Simulation on Heat and Mass Transfer of an Evaporator as Evaporation Chamber In Natural Vacuum Solar Desalination System

Agistyad Ronowikarto1,2*, Ridho E T Siregar1,2 and H Ambarita1,2
1Mechanical Engineering, Faculty of Engineering, University of Sumatera Utara Medan, Indonesia
2Sustainable Energy Research Centre, Faculty of Engineering, University of Sumatera Utara Medan, Indonesia
*agistyad@gmail.com

Abstract. Natural Vacuum Solar Desalination technology was designed and built for one of solutions for water scarcity issues right now. An evaporator as the main part of this system should be analyzed to achieve a good result (fresh water productivity). The evaporator is used as an evaporation chamber to evaporate seawater which will be condensed to water storage. The commercial code CFD ANSYS Fluent 16.1 has been used to simulated and analyzed of heat and mass transfer inside the evaporator. There is no laminar model was used in this work with the temperature of heating coil was 70℃ and the operation pressure was under atmospherical pressure. In this work, the results of simulation and experimental will be compared to be a validation. The highest temperature 72℃ was in heating coil and the lowest temperature 38℃ was in a body of evaporator based on the simulation result. The temperature of heating coil was increasing during evaporation processes. From these comparisons, shows there’s a little discrepancy with a magnitude of error between 0.02% to 33%. It also shows the method has been used is right for analyzed and simulated about evaporation-condensation processes.

1. Introduction
Nowadays, there are millions of people all over the world who don’t have access to water, or if they have access, that water is unable to be used [1]. Water scarcity is the lack of sufficient available water resources to meet water needs with a region. Water scarcity involves 3 things, first is water shortage, second water deficit and the third is water stress. It affects every continent and around 2.8 billion people around the world at least one month out every year [2]. The term water crisis lables a situation where the available potable, unpolluted water within a region less than that region’s demand [3]. The effect of water scarcity is hazardous, because as we know, water is the necessity of every living being. According to World Health Organization (WHO), in 2015, 71% of the global population (5.2 billion people) used a safely managed drinking water service – that is, one located on premises, available when needed, and free from contamination while 89% of the global population (6.5 billion people) used at least a basic service is an improved drinking-water source within a round trip of 30 minutes to collect water. Then 844 million people lack even a basic drinking-water service, including 159 million people who are dependent of surface water. Globally, at least 2 billion people use a drinking water source contaminated
with faeces. WHO also has concluded that by 2025, half of the world’s population will be living in water-stressed area [4]. With these facts, solutions for water scarcity is quite important to be sought. Desalination is widely adopted from Middle East, Arab Countries, North America, Asia, Europe, Africa, Central America, South America and Australian to Desalination is widely adopted from Middle East, Arab Countries, North America, Asia, Europe, Africa, Central America, South America and Australia to fulfill the needs of clean water and water treatment needs. Desalination technologies is quietly simple. The technology just like boiling some water and then the vapor will be condensed and stream down to water storage. The main part in this natural vacuum solar desalination is evaporator because water will be boiled in there as an evaporation chamber. The effectiveness of evaporator is not just based on the temperature, but also based on evaporative cooling.

In Cengel and Bajhar 5, they mention about evaporative cooling, if evaporation rate is high and the evaporation heat requirement is higher than the amount of heat which can be supplied from the bottom of the water and beyond. It makes the water temperature in the surface will be decreased as the water evaporates. Computational fluid dynamics (CFD) method should be employed to investigate volume fraction, mass transfer rate, and heat transfer phenomenon inside the evaporator. CFD method has been often used to investigated about evaporation-condensation inside the evaporator for any kind of solar desalination. Then these results from simulation will be compared with an experimental results. These comparisons will be a validation. A two-phase three-dimensional CFD study has been performed to investigate volume fraction and temperature distribution of the evaporator [6]. In their work, they concluded, a two-phase three-dimensional model was conducted for water mixture (air and water vapor) system with the aim CFX 11 software package.

Then they concluded due to solar radiation, water vaporized and model was able to show condensed water droplets on the glass of basin solar still. Accumulated water amount on downcomer were compare with the water quantity produced in experimental set up. Mowla et al [7] calculated the rate of production of fresh water from sea water as a function of different meteorological parameters. Radhwan [8] studied the transient performance of a stepped solar still with built-in latent heat thermal energy storage. In this paper, CFD method for temperature distribution in the evaporator was developed. The result of measurement by HOBO Data Logger and Pace Scientific Vacuum Pressure Gauge will be used for some parameter in simulation. The commercial code ANSYS Fluent 16.1 has been used for this work.

2. Methodology

In this paper the commercial code ANSYS Fluent 16.1 were used with Langrangian and Langrangian-Eulerian approach is implemented to simulate a two-phase three-dimension flows in the evaporator. Turbulence model was used in this work. Temperature distribution was determined. While, Kharangate et al [9] has investigated, volume of fluid method (VOF) is the most popular for simulate two-phase eulerian schemes. From these results were compared with the experimental data which has been done. A computational model was made with dimensions 80 cm × 20 cm. A computational model and the boundary are shown in Figure. 1. Geometry was created with SolidWorks 2016. Heating coil as interface because it directs contact with water and there just one pressure-outlet on the top of evaporator. And grid generation was executed with the Workbench 16.1. The minimum and maximum cell volumes in the domain are 1.9432 × 10⁻⁴ and 3.8865 × 10⁻². The meshing is shown in Figure 2. The walls of the computational domain are modelled as no-slip walls with zero thickness. 9.8 m s⁻² of gravity acts in vertical axis. The adiabatic thermal boundary condition is used for these surfaces. Zero static gauge pressure is applied at the outlet.
Figure 1. Computational Model and The Boundary

Figure 2. Meshing
2.1. Numerical Method

The model equations based on the continuity, momentum, energy and mass transfer conservation principles at transient conditions. In this work, the simulation has been done for more than 1 hour. According to Kharangate et al [9], modelling two-phase flow and heat transfer requires accurate prediction of the behavior of each phase and interactions along the interface between phase. Several numerical methods are available for this purpose. Most popular CFD methods involve solving conservation equations using macroscopic depiction of the fluids, where fluid matter is described as consisting of a sufficiently large number of molecules that continuum hypothesis for properties is valid [9]. Kharagate et al [9] have investigated, the mass, momentum, and energy equations are as follows:

1. The mass equations:

\[
\frac{\partial}{\partial t} (\rho \vec{u}) + \nabla \cdot (\rho \vec{u}) = 0
\]  
\[
\text{Where: } \quad \vec{u} = \sum_{k=1}^{n} \alpha_k \rho_k \vec{u}_k
\]
\[
\rho_m = \sum_{k=1}^{n} \alpha_k \rho_k
\]

2. The momentum equation:

\[
\frac{\partial}{\partial t} (\rho \vec{u}) + \nabla \cdot (\rho \vec{u} \vec{u}) = -\nabla p + \nabla \cdot [\mu_m (\nabla \vec{u} + \nabla \vec{u}^T)] + \rho \vec{g} + \vec{F}_s
\]

\[
\text{Where: } \quad \mu_m = \sum_{k=1}^{n} \alpha_k \mu_k
\]

3. The energy equations:

\[
\frac{\partial}{\partial t} (\rho E) + \nabla \cdot (\rho \vec{u} (E + \rho \vec{u})) = \nabla \cdot (K_{eff} \nabla T) + Q
\]

\[
\text{Where: } \quad E = h - \frac{\rho}{\rho_k} + \frac{V_k}{2}
\]

While most popular methods are based on this macroscopic depiction, there is now increasing interest in computational methods at the mesoscale, where fluid matter is considered a collection of atoms, which is much smaller than macroscale but larger than single atom. In Ambarita10, the heat source could be from an electric heater which implanted in the element or can also from the occurrence of heat due to friction.

2.2. Mass Transfer Models

Lee [11] divided a evaporation-condensation processes model as two conditions, which gas phase and liquid phase, and also with a different equations. Kharangate et al. [9] also investigated the key premise of this model is that phase change is driven primarily by deviation of interfacial temperature from $T_{sat}$ and phase change rate is proportional to this deviation. The model assumes for both phase is transferred at constant pressure.

For condensation:

\[
S_g = -S_f = r_f \alpha_g \rho_g \frac{(T - T_{sat})}{T_{sat}}
\]
Where:

\[ T < T_{\text{sat}} \]

For evaporation:

\[ S_g = -S_f = r_i \alpha_f \rho_f \left( \frac{T - T_{\text{sat}}}{T_{\text{sat}}} \right) \]  \hspace{1cm} (9)

Where

\[ T > T_{\text{sat}} \]

Where \( r_i \) is an empirical coefficient called mass transfer intensity factor. While the Lee model consistently aims to decrease the deviation from \( T_{\text{sat}} \), there is great variability in the choice of \( r_i \) value [9]. Chen et al [12] suggested a substitute version to the Lee model, given by:

For condensation:

\[ S_g = -S_f = r_i m \alpha_g \rho_g \left( T - T_{\text{sat}} \right), T < T_{\text{sat}} \]  \hspace{1cm} (10)

For evaporation:

\[ S_g = -S_f = r_i m \alpha_f \rho_f \left( T - T_{\text{sat}} \right), T > T_{\text{sat}} \]  \hspace{1cm} (11)

\( T_{\text{sat}} \) is eliminated from numerators of the source terms. From all the equations above will be discretized as a linear equations using Second Order Upwind. In this paper, Second Order Upwind is iteratively used to discretized the equations above.

3. Results and Discussions

3.1. Temperature Distribution

This work mainly focuses on the heat and mass transfer inside the evaporator. The diameter of the evaporator is 80 cm with the height is 20 cm. The temperature of the heating coil is 70℃ in the beginning of evaporation-condensation processes. The second order upwind scheme is applied, and the PRESTO scheme is used for the pressure discretization. Pressure, velocity coupling, density, and momentum are all set to 0.3. The temperature of liquid and the inlet temperature are same with the room temperature 27℃. The constant pressure to atmospheric pressure at the outlet. The temperature distribution during evaporation-condensation from the simulation are shown in Figure 3(a)(b). As we see in Figure 3, the temperature inside the top evaporator is starting from 38℃ to 67℃ while the highest temperature is on the heating coil is 62℃ because it has a high energy. The interface temperature near the wall becomes higher than that near the interface center 13. Wang et al 13 have investigated there are two reasons contributing to this phenomenon.
First, due to the hydrophobicity of the wall, the horizontal distance between the heating coil and interface near the wall is smaller than that near the center. In this case, the heat transfer resistance is smaller, which allows more adsorbed heat to be transferred from the heating coil to the interface [13]. As a result, the interface temperature near the top of evaporators is higher. On the other hand, because of such temperature distribution at the interface, the Marangoni convection is caused, which results in the liquid water to flow from the high-temperature region to the low-temperature region [13].

Clearly, due to the Marangoni convection, the liquid water flowed from the interface is heated up. Under such a circumstance, the upward movement of the heated-up water along with continuous heating results in an expanded region with the highest temperature. In Cengel and Bajhar [5] and Wang et al [13], the evaporation rate is lowered due to the increased vapor pressure so that less heat is taken away from the liquid column, leading to the more significant temperature rise and vice versa (evaporative cooling phenomenon). With the heating coil going on, the magnitude of the Marangoni convection becomes strong and then weak again due to the competition between the photothermal effect and increased vapor pressure. The present numerical method is validated by comparing the result with the experimental results. Figure 3(c) shows the results of simulation, the experimental data, and the measurement result. The result of comparison shows a good agreement. Thus, the present method can be used to explore the problem.

4. Conclusion
In this work, the heat transfer inside the evaporator has been numerically studied using commercial code CFD ANSYS Fluent. The numerical results shows, the spontaneous evaporation is dominant in this period. According to Wang et al [13], the photothermal effect induced evaporation becomes dominant on evaporation-condensation processes. The formed Marangoni convection in turn causes higher temperature near the wall (top of the evaporator) and relatively low temperature near the center in the heating coil and the temperature around body of the evaporator also relatively low because of the evaporative cooling phenomenon. The result from simulation has been compared with the experimental results. From the comparison, as shown in Figure 3, there’s a low discrepancy. The lowest error of magnitude is about 0.02% while the highest error of magnitude is about 33% as shown in a graph (Figure 3(c)). It shows the method as mention above can be used to analyze evaporation-condensation processes.
Acknowledgments

The authors owe deep gratitude to Mechanical Engineering Department, University of North Sumatera, Medan, Indonesia who fully supported this recent research. To Dr. Eng. Himsar Ambarita as our project guide and also our thesis supervisor for Solar Desalination team. And the support from our colleagues in the team, Faisal, Gerry, Fattih, Dodi and Mr. Eko.

References

[1] https://www.conserve-energy-future.com/causes-effectssolutions-of-water-scarcity.php (Accessed October 1, 2017)
[2] "Water Scarcity. International Decade for Action 'Water for Life' 2005–2015". un.org. Retrieved 20 October 2013.
[3] https://www.sciencedaily.com/terms/water_scarcity.htm (Accessed October 1, 2017)
[4] http://www.who.int/mediacentre/factsheets/fs391/en/ (Accessed October 1, 2017)
[5] Cengel, A.Y and Gajhar, J. A 2014 Heat and Mass Transfer Fundamentals & Applications Fifth Edition, Singapore: Mc Graw-Hill, Book Company.
[6] Setoodeh N, Rahimi R, and Ameri A, 2010, Modelling and determination of heat transfer coefficient in a basin solar still using CFD, Journal of Desalination 268 (2011), 103-110
[7] D. Mowla, and G. Karimi, 1995, Mathematical modelling of solar stills in Iran, Solar Energy 55 (5) (1995), 389-293.
[8] Radhwan, A.M, 2004, Transient performance of stepped solar still with built-latent heat thermal energy storage, Desalination 171 (2004), 61-67
[9] Kharangate, R.C and Mudawar, I., 2016, Review of computational studies on boiling and condensation, Int. Journal of Heat and Mass Transfer 108 (2017), 1164-1196.
[10] Ambarita, Himsar, 2017, Heat and Mass Transfer Analytic and Numerical Solutions. In: Natural Convection, Intelegensia Media, Malang.
[11] Lee, W.H., 1980, A pressure iteration scheme for two-phase flow modelling, in: T.N. Veziroglu (Ed.), Multiphase Transport Fundamentals, Reactor Safety, Applications, vol 1, Hemisphere Publishing, Washington DC
[12] Chen, S., Yang, Y., Duan, Y., and Chen,D.W., 2013, Simulation of condensation flow in a rectangular microchannel, Chemical Engineering and Processing 76 (2014), 60-69.
[13] Wang, Z., Li, S., Chen, R., Zhu, X. and Liao, Q., 2017, Simulation on the dynamic flow and heat and mass transfer of a liquid column induced by the IR laser photothermal effect actuated evaporation in a microchannel, Int. Journal of Heat and Mass Transfer 113 (2017), 975-983.