FREEGAH, B., ASIM, T., and MISHRA, R. 2013. Numerical analysis of a closed thermo-syphon hot water solar system under steady and transient loading conditions. Presented at *40th National conference on fluid mechanics and fluid power 2013 (NCFMFP 2013), 12-14 December 2013, Hamirpur, India*.

Numerical analysis of a closed thermo-syphon hot water solar system under steady and transient loading conditions.

FREEGAH, B., ASIM, T., and MISHRA, R.

2013

Proceedings of the Fortieth National Conference on Fluid Mechanics and Fluid Power

December 12-14, 2013, NIT Hamirpur, Himachal Pradesh, India

FMFP2013_135

Numerical Analysis of a Closed Thermo-syphon Hot Water Solar System under Steady and Transient Loading Conditions

Basim Freegah
Huddersfield / Al-Mustansiriya University
Huddersfield, UK / Baghdad, Iraq
U1169570@hud.ac.uk

Taimoor Asim
University of Huddersfield
Huddersfield, UK
t.asim@hud.ac.uk

Rakesh Mishra
University of Huddersfield
Huddersfield, UK
r.mishra@hud.ac.uk

ABSTRACT

Solar energy is one of the most important sources of alternative energy. This energy source is inexhaustible and the cleanest of all known energy sources. One of the most important systems which can be used to exploit energy stored in the solar rays is a Thermo-syphon. A thermo-syphon works under the principle of natural convection, in which the working fluid rises, once it heats up, due to reduction in its density. The research regarding the effect of various types of thermal loadings on the performance output of a thermo-syphon is severely restricted, and most of the work available is based on experiments in which the detailed analysis of the complex flow field within the thermo-syphon is extremely difficult to perform. Hence, the present study focuses on the use of advanced Computational Fluid Dynamics (CFD) based techniques to simulate natural convection in a thermo-syphon operating under various loading conditions. The results depict that an increase in the thermal load, for both steady and transient loading conditions, decreases the temperature of the working fluid within thermo-syphon because of loss of thermal energy in the condenser.

Keywords: Computational Fluid Dynamics (CFD), Thermo-syphon, Navier-Stokes Equations, Natural Convection, Thermal Loading.

1. INTRODUCTION

Due to depleting fossil fuels, rapid escalation in the fuel prices globally, the greenhouse effect and environmental pollution, the need to find alternative sources of energy is becoming increasingly significant. One of the most important sources of alternative energy is solar energy. It is play a vital role in various aspects of life. The sun radiates 13 million times the energy that is generated by all the electricity consumed around the world in one year (Khan, 2011). It has more advantages than fossil fuels have; it is friendly to the environment as well as cheap and renewable. A mechanical system is readily available in the market that converts the solar energy into thermal form. This system is commonly known as a Thermo-syphon. There are many advantages of a thermo-syphon hot water system such as; its use avoids using a conventional pump, which keeps the complexity and costs of a thermo-syphon system low.

Dehdakhel et al., 2010 carried out experimental studies to analyse the effects of fill ratio on performance of a thermo-syphon for three different heat fluxes, and validated the results using CFD. The results have shown that the performance of a thermo-syphon gets affected significantly with fill ratio. Furthermore, it has been observed that CFD predicts the temperature profiles within the thermo-syphon with reasonable accuracy. Sato et al., 2012 conducted theoretical studies in order to analyse the effects of heat pipe tilt angle and condenser geometry on the temperature of the working fluid within the condenser. The study shows that the tilt angle has negligible effects, and that the conventional solar collector geometry is optimal as far as the performance of the thermo-syphon is concerned. Iordanou et al., 2012 conducted CFD based study in order to investigate the effects of placing a metallic mesh within the heat pipe of a passive solar condenser, and compared the results with a conventional passive solar condenser. The results have shown that the use of metal grid inside the heat pipe leads to increased temperature within the solar condenser. Freegah et al., 2013 conducted numerical studies using CFD to analyse the effects of collector tilt angle, number of pipes, heat flux and the length-to-diameter ratio of the pipes under no-load conditions. The study shows that the angle of inclination has negligibly small effect, while both the heat flux and length-to-diameter ratio of the pipes have significant effects on the performance of a thermo-syphon. Furthermore, increase in the number of pipes increases the temperature of the working fluid.

This study is the continuation of Freegah et al., 2013 and hence the thermo-syphon model considered in the present study is the same. Computational Fluid Dynamics tools have been used to carry out an extensive study on the effects of various loading conditions on the performance output of a closed-loop solar hot water thermo-syphon system. The effects of thermal loading in a thermo-syphon have been explicitly analysed and hence this study is important for the design process of such systems.

2. NUMERICAL MODELLING

The thermo-syphon model consists of seven pipes connecting the condenser with the collector (Fig. 1). There is a separate pipe, emerging from one end of the condenser, called as recirculation pipe, through which the cold water returns to the collector. The connecting and recirculation pipes have a diameter and thickness of 25mm and 2mm respectively. The diameters of the condenser and collectors are ten and two times the diameter of the pipes respectively. The length of the connecting pipes is such that the length-to-diameter ratio of the connecting pipes is 50. Two practical solar heat fluxes of 250W/m^2 and 500W/m^2 have been chosen for analysis. As mentioned in Freegah et al., 2013, the model is made inclined by 30º to the horizontal as it is the optimal tilt angle for this type of thermo-syphon.

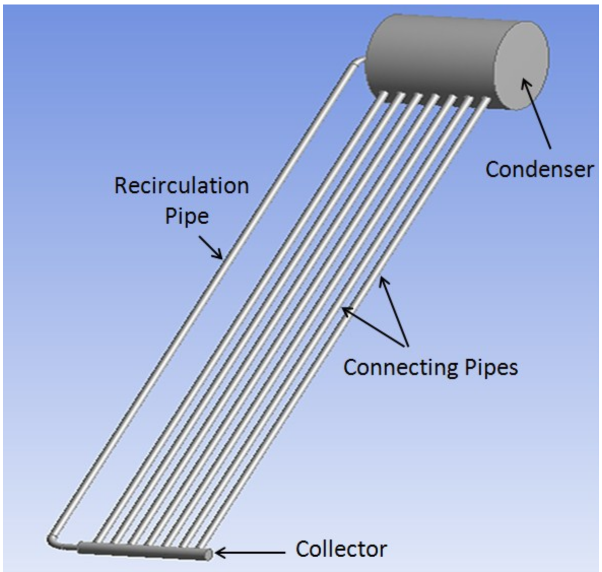

Fig. 1 Thermo-syphon model

Boussinesq approximation has been used to accurately model buoyant forces being generated. This approximation states that the density differences are sufficiently small to be neglected, except where they appear in terms multiplied by g i.e. the acceleration due to gravity. The essence of the Boussinesq approximation is that the difference in inertia is negligible, but gravity is sufficiently strong to make the specific weight appreciably different. It has been observed by Sato et al., 2012 that the Boussinesq approach for the density of the working fluid in a thermo-syphon predicts fairly accurate results and thus has been used in the present study. Three dimensional Navier-Stokes equations, in addition to the continuity and the energy equation, have been numerically solved in an iterative manner to simulate the transient flow of water within the thermo-syphon for one hour of operation.

3. RESULTS AND ANALYSIS

The thermo-syphon model considered in the presented study has been analysed under various thermal loading conditions. These loading conditions correspond to both steady and transient loadings. The following sections describe the effects of these loading conditions on the performance output of the thermo-syphon. The results presented here correspond to one hour of continuous operation of the thermo-syphon.

3.1 Steady Loading

Figure 2 depicts the velocity distribution within the central connecting pipe, the collector and the condenser after one hour of operation at a solar heat flux of 250W/m^2 and under thermal load of 50%. The natural convection phenomena can be clearly seen in the figure, i.e. as the water gets heated, it expands due to increase in volume and density, and rises up in the pipe. The hot water is accumulated in the condenser.

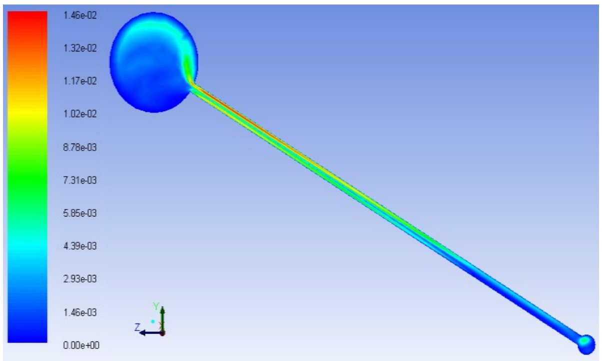

Fig. 2 Velocity variations within the thermo-syphon after 1 hour of operation at Q = 250W/m^2 and a steady loading of 50%

Due to the inclination of the connecting pipes, water propagates along the top wall of the pipe. Furthermore, it can be seen that the velocity of the water increases as it climbs up the pipe. This happens because more thermal energy is being absorbed by the fluid as it propagates along the connecting pipe, further decreasing its density. The water attains highest velocity at the junction of the connecting pipes with the condenser.

Figure 3 further depicts the natural convection phenomena occurring in the thermo-syphon model considered in the present study. As mentioned earlier, the water heats up and rises in the connecting pipe. Along the pipe, more thermal energy of the solar rays is transferred to the internal energy of the water, increasing its temperature further. Highest temperature of water is observed at the junction of the connecting pipes and the condenser. Due to its lower density, the water further rises within the condenser and gets accumulated along the top wall of the condenser, whereas the cold water from the bottom of the condenser is transferred back to the collector via recirculation pipe.

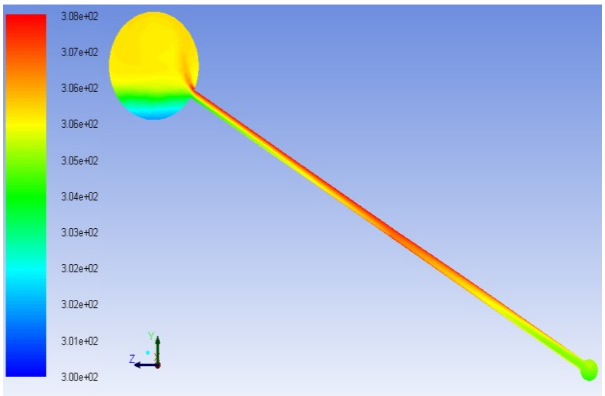

Fig. 3 Temperature variations within the thermo-syphon after 1 hour of operation at Q = 250W/m^2 and a steady loading of 50%

Figure 4 depicts the temperature distribution within the condenser of the thermo-syphon. It can be clearly seen that the hot water occupies the upper section of the condenser while the cold water settles on the bottom of the condenser. After one hour of operation, the difference between maximum and minimum temperature within the condenser is 7.24K at a heat flux of 500W/m^2. Please see Table 1 for complete results.

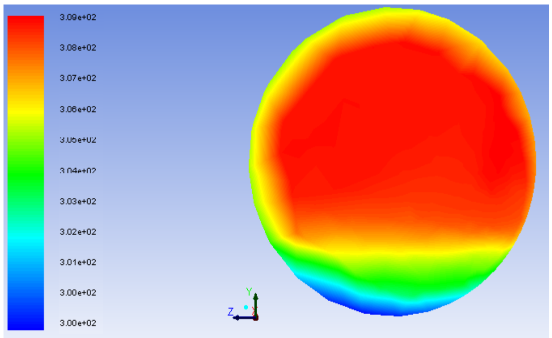

Fig. 4 Temperature variations in the condenser after 1 hour of operation at Q = 500W/m^2 and a steady loading of 50%

Figures 5 and 6 depict the variations in temperature within the condenser for various loading conditions, at a heat flux of 250W/m^2 and 500W/m^2 respectively. It can be clearly seen that after one hour of operation, the temperature within the condenser is highest for no loading condition whereas it is lowest in case of 70% of thermal loading. Hence, increase in thermal load decreases the temperature of water within the condenser. Furthermore, it can be seen in table 1 that the average temperature within the condenser decreases, whereas the difference between the maximum and minimum temperature increases as thermal loading increases.

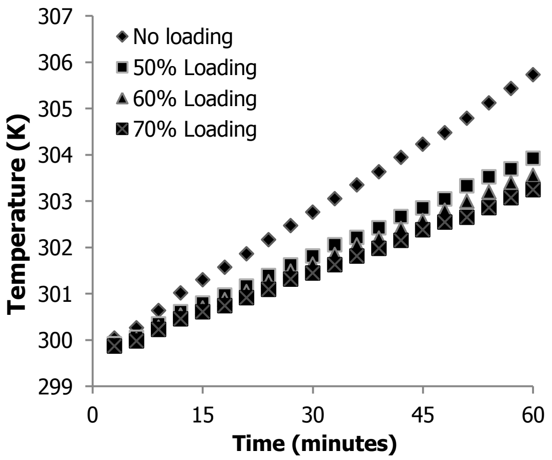

Fig. 5 Variations in temperature within the condenser for one hour of operation at Q = 250W/m^2 under various steady thermal loads

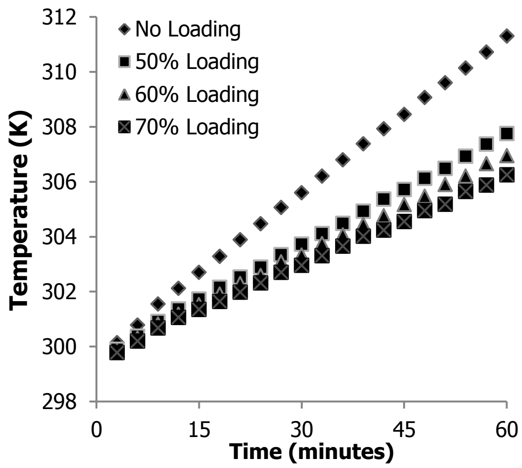

Fig. 6 Variations in temperature within the condenser for one hour of operation at Q = 500W/m^2 under various steady thermal loads

Table 1. Effect of steady loading on condenser's temperature

Heat Flux (W/m^2)	Loading (%)	Average Temperature (K)	Max – Min Temperature (K)
250	No Loading	305.7	4.63
	50	303.9	5.52
	60	303.5	6.06
	70	303.2	6.19
500	No Loading	311.3	5.35
	50	307.7	7.24
	60	306.9	7.75
	70	306.2	7.91

3.2 Transient Loading

Figure 7 depicts the transient thermal loading applied to the thermo-syphon model in the present study. In the first 30minutes, the thermal loading increases from 50% to 70%. In the latter 30minutes, the thermal loading decreases from 70% to 50%. This transient loading has been analysed for both heat fluxes of 250W/m^2 and 500W/m^2.

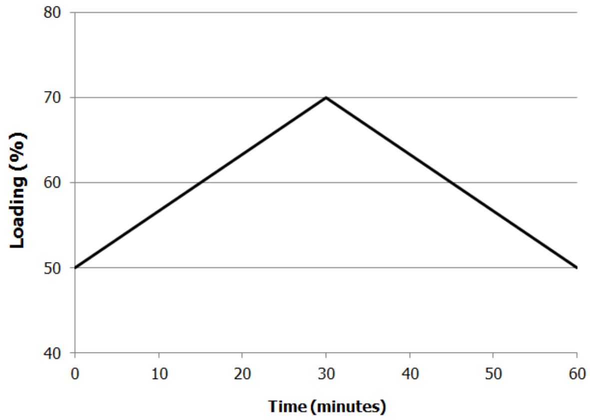

Fig. 7 Transient loading

Figure 8 shows the comparison between the steady and transient thermal loading at a heat flux of 250W/m^2. It can be seen that transient loading results in the same condenser temperature as observed in the case of 60% steady loading. Hence, the transient loading results in the same condenser temperature as that for a steady load equal to averaged transient load, as shown in Eq. (1).

$$T_{Steady} = \overline{T_{Transient}} \qquad (1)$$

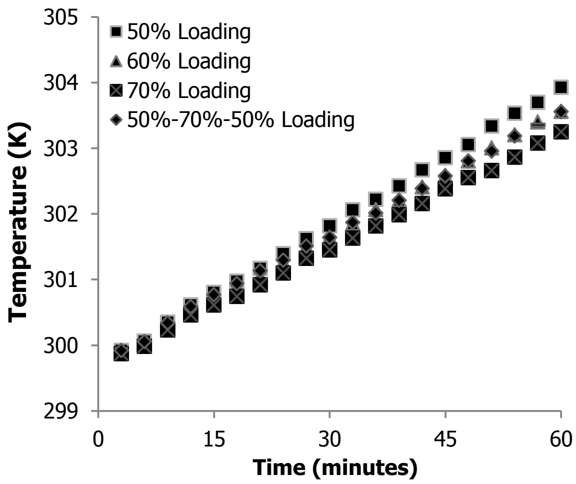

Fig. 8 Variations in temperature within the condenser for one hour of operation at Q = 250W/m^2 under transient loading condition

Figure 9 depicts the variation in temperature within the condenser for transient thermal loading at a heat flux of 500W/m^2. Similar to Fig. 8, it can be seen that the transient loading results in the same condenser temperature as that for a steady load equal to averaged transient load.

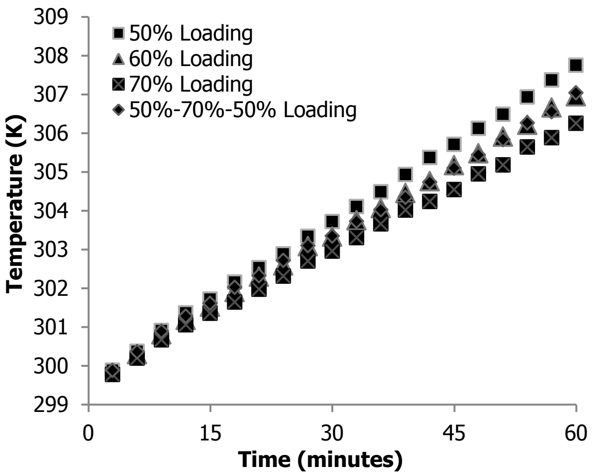

Fig. 9 Variations in temperature within the condenser for one hour of operation at Q = 500W/m^2 under transient loading condition

4. CONCLUSIONS

The effects of various thermal loading conditions on the performance output of a thermo-syphon solar hot water system, has been analysed, where both the steady and transient loads have been taken into consideration. It has been shown that thermal loading decreases the temperature within the condenser. Furthermore, as the thermal loading increases, the temperature within the condenser decreases, while the difference between the maximum and minimum temperature increases. Moreover, transient loading results in the same temperature as that observed for a steady loading, equal to averaged transient load. The information provided in this study can be used to design a thermo-syphon system for various loading conditions, as per the users' demands. It has also been shown that CFD can be used as an effective tool to analyse the performance characteristics of a thermo-syphon, under various loading conditions, with reasonable accuracy.

REFERENCE

[1] Dehdakhel, A. A., Rahimi, M., 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.

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

[3] Iordanou, G., Tsirigotis, G., 2012. Computational Fluid Dynamics (CFD) Investigations of the Effect of Placing a Metallic Mesh in the Channels of a Passive Solar Condenser Model, Electronics & Electrical Engineering 5 69 – 74.

[4] Khan, B. A., 2011. Performance improvement of solar hot water system by reducing solar irradiation from solar collector and solar tank, M.Sc. Thesis, BRAC University, Dhaka, Bangladesh.

[5] Sato, A. I., Scalon, V. L., Padilha, A., 2012. Numerical analysis of a modified evacuated tubes solar condenser. In: International Conference on Renewable Energies and Power Quality, Spain.