Numerical simulation with a Reynolds stress turbulence model of flow and heat transfer in rectangular cavities with different aspect ratios

Fatima MADI AROUS*
*Université des Sciences et de la Technologie Houari Boumediene, Algiers, Algeria
E-mail: madi.fatima@yahoo.fr

Received 2 May 2015

Abstract
The effect of the cavity aspect ratio on the flow and heat transfer characteristics is investigated numerically in this paper. The cavity is part of the channel’s lower wall heated at a uniform temperature. The approach is based on the low Reynolds Stress-omega turbulence model. The study is performed at two different Reynolds numbers. Results show that the flow structure depends heavily on the cavity aspect ratio. The increase of the aspect ratio enhances the heat transfer. In addition, the increase of the Reynolds number does not affect the flow structure but improves the heat transfer. To examine the effect of the cavity presence, we compared the local Nusselt number along the cavity walls to the one along the upstream wall. The heat transfer rate was found to be lower within the cavity but superior in the region located just downstream. In addition, the maximum value of the Nusselt number is observed at the reattachment region of the cavities characterized by the shear layer reattachment. At last, we note a clear correlation between the local Nusselt number and the velocity fluctuations profiles. This demonstrates a close relationship between these quantities.

1. Introduction
Turbulent flows over cavities, steps or obstacles belong to the separated flows class, characterized by the presence of recirculation currents. Those currents improve the blend and can play an important role in heat transfer. This type of flows is encountered in many industries such as mechanical or civil engineering and hydraulics. Many studies were conducted in order to understand and control this type of flows.

Gorin (2007) focuses on the physical models for the transfer processes in turbulent separated flows. These processes differ from those in turbulent flows with zero pressure gradients.

Bouterra et al. (2010) studied numerically using the Large Eddy Simulation methodology the mass and heat transfer in a flow over a backward facing step. The study examines the effect of introducing a sinusoidal blowing/suction into the separated shear layer. The results reveal the existence of an optimum forcing frequency which reduces significantly the reattachment length.

Lancial et al. (2013) investigated experimentally and numerically a heat transfer on a turbulent wall jet over a non-confined backward facing step. It was found that the reattachment length is shorter than that of a confined backward facing step. In addition, the Nusselt number is low in the recirculation region, increases through a maximum value in the reattachment zone and decreases into the recovery region.

Richards et al. (1987) studied experimentally a forced convection heat transfer in a flow over a cavity heated from the bottom. The study examines the effect of the cavity aspect ratio and the inlet boundary thickness. It appears that the heat transfer is very sensitive to the cavity aspect ratio but is less affected by the inlet boundary layer thickness.

The study of Aung (1983) focuses on a laminar forced convection in cavities with aspect ratios of 1 and 4. It was found that the temperature distribution outside the cavity is less affected by the flow inside it. The maximum heat transfer on the cavity floor is located between its midpoint and the downstream wall. Everywhere on the cavity floor, the local heat transfer is substantially less than the upstream cavity region’s.
Batthi and Aung (1984) studied numerically a heat transfer in a laminar flow over rectangular cavities with different aspect ratios. The cavities’ walls were held at a uniform temperature and the study was carried out at different Reynolds numbers. Results show that the average Nusselt number varies as a function of the Reynolds number and the aspect ratio. In addition, the influence of the upstream boundary layer thickness is found to be negligible.

In their study, Aghajani Delavar et al. (2011) examined heat transfer and entropy generation in a square cavity flow in natural convection. The 0.4H long heater was located at the right side wall of the cavity. The study investigates the effect of the heater location. It has been deducted that the heater's location has an impact on the flow’s pattern and on the temperature distribution in the cavity. In addition, it was observed a higher heat transfer from the cold walls when the heater is located on the vertical wall.

Stalio et al. (2011) studied numerically a convective heat transfer in laminar conditions at low Prandtl number. The geometry considered is a channel with a periodic series of shallow cavities. The study investigates the effects of the cavity aspect ratio and the Reynolds number. Results show that the presence of the cavity has a negative effect on the heat transfer. This is due to the presence of a stable vortex downstream the backward step. The insulating effect of the vortex increases with the Reynolds number. The only case where the global Nusselt number increases with the Reynolds number is when the cavity’s AR=10.

Using the standard k-ε turbulence model, Alammar (2006) examined the effect of the cavity aspect ratio on the flow and heat transfer characteristics. The study reveals the presence of several recirculation zones inside the cavity. It was found that cavities with higher aspect ratio enhance heat transfer while increasing pressure drop.

Mesalhy et al. (2010) examined the flow and heat transfer over shallow cavities heated with constant heat flux from the bottom. The presence of two eddies when the aspect ratio increases was observed. Furthermore, the local Nusselt number increases with the increase of the cavity aspect ratio or of the Reynolds number.

Madi Arous et al. (2011) examined the influence of the upstream flow characteristics on a shallow cavity flow. Two different upstream flows were considered: a wall jet flow and a boundary layer flow. The most important result was the earlier reattachment process in the wall jet inflow case. Madi Arous et al. (2012) studied a turbulent wall jet over a cavity of an aspect ratio of 10; they examined the effect of the cavity depth. Results reveal that the cavity depth increase causes a linear decrease of the reattachment length.

A turbulent flow over a rectangular cavity is studied in the present paper. The cavity is part of the channel’s lower wall heated to a uniform temperature. The aim is to examine the influence of the cavity presence on the heat transfer and to investigate the cavity aspect ratio effect on flow structure and on heat transfer. The study is performed at two different Reynolds numbers.

The numerical approach is based on the low-Re stress-omega turbulence model which is based on the LRR model and the specific dissipation rate equation. The low-Re stress-omega turbulence model is a multiscale model that has a wide range of applications. This model has proven to be accurate for wall-bounded flows including separation (Wilcox, 1998). The low-Re stress-omega solves the Reynolds stresses transport equations in addition to an equation for the specific dissipation rate $\omega$. Kolmogorov defined $\omega$ as the energy dissipation rate per unit volume and time (Wilcox, 1998). The numerical procedure is based on the finite volume method. This discretization method is well appropriate for the numerical simulation of conservation laws in fluid mechanics, heat and mass transfer. The FVM may be used on arbitrary geometries, structured or unstructured meshes, and it leads to robust schemes. In addition, The FVM considers the local conservativity of the numerical fluxes from one discretization cell to its neighbour. This last feature makes the FVM appropriate in fluid mechanics schemes (Eymard et al., 2003).

The previous study of Madi Arous et al., (2011) has shown that this turbulence model gives satisfactory results as compared to the experimental ones.

2. Theoretical formulation

The geometrical parameters of the present problem are those considered in Madi Arous et al. (2011). The cavity is inserted at the lower wall of the channel. The cavity has a depth of 20 mm and different aspect ratios: AR=1, 6, 8, 10, 12, 14. The lower wall of the channel is heated to uniform temperature. This study is performed at $Re=8\times10^3$ and $Re=8\times10^4$, to examine the Reynolds numbers effect.
2.1 The Governing Equations

The problem is assumed two-dimensional. The mass, momentum and energy conservation equations, for a steady state incompressible flow, have been averaged in Cartesian tensor notations as follow:

\[
\frac{\partial U_j}{\partial x_j} = 0
\]  

\[
U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \nu \frac{\partial}{\partial x_j} \left[ U_i - u_j u_j \right]
\]  

\[
U_j \frac{\partial T}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \frac{\nu}{\Pr} \frac{\partial T}{\partial x_j} - u_j \theta \right]
\]

Where \( U_j \) and \( u_j \) are respectively the mean velocity and the velocity fluctuation components in \( x_j \) direction, \( T \) is the mean temperature, \( P \) is the static pressure, \( \theta \) is the temperature fluctuation, \( \Pr \) is the Prandtl number equal to 0.71 and \( \rho \) is the fluid density. The fluid considered in the present study is the air.

The modelling of the turbulent heat transport is based in the simple gradient diffusion hypothesis (SGDH):

\[
\overline{-u_j \theta} = \frac{\nu}{\Pr} \frac{\partial T}{\partial x_j}
\]

Where \( \Pr \) is the turbulent Prandtl number equal to 0.85.

2.2 Turbulence Modelling

The closure of the governing equations is realised by the low-Re Stress-Omega turbulence model.

The Reynolds stress tensor equation, for an incompressible and statistically steady flow, is written as follows:

\[
U_k \frac{\partial \tau_{ij}}{\partial x_k} = -P_{ij} + \frac{2}{3} \beta \omega \delta_{ij} - \Pi_{ij} + \frac{\partial}{\partial x_k} \left( \Gamma_{\tau} \frac{\partial \tau_{ij}}{\partial x_k} \right)
\]

The specific dissipation rate equation is given by:

\[
U_j \frac{\partial \omega}{\partial x_j} = \alpha \omega \frac{\partial U_i}{\partial x_j} - \beta \omega^2 + \frac{\partial}{\partial x_j} \left( \Gamma_{\omega} \frac{\partial \omega}{\partial x_j} \right)
\]

Where: \( \Gamma_{\tau} = v + \sigma^* v_t \) and \( \Gamma_{\omega} = v + \sigma v_t \)

\( v_t \) is the turbulent viscosity given by: \( v_t = a^* k / \omega \)

and \( \sigma^* = \sigma = 1/2 \) are closure coefficients.

Introducing the low Reynolds number correction, \( a^* \) and \( \beta^* \) are functions of the turbulent Reynolds number \( Re_T \), given by:
The turbulent Reynolds number \( R_{eT} \) is a function of \( k \) and \( \omega \) as appears in equation (9).

\[
R_{eT} = \frac{k}{\nu} \tag{9}
\]

\( f_{\beta^*} \) verifies the following condition:

\[
f_{\beta^*} = \begin{cases} 
1 & \text{if } \chi_k \leq 0 \\
\frac{1+640\chi_k^2}{1+400\chi_k^2} & \text{if } \chi_k > 0
\end{cases}
\]

Where \( \chi_k = \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \) (10)

\( \alpha \) is also a function of the turbulent Reynolds number as shown in equation (11).

\[
\alpha = \alpha_\infty \left( \frac{\alpha_0 R_{\omega} + R_{eT}}{R_{\omega} + R_{eT}} \right) \left( \frac{3R_{\omega} + R_{eT}}{3\alpha_0 R_{\omega} + R_{eT}} \right) \tag{11}
\]

\[
\beta = \frac{9}{125} \frac{1 + 70\chi_\omega}{1 + 80\chi_\omega}; \quad \chi_\omega = \frac{\Omega_{ij} \Omega_{jk} S_{ki}}{(0.09\omega^3)} \tag{12}
\]

\( \Omega_{ij} \) and \( S_{ki} \) are respectively the mean-strain-rate and the mean-rotation tensors defined as follow:

\[
S_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right); \quad \Omega_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i} \right) \tag{13}
\]

The stress-\( \omega \) model does not require a wall-reflexion term in pressure-strain term \( \Pi_{ij} \) which is written as follow:

\[
\Pi_{ij} = \beta^* C_{i0^*} \left( \tau_{ij} + \frac{2}{3} k \delta_{ij} \right) - \hat{\alpha} \left( P_{ij} - \frac{2}{3} P \delta_{ij} \right) - \hat{\beta} \left( D_{ij} - \frac{2}{3} P \delta_{ij} \right) - \hat{\gamma} \left( S_{ij} - \frac{1}{3} S_{kk} \delta_{ij} \right) \tag{14}
\]

Where: \( P_{ij} = \tau_{im} \frac{\partial U_j}{\partial x_m} + \tau_{jm} \frac{\partial U_i}{\partial x_m} \), \( D_{ij} = \tau_{im} \frac{\partial U_m}{\partial x_j} + \tau_{jm} \frac{\partial U_m}{\partial x_i} \) and \( P = \frac{1}{2} P_{kk} \) (15)

The closure coefficients, when low Re number correction is included, are given by:

\[
\hat{\alpha} = \frac{12 + \hat{\alpha}_\infty R_{eT}}{12 + R_{eT}}, \quad \hat{\beta} = \hat{\beta}_\infty \frac{R_{eT}}{12 + R_{eT}}, \quad \hat{\gamma} = \hat{\gamma}_\infty \frac{12R_{\omega} + R_{eT}}{12 + R_{eT}} \tag{16}
\]

\[
\hat{\alpha}_\infty = \frac{8 + C_2}{11}, \quad \hat{\beta}_\infty = \frac{8C_2 - 2}{11}, \quad \hat{\gamma}_\infty = \frac{60C_2 - 4}{55} \tag{17}
\]
The model constants are given by (18).

\[
\begin{align*}
\alpha_0 &= 0.52, \quad \alpha_0 = 0.21, \quad \alpha_0 = 0.024, \quad R_0 = 6.20, \quad C_1 = 1.8, \quad C_2 = 0.52, \quad \theta_0 = 0.007
\end{align*}
\]

(18)

2.3 Numerical Procedure

Ansys Fluent 14.0 is used as a CFD solver. The discretization of the equations is based on the finite volume method with a co-located scheme. The SIMPLEC algorithm (SIMPLE-Consistent) and the Power Law interpolation scheme (PLDS) are used for pressure-velocity coupling and for the convection–diffusion interpolation term, respectively. The calculation domain is subdivided into two parts: the inner region of the cavity \((y/H<1)\) and the outer region \((y/H >1)\). The mesh is released by Gambit 2.3.16. The grids are non-uniform and refined near the walls where very high gradients prevail in the viscous sub-layer. The convergence of the iteration is based on the scaled residuals value, which is fixed at \(10^{-6}\) for all governing equations, and on the stability of the solution during the iterations. The convergence is considered to be achieved when all these residuals are less than or equal to \(10^{-6}\) and the solution becomes stable.

2.4 Boundary Conditions

The figure 1 shows the computational domain and the boundary frontiers.

- At the inlet boundary, uniform profiles of velocity, turbulent kinetic energy, specific dissipation rate and temperature are imposed:

\[
U = U_{in}, \quad V = 0, \quad k = \frac{1}{2} \left(1 \cdot U_{in}\right)^2, \quad \omega = \frac{k}{1/2 \cdot C_{\mu} \cdot \ell}, \quad T = 300K, \quad \text{where} \quad I \quad \text{is the turbulence intensity rate}, \quad \ell \quad \text{is a turbulent length scale and} \quad C_{\mu} = 0.09 \quad \text{is an empirical constant of the turbulence model.}
\]

- Constant pressure is imposed at the outlet boundary.

- At the wall boundaries, the no-slip condition \((U=V=0)\) is imposed, the turbulent quantities correspond to the wall function approach, the temperature of the lower channel wall is \(T_w = 340K\) (Fig. 2) and that of the upper wall is equal to 300K.

![Fig. 1 Schematic of computational domain](image1)

![Fig. 2 Schematic of the heated walls](image2)

2.5 Heat Transfer Calculation

In addition to isotherm contours, the heat transfer is evaluated in terms of the local Nusselt number at the heated walls as:

\[
N_u = -\frac{L_h}{T_w - T_0} \frac{dT}{dn}, \quad \text{where} \quad L_h \quad \text{is the length of the heated wall and} \quad n \quad \text{is the perpendicular direction to the corresponding wall. Likewise, the heat transfer is evaluated in terms of the average Nusselt number calculated along}
\]
each cavity wall as: \( \text{Nu}_{\text{aver}} = \frac{1}{L_h} \int_{0}^{L_h} \text{Nu}(x)dx \)

### 2.6 Validation of the turbulence model

#### 2.6.1 Characterization of the inlet flow

The figure 3 shows the normalized mean streamwise velocity \( U/U_0 \) profile at \( x=-H \) section. This figure shows that the numerical profile agrees quite well with the boundary layer experimental profile of Klebanoff et Diehl (1951).

![Fig. 3 Inlet streamwise velocity profile](image)

#### 2.6.2 Characterisation of a Shallow Cavity Flow

To validate the numerical approach based on the low Reynolds stress omega turbulence model, we compared the results with the experimental ones of Estève et al. (2000) obtained with the PIV technique.

Figures 4 (a) and 4 (b) display respectively the experimental and the predicted flow field within a cavity of an aspect ratio of 10, described by the streamlines and turbulence intensity. We note a presence of a large recirculation bubble inside the cavity in addition to two corners eddies close to each step face. The experiment reveals a no reattachment of the shear layer to the cavity floor. The numerical approach gives similar results; it permits to find the complex structure of the mean flow with the no reattachment phenomenon. We can conclude that the Low Reynolds Stress Omega Model gives satisfactory results that agree well with the experimental ones.

![Fig. 4 Turbulence intensity and streamlines within the cavity of AR=10 (Re=6,7X10³)](image)
3. Results and Discussions

3.1 Mean cavity flow behavior

The figure 5 shows the streamline contours within rectangular cavities with different aspect ratios at Re=8\times10^3 and Re=8\times10^4. The square cavity (AR=1) is characterized by the presence of a voluminous recirculation bubble turning clockwise. It occupies the most cavity space and a small corner vortex at Re=8\times10^3 which disappeared at Re=8\times10^4. The cavity aspect ratio augmentation causes an elongation of the main recirculation (AR=6, 8) and its separation into two vortices in the cavities with large aspect ratios (AR=10, 12, 14). A similar evolution of cavity flow structure with the increase of the aspect ratio has been proven by Taneda (1979). The cavities with large aspect ratios are characterized by the presence of three vortices. The main vortex is located in the upstream region and the two secondary ones are located close to the corners. We also observe a small vortex above the downstream step. A similar shallow cavity flow structure has been found in Avelar et al.’s (2007) experiments. In addition the figure shows a weak Reynolds number effect; the main flow structure remains unchanged at the two Reynolds numbers considered in this study. However, the size of the upstream corner vortex is slightly smaller. Similar results were found in the backward facing step flow experiment by Eaton and Johnston (1981); when the flow is laminar, the reattachment length increases with the Reynolds number then decreases until it reaches an asymptotic value in a turbulent flow.

![Fig. 5 Evolution of the flow structure as a function of the cavity’s aspect ratio](image-url)
The figure 6 displays the isobar contours within and around the cavities with different aspect ratios and Re=8X10^3. In the square cavity, the highest pressures are near the trailing edge while the depression zone appears in the center. In the cavity with an aspect ratio of 6, the depression moves downstream. The increase of the cavity aspect ratio causes an augmentation of the pressure in the downstream region. Furthermore, we observe a depression behind the upstream step that extends to the upstream region. Another depression, located above the back step, is amplified with the aspect ratio which also causes the pressure to increase within the cavity. At Re=8X10^4, the isobar contours are similar but with much higher pressure values.

![Isobar Contours (Re=8X10^3)](image)

**Fig.6 Isobar Contours (Re=8X10^3)**

### 3.2 Heat transfer

The figure 7 illustrates the isotherm contours within and around the cavities with different aspect ratios at the two Reynolds numbers. We note that the augmentation of the Reynolds number causes a temperature drop within the cavity. Indeed, we remark a thin layer of hot fluid near the cavities’ floor at Re=8X10^3 and that disappears at Re=8X10^4. The cavity aspect ratio increase causes a penetration of the cold outer layers into the cavity. The Reynolds number increase accelerates the penetration.

The evolution of the local Nusselt number along the cavities’ walls and along the upstream and downstream walls (Fig.2) at the two Reynolds numbers is given by Fig.8. The local Nusselt number is normalized with respect to the Nusselt number of the upstream wall Nu₀. This figure shows that the Nusselt number is constant at the upstream wall, decreases along the cavity then increases to reach an asymptotic value equal to Nu₀ downstream the cavity. This figure also shows a sudden increase of the Nusselt number at the trailing edge of the cavities. This peak is more significant for the cavities with a large aspect ratio. This figure permits to examine the effect of the cavity on the heat transfer. Indeed, the ratio of the Nusselt number along the cavity walls to that of the upstream wall indicates a heat transfer enhancement factor. This factor reveals a heat transfer decrease within the cavity followed by a rise in the region located just downstream. However, the increase of the cavity aspect ratio improves the heat transfer which remains lower than that of a flat wall. The Reynolds number increase improves the heat transfer within the cavity and leads to a flattening of the profiles, particularly when the cavities have a large aspect ratio.

The heat transfer between the fluid and the cavity is also evaluated by the sum of the average Nusselt numbers along each cavity wall.
\[ \text{Nu}_{\text{aver}} = \text{Nu}_1 + \text{Nu}_2 + \text{Nu}_3 \]

Where: \( \text{Nu}_1 = \frac{1}{H} \int_0^H \text{Nu}(0,y) \, dy \) is the average Nusselt number calculated along the left vertical cavity wall.

\( \text{Nu}_2 = \frac{1}{H} \int_0^H \text{Nu}(L,y) \, dy \) is the average Nusselt number calculated along the right vertical cavity wall.

\( \text{Nu}_3 = \frac{1}{L} \int_0^L \text{Nu}(x,0) \, dx \) is the average Nusselt number calculated along the cavity floor.

---

**Fig. 7 Isotherm Contours**
Fig. 8 Local Nusselt number distributions

The figure 9 represents the evolution of the average Nusselt number as a function of the cavity aspect ratio at the two Reynolds numbers. The average Nusselt number increases uniformly as the aspect ratio increases; this phenomenon is more significant at Re=8×10^4.

Fig. 9 Average Nusselt number as a function of the cavity aspect ratio
The figure 10 regroups the enhancement factor \( \frac{Nu}{Nu_0} \) and the skin friction coefficient \( Cf \) along the cavity floor in order to examine the relation between the heat transfer and the reattachment phenomenon. The velocity fluctuations profiles are also represented in the same figure.

The many zero \( Cf \) points evidence the presence of more than one vortex. The cavities with an aspect ratio between 1 and 10 are characterized by a no reattachment of the shear layer at the cavity floor. However, the shear layer reattachment occurs at \( x=8,22H \) in the cavity of AR=12 and at \( x=7,63 \) in that of AR=14). This figure shows a close relationship between the Nusselt number and the skin friction coefficient evolution. Indeed, in the cavities characterized by a no reattachment (AR=1 to 10), the skin friction coefficient decrease is accompanied by a Nusselt number increase and the Nu decrease corresponds to a Cf increase. In the cavities characterized by a reattachment (AR=12 and 14), the Nusselt number is maximum at the reattachment point.

This figure also shows a clear correlation between the Nusselt number and the velocity fluctuations. A similar result has been observed by Casarsa and Arts (2002); their experiments revealed that the heat transfer is strongly affected by the velocity fluctuation component normal to the heated wall. However, the present study shows that the heat transfer is also affected by the author components of the velocity fluctuations.

Fig. 10 The enhancement factor \( \frac{Nu}{Nu_0} \), the velocity fluctuation and the skin friction coefficient profiles along the cavity floor (Re=8X10^3)
3. CONCLUDING REMARKS

The comparison of the numerical results with the prior experimental ones proves that the low-Re Stress-Omega Model predicts fairly well the cavity flows. The results show that both the flow field and the heat transfer are very sensitive to the cavity aspect ratio. In addition, the study reveals the presence of a voluminous vortex within the square cavity. The cavity aspect ratio increase causes an elongation of this vortex, giving rise to three vortices within the cavities with a large aspect ratio. Furthermore, the flow structure seems to be less affected by the Reynolds number variation.

High pressures in the vicinity of the trailing edge of the square cavity and a depression in its center were observed in the study. The increase of the cavity aspect ratio causes an augmentation of the pressure in the downstream region and the emergence of two depression zones. The first depression appears just behind the upstream step and the second one above the back step. In addition, the cavity aspect ratio increase causes the pressure to rise within the cavity.

The study also reveals that the heat transfer rate is lowered within the cavity but increased just downstream. So, the insertion of a series of cavities at the canal walls can improve the heat transfer. Furthermore, the aspect ratio increase improves the heat transfer rate which still remains lower than that of the flat wall. Moreover, the average Nusselt number increases linearly as a function of the aspect ratio. It was additionally observed that the Reynolds number increase enhances the heat transfer within the cavity.

The most important results are:
- The Nusselt number is maximal at the reattachment region in the cavities characterized by the shear layer reattachment.
- The close relation between the Nusselt number and the velocity fluctuations components along the heated wall.

Nomenclature

| AR  | Cavity aspect ratio, \( AR=L/H \) |
|-----|-------------------------------------|
| \( C_f \) | Skin friction coefficient, \( C_f = \frac{2\tau_w}{\rho U_0^2} \) |
| \( H \) | Cavity depth (m) |
| \( k \) | Turbulence kinetic energy \((m^2/s^2)\) |
| \( L \) | Cavity length (m) |
| \( Nu \) | Local Nusselt number |
| \( Nu_{ave} \) | Average Nusselt number |
| \( Pr \) | Prandtl number |
| \( Pr_t \) | Turbulent Prandtl number |
| \( Re \) | Reynolds number |
| \( T \) | Flow temperature (K) |
| \( T_W \) | Hot wall temperature (K) |
| \( U \) | Streamwise velocity component \((m/s)\) |
| \( U_0 \) | Maximum of \( U \) at \( x=-H \) \((m/s)\) |
| \( u \) | Streamwise fluctuating velocity \((m/s)\) |
| \( V \) | Vertical velocity component \((m/s)\) |
| \( v \) | Vertical fluctuating velocity \((m/s)\) |
| \( x \) | Streamwise coordinate (m) |
| \( y \) | Vertical coordinate (m) |

Greek symbols

| \( \delta_{ij} \) | Kronecker delta |
| \( \omega \) | Specific dissipation rate \((s^{-1})\) |
| \( \delta \) | Boundary layer thickness (m) |
| \( \nu \) | Kinematic viscosity \((kg/m/s)\) |
| \( \nu_t \) | Turbulent kinematic viscosity \((m^2/s)\) |
| \( \rho \) | Fluid density \((kg/m^3)\) |
| \( \tau_{ij} = -u_i u_j \) | Reynolds stress tensor \((m^2/s^2)\) |
| \( \tau_w = \mu \frac{\partial U}{\partial y} \bigg|_{y=0} \) | Shear wall stress \((kg/m^3 \cdot s^2)\) |
References

Aghajani et al., Effect of discrete heater at the vertical wall of the cavity over heat transfer and entropy generation using LBM, Thermal Science International Scientific Journal, Vol. 15, Issue 2, (2011), pp. 423-435.

Alammar K. N., Effect of cavity aspect ratio on flow and heat transfer characteristics in pipes: a numerical study. Heat and Mass Transfer, 42, (2006), pp. 861-866.

Aung, W., An Interferometric Investigation of Separated Forced Convection in Laminar Flow Past Cavities. Journal of Heat Transfer. vol. 105, (1983), pp. 505-512.

Avelar, A. C. et al. Three-dimensional flow over shallow cavities. 37th AIAA Fluid Dynamics Conference and Exhibit, (2007), 4234

Bhatti, A. and Aung W., Finite difference analysis of laminar separated forced convection in cavities, Journal of Heat Transfer. vol. 106, n°1, (1984), pp. 49-54.

Bouterra, M. et al., Control of local transfer in the separated and reattaching flow by a periodic forcing, Journal of Applied Fluid Mechanics, Vol. 4, No. 2, (2010), pp.63-67.

Casarsa, L. and Arts, T., Aerodynamic performance investigation of a rib roughened channel flow high blockage ratio, 11ème International Symposium on application of Laser Techniques to fluid mechanics, Lisbon, Portugal, (2002).

Eaton J. K. & Johnston J. P., A review of research on subsonic turbulent flow reattachment. AIAA, vol. 19, (1981), pp. 1093-1099.

Estève, M. J. et al., Flow field characterisation within a rectangular cavity, 10th International Symposium on Applications of Laser Techniques to Fluid Mechanics, Portugal, (2000).

Eymard, R. et al., Finite Volume Methods, Handbook of Numerical Analysis, P.G. Ciarlet, J.L. Lions eds, vol 7, (2003), pp. 713-1020

Gorin, A., Turbulent separated flows: Near-Wall behavior and heat and mass transfer, Journal of Applied Fluid Mechanics, Vol. 1, No. 1, (2008), pp. 71-77.

Klebanoff P. S and Diehl Z. W., Some features of artificially thickened fully developed turbulent boundary layers with zero pressure gradient. Supersedes NACA, TN, 2475, (1951).

Lancial, N. et al., Effect of a wall jet on heat transfer over a non-confined backward facing step, International Journal of Heat and Fluid Flow, vol. 44, (2013), pp. 336-347.

Madi Arous et al., Influence of upstream flow characteristics on the reattachment phenomenon in shallow cavities, Thermal Science International Scientific Journal. vol. 15, n° 3, (2011), pp. 721-734

Madi Arous et al., Jet–cavity interaction: effect of the cavity depth, Progress in computational fluid dynamics. vol. 11, issue 5, (2012), pp. 322-332.

Mesalhi, O. M. et al., Flow and Heat transfer over shallow cavities. International Journal of Thermal Sciences, 49, (2010), pp. 514-521.

Richards, R.F. et al., Turbulent forced convection heat transfer from a bottom heated open surface cavity, International Journal of Heat and Mass Transfer. vol. 30, (1987), pp. 2281-2287.

Stalio et al., Numerical simulation of forced convection over a periodic series of rectangular cavities at low Prandtl number, International journal of Heat and fluid Flow, 32, (2011), pp. 1014-1023

Taneda Sadatoshi, Visualization of Separating Stokes Flows. Journal of the Physical Society of Japan, 46, (1979), pp.1935-1942.

Wilcox, D. C., Turbulence Modelling for CFD, DCW Industries, Inc., La Canada, California, (1998).