frac{m}{s}$

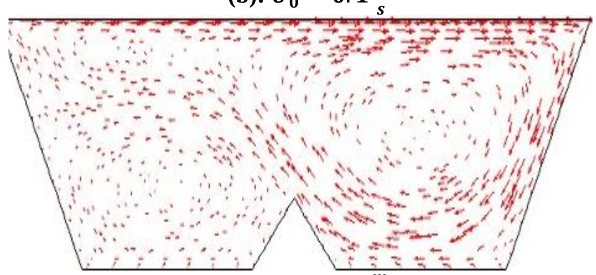
Fig. 4 Impacts of top wall velocity on the distributions of the velocity vectors in the trapezoidal cavity with the rectangular obstacle.



(a): $U_0 = 0$

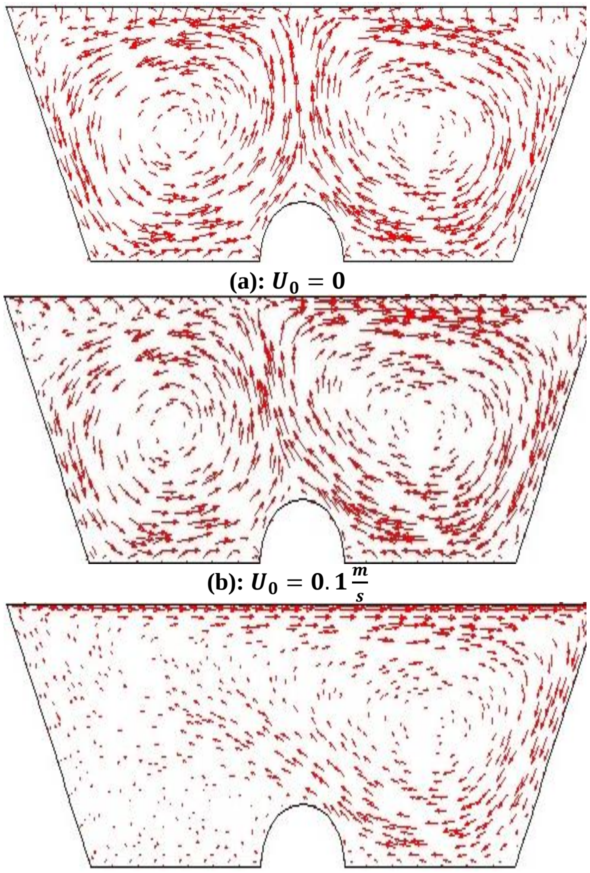


(b): $U_0 = 0.1\,\frac{m}{s}$



(c): $U_0 = 0.5\,\frac{m}{s}$

Fig. 5 Impacts of top wall velocity on the distributions of the velocity vectors in the trapezoidal cavity with the triangular obstacle.



(a): $U_0 = 0$

(b): $U_0 = 0.1\frac{m}{s}$

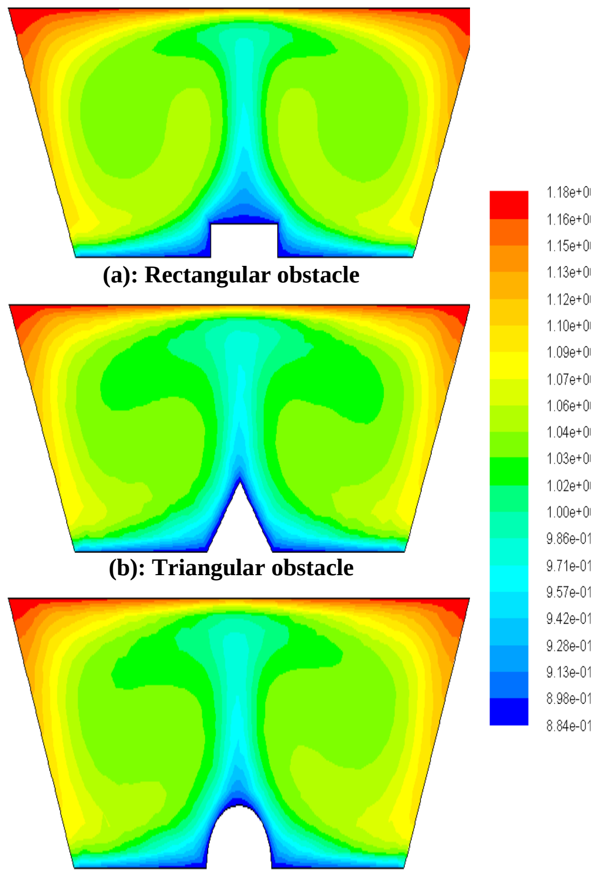
(c): $U_0 = 0.5\frac{m}{s}$

Fig. 6 Impacts of top wall velocity on the distributions of the velocity vectors in the trapezoidal cavity with the semicircular obstacle.

Accurate analysis of these figures clearly shows that for all three shapes of obstacle, the flow pattern is significantly dependent on the magnitudes of top wall velocity. In the case of $U_0 = 0$ (pure free convection), the streamlines are symmetrical such that two equal recirculation zones are formed on the left and right hands of the different obstacles. It should be noted that in this case, the movement of fluid flow in the cavity is affected by the buoyancy force and changes of fluid density in different parts of the cavity.

In "Fig. 7", distributions of fluid density changes in the cavity are shown for all three obstacles geometries.

As it is seen from these Figures, the obstacle shape has a considerable effect on the density changes in the cavity domain. In the cases of $U_0 \neq 0$ (mixed convection), the streamlines are not symmetrical and the size of the right recirculation zones enhances by increasing the top wall velocity. In fact, in these cases, the buoyancy force and the top wall movement lead to the movement of fluid flow in the cavity. Of course, it should be noted that the role of the buoyancy force decreases as the top wall velocity increases. Another noteworthy point in these figures is that the maximum and minimum effect of top wall velocity on the streamlines is related to the cavities with the rectangular and triangular obstacles, respectively.



(a): Rectangular obstacle

(b): Triangular obstacle

(c): Semicircular obstacle

Fig. 7 Distributions of density changes in the trapezoidal cavity for all three obstacles geometries ($\boldsymbol{U_0 = 0}$).

To illustrate the influences of obstacle shape and top wall velocity on the thermal characteristics in the trapezoidal cavity under study, distributions of temperature fields are displayed in "Figs. 8 to 10".

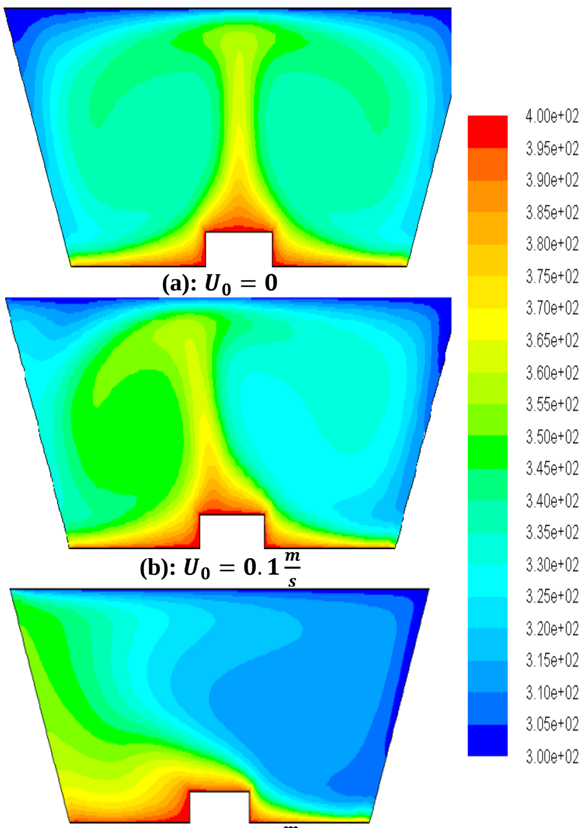
Careful examination of these figures clearly shows that the top wall velocity has an important effect on the trends and values of the temperature distributions inside the cavity. In the absence of forced convection heat transfer ($U_0 = 0$), the temperature distributions inside the cavity are symmetrical for all three shapes of obstacles.

In this case, the highest values of fluid temperature are related to the areas near the cavity bottom wall and the areas above the obstacles. It should be mentioned that in these areas, the fluid density has its lowest values.

In other words, in this case, variations of temperature field are the opposite of the variations of fluid density in the cavity.

In the presence of both forced and free convection heat transfer mechanisms ($U_0 \neq 0$), the temperature field inside the cavity is not symmetrical. In these cases, by moving the top wall from left to right, the hot areas are transferred to the left side of cavity domain. In fact, by enhancing the top wall velocity, more hot areas are located on the left side of the cavity.

Another interesting point in "Figs. 8 to 10" is that the lowest and highest values of temperatures field in the cavities are respectively related to the rectangular and triangular obstacles.

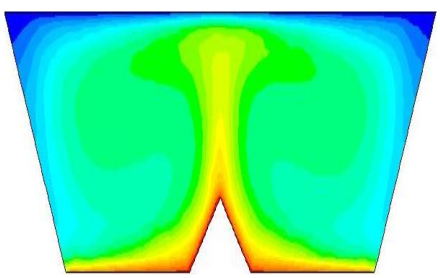


(a): $U_0 = 0$

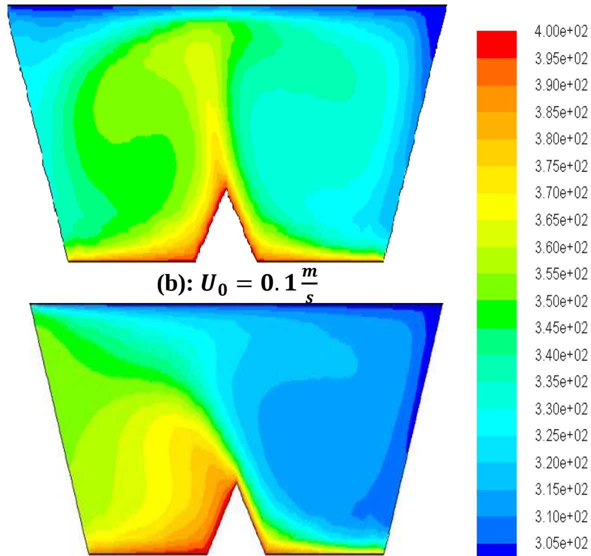
(b): $U_0 = 0.1\,\frac{m}{s}$

(c): $U_0 = 0.5\,\frac{m}{s}$

Fig. 8 Impacts of top wall velocity on the temperature distributions in the trapezoidal cavity with the rectangular obstacle.



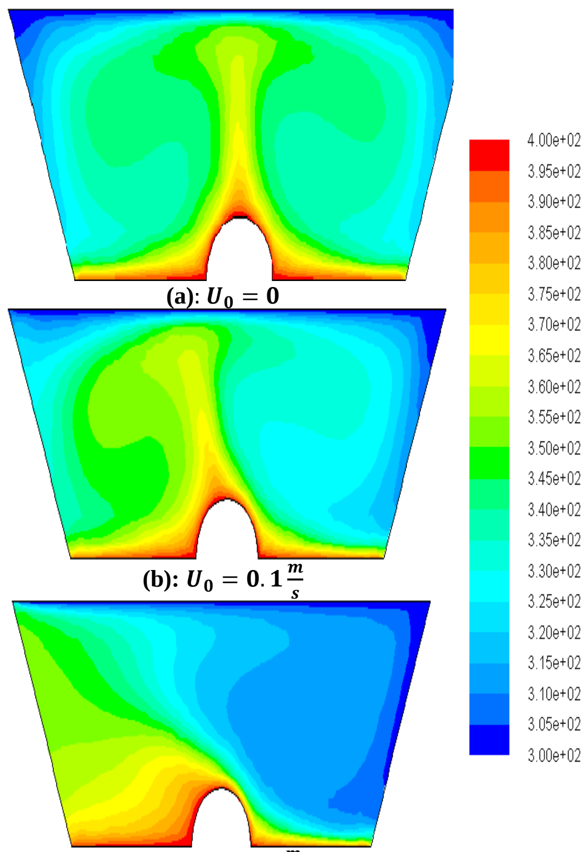
(a): $U_0 = 0$



(b): $U_0 = 0.1\,\frac{m}{s}$

(c): $U_0 = 0.5\,\frac{m}{s}$

Fig. 9 Impacts of top wall velocity on the temperature distributions in the trapezoidal cavity with the triangular obstacle.



(a): $U_0 = 0$

(b): $U_0 = 0.1\,\frac{m}{s}$

(c): $U_0 = 0.5\,\frac{m}{s}$

Fig. 10 Impacts of top wall velocity on the temperature distributions in the trapezoidal cavity with the semicircular obstacle.

To further study the interaction influences of obstacle shape and top wall velocity on the thermal behaviours in the cavities under study, the heat transfer rates along the cavities bottom wall (including the obstacle surfaces) are shown in “Fig. 11”.

A comparison of the data presented in this figure clearly shows that the magnitudes of heat transfer rate increase with augmentation of the top wall velocity. Also, it is quite clear that the highest and lowest values of heat transfer rates on the bottom wall are related to cavities with the rectangular and semicircular obstacles.

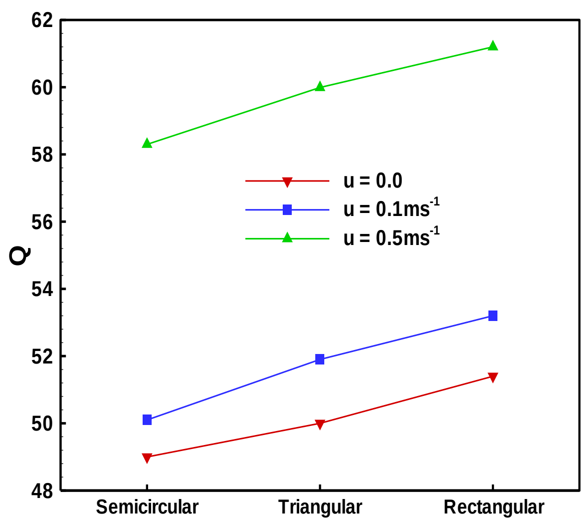


Fig. 11 Interacting influences of obstacle shape and top wall velocity on the heat transfer rates along the cavity bottom wall.

6 CONCLUSIONS

Interacting influences of obstacle shape (rectangular, triangular and semicircular) and top wall velocity on the hydrothermal features of turbulent mixed convection flow in a trapezoidal cavity are analyzed with details. The main results of this study can be summarized as follows:

✓ For the case of pure free convection heat transfer, distributions of velocity vectors and temperature fields inside the cavity are symmetrical for all three shapes of obstacles. Also, an enhancement in the top wall velocity leads to a decrease in the role of buoyancy force on the distributions of velocity vectors and temperature fields.

✓ The maximum and minimum influences of top wall velocity on the velocity vectors occur respectively in the cavities with rectangular and triangular obstacles.

✓ By increasing the top wall velocity, more hot areas are located on the left side of the cavity.

✓ The lowest and highest values of temperatures fields are respectively related to the cavities with rectangular and triangular obstacles.

✓ The highest values of heat transfer rate on the bottom wall occur for the highest values of top wall velocity and for the cavity with the rectangular obstacle.

✓ The lowest magnitudes of heat transfer rate on the bottom wall occur in the absence of forced convection flow (pure free convection, $U_0 = 0$) and for the cavity with the semicircular obstacle.

7 NOMENCLATURE

b	Obstacles length on cavities bottom wall, (cm)
B_i	Buoyancy force
H	Height of triangular obstacle, (cm)
k	Turbulent kinetic energy
P	Pressure, $(N.m^{-2})$
Pr	Prandtl number
Pr_t	Turbulence Prandtl number
Q	Heat transfer rate, (W)
R	Radius of semicircular obstacle, (cm)
Re	Reynolds number
T	Temperature, (K)
U_0	Velocity of top wall, $(m.s^{-1})$
U_i or U_j	Velocity vector, $(m.s^{-1})$
$\overline{u_i u_j}$	Reynolds stress
W	Width of rectangular obstacle, (cm)
x_i or x_j	Cartesian coordinates, (m)

Greek Symbols

ε	Turbulent energy dissipation rate
υ	Viscosity, $(m^2.s^{-1})$
υ_t	Turbulence viscosity
ρ	Density, $(Kg.m^{-3})$

REFERENCES

[1] Yang, G., Huang, Y., Wu, J., Zhang, L., Chen, G., Lv, R., and Cai, A., Experimental Study and Numerical Models Assessment of Turbulent Mixed Convection Heat Transfer in a Vertical Open Cavity, Building and Environment, Vol. 115, 2017, pp. 91-103.

[2] Miroshnichenko, I. V., Sheremet, M. A., Radiation Effect on Conjugate Turbulent Natural Convection in a Cavity with a Discrete Heater, Applied Mathematics and Computation, Vol. 321, 2018, pp. 358-371.

[3] Mahmoodabadi, M. J., Mahmoodabadi, F., and Atashafrooz, M., Development of the Meshless Local Petrov-Galerkin Method to Analyze Three-Dimensional Transient Incompressible Laminar Fluid Flow, Journal of the Serbian Society for Computational Mechanics, Vol. 12, No. 2, 2018, pp. 128-152.

[4] Hegele, J. R, L. A., Scagliarini, A., Sbragaglia, M., Mattila, K. K., Philippi, P. C., Puleri, D. F., Gounley, J., and Randles, A., High-Reynolds-Number Turbulent Cavity Flow using the Lattice Boltzmann Method, Physical Review E, Vol. 98, No. 4, 2018, Article Number: 043302.

[5] Benyahia, N., Aksouh, M., Mataoui, A., and Oztop, H. F., Coupling Turbulent Natural Convection-Radiation-Conduction in Differentially Heated Cavity with High Aspect Ratio, International Journal of Thermal Sciences, Vol. 158, 2020, pp. 106518.

[6] Sun, Y., Liu, Q., Cattafesta III, L. N., Ukeiley, L. S., and Taira, K., Resolvent Analysis of Compressible Laminar and Turbulent Cavity Flows, AIAA Journal, Vol. 58, No. 3, 2020, Article Number: 1046-1055.

[7] Wen, X., Wang, L. P., Guo, Z., and Zhakebayev, D. B., Laminar to Turbulent Flow Transition inside the Boundary Layer Adjacent to Isothermal Wall of Natural Convection Flow in a Cubical Cavity, International Journal of Heat and Mass Transfer, Vol. 167, 2021, Article Number: 120822.

[8] Navarro, J. M. A., Hinojosa, J. F., and Piña-Ortiz, A., Computational Fluid Dynamics and Experimental Study of Turbulent Natural Convection Coupled with Surface Thermal Radiation in a Cubic Open Cavity, International Journal of Mechanical Sciences, Vol. 198, 2021, Article Number: 106360.

[9] Koufi, L., Younsi, Z., Cherif, Y., and Naji, H., Numerical Investigation of Turbulent Mixed Convection in an Open Cavity: Effect of Inlet and Outlet Openings, International Journal of Thermal Sciences, Vol. 116, 2017, pp. 103-117.

[10] Miroshnichenko, I., Sheremet, M., and Chamkha, A.J., Turbulent Natural Convection Combined with Surface Thermal Radiation in a Square Cavity with Local Heater, International Journal of Numerical Methods for Heat & Fluid Flow, Vol. 28, No. 7, 2018, pp. 1698-1715.

[11] Huang, Y. Y., Yang, G., and Wu, J. Y., Large Eddy Simulation and Experimental Study of Turbulent Mixed Convection inside a Cavity with Large Rayleigh Number: Effect of Buoyancy, Building and Environment, Vol. 151, 2019, pp. 268-279.

[12] Olazo-Gómez, Y., Xamán, J., Gijón-Rivera, M., Noh-Pat, F., Simá, E., and Chávez, Y., Mathematical Modelling of Conjugate Laminar and Turbulent Heat Transfer in a Cavity: Effect of a Vertical Glazed Wall, International Journal of Thermal Sciences, Vol. 152, 2020, Article Number: 106310.

[13] Rodrigues, F. A., De Lemos, M. J., Turbulent Flow and Heat Transfer in a Partially Filled Ventilated Cavity using the Local Thermal Non-Equilibrium Method, International Journal of Thermal Sciences, Vol. 164, 2021, Article Number: 106844.

[14] Salari, M., Rashidi, M. M., Malekshah, E. H., and Malekshah, M. H., Numerical Analysis of Turbulent/Transitional Natural Convection in Trapezoidal Enclosures, International Journal of Numerical Methods for Heat & Fluid Flow, Vol. 27, No. 12, 2017, pp. 2902-2923.

[15] Miroshnichenko, I. V., Sheremet, M. A., Turbulent Natural Convection Combined with Thermal Surface Radiation inside an Inclined Cavity Having Local Heater, International Journal of Thermal Sciences, Vol. 124, 2018, pp. 122-130.

[16] Zhang, L., Huang, Y., Yang, G., and Wu, J., Numerical Simulation of Conjugate Turbulent Mixed Convection in an Open Cavity: Evaluation of Different Wall Heat Conduction Models, Numerical Heat Transfer, Part A: Applications, Vol. 74, No. 5, 2018, pp. 1244-1264.

[17] Atashafrooz, M., Sajjadi, H., and Delouei, A. A., Interacting Influences of Lorentz Force and Bleeding on the Hydrothermal Behaviors of Nanofluid Flow in a Trapezoidal Recess with the Second Law of Thermodynamics Analysis, International Communications in Heat and Mass Transfer, Vol. 110, 2020, Article Number: 104411.

[18] Fabregat, A., Pallarès, J., Heat Transfer and Boundary Layer Analyses of Laminar and Turbulent Natural Convection in a Cubical Cavity with Differently Heated Opposed Walls, International Journal of Heat and Mass Transfer, Vol. 151, 2020, Article Number: 119409.

[19] Wu, S., Yaras, M. I., Interaction of a Turbulent Spot with a Two-Dimensional Cavity, Physics of Fluids, Vol. 33, No. 9, 2021, Article Number: 094114.

[20] Atashafrooz, M., Gandjalikhan, Nassab S. A., and Lari, K., Numerical Analysis of Interaction between Non-Gray Radiation and Forced Convection Flow over a Recess using the Full-Spectrum K-Distribution Method, Heat and Mass Transfer, Vol. 52, No. 2, 2016, pp. 361-377.

[21] Alinejad, J., Esfahani, J. A., Taguchi Design of Three-Dimensional Simulations for Optimization of Turbulent Mixed Convection in a Cavity, Meccanica, Vol. 52, No. 4-5, 2017, pp. 925-938.

[22] Kogawa, T., Chen, L., Okajima, J., Sakurai, A., Komiya, A., and Maruyama, S., Effects of Concentration of Participating Media on Turbulent Natural Convection in Cubic Cavity, Applied Thermal Engineering, Vol. 131, 2018, pp. 141-149.

[23] Samantaray, D., Das, M. K., High Reynolds Number Incompressible Turbulent Flow inside a Lid-Driven Cavity with Multiple Aspect Ratios, Physics of Fluids, Vol. 30, No. 7, 2018, Article Number: 075107.

[24] Pinã-Ortiz, A., Hinojosa, J. F., and Hernández-López, I., Computational Fluid Dynamics and Experimental Study of the Effect of Inclination Angle on Turbulent Natural Convection in an Upward Open Cubic Cavity, International Journal of Modern Physics C, Vol. 32, No. 4, 2021, Article Number: 2150056.

[25] Loksupapaiboon, K., Suvanjumrat, C., Assessment of Turbulence Models for Low Turbulent Natural Convection Heat Transfer in Rectangular Enclosed Cavity using Open Foam, In IOP Conference Series: Materials Science and Engineering, Vol. 1137, No. 1, 2021, Article Number: 012044, IOP Publishing.

[26] Lafdaili, Z., El-Hamdani, S., Bendou, A., Limam, K., and El-Hafad, B., Numerical Study of the Turbulent

Natural Convection of Nanofluids in a Partially Heated Cubic Cavity, Thermal Science, Vol. 25, No. 4, 2021, pp. 2741-2754.

[27] Promvonge, P., Heat Transfer and Pressure Drop in a Channel with Multiple 60 V-Baffles, International Communications in Heat and Mass Transfer, Vol. 37, No. 7, 2010, pp. 835-840.

[28] Koolnapadol, N., Hoonpong, P., Skullong, S., Kammul, P., and Promvonge, P., Turbulent Heat Transfer and Pressure Loss in a Square-Duct Heat Exchanger with Inclined-Baffle Inserts, Engineering Journal, Vol. 21, No. 7, 2017, pp. 485-497.

[29] Fawaz, H. E., Badawy, M. T. S., Abd Rabbo, M. F., and Elfeky, A., Numerical Investigation of Fully Developed Periodic Turbulent Flow in a Square Channel Fitted with 45 in-Line V-Baffle Turbulators Pointing Upstream, Alexandria Engineering Journal, Vol. 57, No. 2, 2018, pp. 633-642.

[30] Li, Z., Hussein, A. K., Younis, O., Afrand, M., and Feng, S., Natural Convection and Entropy Generation of a Nanofluid Around a Circular Baffle inside an Inclined Square Cavity under Thermal Radiation and Magnetic Field Effects, International Communications in Heat and Mass Transfer, Vol. 116, 2020, Article Number: 104650.

[31] Singh, S. K., Raushan, P. K., Debnath, K., and Mazumder, B. S., Higher Order Turbulent Flow Characteristics of Oscillatory Flow over a Wall-Mounted Obstacle, ISH Journal of Hydraulic Engineering, Vol. 26, No. 1, 2020, pp. 84-95.

[32] Du, R., Gokulavani, P., Muthtamilselvan, M., Al-Amri, F., and Abdalla, B., Influence of the Lorentz Force on the Ventilation Cavity Having a Centrally Placed Heated Baffle Filled with The $Cu-Al_2O_3-H_2O$ Hybrid Nanofluid, International Communications in Heat and Mass Transfer, Vol. 116, 2020, Article Number: 104676.

[33] Levchenya, A. M., Smirnov, E. M., Smirnovsky, A. A., and Zasimova, M. A., Disturbing Action of a Cubical Obstacle on the Turbulent Vertical-Plate Free Convection Boundary Layer: RANS-Based Simulation, In Journal of Physics: Conference Series, Vol. 1697, No. 1, 2020, pp. 012215, IOP Publishing.

[34] Farsani, R. Y., Mahmoudi, A., and Jahangiri, M., How a Conductive Baffle Improves Melting Characteristic and Heat Transfer in a Rectangular Cavity Filled with Gallium, Thermal Science and Engineering progress, Vol. 16, 2020, Article Number: 100453.

[35] Lee, J. R., Numerical Simulation of Natural Convection in a Horizontal Enclosure: Part I. On the Effect of Adiabatic Obstacle in Middle, International Journal of Heat and Mass Transfer, Vol. 124, 2018, pp. 220-232.

[36] Rahmati, A. R., Tahery, A. A., Numerical Study of Nanofluid Natural Convection in a Square Cavity with a Hot Obstacle using Lattice Boltzmann Method, Alexandria Engineering Journal, Vol. 57, No. 3, 2018, pp. 1271-1286.

[37] Bhattacharyya, S., Benim, A. C., Pathak, M., Chamoli, S., and Gupta, A., Thermohydraulic Characteristics of Inline and Staggered Angular Cut Baffle Inserts in the Turbulent Flow Regime, Journal of Thermal Analysis and Calorimetry, Vol. 140, No. 3, 2019, pp. 1519-1536.

[38] Mahmood, R., Bilal, S., Khan, I., Kousar, N., Seikh, A. H., and Sherif, E.S.M., A Comprehensive Finite Element Examination of Carreau Yasuda Fluid Model in a Lid Driven Cavity and Channel with Obstacle by Way of Kinetic Energy and Drag and Lift Coefficient Measurements, Journal of Materials Research and Technology, Vol. 9, No. 2, 2020, pp. 1785-1800.

[39] Kang, S., Khosronejad, A., and Yang, X., Turbulent Flow Characteristics Around a Non-Submerged Rectangular Obstacle on the Side of an Open Channel, Physics of Fluids, Vol. 33, No. 4, 2021, Article Number: 045106.

[40] Banihashemi, S., Assari, M. R., Javadi, S., and Vahidifar, S., Experimental Study of the Effect of Disk Obstacle Rotating with Different Angular Ratios on Heat Transfer and Pressure Drop in a Pipe with Turbulent Flow, Journal of Thermal Analysis and Calorimetry, Vol. 144, No. 4, 2021, pp. 1401-1416.

[41] Shahid, H., Yaqoob, I., Khan, W. A., and Aslam, M., Multi Relaxation Time Lattice Boltzmann Analysis of Lid-Driven Rectangular Cavity Subject to Various Obstacle Configurations, International Communications in Heat and Mass Transfer, Vol. 129, 2021, Article Number: 105658.

[42] Saha, S., Numerical Simulation of Turbulent Airflow and Heat Transfer Through a Rectangular Channel along with Two Trapezoidal Baffle Plates: Comparison between Plane and Trapezoidal Shape Baffles, In AIP Conference Proceedings, Vol. 2341, No. 1, 2021, Article Number: 030005, AIP Publishing LLC.

[43] Kareem, A. K., and Gao, S., Mixed Convection Heat Transfer of Turbulent Flow in a Three-Dimensional Lid-Driven Cavity with a Rotating Cylinder, International Journal of Heat and Mass Transfer, Vol. 112, 2017, pp. 185-200.

[44] Motlagh, S. Y., and Sarvari, P., Large Eddy Simulation of Three-Dimensional Mixed Convection Flow inside the Ventilated Cavity Containing Obstacle and Extraction of Coherent Structures using Proper Orthogonal Decomposition (Pod), Journal of Solid and Fluid Mechanics, Vol. 7, No. 3, 2017, pp. 199-212.

[45] Barman, A., Dash, S. K., Effect of Obstacle Positions for Turbulent Forced Convection Heat Transfer and Fluid Flow over a Double Forward Facing Step, International Journal of Thermal Sciences, Vol. 134, 2018, pp. 116-128.

[46] Menni, Y., Chamkha, A., Zidani, C., and Benyoucef, B., Baffle Orientation and Geometry Effects on Turbulent Heat Transfer of a Constant Property Incompressible Fluid Flow inside a Rectangular Channel, International Journal of Numerical Methods for Heat & Fluid Flow, Vol. 30, No. 6, 2019, pp. 3027-3052.

[47] Menni, Y., Azzi, A., and Chamkha, A.J., Computational Thermal Analysis of Turbulent Forced-Convection Flow in an Air Channel with a Flat Rectangular Fin and Downstream V-Shaped Baffle, Heat Transfer Research, Vol. 50, No. 18, 2019, pp. 1781-1818.

[48] Menni, Y., Azzi, A., Chamkha, A. J., and Harmand, S., Effect of Wall-Mounted V-Baffle Position in a Turbulent Flow Through a Channel: Analysis of Best Configuration for Optimal Heat Transfer, International Journal of Numerical Methods for Heat & Fluid Flow, Vol. 29, No. 10, 2019, pp. 3908-3937.

[49] Siba, M. A. A., and Jehhef, K. A., Numerical Study of Turbulent Forced Convection Flow Over Sudden Expansion with Triangular Obstacle, Journal of Mechanical Engineering Research and Developments, Vol. 43, No. 3, 2020, pp. 125-143.

[50] Wang, J. Y., Hu, X. J., Application of RNG $k-\varepsilon$ Turbulence Model on Numerical Simulation in Vehicle External Flow Field, In Applied Mechanics and Materials, Vol. 170, 2012, pp. 3324-3328, Trans Tech Publications Ltd.

[51] Koutsourakis, N., Bartzis, J. G., and Markatos, N. C., Evaluation of Reynolds Stress, $k-\varepsilon$ and RNG $k-\varepsilon$ Turbulence Models in Street Canyon Flows using Various Experimental Datasets, Environmental Fluid Mechanics, Vol. 12, No. 4, 2012, pp. 379-403.

[52] Zhao, X., Chen, Q., Inverse Design of Indoor Environment using an Adjoint Rng $k-\varepsilon$ Turbulence Model, Indoor air, Vol. 29, No. 2, 2019, pp. 320-330.

[53] Zabihi, M., Lari, K., and Amiri, H., Comparison of the Blocked-Off and Embedded Boundary Methods in Radiative Heat Transfer Problems in 2d Complex Enclosures at Radiative Equilibrium, Journal of Mechanical Science and Technology, Vol. 31, No. 7, 2017, pp. 3539-3551.

[54] Atashafrooz, M., Shafie, M., Analysis of Entropy Generation for Mixed Convection Fluid Flow in a Trapezoidal Enclosure using the Modified Blocked Region Method, Journal of the Serbian Society for Computational Mechanics, Vol. 14, No. 2, 2020, pp. 97-116.

[55] Atashafrooz, M., Gandjalikhan, Nassab S. A., and Ansari, A. B., Numerical Study of Entropy Generation in Laminar Forced Convection Flow Over Inclined Backward and Forward-Facing Steps in a Duct, International Review of Mechanical Engineering, Vol. 5, No. 5, 2011, pp. 898-907.

[56] Atashafrooz, M., Gandjalikhan, Nassab S. A., and Ansari, B. A., Numerical Investigation of Entropy Generation in Laminar Forced Convection Flow Over Inclined Backward and Forward-Facing Steps in a Duct under Bleeding Condition, Thermal Science, Vol. 18, No. 2, 2014, pp. 479-492.

[57] Atashafrooz, M., Gandjalikhan Nassab, S. A., and Sadat Behineh, E., Effects of Baffle on Separated Convection Step Flow of Radiating Gas in a Duct, International Journal of Advanced Design and Manufacturing Technology (ADMT), Vol. 8, No. 3, 2015, pp. 33-47.

[58] Aminzadeh, N., Sotoodehnia, S., and Atashafrooz, M., Geometrical Effects of Duct on the Entropy Generation in the Laminar Forced Convection Separated Flow, International Journal of Advanced Design and Manufacturing Technology (ADMT), Vol. 11, No. 3, 2018, pp.25-34.