SEMARAK ILMU PUBLISHING

Semarak Engineering Journal

Journal homepage:
https://semarakilmu.my/index.php/sej/index
ISSN: 3036-0145

Numerical Analysis for Assessing the Influence of a Deflector on Heat Transfer and Flow inside a Cavity

Reyam J. Kazem[1], Nabil J. Yasin[1], Salah H. AbidAun[1,*]

[1] Middle Technical University, Technical Engineering College -Baghdad, Iraq

ARTICLE INFO

Article history:
Received 14 January 2026
Received in revised form 29 March 2026
Accepted 22 April 2026
Available online 16 July 2026

Keywords:
Convection heat transfer, cavity, heated block, deflector

ABSTRACT

Numerical investigations have been conducted on convective heat transfer within cavities. This study employs a deflector to improve heat transfer, taking into account variations in Reynolds number (Re) and the position of a heated block (H). All cases are evaluated with a steady heat flux of 2000 W/m² applied to the block, and differing Re and H values. Specifically, Re values of 4000, 8000, 12000, and 16000, and H values of 0, 5, 10, and 15 cm are considered. Using ANSYS Fluent 21, finite volume analysis was carried out with employing the k-ε realizable model for the simulation. The results indicate that the block's position significantly affects the heat transfer rate and velocity distribution within the cavity. Furthermore, it was noted that as the distance between the block and the cavity's bottom surface (H) increases, the Nusselt number (Nu) also increases. Additionally, the deflector positioned at H = 15 cm and Re = 12000 enhances the Nusselt number by approximately 242.28% compared to a channel without a deflector.

1. Introduction

The thermal analysis of cavities has been extensively studied for many years due to its significance in numerous engineering systems, such as cooling systems for electronic components, industrial processing, lubricating systems, drying technology, and chemical processing units [1]. Due to the geometry of the cavity and the dead zone, particularly in the case of cavities with heat blocks where air cannot reach to cool these elements, problems arise with the amount of heat in these components, leading to a decline in device life, safety, and performance. Therefore, a deflector was used to improve the heat rate in a straight passage containing a cavity. Kadhum *et al.*, [2] conducted a numerical study on a cavity filled with nanofluid, featuring an upper oscillating surface and a baffle. Their findings indicated that a baffle attached to the oscillating wall enhances the distribution of vorticity in the enclosure and increases fluid movement within the cavity. Eman and Falah [3] numerically studied mixed convection heat transfer within a duct containing a square cavity incorporating a rotating cylinder. According to the results, the buoyancy force's effect is negligible

[*] *Corresponding author.*
E-mail address: dr.salah_haji102@mtu.edu.iq

https://doi.org/10.37934/sej.15.1.6373

for low Richardson values (Ri = 0.1). When Reynolds numbers are low (Re = 50), the impact of increasing cylinder speed is very evident. For every Richardson value, there is an increase in average Nusselt values as the cylinder rotates faster.

Meenakshi and Manikandan [4] numerically investigated heat transfer and Laminar flow at steady-state in a lid-driven cavity containing two heated blocks. In this simulation, the block temperature is 573 K, and the Reynolds numbers are 100, 200, 500, 1000, and 2000. The findings will be advantageous in similar cases encountered in many applications. Oyewola *et al.*, [5] numerically studied natural convection heat transfer featuring a circular heated block embedded within a square cavity. The heat transfer coefficient increases dramatically at a fixed Rayleigh number. Moreover, when the block is positioned at the top of the cavity, the Nusselt number attains its peak value around the block.

Piña-Ortiz *et al*. [6] studied heat transfer findings from experiments and numerical calculations in a vented cavity with an interior heat. Its vertical wall is continuously and uniformly heated, while the temperature of the opposing surface is maintained constant. Using the renormalized k–ε turbulence model, the lowest deviation between the experimental and numerical Nusselt numbers was 4.1%. In their 2021 study, Bouras *et al*., [7] conducted a numerical investigation of mixed convection heat transfer within a cavity with a centrally located heat source. The upper right face section contains an output port, while the bottom is where the inlet port is situated. The obtained results indicate a significant dependence on both the rate of heat transfer and geometric parameters, as well as an interaction between buoyancy and pressure forces throughout the analyzed interval. Moussaoui *et al*., [8] employed the lattice Boltzmann technique to investigate numerically the 2D free convection heat transport in a cavity. with a hot square obstruction positioned in the center. The Rayleigh numbers range from 10^3 to 10^6 and the heat transmission and flow behaviors are generally disrupted when the square block's geometrical dimensions and horizontal location are changed. Due to the bottom block's higher surface area, the larger block dimensions have improved heat transfer. Hicham *et al*., [9] numerically analyzed mixed convection in a heating source inside a 3D vented cavity with varying positions. The results demonstrate that by optimally selecting the mentioned parameters for various inlet-outlet locations, the heat transfer and flow intensity can be significantly enhanced.

From the above review, the Numerical studies are employed to study convective heat transfer in enclosures and cavities. for either closed cavities or open cavities with hot surfaces. Previous research has examined heat transfer involving different blocks inside various shapes of cavities, as well as the use of deflectors and baffles. However, for the type of cavity used in the present study, none of the literatures have determined the optimal position for a block with a deflector. The current work involves a numerical investigation of heat transfer in a cavity with a heated block, with and without a deflector, under varying Reynolds numbers (Re) and heated block positions (H). Additionally, this work identifies the high and low temperatures inside the cavity, which are used to investigate the optimal position of the heated block under varying Reynolds numbers.

2. Numerical Simulation

2.1 Geometry

In the present work, the geometry considered is a two-dimensional channel containing a cavity, with the duct having dimensions of h = 25 cm. The cavity's depth is equal to the dimensions of the inlet passage, as shown in Figure 1. The cavity contains a heated block, which serves as a heat source. The block's dimensions are 5×5×25 cm, and it is fixed at the center in four various positions (H = 0, 5, 10, 15 cm). A deflector with a radius of R=12.5 cm is fixed in the channel. The deflector's main purpose is to direct flow towards the channel's stagnation points to enhance fluid motion there. The working fluid is assumed to be incompressible and turbulent. All the fluid's thermal properties are considered constant with the exception of density, which changes with temperature. The fluid's thermophysical characteristics are assessed at a fixed surrounding temperature Ti of 300 K. FLUENT was employed for solving the two-dimensional governing equations of energy, continuity, and the Navier-Stokes to determine the turbulent flow field, employing the Boussinesq approximation to account for buoyancy forces. The specifics of the cases are outlined in Table 1.

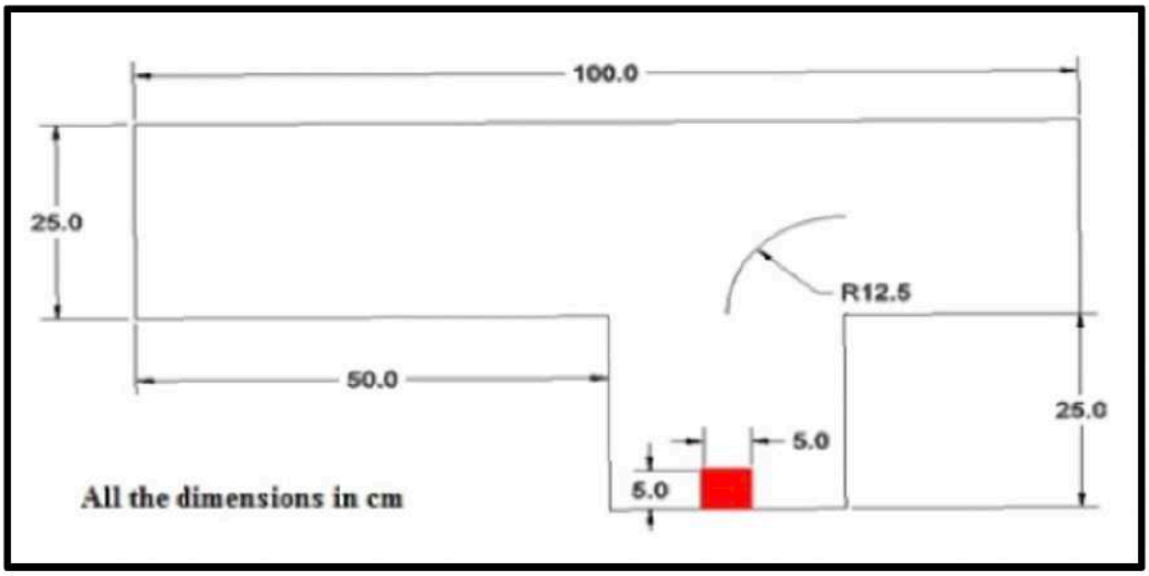


Fig. 1. Physical domain and test section dimensions

Table 1
cases under the numerical simulation

Cases without deflector			
Case No.	H(cm)	U (m/s)	q (W/m^2)
1	0	0.25,0.5,0.75,1	2000
2	5	0.25,0.5,0.75,1	2000
3	10	0.25,0.5,0.75,1	2000
4	15	0.25,0.5,0.75,1	2000
Cases with deflector			
Case No.	H(cm)	U (m/s)	q (W/m^2)
1	0	0.25,0.5,0.75,1	2000
2	5	0.25,0.5,0.75,1	2000
3	10	0.25,0.5,0.75,1	2000
4	15	0.25,0.5,0.75,1	2000

2.2 Governing equation

The governing equations used to solve the flow field include the momentum, continuity, and energy equations, as follows [10].
Continuity equation:

$$\frac{\partial \rho}{\partial t} + u\frac{\partial \rho}{\partial x} + v\frac{\partial \rho}{\partial y} + \rho\left[\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}\right] = 0 \quad (1)$$

Momentum equation:

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}\right)=\rho gx-\frac{\partial p}{\partial x}+\mu(\frac{\partial 2u}{\partial x2}+\frac{\partial 2u}{\partial y2}) \quad (2)$$

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}\right)=\rho gy-\frac{\partial p}{\partial y}+\mu(\frac{\partial 2v}{\partial x2}+\frac{\partial 2v}{\partial y2}) \quad (3)$$

Energy equation:

$$\rho\left(\frac{\partial T}{\partial t}+u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}\right)=K\left(\frac{\partial 2T}{\partial x2}+\frac{\partial 2T}{\partial y2}\right)+\mu\emptyset \quad (4)$$

2.3 Mesh Generation

A significant challenge in many computational fluid dynamics (CFD) applications is mesh modeling, as it directly affects the accuracy and convergence of CFD simulation results [11]. Before creating the mesh, the SolidWorks application program is used to draw all the geometrical elements of the block, deflector, and duct. The two most important considerations when designing a mesh are the type of mesh (unstructured or structured) and the relative size of each computational cell. An unstructured grid (3D) can consist of any polygon (2D) or polyhedron. Cells in two-dimensional unstructured grids are usually triangular, whereas cells in structured grids can be either quadrilaterals (2D) or hexahedra (3D). Structured grids, exclusively composed of quadrilaterals or hexahedra, can result in faster flow solutions as they effectively represent neighboring cells in computer memory. Conversely, unstructured grids have the benefits of being rectilinear and having a high capacity to handle complicated geometries, both of which contribute to a degree of simulation accuracy. To simulate any system on a computer, the partial differential equations (PDEs) must be discretized, yielding a finite number of points in space. One approach is to perform the calculation on a mesh or grid. Common discretization techniques, such as the finite volume method, use nearby points to calculate derivatives. Hexahedral, tetrahedral, pyramidal, and wedge-shaped mesh components are commonly used in CFD solvers. For complex geometries, creating such a mesh takes a long time and often requires modifying the model geometry. Figure 2 shows the mesh geometry, where a tetrahedral mesh with 1,054,000 elements of the finite volume method has been selected.

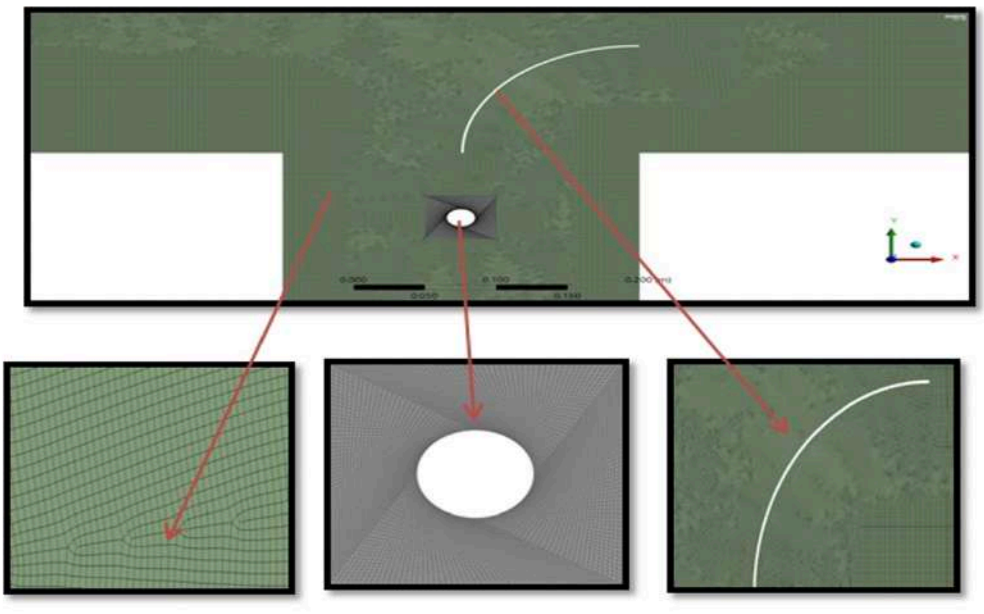


Fig. 2. Mesh generation of the channel

2.4 Boundary Conditions

The domain, physics, and boundary conditions for the cases under study are shown in Figure 3. The block with the heater is at the center of the cavity, with a fixed heat flux of 2000 W/m², and the block's height (H) is variable. At the duct surface, the velocities are Ux = Uy = 0. At the entrance, the inlet domain is represented as an inlet velocity with an inlet temperature of 300 K, and a pressure outlet is selected for the outlet with an initial pressure equal to zero. The inlet velocity is varied according to the Reynolds number values. All the channel walls are specified as adiabatic.

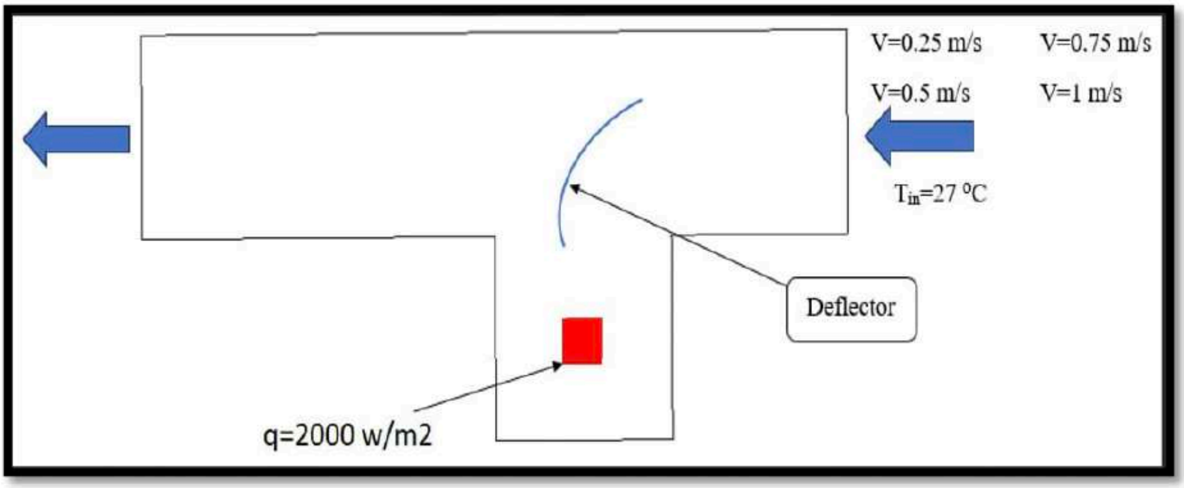


Fig. 3. Boundary condition

3. Results & Discussion

This numerical study utilizes a cavity with a heated block and deflector to compare and investigate the impact of using a deflector and the hot block's location on the fluid behavior and thermal properties of the fluid. As previously mentioned, the distance between the block and the bottom surface of the cavity is variable (H = 0, 5, 10, 15 cm).

Figures 4 and 5 illustrate how air velocity affects the distribution of airflow temperature. along the cavity for the cases with and without a deflector at two positions of the block (H = 0 and 15 cm). In general, for both heated block locations, one can observe that increasing the airflow velocity leads to decreasing temperature. This is due to the increased turbulent flow inside the passage, which increases the quantity of flow entering the cavity. The results also show the effect of adding a deflector inside the channel on temperatures, where the addition of a deflector tends to decrease the temperature. This is due to the incoming cold air pushing the hot air within the cavity, creating relationship between the hot and cold inlet air and carrying it in the downstream direction.

The temperature contour inside the channel with a cavity, with and without a deflector, is shown in Figure 6. The simulation is achieved with Re = 4000 and H = 0 and 15 cm. For all the cases, high values of temperature are found in the entire square cavity, with the maximum values recorded near the heated block compared to other regions of the channel. This is because the flow does not influence the cavity zone as much as other regions where the air momentum is directed downstream. As a result, forced convection is not effective in the cavity zone. To overcome this problem, the deflector is implemented inside the channel to bring cold air to these areas, which is clearly visible through the contour.

Figure 7 shows the velocity contour for cases with and without a deflector at different hot block positions (H = 0 and 15 cm), depending mainly on the velocity of flow at q=2000q = 2000q=2000 W/m² and Re = 4000. From the velocity contour, it is noticed that the velocity contour and flow field show a significant augmentation with the deflector inside the cavity as the Reynolds number increases. The deflector impacts the flow by directing air to the dead zone, which increases convection heat transfer between the heated block and the cold air.

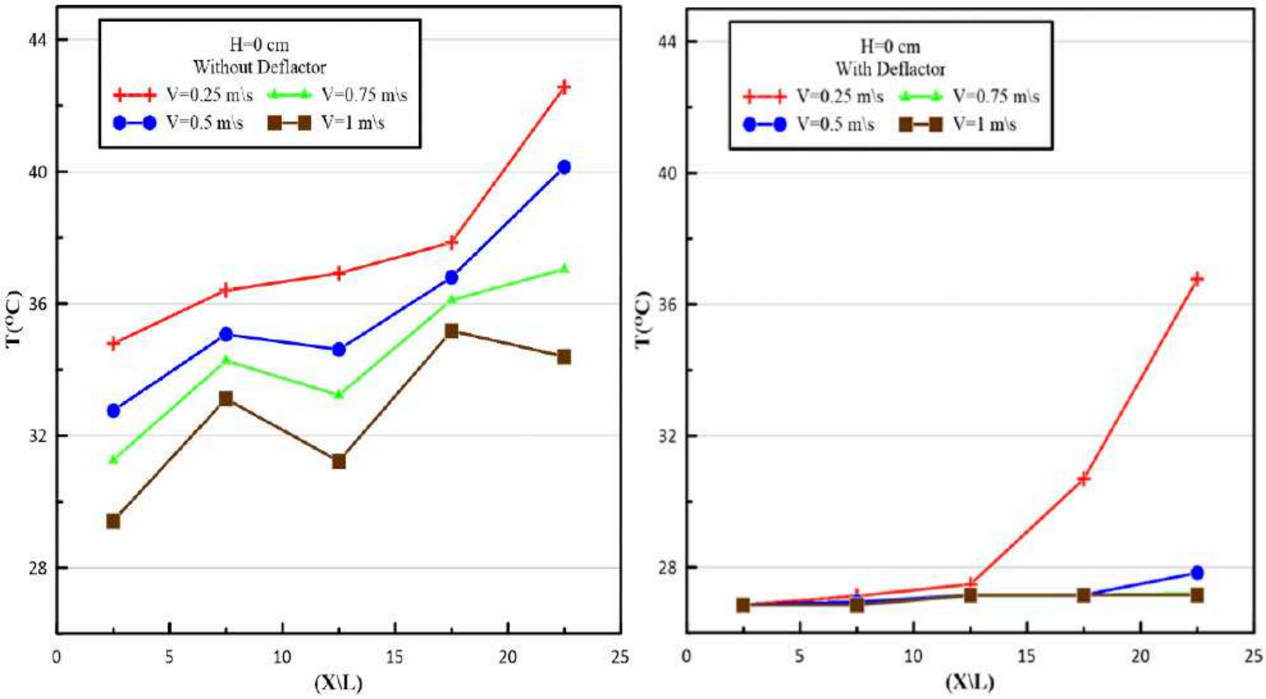


Fig. 4. Impact of velocity on the temperature inside the cavity for (H =0 cm)

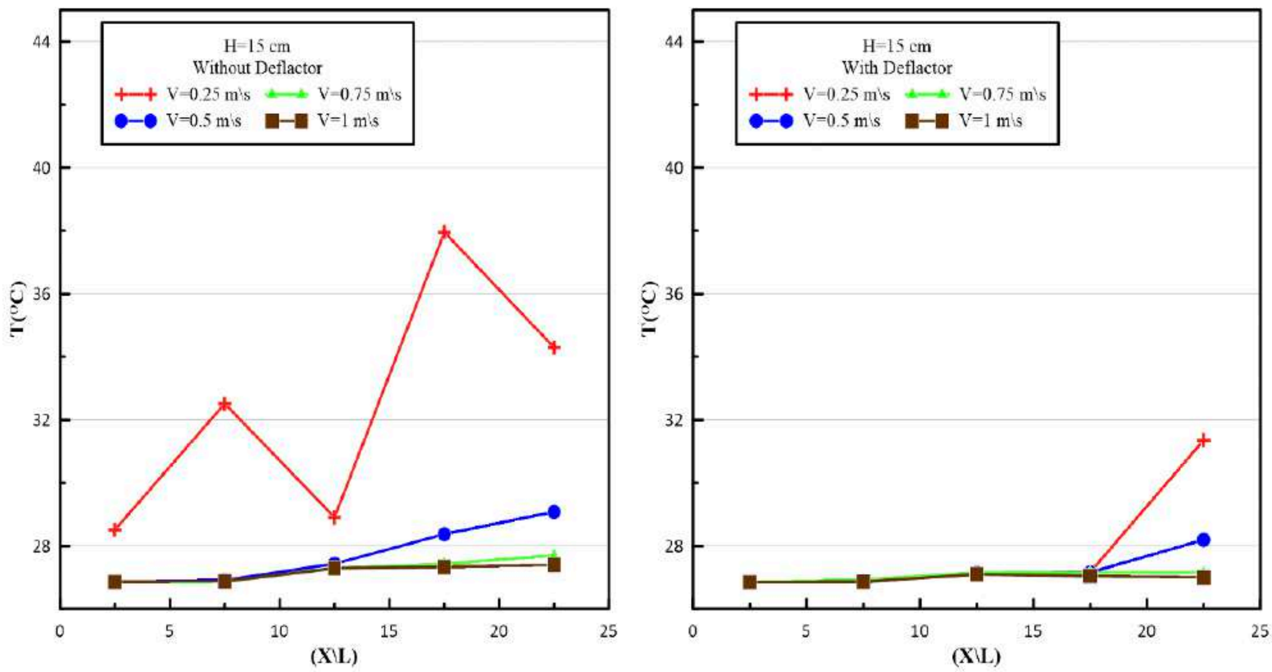
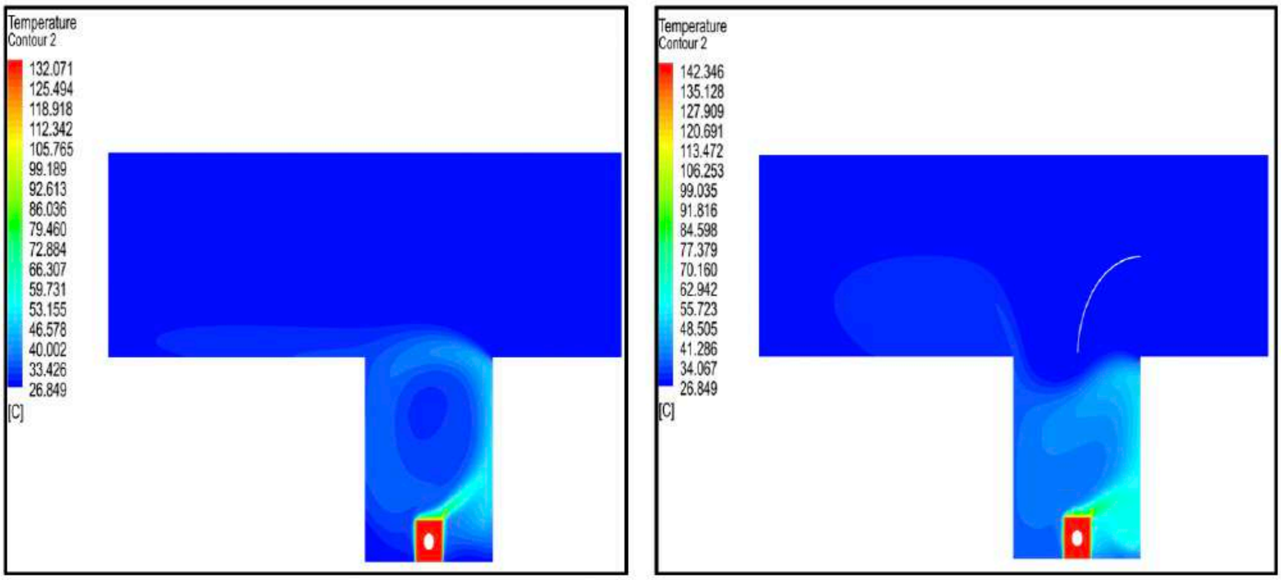
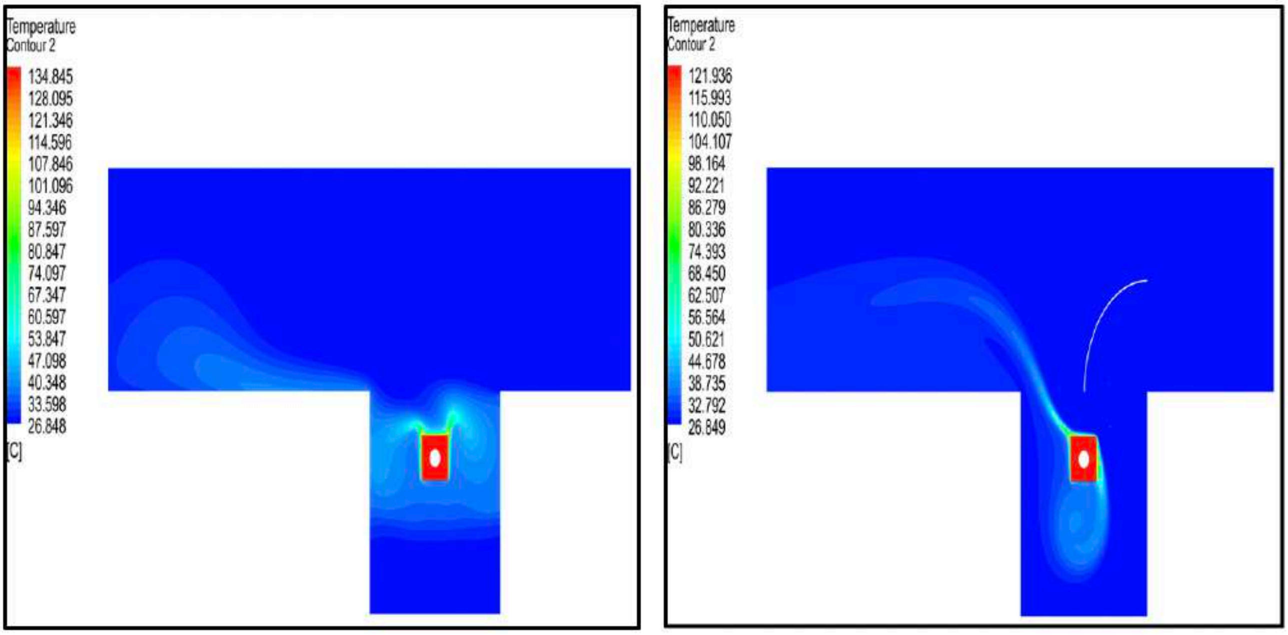


Fig. 5. Effect of air flow velocity on the temperature inside the cavity for (H =15 cm)

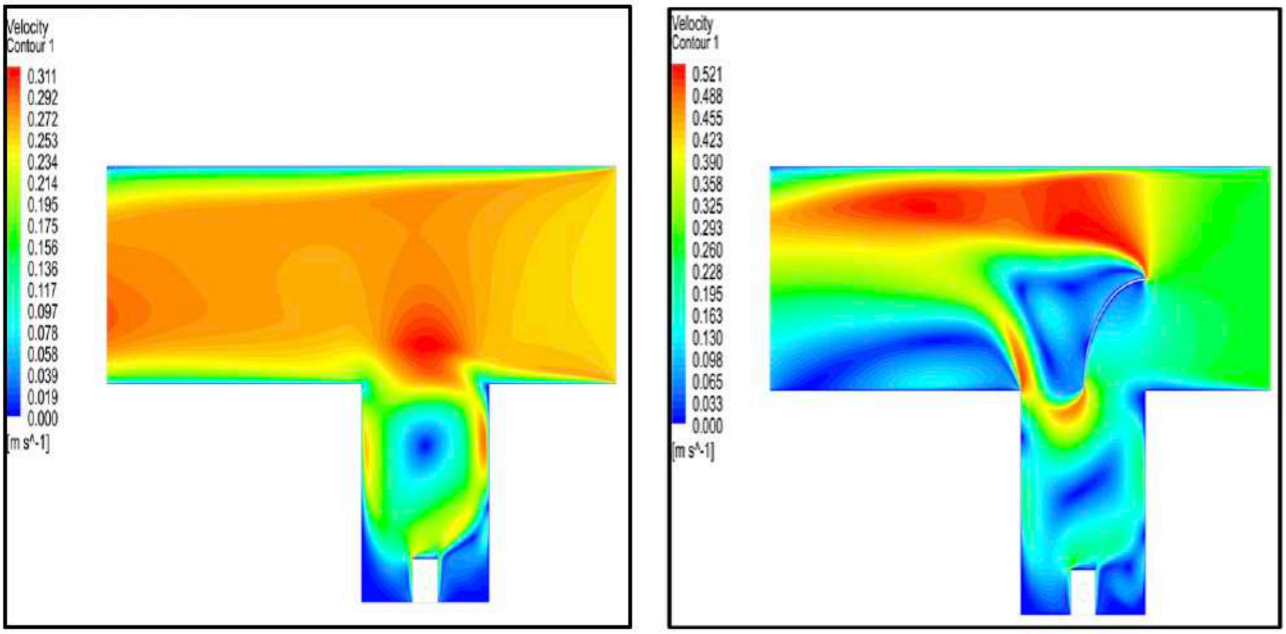


(a) with and without deflector at (H=0 cm)

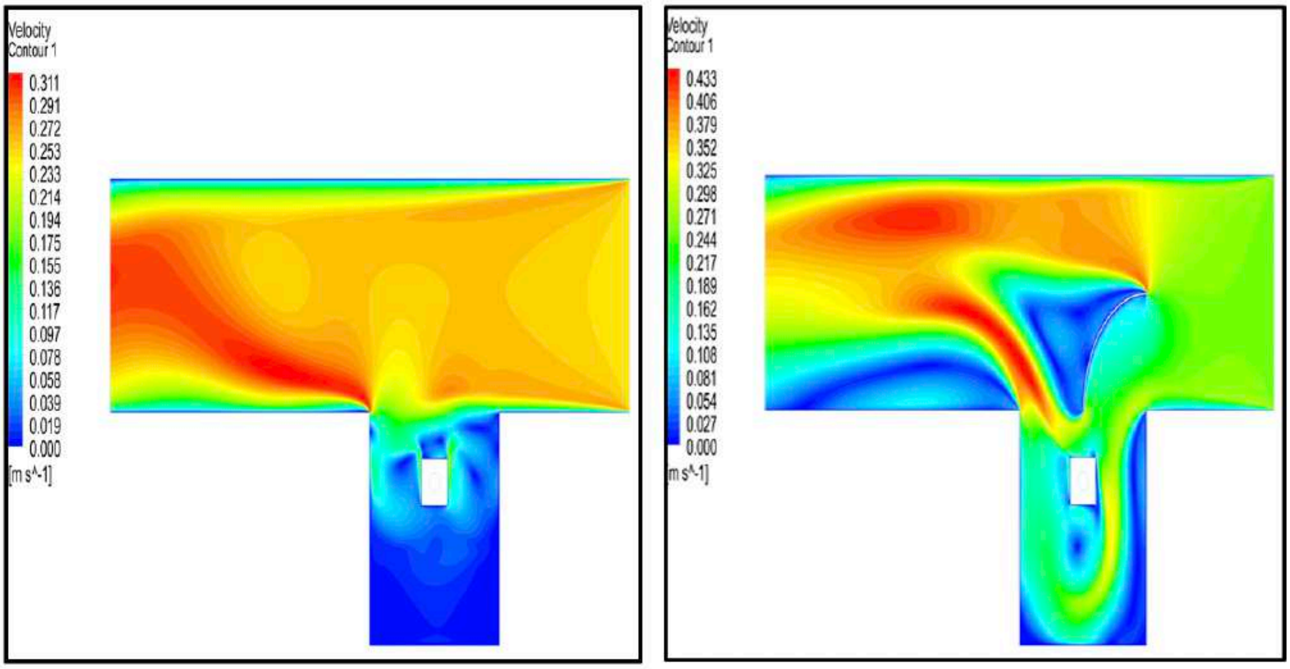


(b) with and without deflector at (H=15 cm)

Fig. 6. Thermal field contour for different (H) at (Re= 4000)



(a) with and without deflector at (H=0 cm)



(b) with and without deflector at (H=15 cm)

Fig .7. Velocity filed contour for different (H) at (Re= 4000)

Figures 8, 9, and 10 show the effect of the block's location inside the cavity on the Nusselt number with varying Reynolds numbers for two cases: with and without a deflector at q = 2000 W/m² and H = 0, 5, 10, 15 cm. It was observed from the figures that when the distance between the block and the lower surface of the cavity (H) increases, the Nusselt number (Nu) increases due to the proximity of the heated block to the air stream, enhancing the heat transfer between the block and the flow. Generally, for the case without a deflector at Re = 16000, a high value of Nux is observed at H = 15 cm compared to other cases, with a 147.57% increase in Nux recorded as H increases from 0 to 15 cm.

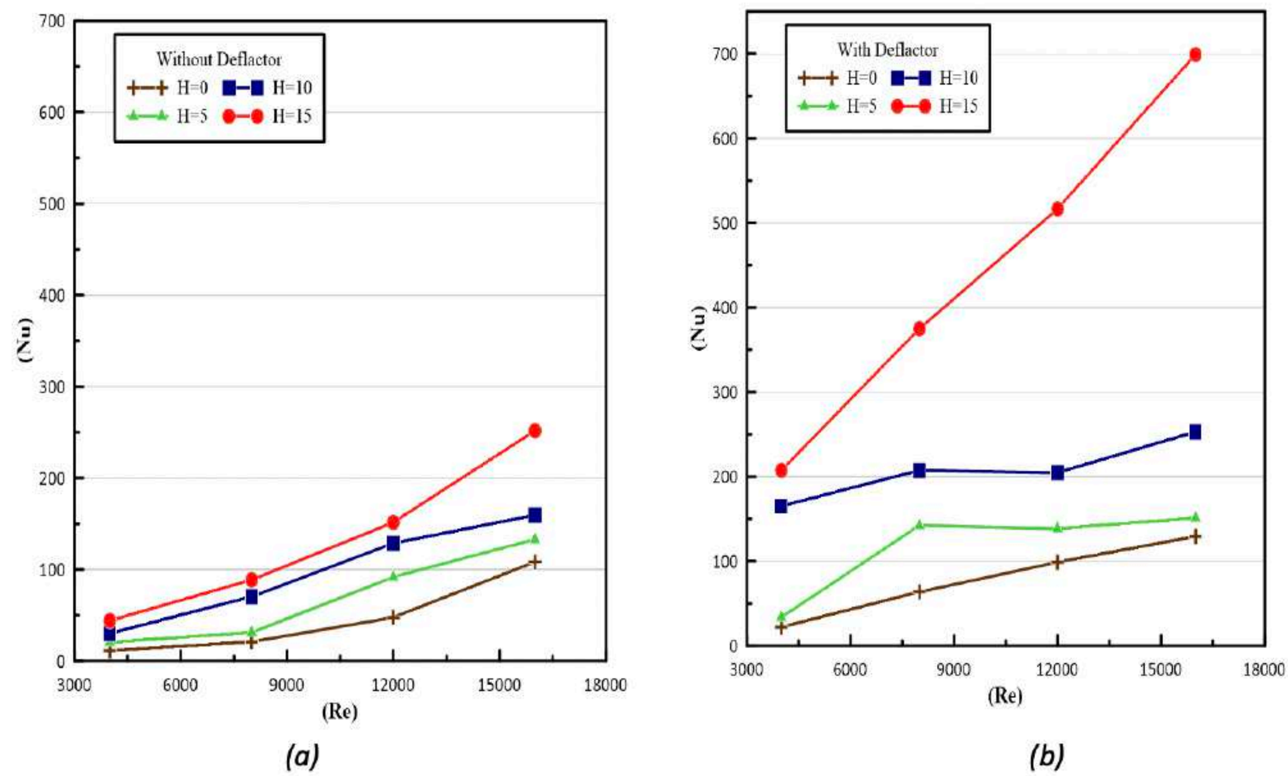


Fig. 8. (Nu) versus (Re) for different positions of the block (with and without deflector)

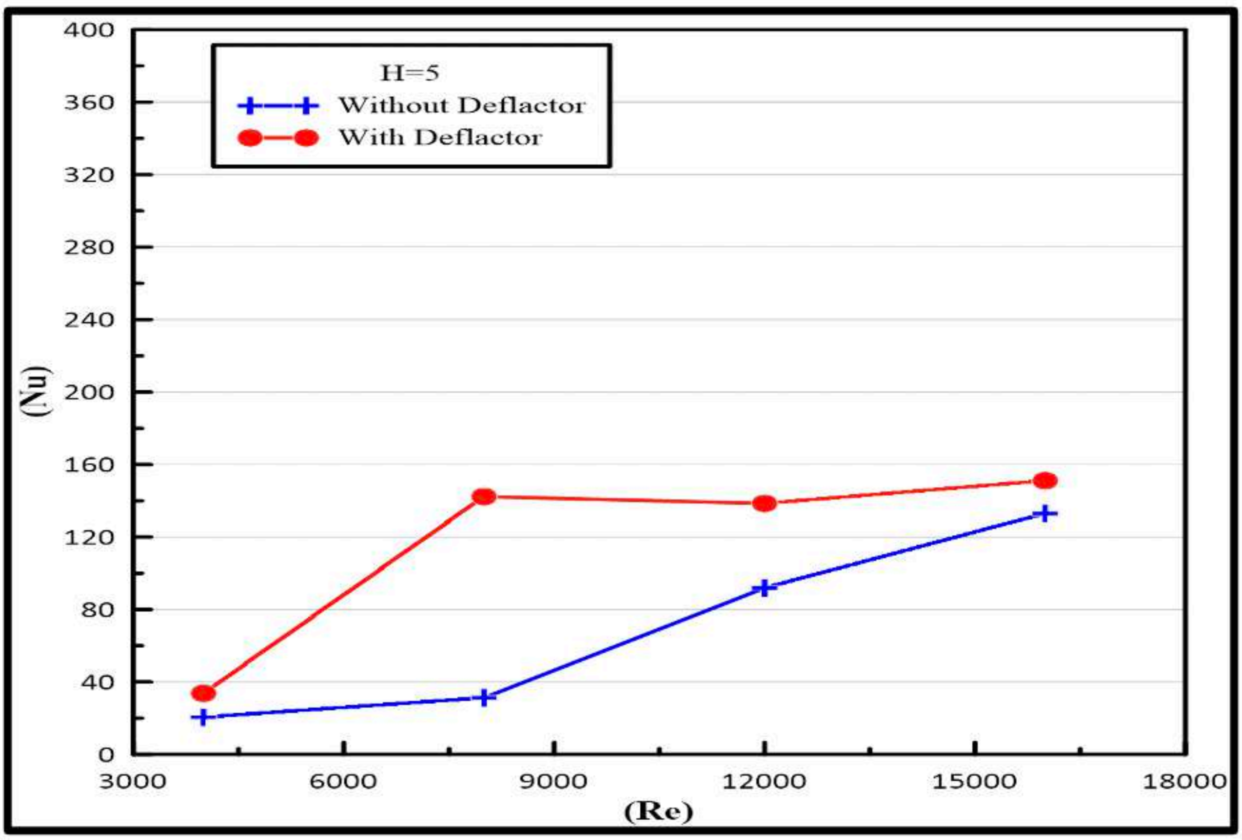
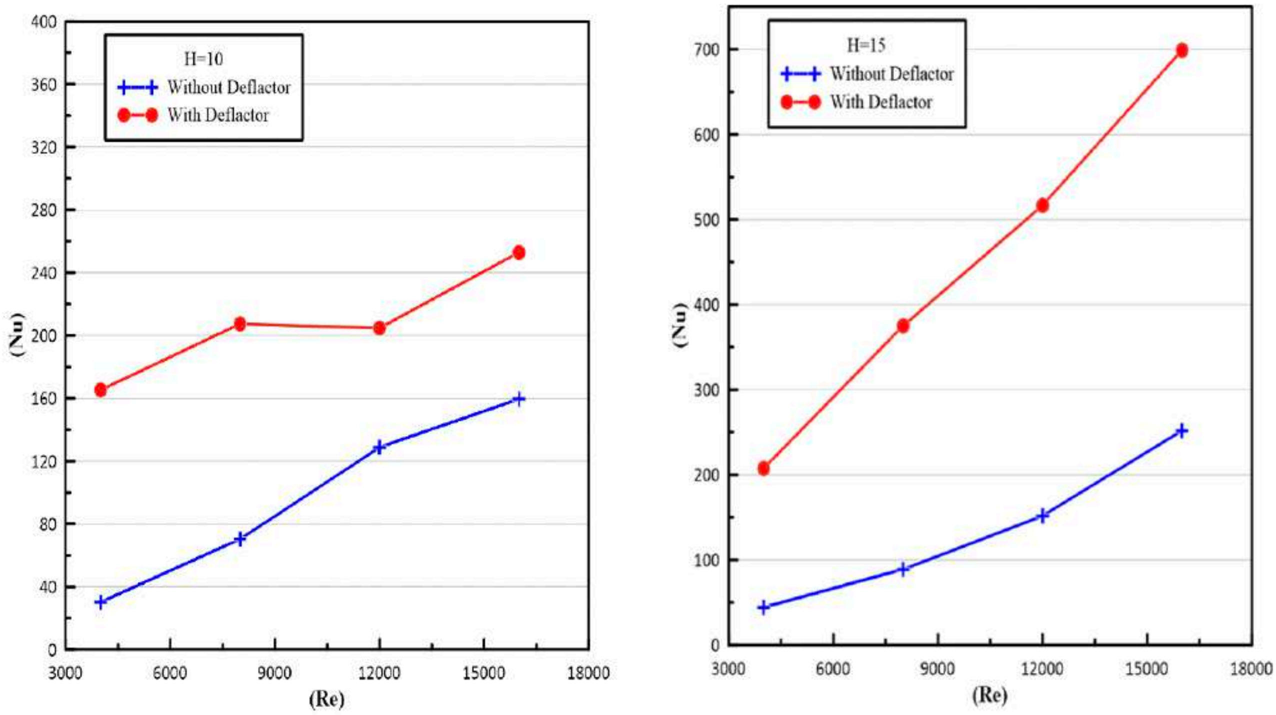


Fig. 9. (Nu) versus (Re) at (H=5 cm) for with and without deflector



(a) H=10 cm (b) H=15 cm

Fig. 10. (Nu) versus (Re) at (H = 10, 15 cm) for with and without deflector

4. Conclusions

The following conclusions can be drawn from the current study:

i. When air speed increases, its temperature decreases. This occurs because an increase in turbulent flow leads to reduced interconnections between air molecules.

ii. The presence of a deflector in a cavity enhances the process of heat transfer within it, causing the flow temperature in the cavity to decrease as the Reynolds number increases.

iii. The height of the hot block (H) inside the cavity plays an important role in determining the Nu value, where the Nu increases with increasing H. Generally, at H = 15 cm, the highest value of Nu is recorded compared to other cases.

iv. The deflector with H = 15 cm and Re = 12000 for q = 2000 W/m² provides a better enhancement in the Nusselt number by approximately (242.28%) compared with the case without a deflector.

v. It was noted that as the distance between the block and the cavity's bottom surface (H) increases, the Nusselt number (Nu) also increases

References

[1] Horikiri, K., Y. Yao, and J. Yao. 2015. "Numerical Optimisation of Thermal Comfort Improvement for Indoor Environment with Occupants and Furniture." *Energy and Buildings* 88: 303–315. https://doi.org/10.1016/j.enbuild.2014.12.015.

[2] Audaa, K., A. J. Ali, S. H. Abid Aun, and A. H. Abed. 2024. "Numerical Examination of a Cavity Containing Nanofluid with an Upper Oscillating Wall and Baffle." *Frontiers in Heat and Mass Transfer* 22 (2).

[3] Mohammed, E., and F. Abood. 2022. "Numerical Study of Mixed Convection in a Channel with an Open Cavity Involving Rotary Cylinder." *Basrah Journal of Engineering Sciences* 22 (1): 20–28. https://doi.org/10.33971/bjes.22.1.3.

[4] Kannan, M. S., and M. Gurunathan. 2022. "Numerical Investigation of Forced Convective Heat Transfer and Fluid Flow Analysis in a Double Lid Driven Staggered Cavity with Various Blocks." *Periodico* 91 (4): 323–334. https://doi.org/10.37896/pd91.4/91421.

[5] Oyewola, O. M., S. I. Afolabi, O. S. Ismail, and M. O. Olasinde. 2022. "Convective Heat Transfer in a Square Cavity with Embedded Circular Heated Block at Different Positions." *Frontiers in Heat and Mass Transfer* 18: 1–7. https://doi.org/10.5098/hmt.18.31.

[6] Hinojosa, J. F., and J. M. A. Navarro. 2017. "Experimental and Numerical Study of Turbulent Mixed Convection in a Cavity with an Internal Heat Source." https://doi.org/10.1177/1744259117735451.

[7] Bouras, A., S. Bouabdallah, B. Ghernaout, M. Ariel, Y. Cherif, and E. Sassine. 2021. "3D Numerical Simulation of Turbulent Mixed Convection in a Cubical Cavity Containing a Hot Block." *Journal of Applied Fluid Mechanics* 14 (6): 1869–1880. https://doi.org/10.47176/jafm.14.06.26043.

[8] Moussaoui, M. A., E. B. Lahmer, Y. Admi, and A. Mezrhab. 2019. "Natural Convection Heat Transfer in a Square Enclosure with an Inside Hot Block." In *2019 International Conference on Wireless Technologies, Embedded and Intelligent Systems (WITS)*, 1–6. Piscataway, NJ: IEEE.

[9] Doghmi, H., B. Abourida, L. Belarche, M. Sannad, and M. Ouzaouit. 2018. "Numerical Study of Mixed Convection Inside a Three-Dimensional Ventilated Cavity in the Presence of an Isothermal Heating Block." *International Journal of Heat and Technology* 36 (2): 447–456. https://doi.org/10.18280/ijht.360209.

[10] de Sousa, J. V. N., et al. 2020. "On the Study of Autonomous Underwater Vehicles by Computational Fluid-Dynamics." *Open Journal of Fluid Dynamics* 10 (1). https://doi.org/10.4236/ojfd.2020.101005.

[11] Hadday, A. N. 2022. *Assessment of the Enhancement of Forced Convection Heat Transfer over Heated Blocks with Slot Jets and Baffles*.