3D Buoyancy Induced Heat Transfer in Triangular Solar Collector Having a Corrugated Bottom Wall

Walid Aich
Mechanical Engineering Department
University of Hail, Saudi Arabia
Research Unit of Materials, Energy and Renewable Energies (MEER)
Faculty of Sciences of Gafsa, Tunisia
aich_walid@yahoo.fr

Abstract—A numerical study of the natural convection heat transfer and fluid flow in 3D triangular solar collector having a corrugated bottom wall has been carried out using finite volume method. The aim of the study is to investigate how buoyancy forces influence airflow and temperature patterns inside the collector heated from below and cooled on its inclined walls while vertical ones are assumed to be perfect thermal insulators. Rayleigh number is the main parameter which changes from $10^3$ to $10^5$ and Prandtl number is fixed at $Pr=0.71$. Results are reported in terms of particles trajectories, iso-surfaces of temperature, velocity magnitude and mean Nusselt number. It has been found that the flow structure is sensitive to the value of Rayleigh number and that heat transfer is enhanced with increasing of this parameter.

Keywords—natural convection; heat transfer; solar collector; Rayleigh number; Nusselt number

I. INTRODUCTION

A prodigious importance has been given to the natural convection heat transfer phenomena which can be seen in many energy-related applications, such as thermal insulation of buildings using air gaps, solar energy collectors, furnaces and fire control in buildings and so on. These applications are reviewed by many authors in the literature [1–4]. The shape of the enclosure is the most effective parameter for natural convection studies in enclosure. The enclosures encountered in these applications are highly diverse in their geometrical configuration and the most investigated enclosures include the annulus between horizontal cylinders, the spherical annulus, the hollow horizontal cylinder, the closed rectangular cavity, and the closed triangular cavity. Pioneers studied natural convection in triangular cavities in [5–7]. Authors in [8, 9] modeled the winter and summer day boundary conditions inside a roof of triangular cross-section. Assuming a geometric symmetry through the midplane of an isosceles triangular cavity, author in [10] examined a right angled triangular cavity with a heated bottom wall, cooled hypotenuse, and insulated vertical wall. Various aspect ratios and preselected Rayleigh numbers were used for the reduced right-angled triangular cavity filled with air or water. Solutions of the time-dependent conservation equations were obtained using two different numerical techniques, which while yielding different numerical values for the velocity and temperature fields, did not alter the flow structure pertinent to a single convective cell for low Rayleigh number and to a multicellular regime for high Rayleigh number. Authors in [11] carried out the natural convection problem with flush mounted heater on one wall of a triangular cavity. Governing parameters on heat transfer and flow fields are triangle aspect ratio, heater location, heater length and Rayleigh number. They observed that the most important parameter on heat transfer and flow field is the heater position which can be a control parameter for their system.

Authors in [12] analyzed a numerical study of heat and mass transfer due to natural convection inside a triangular enclosure. The governing equations of the two-dimensional flow problem consist of a velocity–pressure (UVP) formulation along with the energy and concentration flow equations. These equations are solved numerically by control volume based finite element method using the equal-order method without pressure correction. They found that the buoyancy ratio and the Lewis number values have a profound influence on the thermal, concentration and dynamic fields. Steady state laminar natural convection in right triangular and quarter circular enclosures was investigated in [13]. The presented results showed that both aspect ratios and Rayleigh number have a strong influence on the streamline patterns and isothersms. A sizable amount of other related studies can be found in the literature review [14–25]. Only limited attention has been paid to the study of three-dimensional transverse flow which is predominant when dealing with the enhancement of heat transfer. The paramount aim of this work has been to numerically investigate the diffusive natural convection heat transfer and fluid flow in a three-dimensional triangular solar collector heated from its corrugated bottom wall.

II. MATHEMATICAL FORMULATION

A. Physical Model

The considered physical model is presented in Figure 1 with its specified coordinate system and boundary conditions. The model is a triangular shape solar collector with a corrugated bottom wall. The gravity acts in the negative y axis...
and the cavity is formed by the heated corrugated absorber plate, the two cooled inclined glass covers and the two vertical adiabatic walls.

![Diagram of the problem with domain and boundary conditions.](image)

**B. Governing Equations and Numerical Solution**

As numerical method we used the vorticity-potential vector formalism \(\vec{\omega} - \vec{\omega}\) which allows, in a three-dimensional configuration, the elimination of the pressure, which is a delicate term to treat. To eliminate this term one applies the rotational to the equation of momentum. More details on this 3-D formalism can be found in [26]. The potential vector and the vorticity are respectively defined by the two following relations:

\[
\vec{\omega} = \vec{\nabla} \times \vec{V} \quad \text{and} \quad \vec{V} = \vec{\nabla} \times \vec{\psi}
\]

(1)

After nondimensionalization, the system of equations controlling the phenomenon becomes as:

\[
-\vec{\omega} = \vec{\nabla}^2 \vec{\psi}
\]

(2)

\[
\frac{\partial \vec{\omega}}{\partial t} + (\vec{V} \cdot \vec{\nabla}) \vec{\omega} - (\vec{\omega} \cdot \vec{\nabla}) \vec{V} = \Delta \vec{\omega} + Ra \cdot Pr \cdot \left[ \frac{\partial T}{\partial z} \right] \quad \text{(3)}
\]

\[
\frac{\partial T}{\partial t} + \vec{V} \cdot \vec{\nabla} T = \Delta T
\]

(4)

with: \(Pr = \frac{v}{\alpha}\) and \(Ra = \frac{g \beta \Delta T L^3}{\nu \alpha}\)

Boundary conditions for considered model are given as follows:

- **Temperature:** \(T = 1\) at the corrugated absorber, \(T = 0\) at inclined glass covers and \(\frac{\partial T}{\partial z} = 0\) on vertical walls (adiabatic).

- **Velocity:** \(V_x = V_y = V_z = 0\) on all walls. The average Nusselt at the hot corrugated wall is given by:

\[
Nu_{av} = \frac{2}{\beta} \int_0^1 \frac{\partial T}{\partial n} dx dz
\]

(5)

where \(n\) is the unit vector normal to the hot corrugated wall.

The mathematical model described above was written in a FORTRAN program. The control volume finite difference method is used to discretize governing equations. The central-difference scheme is used for treating convective terms while the fully implicit procedure is used to discretize the temporal derivatives. The grids are considered uniform in all directions with clustering nodes on boundaries. The successive relaxation iteration scheme is used to solve the resulting non-linear algebraic equations. A computer program written for a regular grid was improved to handle the irregularly shaped computational domain using the blocked-off method as described in [27]. In this technique, the whole region is divided into two active and inactive (blocked-off regions) parts. Thus the surface of inclined step in present analysis is approximated by a series of fine cubic steps. It is obvious that using fine grids in the interface region between active and inactive zones causes to have an approximate boundary which is more similar to the true boundary. According to the blocked-off technique, known values of the dependent variables must be established in all inactive control volumes. If the inactive region represents a stationary solid boundary as in this case, the velocity components in that region must be equal to zero, and a known temperature (isothermal boundaries) must be established in the inactive control volumes. The control volumes, which are inside the active region, are designated as 1 and otherwise they are 0. The time step \((10^{-4})\) and spatial mesh \((81 \times 81 \times 81)\) are utilized to carry out all the numerical tests. The solution is considered acceptable when the following convergence criterion is satisfied for each time step:

\[
\sum_i \left[ \max \left| \psi_i^{n+1} - \psi_i^n \right| + \max \left| T_i^{n+1} - T_i^n \right| \right] \leq 10^{-4}
\]

(6)

**III. RESULTS AND DISCUSSION**

Particle trajectories for different Rayleigh number values are illustrated in Figure 2. It is noted that Prandtl number is fixed at \(Pr=0.71\) for the whole work and the Rayleigh number which is the main parameter changes from \(10^5\) to \(10^7\). As it can be seen, the flow strength is very low for \(Ra=10^5\) and two opposite rotating vortices are formed showing poor convective heat transfer. Flow strength increases with increasing of Rayleigh number. Indeed, the fluid takes up heat energy from the bottom heated absorber and becomes lighter to induce a convective current. Lighter fluid goes up and due to the symmetric nature of colder glass covers, the colder portion of fluid follows the line traced by the colder walls. As a consequence, a significant change in convection is observed and strongest flow pattern is achieved for \(Ra=10^7\). In Figure 3 shows the effect of varying Rayleigh number on iso-surfaces of temperature inside the enclosure. At \(Ra=10^5\), convection is not significant at all and the iso-surfaces of temperature are parallel to each other and are densely distributed near the bottom heated absorber. The parallel nature of iso-surfaces of temperature shows the domination of conductive heat transfer. However, plumelike distribution on temperature field is observed at \(Ra=10^7\) due to the starting of convection regime. Iso-surfaces are seen to be densely packed along the boundaries showing that convection is very strong in those regions. At the corners these iso-surfaces are more and more densely packed showing the presence of a strong conduction heat transfer. The temperature distribution assumes a distorted shape near the
middle and the top due to increasing of effectiveness of convection heat transfer. Strong convection occurred for $Ra=10^5$ and stronger plumelike distribution is observed.

![Particles trajectory for different Ra values: (a) 10^3, (b) 10^4, (c) 10^5](image)

The velocity magnitude of the fluid flow inside the enclosure is presented in Figure 4 for different Rayleigh number values. At the lowest value of this parameter and due to the domination of conduction regime of heat transfer, the fluid is almost at rest in the enclosure with negligible motion at the middle far from boundaries. By increasing $Ra$, the heated and rising fluid leads to formation of the thermal boundary layer at the upper part of the inclined walls and there is a strong convective current in the middle of the cavity while corners remain as stagnant zones. This result supports our observations made during the discussion on the particles trajectory and temperature distribution. The highest magnitude of velocity occurred for $Ra=10^5$ and a very strong double plumelike distribution is observed due to increasing of effectiveness of convection heat transfer and fluid flow at the middle of the collector. It is noticed that the heat transfer rate inside the enclosure is measured in terms of the overall Nusselt number. Figure 5 shows the variation of the average Nusselt number with the Rayleigh number $Ra$. It is obvious that for low values of $Ra$ and when the conduction is the dominant mode of heat transfer, this variation is insignificant. However, for $Ra>10^4$ heat removal from the heated corrugated absorber increases by means of increasing $Ra$ and the maximum rate is obtained for the highest $Ra$ as expected. The corrugated form of the bottom of the collector is found to enhance heat transfer due to increasing of heated surface when compared with simple plane wall.

![Iso-surfaces of temperature for different Ra values: a) 10^3, (b) 10^4, (c) 10^5](image)

IV. CONCLUSION

Three-dimensional numerical investigation has been carried out to simulate buoyancy induced heat transfer and fluid flow inside a triangular enclosure with corrugated bottom wall. Results are presented for different values of Rayleigh number which is the main parameter of the study. Some of the drawn conclusions are:

- For lower values of $Ra$ conduction is the primary mode of heat transfer and the flow strength is very low due to poor convective heat transfer.
- Flow strength increases with the increasing of $Ra$ and a strong convective current is noticeable in the middle of the enclosure far from boundaries.
- The flow structure and the magnitude of velocity are sensitive to the value of $Ra$.
- Overall Nusselt number at the heated surface increases with increasing value of $Ra$ indicating a maximum heat transfer rate at $Ra=10^5$.
- Even at high $Ra$ the symmetry of the flow is not broken contrary to results of many previous studies in 2D cases.

Further study may include the effect of filling the enclosure with nanoparticles in water based fluid under the effect of magnetic field.
ACKNOWLEDGMENT

Author would like to express his deepest gratitude to Dr. Ali Amri and his institution, “The English Polisher” for their meticulous and painstaking review of the English text of the present paper.

REFERENCES


