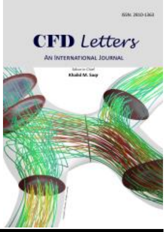
CFD Letters

Journal homepage:
https://semarakilmu.com.my/journals/index.php/CFD_Letters/index
ISSN: 2180-1363




# Application of Extended Eddy-viscosity and Elliptic-Relaxation Approaches to Turbulent Convective Flow in a Partially Divided Cavity

Gunarjo Suryanto Budi[1,*], Sasa Kenjeres[2]

[1] Department of Physics Education, University of Palangka Raya, Palangka Raya, Central Kalimantan, Indonesia
[2] Department of Chemical Engineering, Faculty of Applied Sciences, Delft University of Technology, Van der Maasweg 9, 2629 HZ Delft, Netherlands

**ARTICLE INFO**

***Article history:***
Received 13 December 2023
Received in revised form 12 January 2024
Accepted 11 February 2024
Available online 31 May 2024

***Keywords:***
Buoyancy; convective flow; eddy-viscosity; elliptic relaxation; algebraic approach

**ABSTRACT**

The paper reports on the numerical turbulence model in predicting mass, momentum and heat transfer in a partially divided cavity heated from the side using buoyancy-extended eddy-viscosity and elliptic relaxation approach with the algebraic expressions for the Reynold stress tensor and turbulent heat flux vector. The CDS (central differencing scheme) and LUDS (linear upwind differencing scheme) were used as the discretization method and the governing equations were solved using the finite volume method and Navier-Stokes solver. Validation of the model has been carried out by experimental data of convective flow in the cavity as well as by numerical data DNS (direct numerical simulation). The model agrees very well with the experiment and DNS and it is also able to demonstrate the performance which is comparable to that of the previous advanced second-moment closure model (SMC) in the literature. The results show that the model is suitable for use in simulations of the turbulent convective flow in a cavity with partition and it has the potential to be applied to more complex cavities and a wide range of turbulence levels.

## 1. Introduction

Buoyancy convective flow in a side heated cavity has been extensively studied by Gunarjo [1], Mayeli and Sheard [2], and others. All researchers found that the flows consist of very thin layers in the near conducting walls with intensive heat and mass transfer, characterized by the strong jet with steep gradients of all properties. For turbulent regimes, which occur in the range of Rayleigh numbers higher than $10^9$, the convective flow carries an anisotropic feature that is challenging to model Ampofo [3], Dol *et al.,* [4]. Additional complexity lies in the fact that the flow in the proximity of the thermally active walls is characterized by a coexistence of the laminar, transitional, and finally, fully developed turbulent regimes, which are notoriously difficult to model.

Description of complete features of turbulent flow can be represented by Reynolds-Averaged Navier-Stokes equations (known as RANS). The method has been studied extensively leading to turbulence models that are reasonably simple, numerically robust, applicable for industrial and

* *Corresponding author.*
*E-mail address: gunarjo.budi@chem.upr.ac.id (Gunarjo Suryanto Budi)*

https://doi.org/10.37934/cfdl.16.10.111

technological applications and reliable for a wide range of turbulence levels. Altac and Ugurlubilek [5] assessed the turbulence model in natural convection within 2D and 3D rectangles and the research revealed that the 3D approach using the RANS model revealed a higher accuracy in predicting the mean Nusselt number $(Nu)$. In addition, an assessment of turbulence models has been conducted by Pina-Ortiz *et al.,* [6] for natural convective flow in an open tilted cavity. The standard *k-ε* and its modifications were applied and it was demonstrated that the models fit experimental data quite well. Similarly, Karimpour and Venayagamoorthy [7] simulated the stratified flow of a 1-D channel using a *k-ε* model with parameterization of the turbulent Prandl number, which takes into account the presence of wall boundaries. The author evaluated the profiles of velocity and density for various Richardson numbers. It was found that the proposed Prandl number produced good agreement with direct numerical simulation. In addition, Lazerom *et al.,* [8] have derived a formula for Reynold stress and heat flux for stratified flows. Although they are a mutual couple between Reynolds stress and heat flux and contain a nonlinear form with many coefficients, it turned out to be an explicit algebraic approach that is applied in homogeneous shear flow and turbulent channel flow. The model has demonstrated relatively good performance in terms of accuracy, robustness and reliability.

Another simulation technique is Large Eddy Simulation (LES), which is based on the spatial filtering of the governing equations, resulting in additional sub-grid (SGS) models that need to be introduced (similar to the RANS). This approach is between the Direct Numerical Simulations (DNS), where all spatial and temporal scales of the flow need to be captured, resulting in very high computational costs, and already mentioned RANS. Ma and He [9] studied convective flow, heat, and mass transfer around a horizontal cylinder and it was found that the results were acceptably accurate despite the sensitivity toward the size of the mesh. A good performance of the LES approach was also reported by Ortiz *et al.,* [10] in assessing the correlation of Prandtl number in momentum, heat and mass transfer.

Due to the steep temperature gradient in the thin boundary layer along the thermally active walls, modeling the flow in this region may fail to properly capture the real features of the turbulence. To correctly calculate the convective flow in these regions, many researchers have applied dimensionless wall distance, namely $y^+$as a damping function. However, the success of using the damping function may be due to the fine-tuning of the model coefficients and is associated with a lack of generality. In addition, the damping function was not able to predict the effect of wall blockage which is mostly present in the convective flows. A similar approach was proposed by Jones and Launder [11] who studied the characteristic of near-wall flow using the damping function of turbulent Reynolds number. Since the turbulent Reynolds number is a function of kinetic energy, viscosity, and dissipation, this damping function has a more physical basis than the wall-distance one.

In this study, the wall modelling treatment is based on the advanced elliptic relaxation approach of Durbin [12]. It was also demonstrated by Gunarjo [1] that the elliptic relaxation approach worked very well for various flow situations where thermal buoyancy plays a significant role. Dehoux *et al.,* [13] derived algebraic heat flux from a differential flux model and blended it with an elliptic relaxation approach. A comparison was made between the Generalized Gradient Diffusion Hypothesis (GGDH) and the Algebraic Flux Model (AFM) with and without elliptic blending. Models with elliptic blending performed better, especially in the near-wall region. The elliptic relaxation approach with second-moment closure was also studied by Das [14] for stratified flow in channels. The introduction of a new parameter, namely buoyancy length scale, has produced a good agreement with DNS; however, the model was quite complex, and in the current study, Das [15] simplified the previous model by discarding the elliptic relaxation of scalar flux and elliptic relaxation only in the Reynolds stress part.

introduced a new parameter, namely the buoyancy length scale, which produced a good agreement with DNS. It is very complex for the industry.

The introduction of a partition in different configurations has encouraged researchers to elaborate more research for convective flow in the cavity. In reality, it has a wide range of applications, such as cooling systems of devices for electronic instruments with protected regions from excessive heat, building insulation, solar collectors, double-glazed windows, etc. One advantage of the presence of partition, which could be oriented and located accordingly, is its ability to affect flow, heat transfer, and turbulence, making it an efficient way of passive heat transfer control

The presence of partitions with various orientations on convective flow in the cavity was investigated numerically by Kruger and Pretorius [16]. The authors applied two types of RANS models, namely the basic $k$-$\varepsilon$ and low-$Re$ $k$-$\varepsilon$ models. A similar result was found between the Low-$Re$ $k$-$\varepsilon$ models and the experiment, while the basic $k$-$\varepsilon$ model produced a discrepancy when compared to the experiment. Observation of the presence of a partition on convective flow in a cavity was done by Al-Krmah *et al.,* [17]. The partial partition was attached to a vertical non-conducting wall of the cavity and it was horizontally oriented. It was observed that the partition could be used as a control to the heat transfer to keep the designed temperature range for different power output. Al Amiri *et al.,* [18] carried out a numerical study of convective flow and heat transfer in a square enclosure with partitions over an extensive range of Rayleigh numbers, i.e from of $10^4$ to $5 \times 10^7$. It was demonstrated that the position and dimension of the partition have an important effect on the heat transfer and the levels of turbulence.

The main purpose of the present research is to apply the RANS type turbulent model that is based on buoyancy-extended eddy-viscosity and elliptic relaxation approaches to turbulent convective flow in a partially divided cavity. The resulting velocity, temperature, and temperature fields are analysed, as well as the resulting the heat transfer (in the forms of the local and integral Nusselt numbers). In order to accurately solve turbulent convective flows in a cavity it requires a model which is robust, simple, easy to be implemented in any computer code and applicable for industry and therefore this research is aimed to meet the need.

## 2. Methodology

### *2.1 Numerical Method*

The set of differential equations was solved numerically by finite-volume based Navier-Stokes solvers in the structured, non-orthogonal geometries, with Cartesian vector and tensor components and the collocated arrangement of Ferziger and Peric [19]. The discretization of the convective parts of the turbulence parameters was carried out using the second-order linear upwind scheme (LUDS) while the discretization of the diffusive terms was done using the second-order central-differencing-scheme (CDS). The coupling between the velocity and pressure fields was done using the SIMPLE algorithm. To obtain a stable simulation and convergent solution, the under-relaxation method and false time steps were applied with a variation of the maximum absolute values between two successive iterations was less than $10^{-6}$. The computer code was developed and employed by Kenjeres [20] for many years and it performance is very good, robust, applicable for a number of turbulent flows and it is insensitive for moderate mesh size.

*2.2 Turbulent Model*

The two-dimensional conservation equation for mass, momentum and energy equations for incompressible fluid in which the variation of density ρ is treated using Boussinesq approximation can be written as:

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

$$\frac{\partial \rho U_i}{\partial t} + U_j \frac{\partial \rho U_i}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \rho\overline{u_i u_j}\right] + \rho\beta g_i\,(\mathrm{T} - T_{ref}) \tag{2}$$

$$\frac{\partial \rho T}{\partial t} + U_j \frac{\partial \rho T}{\partial x_j} = \frac{\partial}{\partial x_i}\left(\frac{\mu}{P_r}\frac{\partial T}{\partial x_i} - \rho\,\overline{\theta u_i}\right) \tag{3}$$

Where $U_i$ is velocity vector, $T$ is temperature, $g_i$ is gravitational vector,$P$ is production, β denotes thermal expansion coefficient, $\overline{u_i u_j}$ is Reynold stress, $\overline{\theta u_j}$ is heat flux and $P_r$ is Prandl number.

The equation of turbulent kinetic energy $k$ and the dissipation ε may be written as:

$$\frac{Dk}{Dt} = -\overline{u_i u_k}\frac{\partial U_i}{\partial x_k} + \rho\beta g_i \overline{\theta u_i} - \varepsilon + \frac{\partial}{\partial x_k}\left[\left(\nu + \frac{\nu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] \tag{4}$$

$$\frac{D\varepsilon}{Dt} = \frac{C_1(Pi+G) - C_2\varepsilon}{\tau} + \frac{\partial}{\partial x_j}\left[\left(\nu + \frac{\nu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] \tag{5}$$

Where $\nu_t$ is turbulent viscosity, ν is kinematic viscosity, τ is characteristic time scale.

For a better wall treatment, $v^2$ the scalar fluctuation of turbulence and elliptic relaxation parameter $f$ of Durbin [12] are introduced. Application of this approach can capture the near wall characteristic, enable of using moderate grid resolution as well as represent a correct anisotropic of the Reynold stress tensors and heat flux vector, as demonstrated by Kenjeres *et al.,* [21]. The governing transport equation for the square of reference velocity of turbulent $v^2$, $f$ and temperature variance $\theta^2$ are expressed as follows:

$$\frac{D\overline{v^2}}{Dt} = kf - \varepsilon\frac{\overline{v^2}}{k} + \frac{\partial}{\partial x_j}\left[\left(\nu + \frac{\nu_t}{\sigma_{v^2}}\right)\frac{\partial \overline{v^2}}{\partial x_j}\right] \tag{6}$$

$$L^2\frac{\partial^2 f}{\partial x_j^2} - f = \frac{(1-C_1)}{\tau}\left(\frac{2}{3} - \frac{\overline{v^2}}{k}\right) - C_2\frac{Pi+G}{k} \tag{7}$$

$$\frac{D\overline{\theta^2}}{Dt} = -2\overline{\theta u}_j\frac{\partial T}{\partial x_j} - 2\varepsilon_\theta + \frac{\partial}{\partial x_j}\left[\left(\nu + \frac{\nu_t}{\sigma_{\theta^2}}\right)\frac{\partial \overline{\theta^2}}{\partial x_j}\right] \tag{8}$$

The dissipation of temperature variance is calculated from the thermal to mechanical time scale ratio which is relatively simple as compared to modelling and solving the transport its transport equation. The Production $P_i$, $G$, dissipation of temperature variance, the turbulent viscosity, length scale and time scale is formulated as

$$P = -\overline{u_i u_j}\frac{\partial U_i}{\partial x_j}, \quad G = -\beta g_i \overline{\theta u_i}, \quad \varepsilon_\theta = \overline{\theta^2}/2\tau_\theta \quad v_t = C_\mu^D v^2 T \quad L = C_L \max\left[\frac{k^{\frac{3}{2}}}{\varepsilon}, C_\eta\left(\frac{v^3}{\varepsilon}\right)^{\frac{1}{4}}\right]$$

and time scal $\tau = \max\left[\frac{k}{\varepsilon}, C_T\left(\frac{v}{\varepsilon}\right)^{\frac{1}{2}}\right]$

The Reynolds Averaged Navier Stokes (RANS) equations introduce the second-moment Reynolds stress tensor the turbulent heat flux. They were modeled by using the buoyancy extended eddy viscosity/diffusivity concept and expressed in terms of averaged velocity and temperature gradient as follows:

$$\overline{u_i u_j} = \frac{2}{3}k\delta_{ij} - C_\mu \tau \overline{v^2}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) + C_\theta \tau \beta g_j \overline{\theta u_i} \tag{9}$$

$$\overline{\theta u_i} = -C_\theta \tau\left(\varsigma \overline{u_i u_j}\frac{\partial T}{\partial x_j} + \xi \overline{\theta u_j}\frac{\partial U_i}{\partial x_j} + \beta g_i \overline{\theta^2}\right) \tag{10}$$

The coefficients of the model are: $C_1 = 1.4, C_2 = 0.6, C_\mu = 0.22, C_L = 0.2, C_T = 6, C_\theta = 0.5, C\eta = 50$. Eq. (9) and Eq. (10) are making RANS model closed. The equations are not only relatively simple but also have a strong physics basis.

The picture of the boundary condition is shown in Figure 1. It can be seen from the figure that the boundary condition of the solid wall for temperature is Dirichlet boundary condition, where the value of $T = T_{wall}$ and for other parameters no slip condition is applied: $U = V = k = \overline{\theta^2} = \overline{v^2} = 0$, for the dissipation rate is $\varepsilon = 2\gamma\frac{k}{y_n^2}$ , and for $f = -20\gamma^2\frac{\overline{v^2}}{\epsilon y_n^4}$ where $y_n = \frac{\Delta y_1}{(y_{max} - y_{min})}$ is a non-dimensional.

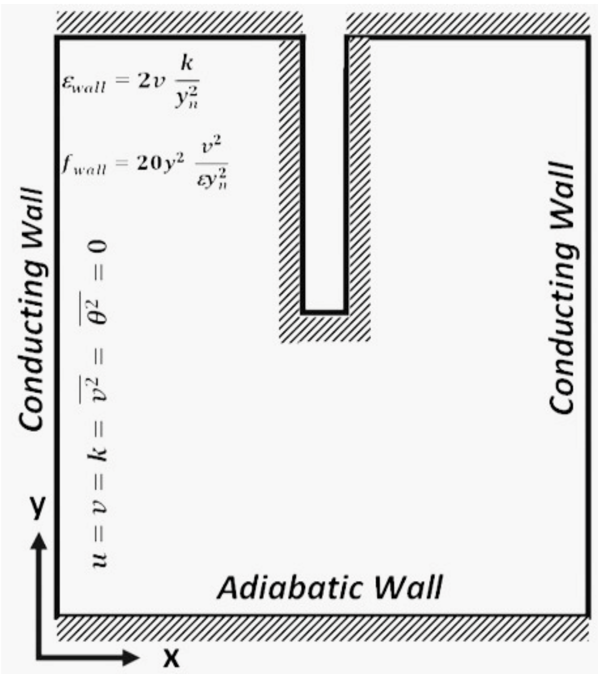



**Fig. 1.** Boundary condition of the solid wall

## 3. Results

The simulation was carried out with collocated grid in which the non-dimensional wall-distance, the distance between the first row of cells and the wall $y_n$ is $10^{-3}$ with the grid expansion factor 1.1. Prandl number for air $P_r$ = 0.71 and the Rayleigh number $R_a$ is 5 x $10^9$. Validation of the pressure-scrambling term of the turbulent heat flux components by the numerical data DNS and the previous model in literature is illustrated in Figure 2. It was shown in the previous study of [1] that the pressure-scrambling contribution, which is decomposed into slow, rapid, buoyant, and wall parts, is dominant in the budget of the turbulent heat flux. It is therefore important to correctly model and calculate the pressure scrambling term to properly capture the characteristic of turbulence. As demonstrated by the present model, it can be seen that the profiles of pressure scrambling of both horizontal and vertical components are well predicted and the model is also able to capture the peak value in the near wall region, Figure 2 and Figure 3. It is concluded that the present model shows a good agreement with the DNS by Versteegh [22]. In addition, the present model improves the performance of the previous model by Dol *et al.,* [4], especially in predicting the peak value in the near wall region.

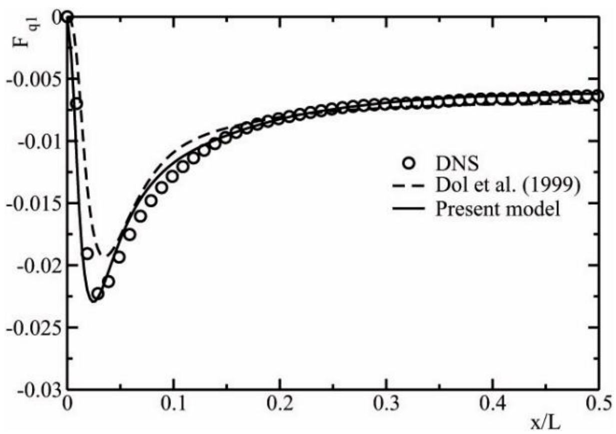



**Fig. 2.** Comparison of DNS with simulation by Dol *et al.,* [4] and the present model for the horizontal component of pressure scrambling

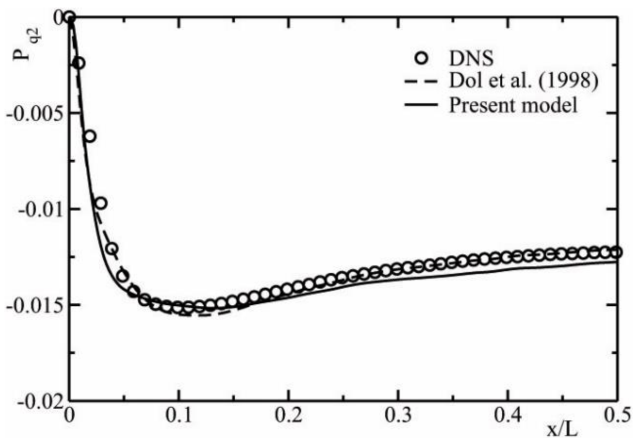



**Fig. 3.** Comparison of DNS with simulation by Dol *et al.,* [4] and the present model for the vertical component of pressure scrambling

The performance of the present model for the forced convection in a cavity is illustrated in Figure 4 in which the calculation is compared with the experiment by Blay [23] for horizontal velocity along the center line. It can be seen from the figure that the present model agrees relatively well with the experiment, especially in the region near the ceiling where the velocity reaches its maximum. The present model is also able to capture the pattern of the velocity in the area where the values are very low. However, slightly different values are observed in the mid-plane and this might be due to the sensitivity of the apparatus in measuring the relatively low velocity.

Figure 5 shows the plots of the vertical velocity of natural convection in a cubical cavity by the present model, by the advanced Second Moment Closure (SMC) of Dol and Hanjalic [25], and experiment by Opstelten *et al.,* [24]. It can be seen that the present model and the second-moment closure (SMC) calculated very well the vertical velocity measured in the experiment, especially in the near-wall region. In addition, the second-moment closure can predict the peak of velocity, while the present model shows a small over-prediction. However, the present model agrees with the experiment in the middle of the cavity while the second moment closure (SMC) under-predicting velocity. Although the present model is quite simple and employs modest grid resolution with low computational effort, its performance is still comparably similar to the advanced second-moment closure (SMC), which is significantly more complex and requires higher computational effort. The reason for the success of here presented model is due to the accurate modelling of the transport terms of the turbulent heat flux, especially the pressure scrambling contributions in the thin layer in the near wall region where is the heat transfer dominant.

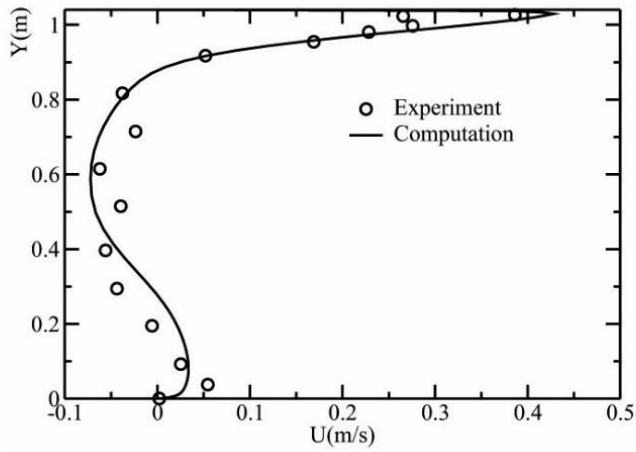



**Fig. 4.** Comparison between the experiment by Blay [23] with the present model for the horizontal velocity along the center of the cavity

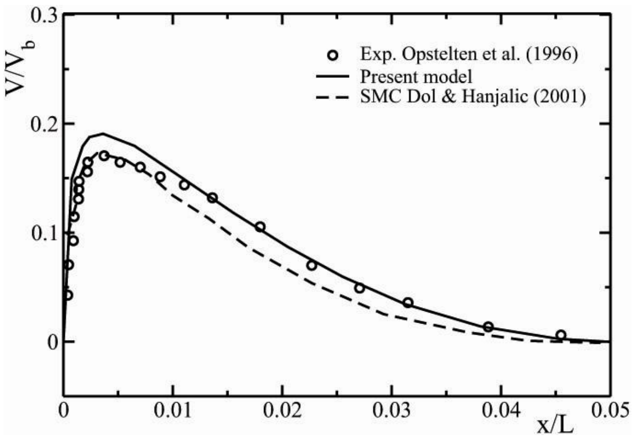



**Fig. 5.** Comparison between the experiment by Opstelten [24], Second Moment Closure (SMC), and the present model for the vertical velocity along the center of the cavity

The results of the convective flow in the partially divided cavity of different partition height ratios $y_H$ in the center of the vertical plane are shown in Figures 6, 7, 8, and 9. Velocity vectors, isolines of temperature, and distribution of the local Nusselt number are presented for different height ratios, namely $y_H$ = 0.25, 0.50, and 0.75. Figures 6a, 6b, and 6c show the characteristic velocity vectors portraying the strong thin jets in the hot and the cold walls due to the presence of buoyancy. Unlike the jet in the cold wall which flows from the ceiling to the floor, the upwards jet in the hot wall flows from the floor to reach a certain position depending on the height of the partition, the jet creates a circulatory flow to the bottom edge of the partition. Only a small part of the flow continues to reach the ceiling and leaves the left corner empty therefore the heat transfer in the region is relatively weak because of the presence of the partition which blocks the upward flow in the hot wall due to the formation of a stable thermal stratification layer, especially for the largest partition height ratio $y_H$ = 0.75. This partition can control the flow which is a similar finding reported by Al Amiri [18]. In addition, turbulence is observed in the near wall region and the bottom edge of the partition. When the ratio $y_H$ is 0.75, an upwards strong jet is created along the partition, while in the case of $y_H$ equals to 0.5 and 0.25 the flow is spread to the top corner of the cavity and splits into two parts, some are directed along the cold wall and the rest are directed towards the ceiling forming a wake region. It is noticeable two symmetrical circulatory flows were created at the bottom part just below the partition.

The isotherms are presented in Figure 7a. – 7c. for the height ratio $y_H$ = 0.25, 0.5 and 0.75. It is observed that vertical isotherm is found in the thin layers near the conducting walls while the remaining part is horizontal showing the mechanism of heat transfer is convection. The pattern is similar to that reported by Kenjeres *et al.,* [26] in the two-dimensional enclosure with partition.

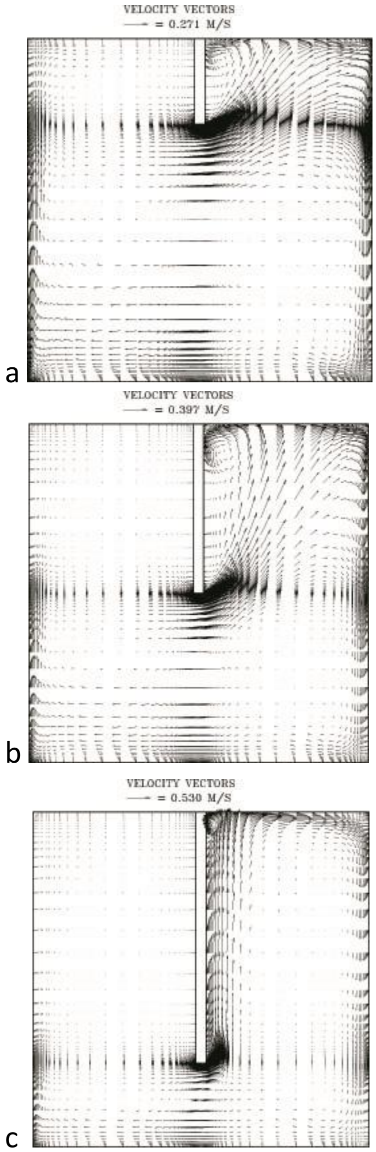



**Fig. 6.** Velocity vector plots at height ratio $y_H$= $y/H$ (a) $y_H$ = 0.25 (b) = 0.50 (c) $y_H$ = 0.75

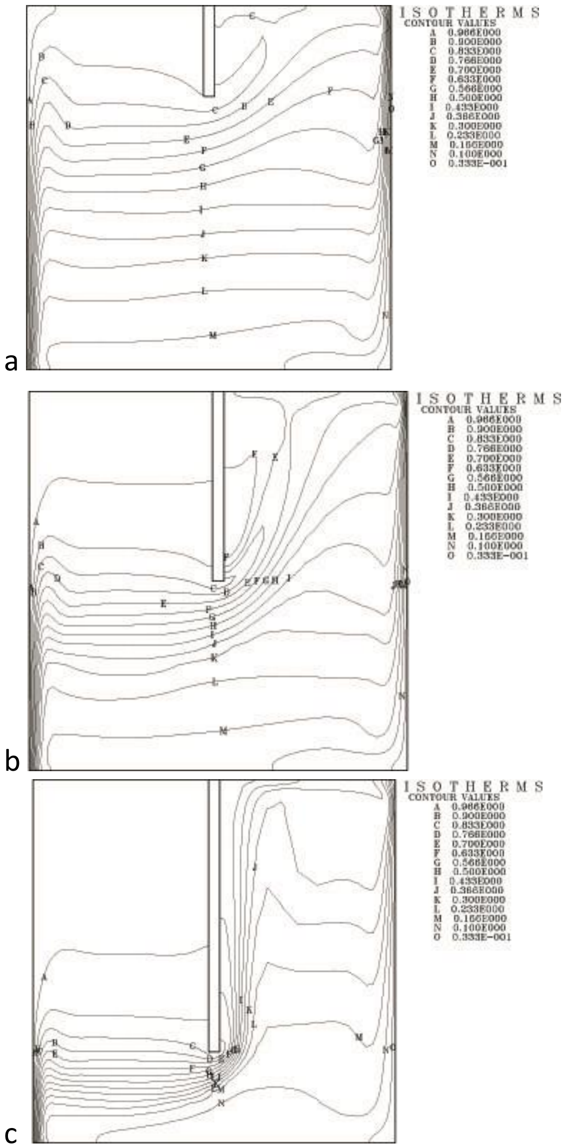



**Fig. 7.** Plot of temperature distribution contour at height ratio $y_H$= $y/H$ (a) $y_H$ = 0.25 (b) = 0.50 (c) $y_H$ = 0.75

Figures 8 and 9 are the 3D contour plots of the local Nusselt number along the conducting walls of different height ratios, respectively. It is noted that the intensity of heat transfer and turbulence occurs in the bottom part of the hot wall and decreases significantly after reaching the point of the bottom part of the partition. A similar pattern is observed for the Nusselt number in the cold wall, in which the maximum numbers occur in the upper part and decline until the lower part of the partition. Peaks in the middle of the wall are observed and this is due to the jet and heat generated in the old wall.

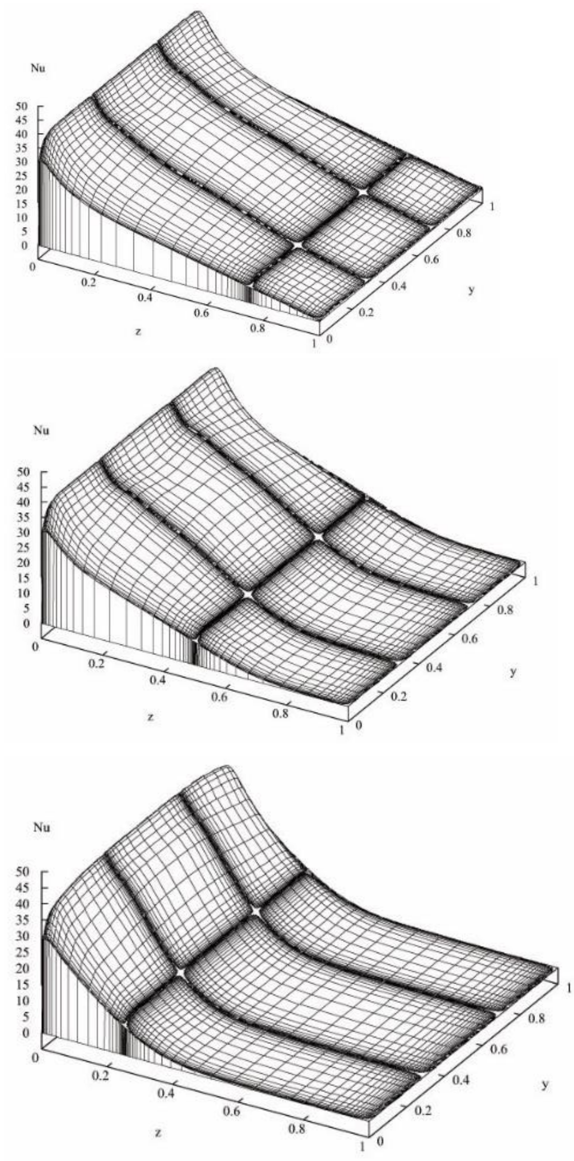



**Fig. 8.** The 3D contour plots of the local Nusselt Number in the hot wall at height ratio $y_H = y/H$ (a) $y_H = 0.25$ (b) = 0.50 (c) $y_H = 0.75$

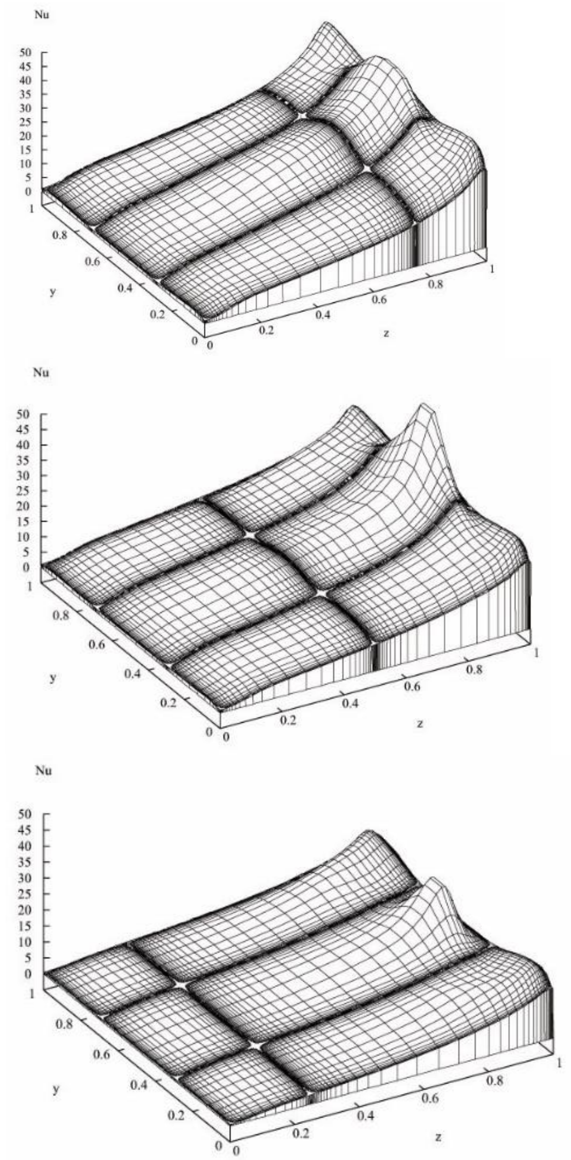



**Fig. 9.** The 3D contour plots of the local Nusselt Number in the cold wall at height ratio $y_H = y/H$ (a) $y_H = 0.25$ (b) = 0.50 (c) $y_H = 0.75$

## 4. Conclusions

The turbulent convective flow in a partially divided two-dimensional cavity was studied numerically using buoyancy extended eddy-viscosity and elliptic-relaxation RANS model. The present model performance in calculating the pressure scrambling, velocity, and temperature is validated with the DNS, experiments, and previous RANS-type models. The results show that the present model performed quite well, and produced a good agreement with the available numerical, as well as experimental data from the literature. The presented model also demonstrated reasonably similar performance compared to that of the advanced second-moment closure (SMC) model reported in the literature. It is shown that the model can capture and reveal realistic features of the turbulent convective flow in a partially divided cavity. It can be concluded that the present buoyancy extended eddy-viscosity and elliptic-relaxation approach works well for turbulent convective flow in a partially divided cavity and this is the novelty of the study and therefore the model has the potential to be applied to more complex cavities and a wider range of turbulence levels.

## References

[1] Gunarjo, S. B. "Contribution to advanced modeling of turbulent natural and mixed convection." Ph.D. Thesis Technische Univ., Delft (2003).

[2] Mayeli, Peyman, and Gregory J. Sheard. "Buoyancy-driven flows beyond the Boussinesq approximation: A brief review." *International Communications in Heat and Mass Transfer* 125 (2021): 105316. https://doi.org/10.1016/j.icheatmasstransfer.2021.105316.

[3] Ampofo, F. "Turbulent natural convection of air in a non-partitioned or partitioned cavity with differentially heated vertical and conducting horizontal walls." *Experimental Thermal and Fluid Science* 29, no. 2 (2005): 137-157.. https://doi.org/10.1016/j.expthermflusci.2004.02.005.

[4] Dol, H. S., K. Hanjalić, and S. Kenjereš. "A comparative assessment of the second-moment differential and algebraic models in turbulent natural convection." *International journal of heat and fluid flow* 18, no. 1 (1997): 4-14. https://doi.org/10.1016/s0142-727x(96)00149-x.

[5] Altaç, Zekeriya, and Nihal Uğurlubilek. "Assessment of turbulence models in natural convection from two-and three-dimensional rectangular enclosures." *International Journal of Thermal Sciences* 107 (2016): 237-246. https://doi.org/10.1016/j.ijthermalsci.2016.04.016.

[6] Piña-Ortiz, A., J. F. Hinojosa, and V. M. Maytorena. "Test of turbulence models for natural convection in an open cubic tilted cavity." *International Communications in Heat and Mass Transfer* 57 (2014): 264-273. https://doi.org/10.1016/j.icheatmasstransfer.2014.08.011.

[7] Karimpour, F., and S. K. Venayagamoorthy. "A simple turbulence model for stably stratified wall-bounded flows." *Journal of Geophysical Research: Oceans* 119, no. 2 (2014): 870-880. https://doi.org/10.1002/2013jc009332

[8] Lazeroms, W. M. J., Geert Brethouwer, Stefan Wallin, and A. V. Johansson. "An explicit algebraic Reynolds-stress and scalar-flux model for stably stratified flows." *Journal of Fluid Mechanics* 723 (2013): 91-125. https://doi.org/10.1017/jfm.2013.116

[9] Ma, Haiteng, and Li He. "Large eddy simulation of natural convection heat transfer and fluid flow around a horizontal cylinder." *International Journal of Thermal Sciences* 162 (2021): 106789. https://doi.org/10.1016/j.ijthermalsci.2020.106789

[10] Ortiz, Agustin Villa, Lilla Koloszar, and Philippe Planquart. "Large Eddy Simulations on a natural convection boundary layer at Pr= 0.71 and 0.025." *Nuclear Engineering and Design* 353 (2019): 110231. https://doi.org/10.1016/j.nucengdes.2019.110231

[11] Jones, W. Peter, and Brian Edward Launder. "The prediction of laminarization with a two-equation model of turbulence." *International journal of heat and mass transfer* 15, no. 2 (1972): 301-314. https://doi.org/10.1016/0017-9310(72)90076-2

[12] Durbin, Paul A. "Near-wall turbulence closure modeling without "damping functions"." *Theoretical and computational fluid dynamics* 3, no. 1 (1991): 1-13. https://doi.org/10.1007/bf00271513

[13] Dehoux, Frederic, Yannick Lecocq, Sofiane Benhamadouche, Remi Manceau, and Laurent-Emmanuel Brizzi. "Algebraic modeling of the turbulent heat fluxes using the elliptic blending approach—application to forced and mixed convection regimes." *Flow, turbulence and combustion* 88 (2012): 77-100. ttps://doi.org/10.1007/s10494-011-9366-8

[14] Das, Sandipan Kumar. "Elliptic relaxation model for stably stratified turbulence." *International Journal of Heat and Fluid Flow* 74 (2018): 173-186. https://doi.org/10.1016/j.ijheatfluidflow.2018.09.007

[15] Das, Sandipan Kumar. "A Reynolds Stress model with a new elliptic relaxation procedure for stratified flows." *International Journal of Heat and Fluid Flow* 83 (2020): 108595. https://doi.org/10.1016/j.ijheatfluidflow.2020.108595

[16] Kruger, Sunita, and Leon Pretorius. "The effect of vertical and horizontal partitions on natural convection in a heated enclosure." In *Heat Transfer Summer Conference*, vol. 55508, p. V004T13A004. American Society of Mechanical Engineers, 2013. https://doi.org/10.1115/ht2013-17439

[17] Al-Krmah, Dhia Al-Deen HA, B. P. Huynh, and Nabil Jamil. "Effects on Natural-Convection Heat Transfer of a Partial Partition in a Cubic Enclosure." In *Australasian Fluid Mechanics Conference*. Australasian Fluid Mechanics Society, 2016.

[18] AlAmiri, Abdalla, Khalil Khanafer, and Ioan Pop. "Buoyancy-induced flow and heat transfer in a partially divided square enclosure." *International Journal of Heat and Mass Transfer* 52, no. 15-16 (2009): 3818-3828. https://doi.org/10.1016/j.ijheatmasstransfer.2009.01.043

[19] Ferziger, Joel H., and Milovan Peric. "Computational methods for fluid dynamics." (2002): 2002-430. https://doi.org/10.1007/978-3-642-56026-2

[20] Kenjeres, Sasa. "Numerical modelling of complex buoyancy-driven flows." *PhD Thesis Delft University of*

*Technology* (1998).

[21] Kenjereš, Sasa, S. B. Gunarjo, and K. Hanjalić. "Contribution to elliptic relaxation modeling of turbulent natural and mixed convection." *International Journal of Heat and Fluid Flow* 26, no. 4 (2005): 569-586. https://doi.org/10.1016/j.ijheatfluidflow.2005.03.007

[22] Versteegh, Theodorus Antonius Maria. "Numerical simulations of natural convection in a differantially heated, vertical channel." (1998).

[23] Blay, D. "Confined turbulent mixed convection in the presence of horizontal buoyant wall jet." *HTD Vol. 213, Fundamentals of Mixed Convection* (1992).

[24] Opstelten, Ivo Johannes. "Experimental study on transition characteristics of natural-convection flow." *Ph. D. Thesis* (1994).

[25] Dol, Henry Simon, and K. Hanjalić. "Computational study of turbulent natural convection in a side-heated near-cubic enclosure at a high Rayleigh number." *International Journal of Heat and Mass Transfer* 44, no. 12 (2001): 2323-2344. https://doi.org/10.1016/s0017-9310(00)00271-4

[26] Kenjeresˇ, S., K. Hanjalic, and S. B. Gunarjo. "A T-RANS/VLES approach to indoor climate simulations." In *Fluids Engineering Division Summer Meeting*, vol. 36150, pp. 947-955. 2002. https://doi.org/10.1115/fedsm2002-31400