ISBN 978-93-84422-22-6
Proceedings of 2015 International Conference on Chemical, Metallurgy and Environmental Engineering (ICMAEE-15)
Istanbul (Turkey), June 3-4, 2015 pp. 272-276

Modeling of Fluid Flow and Temperature Profiles in Solar Stills Using CFD

Muhammad Shakaib, and Masood Ahmed Khan

Department of Mechanical Engineering, NED University of Engineering and Technology, Karachi Pakistan

Abstract—Effectiveness of a solar still depends on fluid flow patterns that takes place due to free convection of moist air. The fluid flow behavior depends on solar still geometry / dimensions (width, depth and slope of condensing surface). In this paper CFD analysis is carried out and results are presented in terms of velocity and temperature profiles and distribution of shear stress and heat transfer coefficient. The work reveals major flow recirculation within the solar still which might be useful for increasing the yield of the device.

Keywords—CFD simulation, solar still, shear rates, Nusselt number

1. Introduction

Solar stills are utilized commonly for desalination when the requirement of pure water is low. Its main components are a bottom basin containing saline water and a top transparent condensing cover of glass or plastic. In this device, incident solar radiation transmits through the top cover and heats the saline water due to which temperature increases. This causes the water to evaporate and rise. The heated water vapor then condenses on the top cover which is relatively at a lower temperature. Finally the condensed water which is free of impurities such as salts and microbiological organisms is collected in the distillate tray for end use. The advantages of such a passive solar still is that it is simple in construction, cheap, no moving parts and needs only sun energy to operate. The factors that affect the performance of this device is the amount of solar irradiation, ambient temperature and humidity, area of water surface / basin, depth of water surface, orientation of the condensing surface and the overall dimensions / geometry. Various papers have been published to investigate the effect of these factors on solar still productivity. Tiwari and Tiwari [1] conducted experiments and found that solar still unit with 30° angle is suitable as it results in maximum heat transfer rates yield. The experiments of Tiwari and Tripathi [2] showed that the semi-cylindrical shape is better than spherical one for solar distillation units. It was also found that product water flows were higher in case of forced convection when compared to natural convection. Porta Gandara et al. [3] visualized the fluid flow experimentally. Vortex formation and destruction was noticed which increased the heat and mass transfer rates. Omri et al. [4] discussed the thermal profiles at different Grashof numbers and inclination of top cover and found that walls were not at constant temperature. The study showed that natural fluid motion within the solar still is determined by the top cover inclination. The same authors numerically investigated the natural convective flow in a triangular cavity of a solar still [5,6]. The work indicated that fluid flow behavior and temperature distribution are influenced by the geometry of the solar still and Rayleigh number. A study of Djebedjian and Rayan [7] revealed several flow features within the solar still cavity. The authors emphasized that theoretical / numerical modeling is important for optimization of solar still geometry. Setoodeh et al. [8] performed multiphase simulation and experiments heat and mass transfer. The water temperatures predicted by CFD were in agreement with the experimental results. In this paper, we perform a three dimensional simulation for natural convection flow in a solar still cavity and report the results in terms of shear stress and heat transfer coefficients.

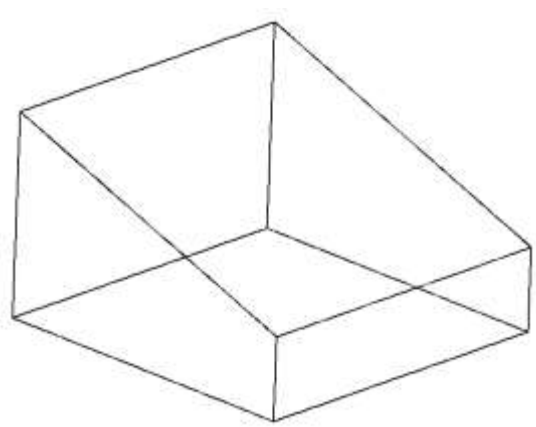

Fig. 1. Solar still geometry and computational domain.

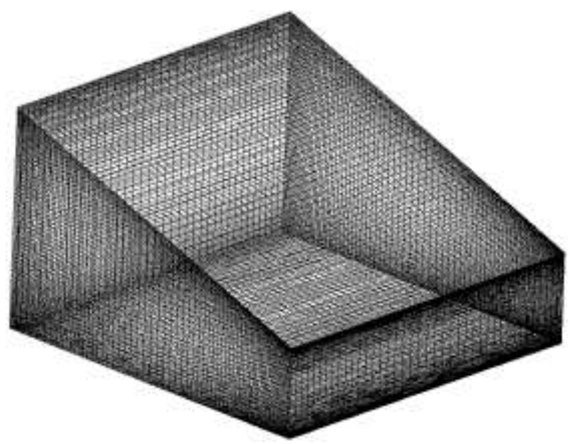

Fig. 2 Computational grid / mesh for CFD simulation.

2. Modeling Procedure

The geometry as shown in Fig. 1 has six faces which are considered as impermeable wall. The bottom and top surfaces are at fixed temperatures of 300 and 350 K respectively and the remaining are adiabatic.

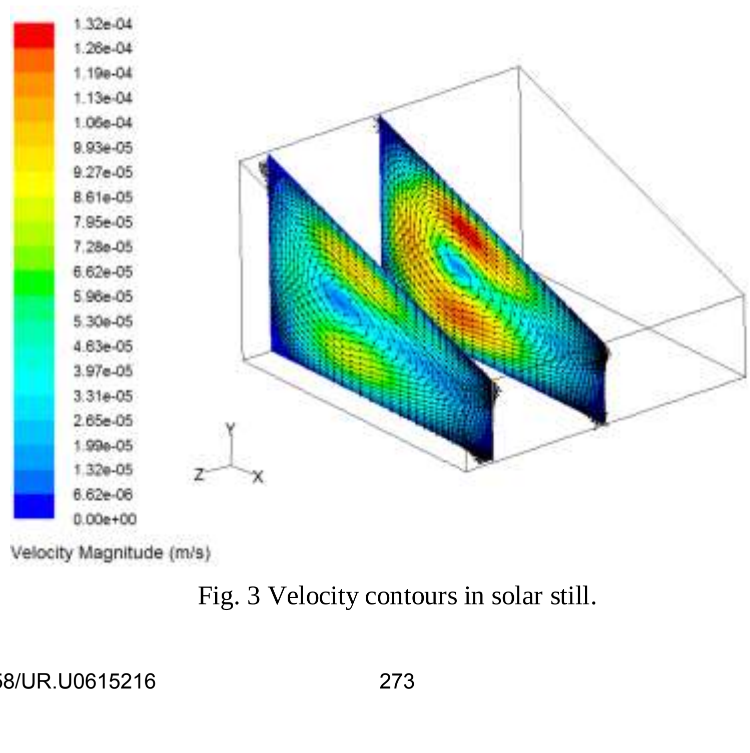

Fig. 3 Velocity contours in solar still.

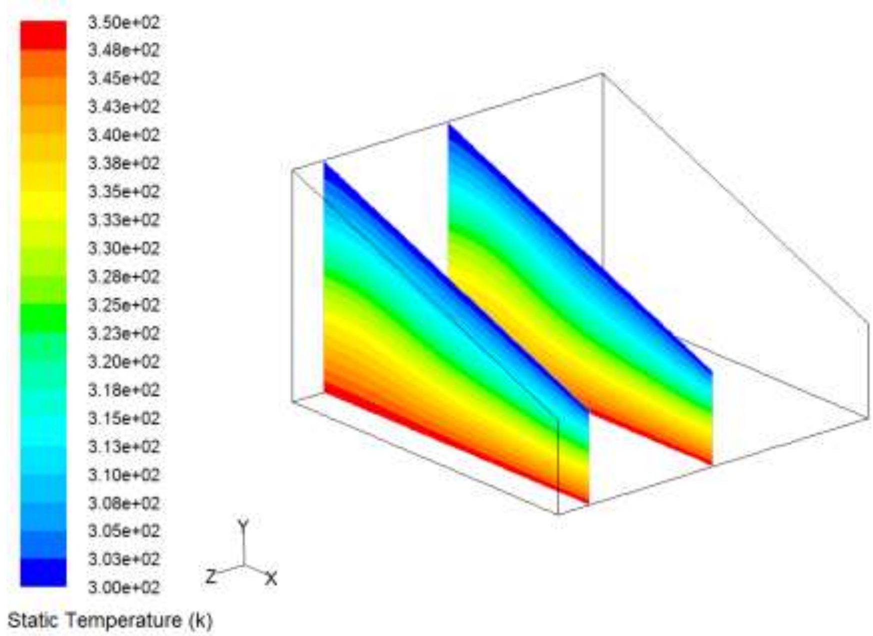

Fig. 4 Temperature profiles on two planes.

The vertical heights of the unit are 0.61 m and 0.25 m which results in a top surface slope of 20°. The width and depths of the unit are 1 m. The geometry is constructed and meshed in Gambit. The mesh contains fine cells as depicted in Fig. 2. It has about 100,000 cells which yields mesh-independent results. The mesh is exported in ANSYS FLUENT 12 which solves the equations of continuity, momentum and energy. The fluid is ideal air of which the thermal conductivity and viscosity varies with temperature. Rayleigh number for the simulation was 1000. Second order upwind scheme is used to discretize the convective terms and SIMPLE algorithm is used to couple the pressure and velocity fields.

3. Results and Discussion

The flow patterns in solar still are shown in Fig. 3 with the help of velocity contour and vectors on two vertical planes. The velocity vectors clearly show flow recirculation along with high velocity zones in the top and bottom portion with an intermediate low velocity region in the central portion. On the plane which is near the wall, the velocities are lower. The flow behavior can be considered suitable as the recirculating air will possibly drive / force the condensed water towards the distillate tray in the solar still device. The temperature contours in Fig. 4 shows variation of temperature in the vertical direction. Due to constant temperatures at top and bottom surfaces the temperature variation appears to be rapid near the vertical wall with less height. The results of shear stress in Fig. 5 profiles show that shear stress is higher somewhere in the center of the top and bottom surfaces. This is due to high velocities as were discussed in Fig. 3. The flow direction is mostly aligned with the two surfaces except near the corners (near the vertical surfaces) where the fluid takes a sharp turn. The flow seems two-dimensional as fluid movement is mostly in x-y plane. Heat transfer coefficient is found to be higher near the vertical wall with shorter height due to high temperature gradients as given in Fig. 6.

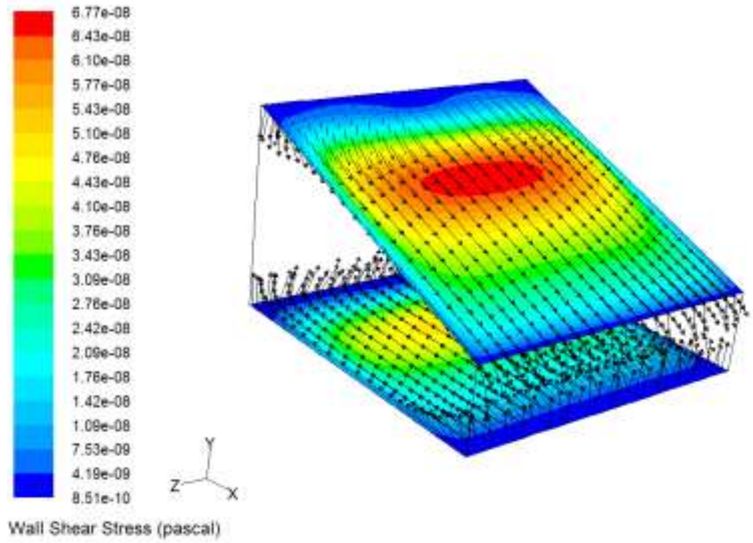

Fig. 5 Shear stress on top and bottom surfaces

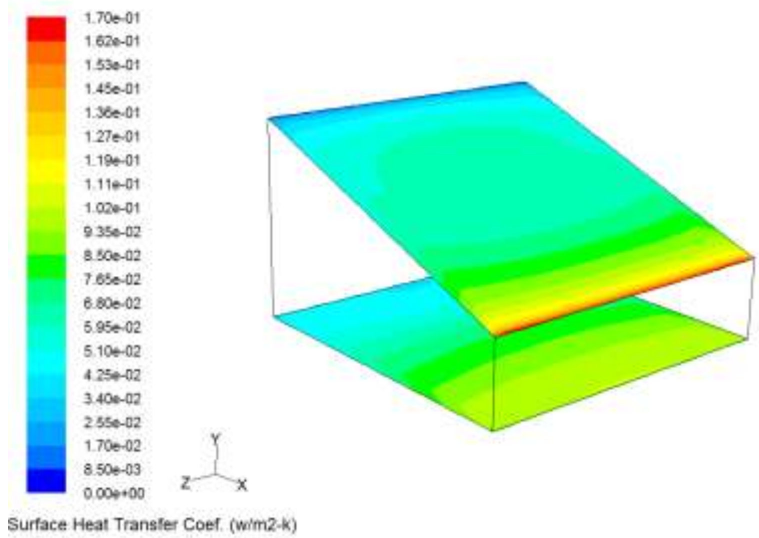

Fig. 6 Distribution of local heat transfer coefficients.

The results are compared with following Dunkle correlation:

$$\mathrm{Nu} = 0.075\mathrm{Ra}^{1/3}$$

Where

$$\mathrm{Nu} = \frac{h_{av} L_c}{k}$$

$$\mathrm{Ra} = \frac{g\beta (T_w - T_\infty) L_c^3}{\nu^2} \mathrm{Pr}$$

In the above equation Nu is Nusselt number, Ra is Rayleigh number, Pr is Prandtl number, and Lc is considered as average height of the unit.

Nusselt number is thus determined from CFD results and compared with the above experimental correlation. The difference of Nusselt numbers was about 30%. This shows satisfactory agreement between present numerical results and available experimental data.

4. Conclusions

Three dimensional CFD modeling is performed to study fluid flow caused due to natural convection in a solar still unit. A major flow recirculation region along with high velocity zones in the top as well as in the bottom portions is observed. Due to these high velocity zones shear stress increases on the top and bottom walls. The CFD results were evaluated against the experimental results and reasonable agreement was found.

5. Acknowledgments

The authors acknowledge the support provided by NED University of engineering and Technology, Karachi and Higher Education Commission of Pakistan.

6. References

[1] A.K. Tiwari, G.N. Tiwari, Effect of the condensing cover's slope on internal heat and mass transfer in distillation: an indoor simulation, Desalination 180 (2005) 73-88.
http://dx.doi.org/10.1016/j.desal.2004.12.029

[2] G.N. Tiwari, R. Tripathi, Study of heat and mass transfer in indoor conditions for distillation, Desalination 154 (2003) 161-169
http://dx.doi.org/10.1016/S0011-9164(03)80017-6

[3] M.A. Porta-Gandara, E. Rubio-Cerda, J.L. Ferna´ndez-Zayas, Visualization of natural convection inside shallow solar stills, Experiments in Fluids 25 (1998) 369-370.
http://dx.doi.org/10.1007/s003480050242

[4] A. Omri, M. Najjari, S. Nasrallah, Numerical analysis of natural buoyancy-induced regimes in isosceles triangular cavities, Num. Heat Transfer - A, 52 (2007) 661-678.

[5] A. Omri, Numerical investigation on optimiztion of a solar distiller dimensions, Desalination 206 (2007) 373-379.
http://dx.doi.org/10.1016/j.desal.2006.03.574

[6] A. Omri, J. Orfi, S. Nasrallah, Natural convection effects in solar stills, Desalination 183 (2005) 173-178.
http://dx.doi.org/10.1016/j.desal.2005.04.025

[7] B. Djebedjian, M. A. Rayan, Theoretical investigation on the performance prediction of solar still, Desalination 128 (2000) 139-145.
http://dx.doi.org/10.1016/S0011-9164(00)00029-1

[8] N. Setoodeh, R. Rahimi, A. Amer, Modeling and determination of heat transfer coefficient in a basin solar still using CFD, Desalination 268 (2011) 103-110.
http://dx.doi.org/10.1016/j.desal.2010.10.004