Proceedings of the 7th World Congress on Momentum, Heat and Mass Transfer (MHMT'22)
~~Lisbon, Portugal~~ Virtual Conference – April 10 – 12, 2022
Paper No. ENFHT 197
DOI: 10.11159/enfht22.197

Numerical Study of the Effects of Humidity on Natural Convective Flows in Building-Integrated Photovoltaic (BIPV) Systems

H. Ahmadi Moghaddam[1*], S. Tkachenko[1], J. Reizes[1], R. Raja[2], C. Menezo[2], S. Giroux–Julien[3], V. Timchenko[1]
[1]School of Mech. & Manuf. Eng., UNSW Sydney, Australia
[2]University Savoie Mont-Blanc, LOCIE UMR CNRS 5271, France
[3]Univ Lyon, CNRS, INSA-Lyon, Université Claude Bernard Lyon 1, CETHIL UMR5008, France
∗Correspondence author. Email: h.ahmadi_moghaddam_dastjerdi@unsw.edu.au

Abstract **-** Due to the importance of employing renewable energy resources, the investigation of building-integrated photovoltaic (BIPV) systems focused on cooling PV panels has received much attention recently. A numerical study has been conducted to evaluate the effects of humidity on turbulent natural convective flow and thermal radiation in an open-ended channel coupled with a room under different values of surface emissivity and heat load with different heating modes, noting that the humidity effects have not been assessed thoroughly in the literature. The potential of generated natural convective flow inside the chimney for cooling the chimney walls (representing PV panels) and ventilating the room space has been investigated. It was found that humid air has a higher cooling potential than dry air. Introducing humidity to the airflow (with the mass fraction of 0.01365) and increasing the surface emissivity (from 0.1 to 0.95) both contributed to an increase in the free convective mass flow rate and a decrease in the temperature of chimney walls. As compared to the most critical case with the highest heat load (600 W/m^2) using dry air and low emissivity under the single heating mode, employing both cooling techniques simultaneously decreased the surface temperature up to 39 K. The numerical results suggest thermal management of BIPV systems through techniques such as high emissivity coatings and humidity control.

Keywords**:** BIPV, Natural Convective Flow, Radiation, Chimney, Surface Emissivity

1. Introduction

Investigating the performance of thermal-fluid systems such as cooling or heating systems and utilizing renewable energy resources have become major concerns by the growth of population worldwide and increase in demands for energy [1-5]. With that in mind, many investigations have been performed for performance assessment of building-integrated photovoltaic (BIPV) systems for producing clean energy. A problem with these systems is that the electrical efficiency of PV panels decreases by an increase in the temperature. Researchers have been investigating different parametric factors in these systems to attain lower temperatures in PV panels by employing the cooling potential of natural convective flow generated in the gap between the outer surface of a building and PV panels called double skin façade. A schematic view of a double skin façade is shown in figure 1. Due to the heat transferred to the air in the gap from the PV panels heated by solar radiation, a natural convective flow is generated which can be used for thermal management of panels [1].

In this regard, Lau *et al.* [6] numerically investigated the heat transfer characteristics of natural convective flow in open-ended channels with uniformly heated walls and inclination angles in the range of 15° to 90°. It was observed that the heat transfer rate enhances by increasing the inclination angle resulting in the temperature reduction of walls. Based on the discrepancies observed between the numerical and empirical data, it was deduced that radiative heat transfer has a significant effect on natural convective flow and should be considered in the numerical studies. Another computational study on turbulent natural convection flow by Lau *et al.* [7] revealed that the width of the channel plays a crucial role in the change of turbulent quantities resulting in varied heat transfer rate for different width values. Tkachenko *et al.* [8, 9] studied the effects of radiation and humidity on natural convective flow in vertical open-ended channels. Through their numerical model, it was found that adding humidity to flow postponed the transition to the turbulent regime close to the heated wall. In another empirical and computational study, Tkachenko *et al.* [10] simulated the operating conditions of a double skin façade by investigating the buoyancy-driven flow in a vertical channel with non-uniform heating patterns. It was found that more free convective mass flow is generated when both walls are heated than one heated wall. Wang and Lei [11] numerically studied the effects of design parameters of a solar chimney coupled with a water fall for building ventilation. Their study showed that the proposed system was successful in ventilation by modifying parameters such as glass panel or water column thickness.

In addition to factors such as heating modes or configuration of double skin façade, the type of the working fluid is another parameter that can affect the thermal performance of BIPV systems. Specifically, the presence of water vapor in the airflow is a parameter that needs to be investigated. Water vapor generally exists in the air at different levels. Considering the presence of humidity in indoor environments in processes such as ventilation, cooling, or heating, the present research numerically investigates a natural convective flow inside a chimney coupled with a room. In this regard, the effects of different parameters such as relative humidity, heating mode, and internal emissivity are revealed through a 3D CFD model. In the next sections, the details of the studied geometry and the numerical scheme are explained. Then, a discussion on the obtained results is presented to reveal the effects of the above-mentioned factors on the natural convective flow inside the chimney coupled with the room.

2. Numerical Procedure

2.1. Geometrical Configuration

The current CFD study is conducted in accordance with an experimental setup assembled at the LOCIE laboratory of Savoie Mont Blanc university in France [12]. The test rig is constructed by coupling a solar chimney with a reduced scale room as depicted in figure 2. The room walls and the chimney side walls are constructed from plexiglass of 1 cm thickness.

A window has been placed on one of the room walls working as the airflow inlet (shown by blue in figure 3). It should be noted that the inlet has been extended, as illustrated in figure 3, to correctly simulate the inflow entrainment. To simulate the effects of PV modules, two electrical heaters (as shown in figure 2) are installed on chimney walls. The walls containing the heaters are made from the insulation of 10 cm thickness, heater elements of 4 mm thickness, and a thin Radflek foil layer to control the wall emissivity by painting.

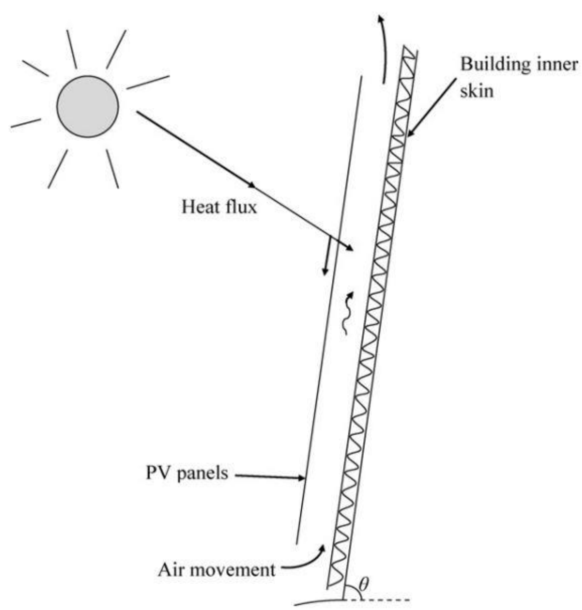


Figure 1. A schematic view of a PV panel installed on a wall (adapted from [6]).

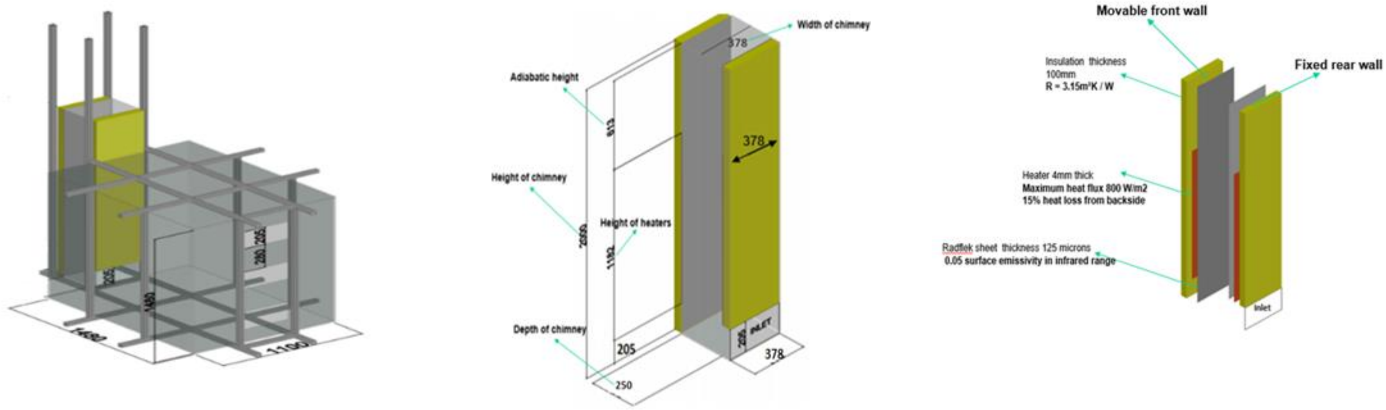


Figure 2. The geometrical dimensions of the studied case (dimensions in mm).

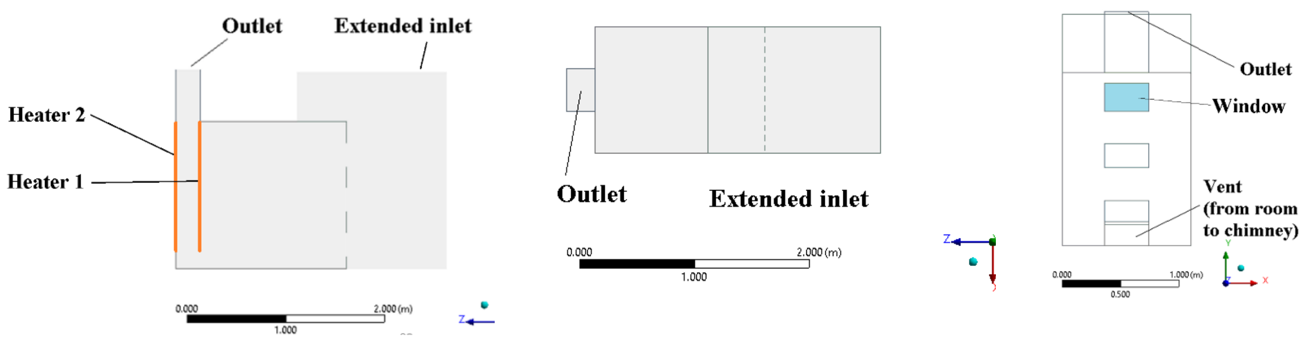


Figure 3. The domain and boundaries of the current CFD model.

2.2. Governing Equations

The current 3D CFD model is developed in ANSYS Fluent 2021R1. As primary steps of this research, steady-state calculations have been performed to evaluate the system performance and mean flow characteristics without a detailed focus on turbulence statistics. The thermophysical properties of air are considered constant expect density which is computed based on the ideal gas relation. The momentum, mass, and energy conversation equations have been solved:

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial \left(\rho u_j\right)}{\partial x_j} = 0 \tag{1}$$

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial \left(\rho u_i u_j\right)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \sigma_{ij}}{\partial x_j} + \left(\rho - \rho_{ref}\right) g_i \tag{2}$$

$$\frac{C_p \partial (\rho T)}{\partial t} + \frac{C_p \partial (\rho u_i T)}{\partial x_j} = k \frac{\partial}{\partial x_j}\left(\frac{\partial T}{\partial x_j}\right) + \frac{\partial q_j}{\partial x_j} \tag{3}$$

Where g, ρ_{ref}, u, T, P, C_p, σ_{ij}, k, and q_j are the gravity vector, reference density, velocity vector, temperature, dynamic pressure, the specific heat capacity of constant pressure, stress tensor, thermal conductivity, and turbulent thermal flux vector, respectively.

For the case of humid air, the working fluid is the mixture of two fluids (air and water vapor). Hence, the conservation relations for the gas species are computed [13].

While in authors' previous studies the LES approach was employed to adequately model the transitional structures of free convective flow in channels [6, 9, 10], RANS modelling, which is much less time-consuming as compared to LES, is chosen in the current research because this work has focused on the prediction of effects of design parameters on free convective flow in a chimney coupled with a room. The SST model utilizes $k-\omega$ formulation adjacent to walls where the viscous forces are considerable and switched to $k-\varepsilon$ formulation in areas far from walls. This characteristic has made it appropriate for use in different engineering applications [14].

The radiation effects were considered using surface to surface radiation model. The energy reflected from a surface m is computed according to the following equation:

$$q_{out,m} = \epsilon_m \alpha T^4 + \Omega_m \cdot \sum_{j=1}^{N} F_{jm} q_{out,j} \tag{4}$$

where $q_{out,m}$ is the energy flux leaving the surface m, α is the Stefan-Boltzmann constant, $\Omega_m = 1 - \epsilon_m$ is the reflectivity of surface m, ϵ_m is the emissivity of the surface m, F_{jm} is the view factor between surface m and surface j. The view factors are computed using ANSYS Fluent.

Structural mesh is used for meshing the domain including the room and chimney space while unstructured mesh is employed for the extended inlet area. Mesh convergence was attained at eight million mesh elements.

2.3. Studied Cases

In this research, a parametric study has been performed. The parameters include different values of heat flux (200, 400, and 600 W/m^2), different heating modes (double or single heating mode), the emissivity of chimney walls (0.1 and 0.95), and relative humidity. Regarding the heating conditions, once both heaters are active (double heating mode), the heat load is divided between them equally. In the single heating mode, the heater adjacent to the room (called heater 1) is inactive and the total heat load is imposed on the heater located at the external wall of the solar chimney (called heater 2). To assess the effects of emissivity, the emissivity has only changed on the two walls containing heaters from bottom to top. Furthermore, two types of working fluids have been used, dry air and humid air. In the second case, humidity is added to the air for which the mass fraction of water vapor is 0.01365 corresponding to the temperature of 300K and relative humidity of 61%.

3. Results and Discussion

3.1. Validation of the Numerical Procedure

The experimental results of Habib et al. [15, 16] have been used for the validation of numerical procedures. Their test rig consisted of two vertical plates with the dimensions of 125 mm×40 mm kept at constant temperatures. The depth of plates was 200 mm to decrease the 3D effects in the system. More details of the experimental setup and operating conditions are provided in [15, 16]. The primary attempts to reproduce the empirical data by modelling only the area between the plates were unsuccessful. Therefore, the domain was extended. A similar approach was considered in the rest of the present simulations. As shown in figure 4 (a), the velocity distribution has been predicted well with respect to the experimental data obtained at the mid-height of the channel. The flow features and velocity magnitudes are in good agreement also considering Y-velocity contours at the midplane of the channel in figure 4 (b and c). Furthermore, the difference in the mass flow rate predicted by the current simulation and the experiments was 4%.

3.2. Effects of Humidity

The variation of buoyancy driven mass flow rate with imposed heat flux is depicted in figure 5 for both humid and dry cases under different values of emissivity. As shown, for the same value of heat flux and emissivity, the free convective mass flow rate inside the chimney is higher when humid air is used in comparison with dry air. The reason could be that the molecular weight of water vapor is smaller than that of the air. Consequently, by adding humidity, the the overall density of the mixture decreases giving rise to the convective currents [17]. Hence, the mass flow rate is higher higher when humid air is used.

It can also be seen in figure 5 that for a given value of heat flux and emissivity and for both humid and dry air cases, double heating mode generates more mass flow rate than single heating mode. When two heaters are active, the buoyancy effects are stronger than in the case when one heater is active because the flow is driven from both sides when both walls are heated. However, when only one wall is heated, the inactive or unheated wall exerts a drag yielding lower buoyancy-driven mass flow.

The variation of the mean temperature of heaters for both humid and dry air with applied heat flux is illustrated in figure 6. It is shown that adding humidity to the airflow reduces the mean temperature of heaters. The temperature contours of the heated wall (heater 2) for dry and humid cases under single heating mode with the heat flux of 600 W/m^2 are shown in figure 7. As can be seen, the temperature of the heater decreases by adding humidity to the air showing that humid air is a better coolant than dry air. When water vapor is introduced, the properties of the mixture including density, thermal conductivity, specific heat capacity, and viscosity change as compared with dry air. These changes contribute to enhanced heat transfer [17] resulting in the decrease of the walls temperature.

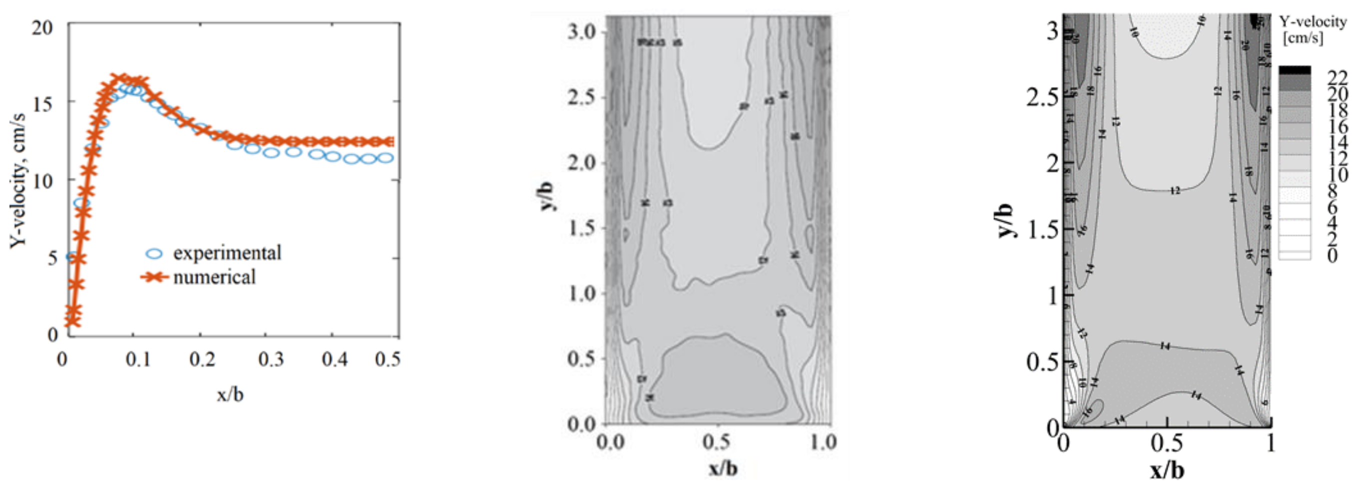


Figure 4. Comparison of the experimental results from Habib et al. [15, 16] with current numerical results (a), experimental (b) and numerical (c) velocity contours of velocity.

Based on the obtained results, it is found that when only one heater (heater 2) is active, the effect of increasing emissivity from 0.1 to 0.95 is considerable for both humid and dry cases. When the emissivity is low, free convection is the main mechanism for heat removal from the chimney heated walls. But the role of thermal radiation becomes significant by increasing the emissivity from 0.1 to 0.95. Indeed, when one heater is active, by increasing the emissivity the temperature of the inactive wall increases significantly. Heating of this inactive wall gives rise to the buoyancy effects while for the lower emissivity the inactive wall is maintained at low temperature. Therefore, it does not contribute to boosting the buoyancy effects.

It should be noted that, compared with two heaters active mode, one heater active mode resembles the real operating conditions in BIPV systems. In these systems, the PV modules (represented by heater 2 here) are heated directly by solar radiation and receive significant heat flux while the outer surface of buildings (represented by heater 1 here) does not receive much heat flux. Therefore, it can be suggested that thermal management of PV panels by increasing the emissivity of surfaces in BIPV systems using methods such as high-emissivity coatings is practical.

From the results in figure 6, for the largest heat load (600 W/m^2) under single heating mode using dry air, it is observed that increasing the surface emissivity from 0.1 to 0.95 reduces the mean surface temperature of heater 2 by about

30 K resulting in the mean surface temperature of about 354 K. However, when the humidity was introduced, under the same heat load and heating mode with the emissivity of 0.95, the surface mean temperature was further decreased by about 9 K resulting in mean surface temperature of about 345 K. It can be observed that simultaneous usage of both techniques, i.e., increasing the surface emissivity and adding humidity to the airflow, shows the most prominent effect on cooling walls by overall temperature reduction of about 39 K.

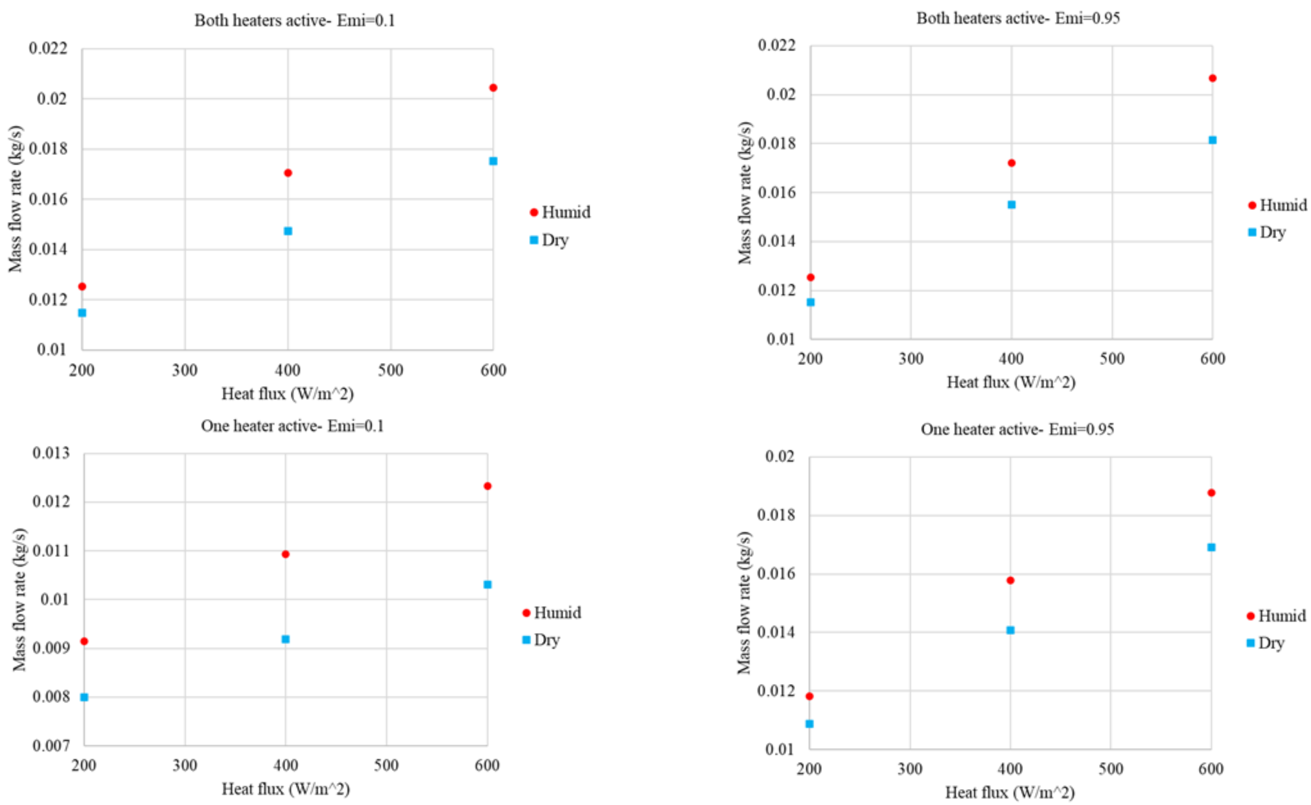
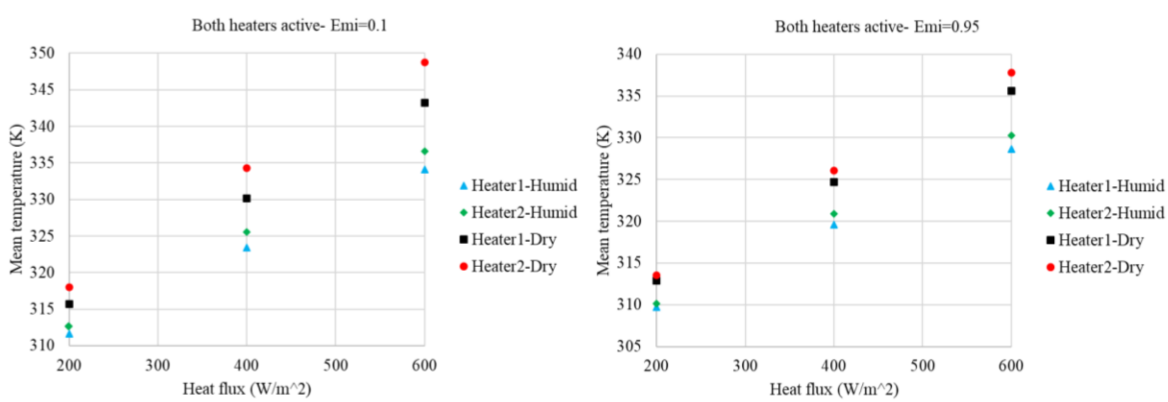


Figure 5. Comparison of free convective mass flow rate for dry and humid air.



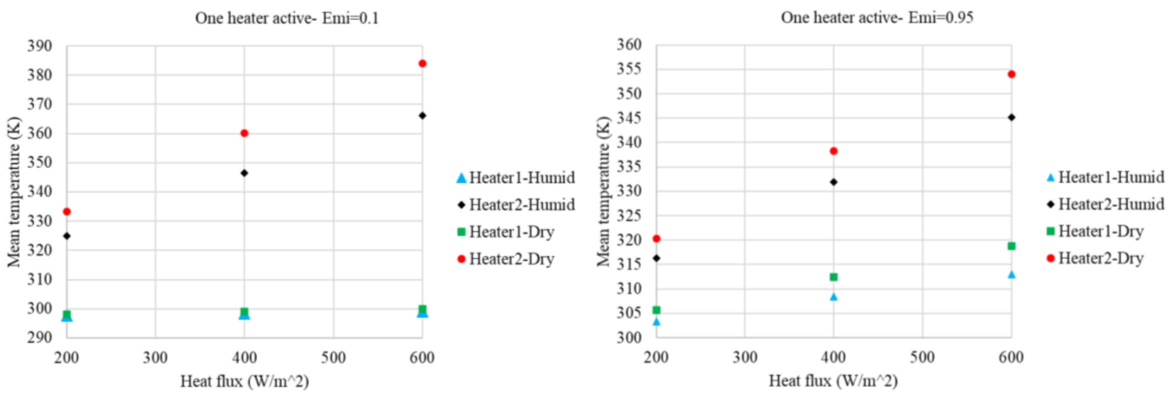


Figure 6. Changes of mean temperature of heaters with heat flux for both dry and humid air.

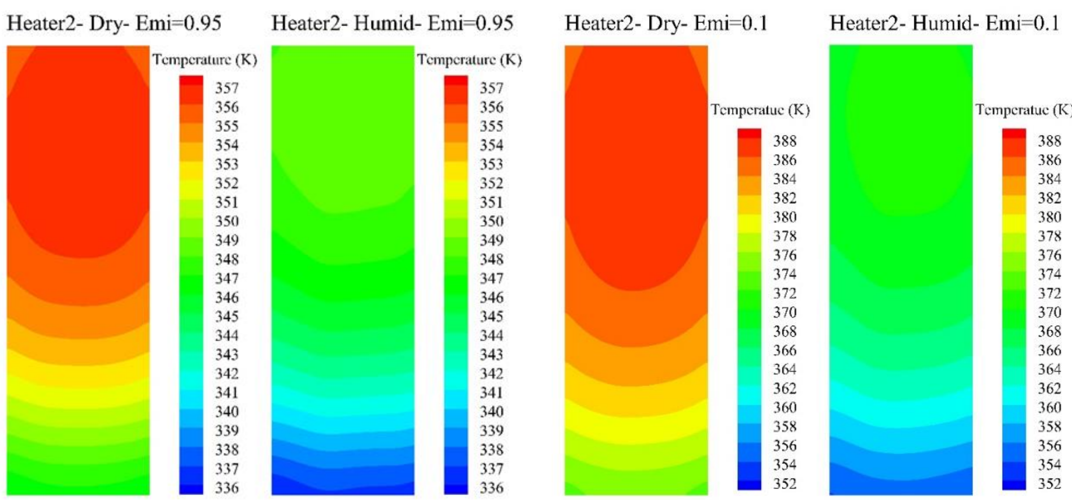


Figure 7. Temperature contours for heater 2 in single heating mode with heat flux of 600W/m^2.

4. Conclusion

This 3D numerical research investigates the heat transfer characteristics of a natural convection flow in a vertical open-ended channel coupled with a room (with application to BIPV systems) to reveal the effects of different parameters such as relative humidity, surface emissivity, and heating mode. The results showed that humid air is a better coolant than dry air. Furthermore, increasing the emissivity of channel walls is effective in the thermal management of surfaces. Considering the most critical case with the highest heat load under single heating mode where the surface emissivity was low and dry air was used (resulting in the mean surface temperature of about 384 K for heater 2), simultaneous usage of both techniques, i.e., increasing the surface emissivity from 0.1 to 0.95 and introducing humidity to the airflow, resulted in the mean surface temperature of about 345 K showing a temperature drop of about 39 K.

Acknowledgements

The funding provided for this research through the Australian Government Research Training Program (RTP) Scholarship is appreciated. This research includes computations using the computational cluster Katana supported by Research Technology Services at UNSW Sydney and the super-computer Gadi with the assistance of services from the National Computational Infrastructure under the support of the Australian Government.

References

[1] G. Asefi, A. Habibollahzade, T. Ma, E. Houshfar, and R. Wang, "Thermal management of building-integrated photovoltaic/thermal systems: A comprehensive review," *Solar Energy,* vol. 216, pp. 188-210, 2021.

[2] A. Sarmadian, H. A. Moghaddam, A. Asnaashari, H. Ahmadi Nejad Joushani, M. Moosavi, M. S. Islam, S. C. Saha, and M. Shafaee, "Flow boiling heat transfer and pressure drop characteristics of Isobutane in horizontal channels with twisted tapes," *International Journal of Heat and Mass Transfer,* vol. 162, p. 120345, 2020.

[3] H. A. Moghaddam, A. Sarmadian, M. Shafaee, and H. Enayatollahi, "Flow pattern maps, pressure drop and performance assessment of horizontal tubes with coiled wire inserts during condensation of R-600a," *International Journal of Heat and Mass Transfer,* vol. 148, p. 119062, 2020.

[4] H. A. Moghaddam, A. Sarmadian, A. Asnaashari, H. Ahmadi Nejad Joushani, M. S. Islam, S. C. Saha, G. Ghasemi, and M. Shafaee, "Condensation heat transfer and pressure drop characteristics of Isobutane in horizontal channels with twisted tape inserts," *International Journal of Refrigeration,* vol. 118, pp. 31-40, 2020.

[5] H. A. Moghaddam, A. Sarmadian, and M. Shafaee, "An experimental study on condensation heat transfer characteristics of R-600a in tubes with coiled wire inserts," *Applied Thermal Engineering,* vol. 159, p. 113889, 2019.

[6] G. E. Lau, E. Sanvicente, G. H. Yeoh, V. Timchenko, M. Fossa, C. Menezo, and S. Giroux-Julien, "Modelling of natural convection in vertical and tilted photovoltaic applications," *Energy and Buildings,* vol. 55, pp. 810-822, 2012.

[7] G. Lau, G. Yeoh, V. Timchenko, and J. Reizes, "Numerical investigation of passive cooling in open vertical channels," *Applied Thermal Engineering,* vol. 39, pp. 121-131, 2012.

[8] S. A. Tkachenko, V. Timchenko, G. H. Yeoh, and J. Reizes, "Effects of Humidity on Natural Convection in a Differentially Heated Cubic Cavity," in *Proceedings of CHT-15. 6 th International Symposium on Advances in Computational Heat Transfer*, A. Leontiev, D. Butterworth, M. D. G. Carvalho, and K. Hanjalic, Eds., 2015: Begel House Inc.

[9] S. Tkachenko, V. Timchenko, G. Yeoh, and J. Reizes, "Effects of radiation on turbulent natural convection in channel flows," *International Journal of Heat and Fluid Flow,* vol. 77, pp. 122-133, 2019.

[10] O. Tkachenko, V. Timchenko, S. Giroux-Julien, C. Menezo, G. H. Yeoh, J. Reizes, E. Sanvicente, and M. Fossa, "Numerical and experimental investigation of unsteady natural convection in a non-uniformly heated vertical open-ended channel," *International Journal of Thermal Sciences,* vol. 99, pp. 9-25, 2016.

[11] H. Wang and C. Lei, "A numerical investigation of combined solar chimney and water wall for building ventilation and thermal comfort," *Building and Environment,* vol. 171, p. 106616, 2020.

[12] S. A. Tkachenko, H. A. Moghaddam, J. Reizes, R. Raja, C. Menezo, S. Giroux-Julien, and V. Timchenko, "COMPUTATIONAL STUDY OF NATURAL CONVECTION FLOW IN AN OPEN-ENDED CHANNEL COUPLED WITH A ROOM: APPLICATION TO BUILDING-INTEGRATED PHOTOVOLTAIC (BIPV) SYSTEMS," in *Proceedings of CHT-21 ICHMT International Symposium on Advances in Computational Heat Transfer*, 2021: Begel House Inc.

[13] M. Shafaee, M. Tavakol, R. Riazi, and N. Sharifi, "An investigation on the supersonic ejectors working with mixture of air and steam," *Journal of Mechanical Science and Technology,* vol. 29, no. 11, pp. 4691-4699, 2015.

[14] H. A. Moghaddam, M. Shafaee, and R. Riazi, "Numerical investigation of a refrigeration ejector: effects of environment-friendly refrigerants and geometry of the ejector mixing chamber," *European Journal of Sustainable Development Research,* vol. 3, no. 3, p. em0090, 2019.

[15] M. Habib, S. Said, S. Ahmed, and A. Asghar, "Velocity characteristics of turbulent natural convection in symmetrically and asymmetrically heated vertical channels," *Experimental Thermal and Fluid Science,* vol. 26, no. 1, pp. 77-87, 2002.

[16] T. Ayinde, S. Said, and M. Habib, "Experimental investigation of turbulent natural convection flow in a channel," *Heat and Mass Transfer,* vol. 42, no. 3, pp. 169-177, 2006.

[17] J. Zhang, A. Gupta, and J. Baker, "Effect of relative humidity on the prediction of natural convection heat transfer coefficients," *Heat transfer engineering*, vol. 28, no. 4, pp. 335-342, 2007.