Numerical Analysis of the Thermal Performance in Traditional and Developed Shapes of Thermosyphon

M S Qasim, B Freegah and G S Ali
Mustansiryah University, College of Engineering, Mech. Eng. Dept., IRAQ, Baghdad
Email: qasim602006@yahoo.com

Abstract. In recent years, the research in renewable energy is remarkably increased to reduce the reliance on fossil fuels. Solar energy took the lead among the other sources of renewable energy due to its availability everywhere and the feasibility of its use for various circumstances. Thermosyphon is one of the most successful systems for solar energy conversion. In the present study, a CFD code is employed to analyze the natural convection phenomenon which is inherent in the performance of a thermosyphon system. A steal pipe of 25 mm internal diameter and 1250 mm length is considered as the condenser of the thermosyphon. Water jacket is used to cool the condenser with an internal diameter that is five times that of the condenser. Two types of pipes are considered in the present study, namely; straight and helical pipes. The case of helical pipe assumes five different numbers to turns, which are; 10, 20, 30, 40 and 45 turns. A solar heat flux of 500 W/m² and above is used in the analysis. The results obtained from the numerical analysis showed that the helical pipe is more efficient than the straight pipe in heating the water inside the condenser jacket for the same operating conditions. The ratios of increase in jacket water temperature of the five cases of the helical pipe to that of the straight pipe were 23.8%, 36.5%, 42.1%, 45.3% and 46.2%, respectively.

1. Introduction
There are several parameters having effect on increase amount of the heat transfer and hence enhancement the thermal performance of thermosyphon closed loop. The surface area of the riser pipe is one of the most important these parameters. The increase surface area of the riser pipe can be achieved by several ways (e.g. fins adding or change of shape of the riser pipe. Helical pipes are efficient heat transfer parts, because they provide large surface area in a compact volume. However, their use in solar energy systems is still limited. El-Din [1] conducted an experimental study to evaluate heat transfer rate to single-phase liquid. In this study; toroidal thermosyphon type was investigated, and the effect of several parameters were examined. The parameters included; the ratio of the heated part to the cooled part of the thermosyphon, length to diameter ratio, torus diameter, torus to tube diameter ratio and the tube inclination angle. The results have been shown that increase both ratio length to diameter and heated length to heated length are lead to decrease in heat transfer rate. However, increasing torus-tube diameter ratio raises the heat transfer rate. The tilt angle range between 30° to 45° gave maximum heat transfer. Comparative investigation between new design solar collector which known accelerated absorber and the traditional absorber collector has been carried out by Amori and Jabouri [2]. All tests have been conducted under same identical conditions for both systems at an angle of inclination facing the south with magnitude of 33°. The results
showed a significant increase of thermal performance for new system approximately 60% as compared to the traditional system. The temperatures of the storage tank of the new and conventional types were 50°C and 37°C respectively. The effect of number of the risers has been studied on the performance of the evaporator side of the thermosyphon by Subramanian et al. [3]. The performance of two patterns was also investigated, namely; the straight and zigzag patterns. In both patterns, the risers and headers were made of copper with the diameter of header is larger than that of risers. Results showed that the maximum collector efficiency increased to 59.09% when the number of risers were increased from 9 to 12. The efficiency was further improved to 62.9% when zigzag configuration was employed. The maximum efficiency in all experiments was achieved around 13:00 p.m. The collector efficiency along the whole experiments time was higher in zigzag arrangement. Freegah et al. [4] considered several influencing parameters of thermosyphon systems. The influencing parameters that have been investigated in this study were solar heat flux system, inclination angle, number of connecting pipes and the ratio of length to diameter of connecting pipes. Results showed that heat flux and length to diameter ratio dominate the performance of the thermosyphon. The inclination angle caused minor effect on the performance while, the working fluid temperature increases with increase the connection pipes. An experimental and numerical study was conducted by Freegah and Al-Tabbakh [5]. Other parameters of thermosyphon solar water heater were studied, namely; number of risers and riser length to diameter ratio. Results showed that an increase both number of risers and ratio of length to diameter improve the system thermal performance. However, ratio of length to diameter has more pronounced effect than number of risers.

The current work is a comparative study of thermosyphon system incorporating two types of risers, namely; straight and helical risers. A computational Fluid Dynamics (CFD) technique has been utilized to analyse a closed loop thermosyphon solar water heater employing for two types of the riser pipe mentioned above. In this study, the effect of turn’s number of the helical riser pipe on the water temperature within the condenser has been investigated.

2. Specifications of New Model Geometry

The collector of the conventional closed-loop thermosyphon solar water heater consists of several inclined risers. The risers are connected to an up riser and down comer at top and bottom sides respectively as shown in figure 1.

![Figure 1. The geometry of the thermosyphon.](image)

The dimensions of the model used in the current study are: 13.6 mm and 0.7 mm for the riser pipe internal diameter and wall thickness respectively, whereas the condenser diameter (5x) that of the riser
pipe. These estimations are according to the copper pipe standard dimensions. The diameter and thickness of both the upriser and downcomer are kept similar to the riser pipe all over the present work. In the new model, different number of turns (namely 10, 20, 30, 40 and 45), have been used for investigations, while the heat flux applied to the riser pipes is kept to 500W/m² to mimic the solar rays effect on the pipes. The latitude of Huddersfield city – that is 53° - is used for the inclination in the present model. Figure 2 shows the numerical model built for comparing the thermosyphon performance of both the helical and straight pipes.

![Figure 2. The geometry of straight and helical model of thermosyphon: (a) straight model; (b) helical model.](image)

3. Numerical Modelling
The performance of the closed-loop thermosyphon is investigated by calculating the temperature distribution of water within the condenser. As stated in the introduction part, and for acquiring a best understanding of the working fluid flow structure within a thermosyphon, Computational Fluid Dynamics (CFD) based technique have been implemented in the present work. Using these techniques, a virtual domain of the working fluid has been developed for investigating the natural convection phenomena within the thermosyphon loop. Such techniques using for predicting the velocity and temperature distributions of the working fluid within the thermosyphon.

4. CFD simulation
Flow simulation of the flow within the thermos-syphon is implemented using a commercial CFD package. The thermosyphon is assumed to work on a no-load condition; hence this is the only modelled part of the thermosyphon where the working fluid is present. Boussinesq approximation is utilized for accurate modelling of the generated buoyant force Dehdakhel et al. [6]. In this approximation, the density differences are assumed as negligibly small, with the exception of the terms multiplied by the acceleration due to gravity. The core idea the Boussinesq approximation is that inertia gradient is negligible, whereas the gravitational force is adequately strong for making the difference in the specific weight.

In the present study, hybrid meshing has been used as type of grids [7, 8]. Furthermore, the semi implicit method for pressure linked equation (SIMPLE) algorithm has been utilized as the pressure-
velocity coupling. In addition, time step size that has been used of 24sec. In order to achieve a high accuracy results, the mesh independence has been conducted to find suitable elements mesh numbers as shown in table 1. It can be seen clearly from this table that does no significant difference between the results which has been obtained from simulations. So, for reducing the computational time, a grid with lower number of elements has been chosen.

Table 1. The mesh independency.

| The applied model   | Number of grid elements | Base temperature (°C) | Temperature difference (%) |
|---------------------|-------------------------|-----------------------|---------------------------|
| Straight model      | 0.72×10^6              | 80.871                | 0.106                     |
|                     | 1.5×10^6               | 80.957                | 0.16                      |
|                     | 2.95×10^6              | 81.093                |                           |
| Helical model       | 2.8×10^6               | 78.85                 | 0.16                      |
|                     | 5.5×10^6               | 78.98                 | 0.14                      |
|                     | 9.4×10^6               | 79.09                 |                           |

5. Governing Equations of Fluid Flow
For simulating the transient flow in the thermosyphon in an hour interval of time, numerical solution of the 3D Navier-Stokes equations, continuity equation, and energy equation have been iteratively carried out.

Law of Conservation of Mass
It can be represented for mass conservation of fluid flow as below Munson et al. [9]

\[
\text{Rate of change of mass in fluid element} = \text{Net rate of flow of mass into the fluid element} \tag{1}
\]

Due to the density of fluid is remain constant, so the mass conservation equation can be written as:

\[
\text{Div} u = 0 \tag{2}
\]

Equation (2) can also be expanded fully as shown in equation (3).

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{3}
\]

Law of Conservation of Momentum
Conservation law of momentum for three directions (x, y and z) can be presented as follows Hoffmann and Chiang [10].

\[
\rho g_x + \frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} = \rho \left( \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) \tag{4}
\]

\[
\rho g_y + \frac{\partial \sigma_{yy}}{\partial y} + \frac{\partial \tau_{yx}}{\partial x} + \frac{\partial \tau_{zy}}{\partial z} = \rho \left( \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right) \tag{5}
\]

\[
\rho g_z + \frac{\partial \sigma_{zz}}{\partial z} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zx}}{\partial x} = \rho \left( \frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right) \tag{6}
\]

Law of Conservation of Energy
Conservation law of energy can be represented mathematically as follows Blazek [11].
\[
\frac{\delta(E_T)}{\delta t} + \frac{\delta(uE_T)}{\delta x} + \frac{\delta(vE_T)}{\delta y} + \frac{\delta(wE_T)}{\delta z} = - \frac{\delta(uP)}{\delta x} - \frac{\delta(vP)}{\delta y} - \frac{\delta(wP)}{\delta z} - \frac{1}{RePr} \left[ \frac{\delta q_x}{\delta x} + \frac{\delta q_y}{\delta y} + \frac{\delta q_z}{\delta z} \right] + \delta \left[ \Delta u_T \frac{\tau}{\Delta x} \tau + \Delta v_T \frac{\tau}{\Delta y} \tau + \Delta w_T \frac{\tau}{\Delta z} \tau \right] + \delta \left[ \Delta u_T \frac{\tau}{\Delta x} \tau + \Delta v_T \frac{\tau}{\Delta y} \tau + \Delta w_T \frac{\tau}{\Delta z} \tau \right]
\]

(7)

where \(E_T\) depict total energy.

6. Results and Discussion

The numerical analysis of the thermosiphon under this study has been conducted for various models of the collector which are helical and straight pipe. This section describes the effects of a new model which known is helical pipe on the thermosiphon thermal performance and comparative with conventional pipe. The results below are corresponding for one an hour of continuous operation of the thermosyphon. The temperature distribution of the working fluid within the condenser is depicted in figure 3. The results have been shown that the hot working fluid occupies the upper section of the condenser while the cold working fluid settles on the bottom of the condenser. Furthermore, the average temperature of water for helical pipe of 45 turns is higher as compared to the straight pipe. This indicated that raise in the collector surface area increases of the average water temperature within the condenser.

Figure 3. Temperature variations at cross-section of the condenser after one an hour of operation under heat flux of 500W/m²: (a) Straight pipe; (b) Helical pipe with 10 turns; (c) Helical pipe with 20 turns; (d) Helical pipe with 30 turns; (e) Helical pipe with 40 turns; (f) Helical pipe with 45 turns.
The temperature variation of water within the condenser for both models helical and straight pipes illustrate in figure 4. According figure 4, can be notice that the temperature of water within the condenser at case of helical pipe is higher than the water temperature at case of straight pipe. After one hour of operation, the different in the condenser temperature between both models of thermosiphon is 13K.

![Figure 4. Temperature variation of the working fluid inside the condenser after an hour operation time for straight and helical pipe.](image)

The variation the working fluid temperature at the cross section of the condenser for helical pipes can be shown in figure 5. The results have been shown that the temperature within the condenser for number of turns of 45 is higher as compared to number of turns of 10. After one hour of operation, the working fluid's temperatures within the condenser are 349.7, 348.35, 344.18, 337.94 and 327.14K to number of turns 45, 40, 30, 20 and 10 respectively. Thus, rise in the number of turns increases the water temperature within the condenser.

![Figure 5. Temperature variation of the working fluid inside the condenser after an hour operation time for helical pipe.](image)
Figure 6 depicts the temperature percentage of the working fluid within the condenser part of a closed loop thermos-syphon at variation heat flux for various number of turns. It can be seen clearly that there is a notice increase in the percentage for increase the working fluid temperature when increase turns number from zero to ten turns. Thereafter, the increase percentage continues with increase the number of turns, but to a lesser extent. The value percentage increase is 25%, 11%, 6.5%, 3.3% and 1.2% for increase number of turns from zero to (10, 20, 30, 40, and 45) respectively.

![Graph showing temperature percentage increase with number of turns.](image)

**Figure 6.** Temperature percentage of the working fluid inside the condenser after an hour operation time for various number of turns.

7. Conclusions

In the present study, conducted a comparison of the thermal performance for two solar water heaters, one of the collectors is a helical pipe and the other collector is a conventional. Both collectors are the primary part of an indirect thermosiphon circulation solar hot water system. Results show that increase number of turns rises the working fluid temperature within the condenser as compared to straight pipe (without turns) although increase the volume of the working fluid within the thermosiphon loop. Hence, considerable enhancement of thermal performance of absorbed heat was obtained for the new model in comparison with the conventional model. However, the increase percentage of temperature for helical pipe model is little when increase number of turns. The increase number of turns from (10 to 20), (20 to 30), 30 to 40) and (40 to 45) leads to increase the percentage of 12.7%, 5.6%, 3.2% and 0.96% respectively.

8. References

[1] El-Din S.H.S. 2005 Heat transfer characteristics of an inclined single-phase toroidal thermosyphon. *Alexandria Engineering Journal* 44(2) 157-171.

[2] Amori, K.E. and N. S.Jabouri 2012 Thermal performance of solar hot water systems using a flat plate collector of accelerated risers *The Journal of Engineering Research* 9(1) 1-10.
[3] Subramanian, J., S. Tamilkozhndu and T. Selvam. 2012 Experimental Studies on Variable Header Solar Water Heating System 2nd International Conference on Mechanical Production and Automobile Engineering (ICMPAE’2012) Singapore 258-261

[4] Freegah, B., Asim, T. and Mishra. R., 2013, Computational Fluid Dynamics based Analysis of a Closed Thermo-Syphon Hot Water Solar System 26th International Congress of Condition Monitoring and Diagnostic Engineering Management Helsinki Finland 14582631.

[5] Basim Freegah, and Aouf A. Al-Tabbakh 2019 Experimental and Numerical Analysis of a Thermosyphon Solar Water Heater for Domestic Applications, U.P.B. Sci. Bull., Series D: Mechanical Engineering, 81(1) 117-132.

[6] Dehdakhel, A. A. Rahimi, M. and Alsairafi, A. A. 2010 CFD modeling of flow and heat transfer in a thermo-syphon International Communications in Heat and Mass Transfer 37 312-318.

[7] Z. Duan, Y.S. Muzychka 2007 Pressure drop of impingement air cooled plate fin heat sinks, Trans. ASME 129(2) 190–194.

[8] A. H. M AlEssa 2012 Augmentation of Fin Natural convection heat dissipation by square perforations Journal of mechanical engineering and automation 2(2) 1-5.

[9] Munson, B.R., D.F. Young, and T.H. Okiishi 1994 Fundamentals of Fluid Mechanics (New York: John Wiley & Sons. Inc.) 2nd edition.

[10] Hoffmann, K.A. and S.T. Chiang 2000 Computational fluid dynamics (USA: Engineering Education System.

[11] Blazek, J. 2015 Computational fluid dynamics: principles and applications, (Oxford: Butterworth-Heinemann).