PEER REVIEWED
ASIA-PACIFIC
SOLAR RESEARCH CONFERENCE

ASIA-PACIFIC
SOLAR RESEARCH CONFERENCE

Numerical and Experimental validation of natural convection inside single slope solar still

Benhadji Djamal[1], Timothy Anderson[1] and Roy Nates[1]

[1] *Department of Mechanical Engineering, Auckland University of Technology, New Zealand*

Introduction

The availability of freshwater is one of the main challenges facing humanity in achieving the sustainable development of societies (Zhang et al. 2018). The fast-economic growth of many countries in the world has led to a drastic escalation in water scarcity and water pollution. Consequently, there has been a decrease in the supply of potable water in many regions in the world (Sharon et al. 2018). In response to this, large amount of energy needs to be deployed in order to produce drinkable water from saline or brackish water. That said solar still technology offers a simple and practical solution for remote zones in the world. This is paramount, particularly where the population relies on the unsafe water from surface or underground for their survival.

In order to accurately assess the potential freshwater that can be produced by a solar still, it is necessary to undertake thermal modelling. Based on the research found in the literature, the effect of the solar still geometry on its performance has been examined extensively; both theoretically and experimentally. However, the results that have been reported yield a wide variety of contradictory results. For example, numerous researchers have examined the effect of the single-slope solar still cover angle on its production. They have suggested that the cover angle should be equal to the latitude of the studied location, though this appears unsubstantiated in light of the conflicting results.

Methodology

To address this, this work investigated the steady-state free convective heat transfer inside single slope solar still using computational fluid dynamics (CFD). For natural convection problems, (Altaç and Uğurlubilek 2016) and (Rincón-Casado et al. 2017) have shown that Realisable k-ε model delivers the best performance among all existing models. Therefore, the previous model was used to resolve the turbulence due to the fluid flow inside the enclosure. Boussinesq's approximation is considered to model the density variation since the temperature difference in single slope solar stills is low. Boundary conditions imposed to simulate steady-state natural convection in this study are shown in Figure 1. In this study, it is considered that the convective heat transfer in solar stills which occurs simultaneously with evaporation and condensation can be modelled as a geometry filled with saturated air. This has been used in many studies to predict natural convection in solar stills and reported good agreement with experimental work.

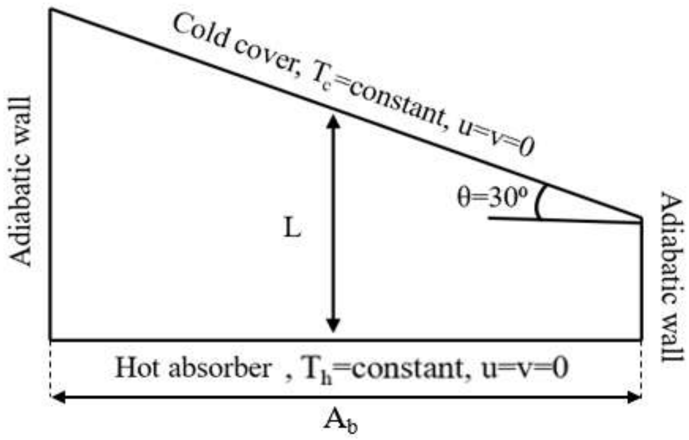

Figure 1: solar still geometry and boundary conditions.

In order to visualize the flow of humid air inside such geometry, silicone oil was used instead of humid air to ease the flow visualization while ensuring the same Rayleigh number was achieved. Particle image velocimetry (PIV) was used to investigate the flow inside the geometry. MATLAB software was used to analyse the 800 pairs of images obtained by the PIV experiment. It is worth to mention that it was not possible to take images of the whole cross-sectional area instantaneously hence the whole area-imaging was done by dividing it into 9 subareas.

Results

Analysis of the experimental results showed the development of similar flow features with the CFD simulation when steady-state conditions are reached. Figure 2 shows the velocity contours for both PIV experiment and CFD simulation at 200 mm at the centre plane in z-direction. In both cases, there is a plume rising at approximately 150 mm, splitting the flow in the enclosure into two cells. The comparison the simulation and the measurements show that near the walls, velocity is very low compared to the velocity at the rising plume. Furthermore, the predicted maximum velocity by the numerical model is 9.3 mm/s, 4% lower than the measured peak velocity of 9.7 mm/s. Even though the predicted velocities are in agreement, the figure shows that the simulation overpredicts the width of the plume. This is mostly due to the limited number of particles in the experimental study. In general, the predicted results from the simulations are in agreement with the PIV measurements at a certain extent which gives confidence in the use of the CFD model.

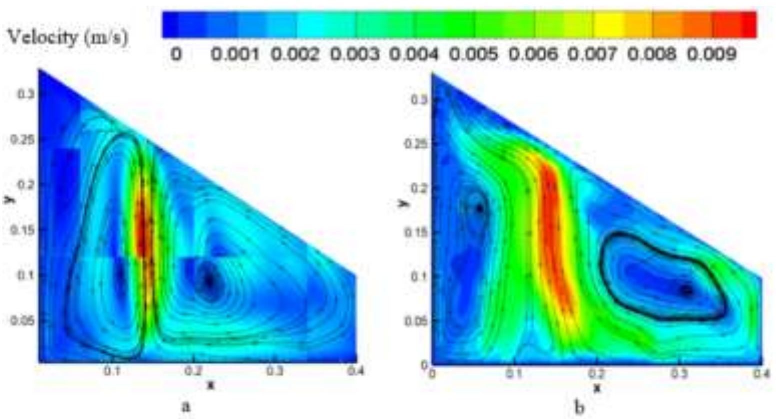

Figure 2: Cross-sectional velocity contour in the middle of the geometry (200 mm, Ra=1.38*108), a: experiment, b: CFD simulation.

The numerical model was used to investigate the effect of geometrical parameters on convective heat transfer from the absorber (hot surface). Figure 3 (a) shows the effect of varying cover angle on convective and evaporative heat transfer from the absorber for a fixed aspect ratio (Ar=2.55). The results show that increasing the cover angle leads to increase in both modes of heat transfer. Figure 4 (a) and (b) show isotherms of two geometries with different cover angles. It can be noticed that the temperature gradient on the right corner at 30˚ is higher than 10˚ cover angle geometry. In the same vein, Figure 4 (c) shows that at approximatively 0.9 m, the high temperature gradient of 30˚ geometry is translated to a higher peak in local heat transfer coefficient. Same trend was found when comparing the evaporative heat transfer since it is proportional to the convective heat transfer. Figure 3 (b) shows the effect of changing aspect ratio on the heat and mass transfer for a fixed cover angle (θ=30º). Results show that for low aspect ratios, heat transfer coefficients are high. However, with the increase in aspect ratio the heat transfer coefficient decreases due to the decreasing distance between the cover angle and absorber which directly lower the velocity of the air inside the cavity. Moreover, with the increase in aspect ratio, heat transfer coefficient increases after approximatively Ar=2.55. This is due to the formation of additional cell near the right corner Figure 5 (a) and (b). It can also be noticed that the heat transfer coefficient starts increasing and reaches its peak value at Ar=3 before Ar=2.55, as shown in Figure 5 (c). This is because of the formation of the cell at the right corner that splits the cold air from the cover into two cells, allowing higher amount of cold air to interact with the absorber (hot) surface, as illustrated in Figure 5 (a) and (b). Although the peak of local heat transfer coefficient at Ar=2.55 is higher than Ar=3 as demonstrated in Figure 5 (c), the trend is reversed for the overall heat transfer coefficient because of multicellular flow pattern of the geometry with high aspect ratio. Evaporative heat transfer coefficient was found to have the same behaviour since the latter is directly proportional the convective heat transfer coefficient.

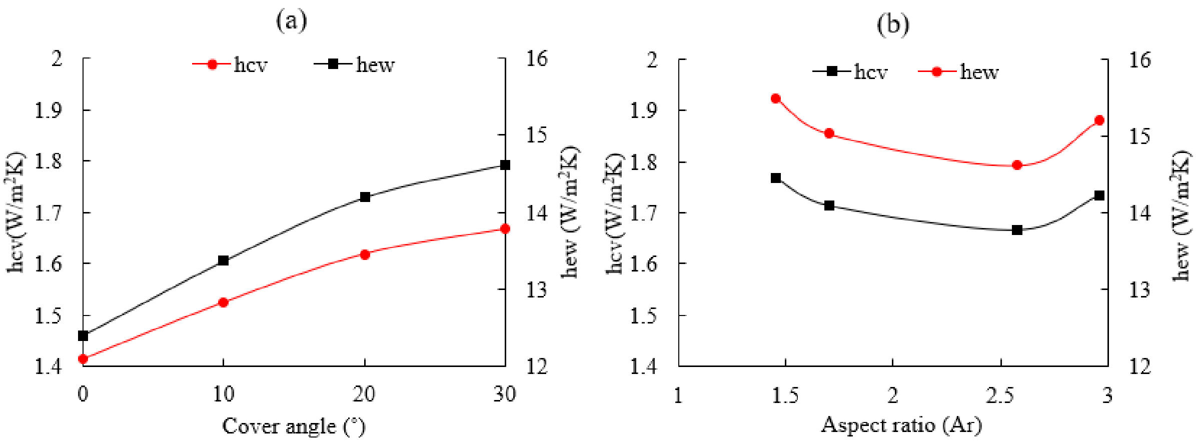

Figure 3: (a) effect of cover angle heat and mass transfer Ar=2.55, (b) effect of aspect ratio on heat and mass transfer for θ=30º.

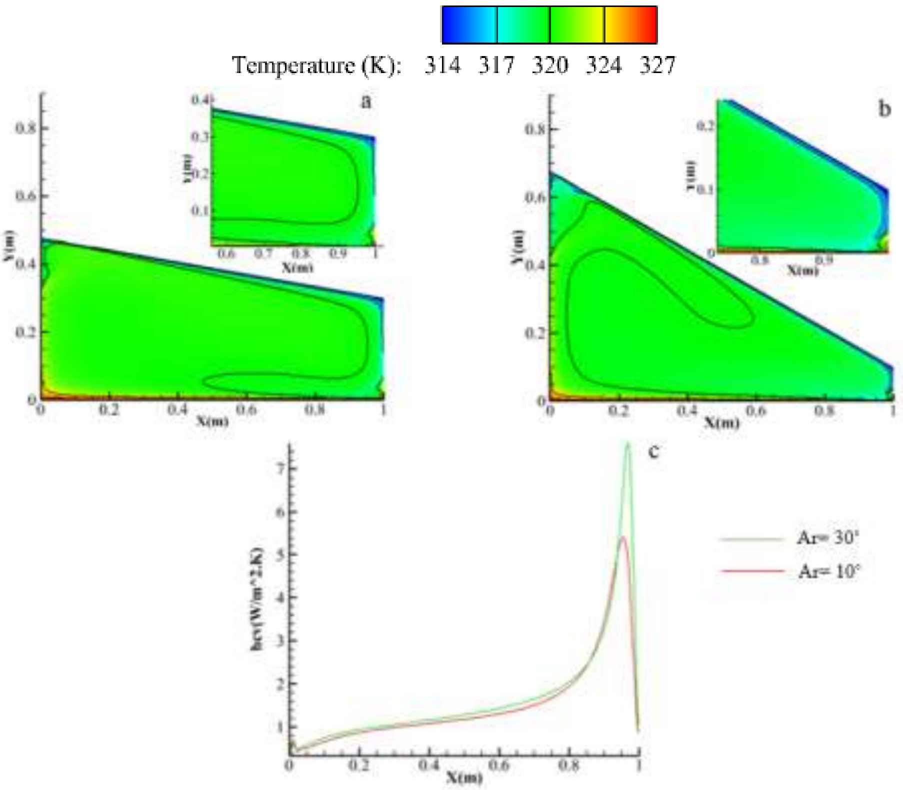

Figure 4: Effect of aspect ratio on heat transfer coefficient θ=30˚, a: Ar=2.55 temperature isotherms, b: Ar=3 temperature isotherms, c: local heat transfer coefficient

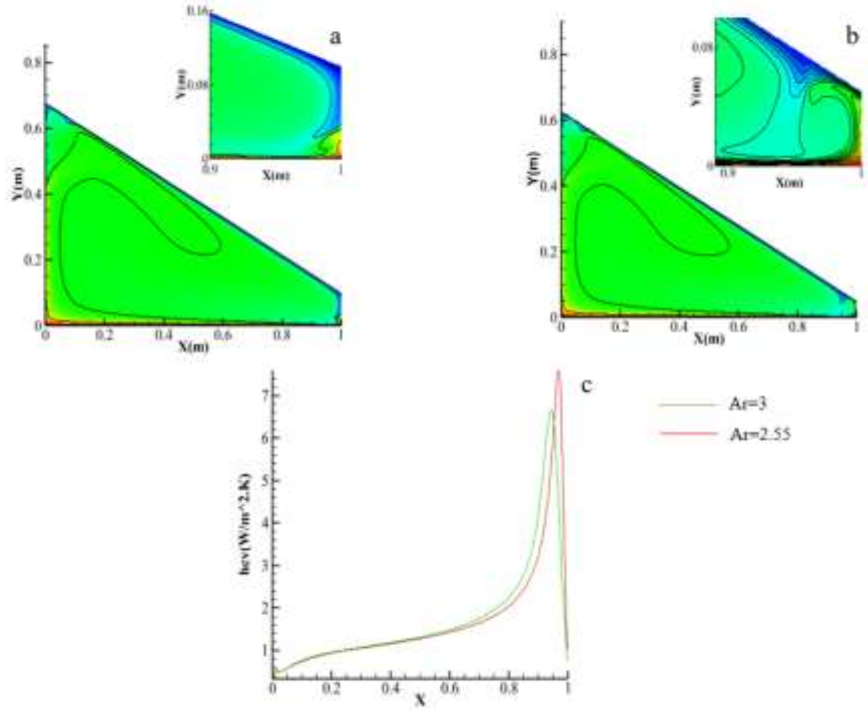

Figure 5: Effect of angle on heat transfer coefficient for Ar=2.55, a: θ=10˚ temperature isotherms, b: θ=30˚ temperature isotherms, c: local heat transfer coefficient

Conclusion

Previous results show the direct effect of the solar still geometry (cover angle and aspect ratio) on heat and mass transfer phenomenon. It was found that recirculation and flow patterns play an important role in maximizing the heat transfer exchange between the absorber and cover. Thus, for optimum solar still design, the author suggests a wider and more generalizable investigation on the effect of aspect ratio and cover angle on heat transfer in single slope enclosure.

Nomenclature

A_b: Absorber length.
Ar: aspect ratio (A_b/L).
Ra: Rayleigh number.
hcv: convective Heat transfer coefficient from absorber.
Hew: evaporative Heat transfer coefficient from absorber.
Th: Absorber temperature (hot).
Tc: Cover temperature (cold).
u: x-component of velocity.
v: y-component of velocity.
θ: Cover angle.

References

Altaç, Z. and Uğurlubilek, N, 2016, 'Assessment of turbulence models in natural convection from two- and three-dimensional rectangular enclosures', International Journal of Thermal Sciences, 107, p 237-246.

Rincón-Casado, A, Sánchez de la Flor, F.J, Chacón Vera, E. and Sánchez R, 2017, 'New natural convection heat transfer correlations in enclosures for building performance simulation', Engineering Applications of Computational Fluid Mechanics, 11(1), p 340-356.

Sharon, H., Reddy, K.S., Krithika, D. and Ligy, P, 2017, 'Experimental performance investigation of tilted solar still with basin and wick for distillate quality and enviro-economic aspects', Desalination, 410, p 30-54.

Zhang, Y., Sivakumar. M., Yang S. and Enever, K, 2018, 'Application of solar energy in water treatment processes: A review', Desalination, 428, p 116-145.