ASIA-PACIFIC SOLAR RESEARCH CONFERENCE

PEER REVIEWED
ASIA-PACIFIC
SOLAR RESEARCH CONFERENCE

The effect of a passive baffle on the performance of a single slope solar still

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

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

Abstract

Access to fresh water has become the major challenge facing humanity in attaining the goals of sustainable development. However, the exploitation of solar energy by conventional single slope solar stills serves as one the most viable solution for water purification. Several researchers have attempted to supress natural convection in such geometries using mounted baffles. However, only few investigated ways to increase natural convection by including judicious configuration of baffles. Therefore, this study examined the effect of mounting baffles on the cover surface of single slope solar still. Results of the study have shown that the inclusion of a single baffle mounted vertically downwards from the cover can alter the flow patterns in a single slope solar still. This in turn can increase the natural convection heat transfer which would lead to an increased yield in fresh water beyond that of a simple single slope solar still.

1. Introduction

Studies have been performed with a view to increasing their efficiency by varying solar still geometry. An alternative to previous research, that has not been fully explored, is the use of baffles to alter the fluid flow in a single slope solar still to increase its yield. In their study, [1] computationally investigated the use of baffles on the upper inclined surface of an attic-like trapezoidal enclosure, with an angle of 15˚. It was found that the heat transfer coefficient increased with the increase in distance of the baffle from the lower side wall. Similarly, [2] numerically examined the effect of using thin fins on natural convection inside porous triangular enclosures. Results of the simulations have shown that including fins changes the rotational direction vortices inside the cavity. Furthermore, it was concluded that increasing the length of the baffle blocked the flow inside the geometry thus decreasing the heat transfer coefficient. More recently, [3] numerically and experimentally investigated the effect of using baffles on the yield of single slope solar still with a cover angle of 12.5˚. The results of the study showed that the effect of the baffle varies with the ambient conditions to reach a maximum improvement of 8% compared to a conventional solar still.

Despite the work that has been undertaken, it is evident that baffles in single slope solar still may offer scope to increase their yield. However, the studies on the effect of these passive baffles in similar geometries appears to be contradictory. Therefore, this study aims create an understanding of this problem with a view to developing a firm conclusion on the use of baffles to increase distilled water production from a solar still.

2. Method

2.1 Numerical modelling

To understand the flow in a single slope solar still analysis was carried out using ANSYS's commercial computational fluid dynamics (CFD) software, FLUENT 19.2. In this study, a 2-D trapezoidal enclosure (representative of a typical single slope solar still, which is long and has uniform cross-section) was modelled, with the boundary conditions being a cold top and hot bottom wall. As the side walls in normal solar stills are heavily insulated, these sides were assumed to be adiabatic, as was the baffle/fin attached to the top wall (as shown in Figure 1).

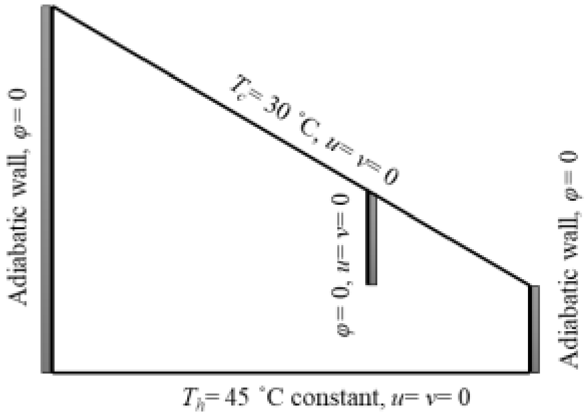


Figure 1: Boundary conditions

Since the nature of the flow inside this geometry has been shown to be turbulent [3,4], the realisable *k-ε* model was selected to capture this due to its high performance when dealing with natural convection inside enclosures.

The fluid modelled in the simulation was assumed to be an incompressible ideal gas and to have the physical properties of humid air. To resolve the discretized form of the continuity equations, second order upwind approach has been chosen because of its higher accuracy. Moreover, since the temperature gradient inside the geometry is low, the Boussinesq approximation was used to model the density variation. the pressure-velocity coupling was solved using a coupled algorithm, as it offers accurate solution for steady state flows. A mesh sensitivity study was performed, and the results showed that using an element size of less than 2 mm did not affect heat transfer coefficient, therefore, the problem became independent from the grid.

2.2 Experimental setup

To validate the selected numerical setup, it was decided to develop an experiment to compare in detail the flow features. As such, an acrylic enclosure was designed with back and front wall heights of 0.331 m and 0.1 m respectively, resulting in a cover angle of θ= 30˚. Moreover, a baffle made from acrylic with a length (d) of 0.17 m (of the total length) was fixed 0.13 m (~1/3 of the base) from the front wall (b= 0.13 m). To replicate the boundary conditions of convetional solar stills, the bottom surface (0.4 m×0.4 m) was made of a thin aluminium sheet adhered with an electrical heating element (water layer in SS). The bottom of the surface was than insulated to avoid heat losses from the electrical element. Moreover, the top surface was made from aluminium facing an electric fan in order to maintain the temperature difference with the absorber surface.

To ease the flow visualization during the validation experiment, the humid air (that would be present in a solar still) was replaced by silicon oil, but with the Rayleigh number matched to fall within the range of normal working conditions of a solar still. To visualise the flow, the silicon oil was seeded using neutrally buoyant round polyamide particles (50 μm diameter) and particle image velocimetry (PIV) was undertaken when the system reached steady state thermal conditions.

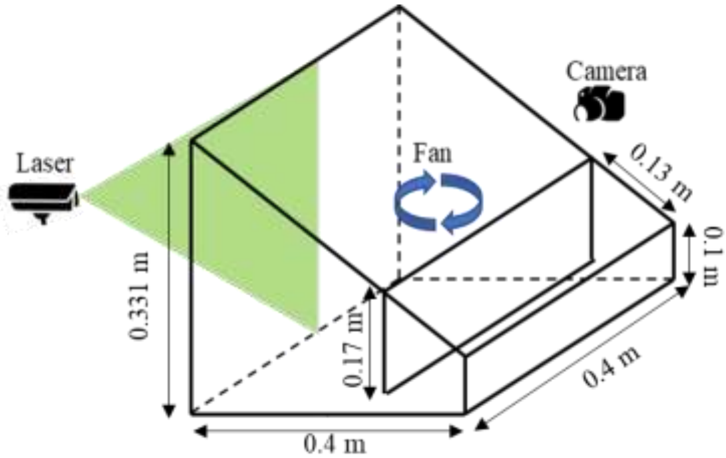
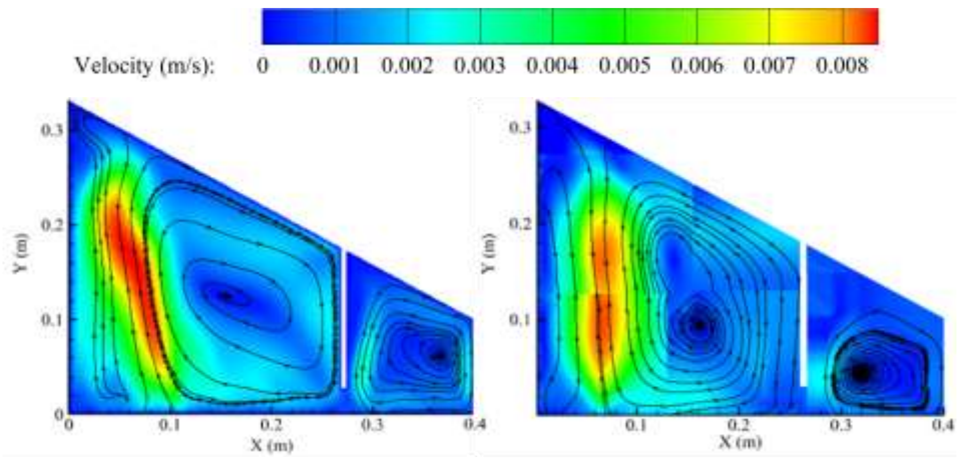


Figure 2: Experimental setup

3. Results and discussion

Before proceeding to the effect of the baffle length on the heat transfer coefficient from the bottom hot surface, it is essential to validate the selected CFD model. For that, steady state boundary conditions encountered in the experiment were applied in a CFD simulation with the same geometry and fluid properties. As can be seen from Figure 3, both CFD and experimental velocity contour show the existence of two cells, and this is because of the baffle forcing the flow to break. Moreover, at approximatively 0.1 m, a rising plume with similar velocity magnitude can be noticed in both contours near the left wall, the flow again drops at the baffle carrying cold fluid from the upper surface of the geometry. Similarly, the right cell is created but with lower velocity. in general, the results from the simulations agree very well with the PIV measurements, which gives confidence in the use of the CFD model.



a: CFD b: Experiment

Figure 3: Cross-sectional contour of velocity magnitude in the middle of the geometry (0.2 m,* AR= *2.6,* θ= *30º,* Ra= *1.38×10^8), a: CFD simulation, b: experiment.

Having seen that the baffle plays a role in the creation of multicellular flow, it is important explore the effect of this flow on the heat transfer coefficient for different baffle lengths (1/6)*h*, (3/6)*h* and (5/6)*h* compared to a conventional geometry (no baffle). Figure 4 shows the velocity contours for an enclosure with a 30˚ cover angle without the baffle and with different lengths of baffle. It is apparent that the inclusion of baffles results in the creation of multicellular flow. However, it is noticeable from Figure 4 (b, c and d) that a longer baffle leads to stronger partitioning of the flow. This is because of the opposite rotational direction of the two generated cells leading to a higher resistance at the baffle position.

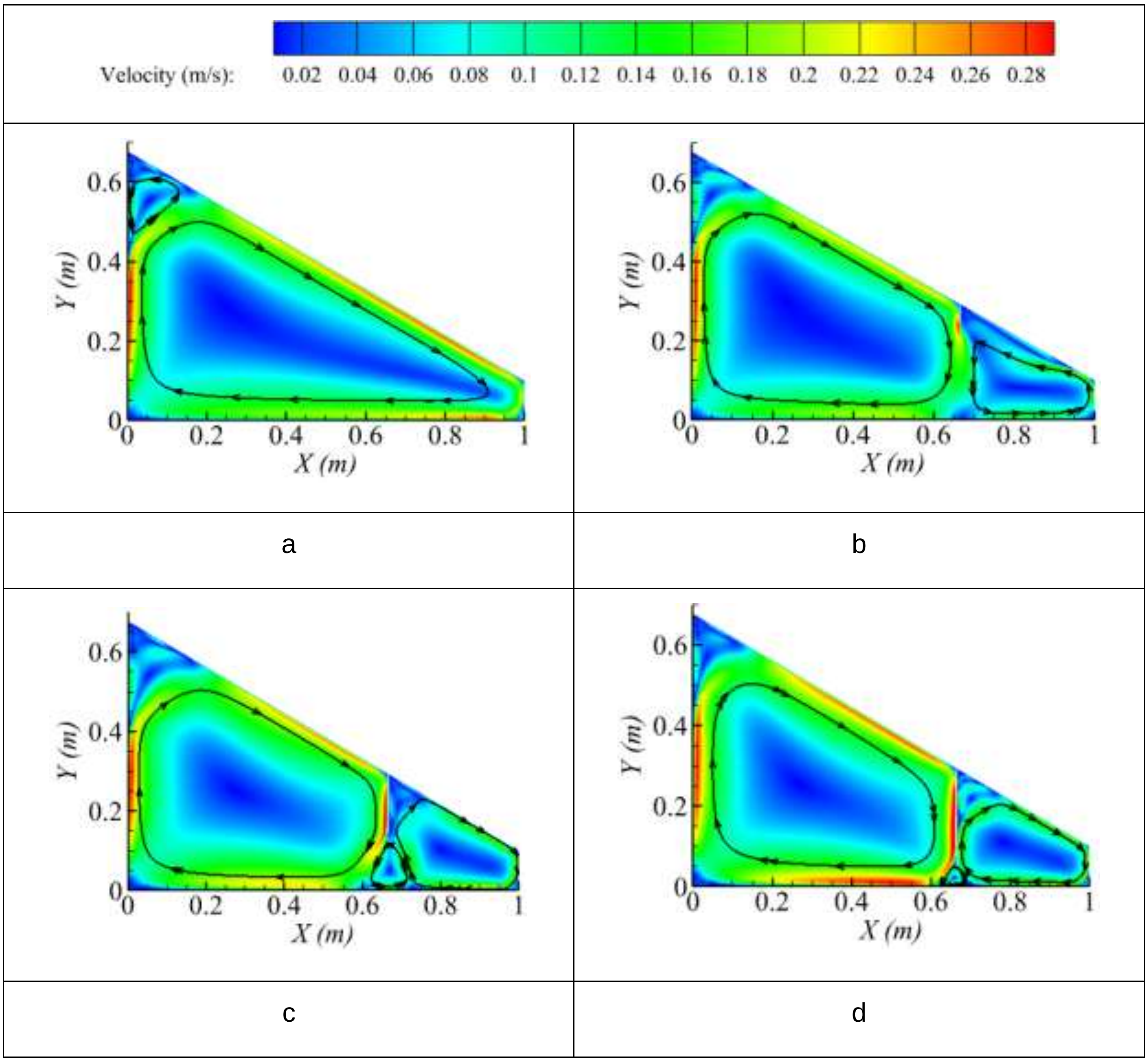


Figure 4: Contour of velocity magnitude for different considered cases a: no baffle, b: (1/6)*h *baffle, c: (3/6)*h *baffle and d: (5/6)*h *baffle

Looking at the effect of the baffle on the local heat transfer coefficient (**Error! Reference source not found.**), one can notice that the inclusion of baffles leads to an additional peak in the heat transfer coefficient at around x=0.66 m. This is due to the downward flow from the baffle carrying cold fluid from the cover surface to the bottom hot surface. However, the magnitude of the latter mentioned peak is proportional to the baffle length, this is because of the flow suppression caused by the collision of the two counter rotating cells. As expected, heat transfer coefficients for all cases except for (1/6)*h* baffle exhibit a sharp increase near the right side wall because of the similar downward flow from the cover.

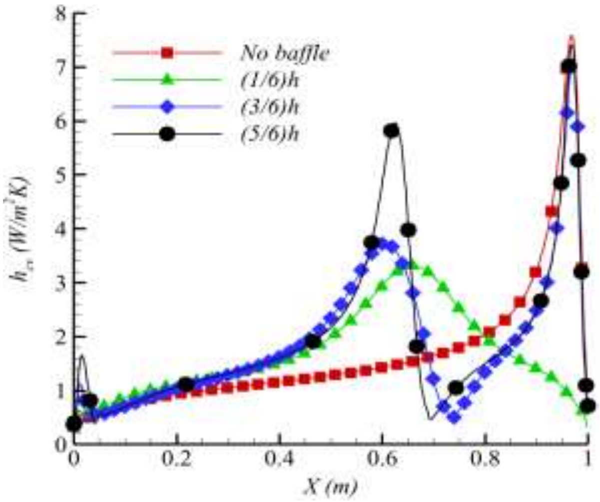


Figure 5: Local heat transfer coefficient for different baffle lengths

Conclusion

The performance of a single slope solar still is mainly driven by natural convection inside the enclosure, nevertheless due to their low yield, improving heat transfer inside these devices is a matter of the utmost importance. One of the ways to increase natural convection in these geometries is the inclusion of baffles. This work has demonstrated experimentally and numerically that the flow inside single slope solar still enclosure can be strongly influenced by the addition of baffles. Consequently, the heat transfer coefficient was found to increase with the baffle length leading to a stronger natural convection. Having said that, the findings of this study demonstrate the opportunity to use the baffles as a passive way to increase the yield of single slope solar still through the analogy between heat and mass transfer.

Reference

[1] Acharya, F.M.S, 2000, Natural convection in trapezoidal cavities with baffles mounted on the upper inclined surfaces, Numerical Heat Transfer, Part A: Applications, 37(6): p. 545-565.

[2] Rashidi, S., J. Abolfazli Esfahani, and Rahbar N, 2017, Partitioning of solar still for performance recovery: Experimental and numerical investigations with cost analysis, Solar Energy, 153: p. 41-50.

[3] Poulikakos, D. and Bejan A, 1983, Natural Convection Experiments in a Triangular Enclosure, Journal of Heat Transfer, 105(3): p. 652-655.

[4] Flack, R.D, 1980, The Experimental Measurement of Natural Convection Heat Transfer in Triangular Enclosures Heated or Cooled from Below, Journal of Heat Transfer, 102(4): p. 770-772.