Mixed Convection Enhancement in a Rectangular Cavity by Triangular Obstacle

Shaalan Ghanam Afluq¹, Mohamed Abed Al Abas Siba² and Kadhum Audaa Jehhef³
¹Department of Mechanics, Institute of Kut, Middle Technical University, Kut, Iraq, Email: shgh61@yahoo.com
²Department of Machines and Equipment, Institute of Technology, Middle Technical University, Baghdad, Iraq, Email: Moh_siba@yahoo.com
³Department of Machines and Equipment, Institute of Technology, Middle Technical University, Baghdad, Iraq, Email: kadhum.audaa@yahoo.com

Abstract. Numerical modeling analysis of mixed convection heat transfer for air flow in a cavity with bottom local heat source and top inlet and outlet sections is studied. Also, the cavity equipped by vertical triangular obstacle on the top wall in order to enhancement the convection inside the cavity are researched. System of dimensionless stationary Navier-Stokes equations is solved numerically by discretizing the compositional domain into small grids. Mixed convection regimes are the viscous incompressible Newtonian fluid with Reynolds number range (800-1400) and the Grashof number ranged (10⁵ - 10⁸) and at fixed Prandtl number at (7.1). The pressure, temperature and velocity distributions characterizing the basic laws of viewed process. The results showed that the main circulating currents in different zones of the cavity is due to the effect of number of blocks and the presence vertical triangular obstacle on the top boundary of the cavity. The formation of thermal conditions in the region under study and the effect of low temperatures above the heated bottom wall on the circulation fluid flow in a cavity. Distribution pattern established velocity and temperature profile in centered sections depending on the height of vertical triangular obstacle. Also, the results showed that the Nusselt number increased by about 26% when increasing the height ratio of triangular obstacle to (h/H=0.5) with three blocks.

1. Introduction
Convection heat transfer belongs to the field of energy transferred technologies and finds everything in the greater application [1]. The mixed convection researches are significantly more complicated due to bridges accounting for mixed convection, recently, the study of mixed convection in electronic equipment installation has much attention, but models describing similar processes are quite rare. In many cases, significant role in the formation of thermal conditions in the considered area and the behavior of heat sink is played by external faces of the cavities. Until now, many convective heat transfer deletions, taking into account the influence of the external environment on nature of the flow and temperature field object which had not carried out in the studies [2]. The heat transfer convection flow in cavities have been interest subject for many applications such as electronic cooling, solar collectors, lubrication technologies, food processing, and nuclear reactors [3]. When the both natural and forced convections simultaneously exist,
the heat transfer convection phenomenon is called mixed convection, thus, the mixed convection occurs when the buoyant flow matters in a forced flow effects [4]. However, [5] studied numerically the effect of using two moving walls on the mixed convection heat transfer in square cavities. Their results showed that the heat transfer decreased significantly when the vertical walls moved upwards in the same direction.

Also, [6] used two baffles attached to the cavity’s plane horizontal surface in order to study the free convection in trapezoidal cavities. It assumed that the floor and upper inclined walls of the cavity are both adiabatic while the vertical surfaces are isothermal with Rayleigh number range about $10^3 \leq Ra \leq 10^6$. Results indicated that the second baffle will cause to decrease the cavity’s heat transfer and fluid flow and the heat transfer dropped drastically as the height of the baffle rises. In addition, [7] performed a study on a cavity with a heated square blockage. They found that size of heater eccentricities, location of heater eccentricities and Richardson number ($Ri = Gr/Re^2$) would effect on the average Nusselt number of the heater. The vented cavity with a centered heat generating element in order to study the effects of various parameters on the combined free and forced convection problem was investigated by [8]. The influence of Hartmann number ($Ha = BL\sqrt{\sigma/\mu}$, where $B$ is the magnetic field intensity, $L$ is the characteristic length scale, $\sigma$ is the electrical conductivity and $\mu$ is the dynamic viscosity and Prandtl number and Reynolds number on the flow and thermal fields had carried out. They noted that the studied parameters played a significant role on both the flow and thermal field.

A natural convection of nanofluid in a square cavity including a square heater was studied in [9]. They found that by increasing size of the heater, the heat transfer rate enhances. As showing above, there are few studies have been presented in order to investigate the problem of mixed convection in cavities with three heated blocks and use triangular obstacle on the top wall. [10] studied numerically a fin-cavity system under mixed convection flow. The system was composed of a heat triangular fin inserted in a squared cavity. They investigated the effect of Reynolds ($Re_H$) and Rayleigh ($Ra_H$) numbers. Their results showed that the thermal performance increased with the decrease of fin area ratio ($\phi$). Also, the results showed a better performance (up to 8%) of the triangular fin for low Reynolds numbers ($Re_H < 200$). A numerical study of a heat transfer and fluid flow by mixed convection for incompressible, two-dimensional and laminar flow presented by [11]. The cavity had two rectangular fins inserted in the lower surface. The geometry has three degrees of freedom: the ratio between height and cavity length ($H/L$) and the ratio between height and length of each fin ($H_1/L_1$ and $H_2/L_2$) at Richardson ($Ri = 0.1$and Reynolds number ($Re_H$)=400. Their results indicated that the lower $H_2/L_2$ ratios resulted in higher $Nu_H$ values. Also, the results found highlight the importance of the geometric evaluation for the purpose of theoretical recommendation on the geometric configurations that lead to the best thermal performance.

A numerical investigation on mixed convection in open ended cavity with different aspect ratios was presented by [12]. The simulation performed for a wide range of Reynolds numbers ($Re = 100–1000$) and used several aspect ratios of the cavity ($L/D = 0.5–4.0$) and $H/D = 0.1$. Results showed that the enhancement of heat transfer rate was generated principally by the increasing the assisting configuration was thermally more efficient when compared to the opposing one. The effects of magnetic field inclination on the heat transfer and fluid flow in square cavity was studied by [13]. The bottom wall was maintained at a cold temperature and vertical walls were well insulated while the top moving lid was kept at a hot temperature. The results showed that the mixed convection flow had retarded by the presence of the magnetic field and the average Nusselt number is an increasing function of the magnetic field angle.

Thus, the main aim of this study is presented a mathematical mixed convection modeling airflow in a cavity under various conditions and to explore the heat transfer rate by mixed convection in a lid driven cavity with rectangular blocks. Also, it is studied the effects of number of the three rectangular blocks, the height of the triangular obstacle on the heat transfer and fluid flow by the mixed convection.
2. Mathematical model
The schematic of the physical domain geometry of the two dimensional of rectangular cavity is presented in the Figure 1. The cavity includes top inlet and outlet ports, the top wall equipped with a variable height triangular obstacle, adiabatic (left, right, bottom) walls has and three variable height blocks but with constant pitch between them.

![Figure 1. Physical schematic diagram of the cavity.](image)

By considering viscous fluid motion within a rectangular cavity with obstacle triangular attached to the top wall with dimension (h) and three vertical rectangular blocks with dimension of height (h) and width (w). The boundary condition is assumed as heated top wall with constant heat flux (q'') that varied by 200-800 W/m² as shown in Figure 2. The adiabatic left, right and bottom walls have is mentioned in this study.

![Figure 2. Schematic diagram of the physical boundary conditions of the cavity.](image)

| Models | Configuration |
|--------|---------------|
| Model-1 | h/H=0         |
| Model-2 | h/H=0.15      |

Table 1. Models employed in the present numerical study.
The air is assumed the working fluid with constant properties except for the density that varied and treated according to Boussinesq approximation. The viscous air flow is assumed steady, laminar, incompressible, Newtonian and 2D flow and described by using the Navier–Stokes equations as follow [14]:

Continuity equation:
\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0
\]  

(1)

x- momentum equation:
\[
\frac{u}{u} \frac{\partial u}{\partial x} + \frac{v}{u} \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \left[ \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right]
\]

(2)

y- momentum equation:
\[
\frac{u}{u} \frac{\partial v}{\partial x} + \frac{v}{u} \frac{\partial v}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial y} + \nu \left[ \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right] + g\beta(T - T_c)
\]

(3)

Energy equation:
\[
\frac{u}{u} \frac{\partial T}{\partial x} + \frac{v}{u} \frac{\partial T}{\partial y} = \alpha \left[ \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right]
\]

(4)

In order to dimensionless the above equations by using the non-dimensional variables presented as:

\[
X = \frac{x}{L}, Y = \frac{y}{L}, U = \frac{u}{u_i}, V = \frac{v}{u_i}, P = \frac{p}{\rho u_i^2}, \theta = \frac{T - T_i}{T_h - T_i}, \theta = \frac{T_s - T_i}{T_h - T_i}
\]

(5)

Also, the non-dimensional parameters such as Re, Ri and Pr can be defined as:

\[
Re = \frac{u_i E}{v}, Ri = \frac{g\beta(T - T_c)Le}{u_i^2}, Pr = \frac{v}{\alpha}
\]

Thus, the above governing equations of the present problem are obtained as follows:

Dimensionless continuity equation:
\[
\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0
\]

(5)
Dimensionless x- momentum equation:

\[
\frac{U}{Re} \frac{\partial U}{\partial X} + \frac{V}{Re} \frac{\partial U}{\partial Y} = -\frac{\partial P}{\partial X} + \frac{1}{Re} \left[ \frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 U}{\partial Y^2} \right]
\]  

(6)

Dimensionless y- momentum equation:

\[
\frac{U}{Re} \frac{\partial V}{\partial X} + \frac{V}{Re} \frac{\partial V}{\partial Y} = -\frac{\partial P}{\partial Y} + \frac{1}{Re} \left[ \frac{\partial^2 V}{\partial X^2} + \frac{\partial^2 V}{\partial Y^2} \right] + Ri \theta
\]  

(7)

The dimensionless energy equation:

\[
\frac{U}{Pr Re} \frac{\partial T}{\partial X} + \frac{V}{Pr Re} \frac{\partial T}{\partial Y} = \frac{1}{Re Pr} \left[ \frac{\partial^2 \theta}{\partial X^2} + \frac{\partial^2 \theta}{\partial Y^2} \right]
\]  

(8)

The appropriate boundary conditions are used to solve equations (1) to (5) inside the cavity, at the inlet boundary condition is given as:

\[ U=1, \ V=0, \ \theta=0 \]

Convective boundary condition (CBC) at the outlet is given by:

\[ P=0, \ \frac{\partial \theta}{\partial X}|_{X=0} = \frac{\partial \theta}{\partial Y}|_{Y=1,0} \]

At all other adiabatic (left, right and top) walls boundaries is presented as follow:

\[ U=0, \ V=0, \ \frac{\partial P}{\partial n} = 0 \]

At the heated bottom wall is given by:

\[ \theta=1 \]

The average Nusselt number (Nu) at the heated top wall is formatted as:

\[ Nu = \frac{1}{L} \int_0^L \left. \frac{\partial \theta}{\partial Y} \right|_{Y=1} \, dX \]  

(9)

And the bulk average temperature in the cavity is defined as:

\[ \theta_{av} = \frac{1}{V} \int \theta \, dV \]  

(10)

3. Numerical Technique

In this study, the governing equations are transformed to the partial differential equations by using the implicit line by line Gauss elimination scheme and solved by ANSYS-Fluent software that based on the finite volume method. The present problem is developed to attain the results by using the pressure velocity coupling (SIMPLC algorithm) [15]. The relaxation factors with best results that used for velocity, temperature and pressure are 0.3, 0.6 and 0.8 respectively. Also, the model is terminated when the mass, momentum, and energy for each simulation evaluated over the course, residuals drop below $10^{-7}$. The computational domain nods are presented in Figure 3.
4. Test of Grid Independence
The present numerical computations have calculated for three selected grid sizes (i.e., 150×200, 180×220, 200×250). However, the results of these tests of the grids plotted in Figure 4 that shows the local Nusselt number distribution along the heated wall subjected to a constant heat flux of 1000 W/m². The results for these grid sizes show very good agreement coincident with each other. Maximum grid (200×250) is presented throughout this study in order to obtain an optimum simulation accuracy.

![Figure 4](image_url)

**Figure 4.** The test of grid independence of the local Nusselt number at Ra= 6.4×10⁷.
5. Results and Discussion

In the simulation, it was assumed that the thermos-physical properties of an air fluid and wall material are independent of temperatures. The fluid was considered thermally conductive, viscous, Newtonian and satisfying Boussinesq approximation and flow mode is considered as laminar flow and the outflow of mass to the atmospheric. At the initial time, it was taken that the cavity temperature in all considered constant and the same to simplify the present solution. The other outer walls were assumed heat insulated. The influence of conditions on mixed convection of the air in a cavity with bottom wall heat source. The numerical solution of the problem is carried out at the following values dimensionless quantities and temperatures: \( \text{Re} = 800-1400, \text{Gr} = 10^5-10^8, \text{Pr} = 7.1. \)

Note that in real airflow in three-dimensional distribution is possible temperature and velocity components, but in the problem under consideration transverse speed component will be less than two other components corresponding to plane of motion introduced into the cavity. Therefore, the assumption of two-dimensional formulation tasks is justified. The process of heat transfer in air is presented as contours as shown in Figure 5 for the accepted physical model described by a system of unsteady two-dimensional Navier-Stokes equations in Boussinesq approximation with nonlinear trivial condition. The figure presented the effect of using triangular obstacle on the temperatures and velocity distribution into the cavity. The results showed that the increasing the height of the obstacle will lead to induced the air flow inward the heated bottom wall and increased the mixed convection process in the whole domain of the cavity. And this lead to increase the airflow circulating inside the cavity and increased the enhancement of the heat transfer of the mixed convection. The figures showed that the maximum velocity increased by about 10% when used a triangular vertical obstacle that attached on the top wall with height ratio of \( h/H=0.5 \) but the maximum temperatures will have decreased by about 22% in this case. Also, the using three blocks inside the cavity will lead to redistributed the heated air inside the cavity but it decreased the maximum velocity from 1.215 to 1.111 m/sec due to increasing the air flow resistance inside the cavity, but the maximum temperatures stay decrease from 474 to 417 K in the same conditions.

The profiles of pressure, velocity and temperatures is plotted in Figures 6, 7 and 8 respectively for the mixed convection in a cavity for three locations along the y-axis at \( \text{Re} = 1000, \text{Gr} = 10^6. \) It should be noted that the pressure profile will be affected by the using the triangular obstacle and the pressure in the location of the upstream \( (x/L=0.4) \) will increased with increasing the obstacle height ratio and with locate the three blocks inside the cavity. However, the velocity profiles will differ from point to point in these locations, due to changing the area of the air flow when using the triangular obstacle and the three blocks inside the cavity, for example the maximum velocity will increased from 0.62 to 0.74 at \( x/L=0.5 \) when increasing the obstacle height ratio from \( h/H=0 \) to \( h/H=0.5. \) The temperatures profile will slightly be decreasing from the bottom heated wall \( (y=0 \text{ m}) \) to the adiabatic top wall \( (y=0.35 \text{ m}) \) due to the stream of the air flow at the top region, but when using the triangular obstacle along the top wall, the temperature decreased significantly, due to increasing the cooling region inside the cavity when the heat source is located in the bottom wall of the cavity to investigated the main vortex in the right bottom corner and a series of low-temperature tour circulation zones in corner on both sides of the cavity. In both cases, well expressed depth temperature heterogeneity cavity and there is a probability of measurement knowledge of the heat transfer leads to the formation of one broad circulation vortex and, accordingly and intensive air mixing of air throughout the cavity. Analysis of the results (Figures 7 and 8) allows circulation air currents identity flows in the fall, while remote temperatures are significantly being different.
Figure 5. Velocity and temperatures counters for three models of the cavity.

- a) Without obstacle
- b) With obstacle $h/H=0.25$
- b) With blocks and obstacle $h/H=0.25$
Figure 6. Pressure profile along three sections passing through the center of the cavity.
Figure 7. Velocity profile along three sections passing through the center of the cavity.
Figure 8. Temperatures profile along three sections passing through the center of the cavity.
Analysis of the influence of the various parameters such as obstacle height ratio (h/H) Reynolds number (Re) of blocks inside the cavity on the local and average Nusselt number at the heated bottom wall was carried using the equation 9. In the analysis of the dimensionless coefficient heat transfer agent at all boundaries of the area to be established that heat exchange between hot bottom wall and the surrounding environment of the air is carried out predominantly through the upper. The results showed that Nusselt numbers are significantly affected by obstacle height ratio (h/H) Reynolds number (Re) and number of blocks as presented in Figures 9 to 14.

In Figure 9 shows the profiles of Nusselt number along the heated bottom wall of the cavity at y = 0. By comparing the Nusselt number profiles for four cases of height ratio of the triangular obstacle, it can be seen that when increasing obstacle height ratio from h/H=0 to 0.5 the Nusselt number increased. By analyzing the mixed convection Nusselt number mode, should note that the Nusselt number decreasing upward the axial distance of the cavity varies significantly. But when located the triangular obstacle in the medium part of the top wall, the Nusselt number will increased with increasing the height of this obstacle. Moreover, the Nusselt number increased with increasing the Reynolds number as shown in Figure 10 in all conditions. Also, it is noticeable that with increasing the numbers Re are the values of the dimensionless coefficient heat transfer rate for all options. For example, the dependence $\text{Nu}_{\text{avg}} = f(\text{Re})$ for solution areas with central placement of the triangular obstacle is distinguished.

Figures 11 and 12 presented the effect of locate one or more blocks inside the cavity on the local and mean Nusselt number respectively, the results showed that the increasing the number of blocks inside the cavity will lead to increasing the Nusselt number but with rundum order of the Nusselt number profile. It can mark it should be noted that the dependences $\text{Nu}_{\text{avg}} = f(\text{Re})$ change at different layouts of cavity and different conditions of heat transfer on the top wall boundary of the cavity. Figure 13 presented the variation of average Nusselt number with Reynolds number for various number of blocks. The results showed that the Nusselt number increased by about 65% when increasing triangular obstacle height ratio from (h/H=0 to 0.5), at Re=1100. Finally, Figure 14 plotted the relation between the average Nusselt number and Rayleigh number for various number of blocks. the results showed that the variation of the Nusselt number with Rayleigh number is less than the average Nusselt number with Reynolds number, due to the lower heat flux is considered in this study, but increasing the number of blocks inside the cavity has significantly more complicated due to bridges accounting for mixed convection.

Different triangular obstacle height ratio also will lead to a change in structure currents and temperature fields in a cavity with number of blocks. Location of the adiabatic blocks inside the cavity affects the heat transfer between air and a heated bottom wall boundary. It can be concluded that with increasing the triangular obstacle height in the cavity, the temperature distribution will change modes also. Accordingly, for various triangular obstacle height it is needed to change the number of blocks in the cavity for increasing the intensity of exchange and reduce its likelihood heating.
Figure 9. Variation of local Nusselt number with axial distance for various obstacle height ratio.

Figure 10. Variation of local Nusselt number with axial distance for various Reynolds number.
Figure 11. Variation of local Nusselt number with Reynolds number for various number of blocks.

Figure 12. Variation of average Nusselt number with Reynolds number for various number of blocks.
Figure 13. Variation of average Nusselt number with Reynolds number for various number of blocks.

Figure 14. Variation of average Nusselt number with Rayleigh number for various number of blocks.
6. Conclusions
The mixed convection heat transfer of air flow in a cavity with bottom local heat source with vertical triangular obstacle on the top wall in order to enhancement the convection inside the cavity is considered in numerical modeling. The system of governing equations is solved numerically by discretizing the compositional domain into small grids. The results are presented in form of the pressure, temperature and velocity contours distributions. The results showed that the main circulating currents in different zones of the cavity is due to the effect of number of blocks and the presence vertical triangular obstacle on the top boundary of the cavity. The formation of thermal conditions in the region under study and the effect of low temperatures above the heated bottom wall on the circulation fluid flow in a cavity. The results showed that Nusselt number increased by 65% when increasing triangular obstacle height ratio from \((h/H=0)\) to \((0.5)\). at \(Re=1100\). Distribution pattern established velocity and temperature profile in centered sections depending on the height of vertical triangular obstacle. Also, Nusselt number increased at 26% when increasing the height ratio of triangular obstacle to \((h/H=0.5)\) with three blocks.

References
[1] Elistratov, S.L., 2010 Comprehensive study of the effects of the activity of heat pumps, dis. doc tech. Sciences - Novosib-Birs, 383 p.
[2] Kuznetsov, G.V., Maksimov V.I., 2006 Mixed convection in a rectangular area with local sources mass input and output in conditions of inhomogeneous heat transfer, Bulletin of the Tomsk Polytechnic, 5, pp.114–118.
[3] Cong, R., Zhou X. and B. Machado, 2016 Mixed Convection Flow of Nanofluid in a Square Enclosure with an Intruded Rectangular Fin, International Conference on Mechanical Engineering (ICME 2015), Dhaka, Bangladesh: AIP Publishing.
[4] Goshayeshi, H. and Safaiy, M., 2011 Investigation of turbulence mixed convection in air filled enclosures, Journal of Chemical Engineering and Materials Science 2(6), pp. 87-95.
[5] Oztop, H., Dagtekin, I., and Bahloul, A., 2004 Comparison of position of a heated thin plate located in a cavity for natural convection, Int. Commun. Heat Mass Transfer 31, pp. 121–132.
[6] Adriano, S., Fontana, E., Mariani V. and Marcondes, F., 2012 Numerical investigation of several physical and geometric parameters in the natural convection into trapezoidal cavities, International Journal of Heat and Mass Transfer 55, pp. 6808–18.
[7] Islam, A., Sharif M. and Carlson, E. 2012 Mixed convection in a lid driven square cavity with an isothermally heated square blockage inside, Int. J. Heat Mass Transf. 55, pp. 5244–55.
[8] Ahammad, M., Rahman M. and Rahman, M., 2014 A Study on the Governing Parameters of MHD Mixed Convection Problem in A Ventilated Cavity Containing A Centered Square Block, International Journal of Scientific & Technology Research 3(5).
[9] Boulahia, Z., and Sehaqui, R., 2015 Numerical Simulation of Natural Convection of Nanofluid in a Square Cavity Including a Square Heater, International Journal of Science and Research (IJSR), ijsr.net, 4(12), pp.1718-22.
[10] Razera, A., Fagundes, T., Seibt, F., da Fonseca, R., Varela, D., Ortiz, P., Coelho, F., Lessa, L., Schmidt, A., Furtado, G., dos Santos, E., Isoldi, L., Rocha, L., 2017 Constructal Design of a Triangular Fin Inserted in a Cavity with Mixed Convection Lid-Driven Flow. Defect and Diffusion Forum 372, pp.188–201.
[11] Rodrigues, P., de Escobar, C., Rocha, L., Isoldi, L., dos Santos, E., 2019 Geometry Evaluation of Heat Transfer by Mixed Convention in Driven Cavities with Two Inserted Fins. Defect and Diffusion Forum 396, pp.164–173.
[12] Carozza, A., 2018 Numerical Study on Mixed Convection in Ventilated Cavities with Different Aspect Ratios, Fluids 3(11).
[13] Bakar, N., Roslan, R., Karimipour A., and Hashim, I., 2019 Mixed Convection in Lid-Driven Cavity with Inclined Magnetic Field, *Sains Malaysiana* 48(2), pp. 451–471.

[14] Bejan A., 1993 *Heat Transfer*, John Wiley and Sons.

[15] Verstege, H. and Meer, W. 1995 *An introduction of computational fluid dynamics*, Hemisphere Publishing Corporation, United State of Americas.

**Nomenclatures**

- $c_p$: specific heat, J/kgK
- $g$: gravitational acceleration, m·s$^{-2}$
- $H$: cavity height, m
- $h$: heat transfer coefficient, W/m$^2$·K
- $H$: triangular obstacle height, m
- $k$: thermal conductivity, W/m·K
- $L$: cavity length, m
- $Le$: characteristic length, m
- $Nu$: Nusselt number
- $P$: dimensionless pressure
- $p$: pressure, N/m$^2$
- $p$: blocks pitch, m
- $Pr$: Prandtl number
- $q''$: applied heat flux, W/m
- $Re$: Reynolds number
- $s$: blocks height, m
- $T$: dimensional temperature, K
- $u$, $v$: dimensional velocities components in x and y directions, m/s
- $U$, $V$: dimensionless velocities components in X and Y directions
- $w$: block width, m
- $x$, $y$: dimensional Cartesian coordinates, m
- $X$, $Y$: dimensionless Cartesian coordinates

**Greek Symbols**

- $\alpha$: thermal diffusivity, m$^2$·s$^{-1}$
- $\beta$: thermal expansion coefficient, 1/K
- $\theta$: dimensionless temperature
- $\mu$: dynamic viscosity, kg/m·s
- $\nu$: kinematic viscosity, m$^2$/s
- $\rho$: density, kg/m$^3$