Analytical and CFD Analysis of Single Phase Thermosyphon

Parth Patel[1] Keval Brahmbhatt[2] Deep Patel[3] Malay Patel[4]

[1,3,4]Pandit Deendayal Petroleum University, Gandhinagar [2]Government Engineering College, Patan

Abstract— Thermo siphon is a method of passive heat exchange. It is based on natural convection, which circulates a fluid without the necessity of a mechanical pump. Thermo siphoning is used for circulation of liquids and volatile gases based on natural convection in heating and cooling applications such as heat pumps, water heaters, boilers and furnaces. In this paper experimental analysis results are compared by calculating buoyancy force generated by available temperature difference. Based on buoyancy force it is easy to calculate mass transfer rate and heat transfer rate. For see local effect of thermosyphon CFD analysis is carried out in ANSYS Fluent and compared these experimental, theoretical and CFD analysis results for different heat transfer rate.

Key words: Thermo-Siphon, Simulation, CFD Analysis

I. INTRODUCTION

Water heating characteristically represents a significant percentage of energy consumption in domestic, industrial applications. Carbon baseline fuels water heating produce emissions of greenhouse gases and other pollutants. Due to high cost primary energy resources and their associated with serious environmental issues, solar energy is an alternative viable source of energy for water heating. Solar water heating can be characterized as active or passive [1-5]. An active system is based on an electric pump to circulate the working fluid through the solar collector. In passive solar water heating, heat-transfer fluid uses thermo syphoning phenomenon to circulate the water by the buoyancy forces and is replaced by colder water from the bottom of the tank [5]. This constant circulation continues until heats up water in the storage tank.

The Thermosiphon solar system, or water heating solar system with natural circulation, is the most exciting, simplest and most well-known technological device used for solar energy exploitation. Its remarkable effectiveness coupled with its simplicity in design, its autonomy in its operation and its minimization of maintenance makes it an interesting alternative to any system using an auxiliary pump.

Same as the thermo-syphon cooling system operates on the principle of natural convection caused by variation in density of water, and hence does not use a pump. The heated water expands, due to which the density decreases. When it cools down, its volume decreases and hence density increases. This variations in density sets up convection currents so that circulation of water takes place. All components of water-cooling systems except the circulating pump are used in this case.

Thermosiphon effect is not only limited to the solar water heating system but now a time it's become a applicable to the different systems such as the Avionics cooling, enclose cooling, Electronics cooling, Led thermal management, Material processing, Power electronics etc.

But many studies have been conducted on the solar water heating system. Venkatesh [1]1994 presented experimental and theoretical study for domestic solar water heater with collector area of 1m^2 and storage tank of 0.16 m^3. The maximum solar intensity is taken as 1000 W/m^2 and the overall heat loss coefficient is 5.5 W/m^2 ℃.He founds that the predicted maximum temperature of the storage water is 59.5℃.

This paper is mainly focusing on theoretical and CFD analysis of thermosyphon. And try to develop its theoretical model for future cooling and heating application.

II. MATHEMATICAL MODEL

Analysis of single phase thermo-syphon is presented based on theoretical analysis and CFD approach. Results are compared with available experimental data.

A. Experimental Analysis

G.L.Morrison and D.B.J.Ranatunga [9] have been conducted the experiment of single phase thermosyphon loop. An outline of the flow circuit used for the tests is shown in Fig. 1. A single riser 0.011 m dia. and 1.1 m long was connected between two headers and a storage tank by 0.022 m dia. piping. This model was not designed to simulate the flow conditions in an actual collector but to provide a simple flow circuit in which a test of thermosyphon flow calculations could be made.

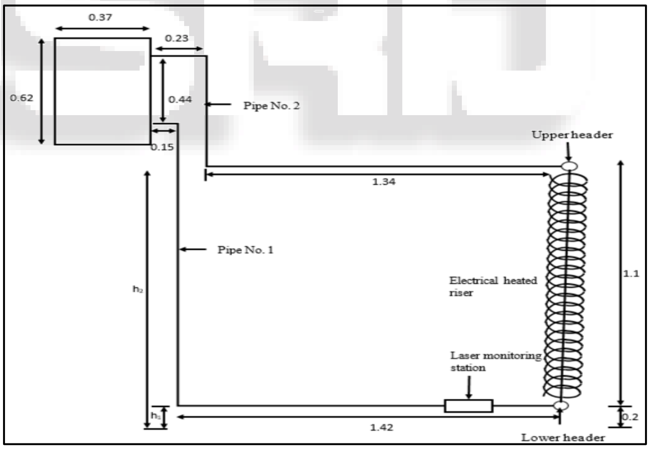

Fig. 1: Electrically heated test circuit

The heat input was provided by an electric heating cable wrapped around the full length of the riser, with 20 mm of insulation around the heater assembly. Temperatures were measured using calibrated copper-constantan thermocouples inserted through the pipe wall and into the flow. The temperature in the tank between the inlet and outlet pipes was kept uniform by stirring the contents approximately 30 rain before each test, and the temperature distribution was checked by traversing a thermocouple down the tank. The tank depth above the return pipe connection was sufficient to ensure that hot return water was always above the upper connection. These factors were carefully monitored to ensure that the thermosyphon head did not vary during the measurements due to density variations in the tank between the inlet and outlet points.

The flow rate was measured with a laser Doppler anemometer to avoid introducing any restriction into the circuit. The accuracy of the flow measurements was checked by blocking the return pipe to the tank and bleeding flow from the top header.

B. Theoretical Analysis

A. Zerrouki, A. Boumedien , K. Bouhadef [10] have been presented theoretical analysis of thermo-syphon for solar water heater. In this report, theoretical analysis has been done for above experimental circuit.

1) Buoyant pressure in the thermo-syphon loop

The effective pressure (ΔP_e) responsible for the flow can be considered to be made up of two parts, one representing the pressure drop across the heater (ΔP_c) and the other between the top and bottom of the tank (ΔP_t) such that

$$\Delta P_e = \Delta P_c + \Delta P_t \quad (1)$$

Let T(x) be the temperature at distance x from the inlet of the collector, the pressure due to buoyancy forces (ΔP_c and ΔP_t) can be evaluated by the following equations,

$$\Delta P_c = g\,\beta\,\rho_0\,(\sin\theta)\int_0^L T(x) - T\;dx \quad (2)$$

$$\Delta P_t = g\,(T_o - T_i)\,H \quad (3)$$

Then,

$$\Delta P_e = g\,\beta\,\rho_0\,(\sin\theta)\int_0^{Lc} T(x) - T\;dx + g\,(T_o - T_i)H \quad (4)$$

The variation of temperature in the tubes can be described by a linear equation,

$$T(x) - T_i = \frac{(T_O - T_i)}{L_c}x \quad (5)$$

So,

$$\Delta P_c = g\,\beta\,\rho_0\,(T_o - T_i)\left[\frac{L_c(\sin\theta)}{2} + H\right] \quad (6)$$

Where,

L_C= length of heater (m)
β = coefficient of expansion of water (–)
θ = heater's inclination to the horizontal (deg)
H = height of tank (m)
g = acceleration due to gravity (m / s^2)
ρ_0 = water density at heater outlet (kg / m^3)

2) Frictional pressure losses in the system

The total frictional pressure loss (ΔP_L) in the circuit can be separated into the contribution (ΔP_c) from the heater and that (ΔP_{ct}) from the piping components outside the heater:

$$\Delta P_L = \Delta P_c + \Delta P_{ct} \quad (7)$$

$$\Delta P_L = \Delta P_c\,(1+\emptyset) \quad (8)$$

$$\emptyset = \frac{\Delta P_{ct}}{\Delta P_c} \quad (9)$$

Where the ratio ∅ characterizes the flow resistance in the heater and connecting Tubes.

At the equilibrium, the buoyancy forces (ΔP_e) are equal to the friction resistance (ΔP_L) in the flow path. Then:

$$\Delta P_L = \Delta P_e \quad (10)$$

$$\Delta P_c\,(1+\emptyset) = g\,\beta\,\rho_0\,(T_o - T_i)\left[\frac{L_c(\sin\theta)}{2} + H\right] \quad (11)$$

3) Mass Flow Rate

Mass flow rate of the system can be obtained by following method:

a) For the heater:

$$\frac{\Delta P_c}{L_c} = \frac{\rho\,f\,V_c^2}{2\,d_c} \quad (12)$$

Where f is the fluid friction factor in the heater.

$$f = \frac{64\,\upsilon}{d_c\,V_c} \quad (13)$$

Where υ = kinematic viscosity of water (m^2/s)

$$\Delta P_c = \frac{32\,\rho \times V_C \times L_c \times \upsilon}{d_c^2} \quad (14)$$

Similarly, the pressure drop in the connecting tubes is given by:

$$\Delta P_{ct} = \frac{32\,\rho \times V_{ct} \times L_{ct} \times \upsilon}{d_{ct}^2} \quad (15)$$

$$\emptyset = \frac{\Delta P_{ct}}{\Delta P_c} \quad (16)$$

Putting the values of ΔP_{ct} and ΔP_c in above equation

$$\emptyset = \left(\frac{L_{ct}}{L_c}\right) \times \left(\frac{d_c}{d_{ct}}\right)^4 \quad (17)$$

The water velocity in the flow path can be obtained by the following equation:

$$V_c = \frac{4\dot{M}}{\rho \Pi d_c} \quad (18)$$

So, pressure losses in heater,

$$\Delta P_c = \frac{128\,\upsilon\,Lc\,\dot{M}}{\Pi {d_c}^4} \quad (19)$$

$$\Delta P_e = \Delta P_c \quad (20)$$

$$\frac{128\,\upsilon\,Lc\,\dot{M}}{\Pi {d_c}^4}[1+\emptyset] = g\,\beta\,\rho_0\,(T_o - T_i)\left[\frac{L_c(\sin\theta)}{2} + H\right] \quad (21)$$

$$Q = \dot{M} C_p\,(T_o - T_i) \quad (22)$$

From equations (21) and (22) mass flow rate can be calculated.

C. CFD Analysis

Thermo-syphon is being investigated using commercial CFD fluent 17.2 software. Transient CFD calculations are performed with a density, viscosity and specific heat of water as functions of temperature (piecewise linear). The second order upwind method is used for discretization of pressure, momentum and energy equations. The simulation runs with a time step of 1 second.

Basically problem consists of solid area in which incompressible fluid water is there. So pressure based solver is selected to solve transient heat transfer flow. CFD SIMPLE scheme is adopted to solve the problem. Unsteady temperature behavior of water need to be obtained from the problem and velocity-density changes need to obtained mass flow rate.

The buoyancy effect is modeled with piecewise linear approximation during CFD simulation. Transient CFD calculations are performed with a density, viscosity and specific heat of water as functions of temperature (piecewise linear). Laminar model is used. And the buoyant force should be enough capable to provide hot water to the storage tank.

Heat flux is applied to the heater and water tank temperature is taken as reference temperature.

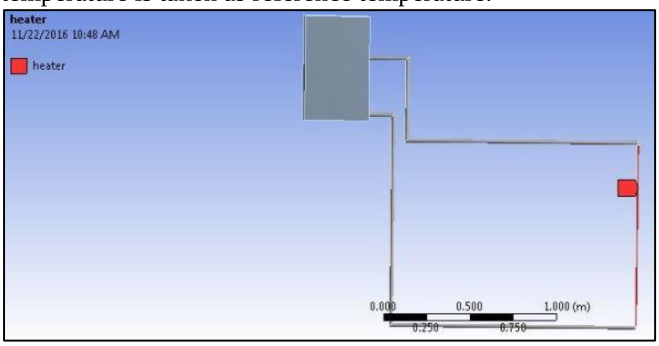

Fig. 2: Boundary condition to heater

III. RESULT AND DISCUSSION

A. Results

Following are the results of single phase thermo-syphon analysis based on theoretical and CFD approach and figures shows the two different diagrams based on the density change and based on the velocity change.

Figure 3 shows the density change is just only occur at the portion where the heat is supply and is shown by the blue colour and the portion of pipe coming from the storage tank is shown by the red colour which shows that the density of the water is the highest due to law temprature but the hot water carrying in the pipe which enters in the storage tank has density is nearest to the highest density water in the pipe so that it also is shown as the rad colour in the figure. And there is no too much density difference between the cold water and the hot water for these small dimensions. It may be differ for the large dimensions.

Figure 4 shows that the velocity change in the pipe during the heat transfer in the water. When the water comes from the storage tank at that time the velocity will be law because of the high density of the water and the velocity goes on increasing as temperature of water increases and density reduction and due to this temperature increasing mass flow rate of hot water in the storage tank is also goes on increasing.

1) Result for water thermo syphon system

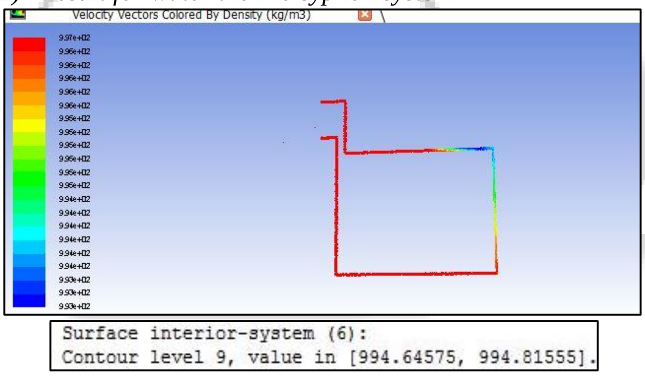

Fig. 3: Density change in water thermosyphon loop

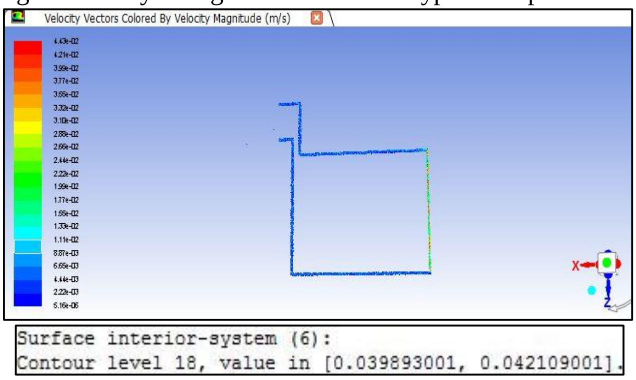

Fig. 4: Velocity change in water thermosyphon loop

Fig. 3 & 4 shows velocity and density changes in single phase thermo-syphon for water system. From that mass flow rate can be obtained. As temperature of Water increases due to heat load, density will be decrease because density is linearly varies with temperature. There is a negligible change in velocity. From density and velocity mass flow rate (m = ρAV) can be obtained. Example, CFD result for Q = 100 W

$$\rho = 994 \text{ [kg/m}^3\text{]}$$

$$V_c = 0.04 \text{ [m/s]}$$

$$A = \frac{\Pi}{4}\,(d_c^2) \text{ [m}^2\text{]}$$

$$\dot{M} = \rho A V_c = 0.00377 \text{ [kg/s]}$$

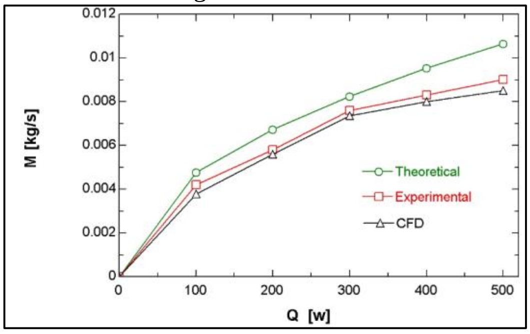

Fig. 5: Mass flow rate vs heat load in case of water thermosyphon loop

Fig. 5 shows variation in mass flow rate as load is varied. As the Heat transfer increase mass flow rate also increase. Graph shows comparison of results by theoretically, experimentally and CFD approach. Variations in experimental and CFD mass flow rate is Less compare to theoretically because in CFD and experiment all losses are considered like losses due to sudden expansion, sudden contractions, elbow losses etc. while in theoretical design these losses are not considered and smooth pipe is considered.

Above figure shows the difference between the results of Theoretical, experimental, and CFD is very close at the law heat transfer in water and as increase in the heat transfer theoretical results are somewhat differ from the experimental as well as the CFD results. And that's the reason to do CFD analysis at high temperature heat transfer to water for getting the actual value.

IV. CONCLUSION

The results obtained by theoretically is nearest to the results obtained by experiment and CFD analysis only at low temperature heat transfer because at that time losses generated in the system like pipe losses, elbow losses etc. are less compared to the high temperature heat transfer and that is the reason result obtained by theoretically at high temperature is too much differ from the result obtained by the experiments and CFD analysis

Above results also conclude that as heating load increases to the cooling loop, mass flow rate of the fluid will be also increased. There is a variation between theoretical and CFD results because in theoretical analysis losses like losses due to sudden expansion, sudden contractions, elbow losses are not considered and smooth pipe is considered.

REFERENCES

[1] Close DJ. The performance of solar water heaters with naturalcirculations. Sol Energy 1962;6:30–40.
[2] Nahar NM. Year round performance and potential of a natural circulation type of solar water heater in India. Energy Buildings 2003;35:239–47.
[3] Pierson P, Javelas R. Theoretical and experimental study of a natural circulation type solar water heater with exchanger. Re´v Ge´n Thermique 1983;258–259:467–82.

[4] Ammari HD, Nimir YL. Experimental and theoretical evaluation of the performance of a tar solar water heater. Energy Convers Manage 2003;44(19):3037–55.
[5] J. Huang, S. Pu, W. Gao, and Y. Que, "Experimental investigation on thermal performance of thermosyphon flat-plate solar water heater with a mantle heat exchanger", Energy, Vol. 35, pp.3563-3568, 2010.
[6] A. Manickavasagan, A. Sampathrajan, P. R. Manigandan, S. Manivannan, F. M. E. Emerald, R. Malarkodi, and S. Mangalam, "An experimental study on solar flat plate collector using an alternative working fluid", Pertanika Journal of Science & Technology, Vol. 13, Issue 2, pp. 147–161, 2005.
[7] B. Sitzmann, "Solar Water Heater with Thermosyphon Circulation", Appropriate Technology, Vol. 31, Issue: 1, pp. 66-70, 2004.
[8] K. Lovegrove, and M. Dennis, "Solar thermal energy systems in Australia", International Journal of Environmental Studies, Vol. 63, Issue: 6, pp.791-802, 2006.
[9] G. L. Morrison, "THERMOSYPHON CIRCULATION IN " ~ Cottector," vol. 24, pp. 191–198, 1980.
[10] K. Bouhadef, A. Zerrouki, and A. Boume, "The natural circulation solar water heater model with linear temperature distribution," vol. 26, pp. 549–559, 2002.

