CFD Modelling for flow and heat transfer in a closed Thermosyphon charged with water–A new observation for the two phase interaction

Alaa A B Temimy¹, Adnan A Abdulrasool²
¹Training and Power research office, Ministry of Electricity, Baghdad, Iraq.
²Mechanical Engineering Department, Faculty of Engineering, Al-Mustansiriyah University, Baghdad, Iraq

Abstract. A three dimensional transient numerical model is done for a Thermosyphon heat pipe to visualize the steam and condensate flow patterns and interaction. The Computational Fluid Dynamics (CFD) model was arranged using Volume of Fluid solution model (VOF) in Fluent. A copper vertical pipe with 14.2mm inside diameter, 400 mm length, and 0.9mm thickness is selected. The heat input varies from (20-120) Watt. The cooling is made with water jacket heat exchanger around the condenser with outer diameter of 55 mm. Filling ratios of (40,50,60,70 and 100) % are tested. All cases show that the classical known flow pattern where the steam flow upward in the core and the condensate flow downward attached to the inner surface of the pipe do not occurs. The simulation results clear out a behavior that both the rising hot steam and the downward condensate flows near the inner pipe wall. So that, both phases face each other near the wall. The high momentum phase continue it flow in its way and shift the other phase from the wall. This flow behavior leads the steam and condensate to flow in a spatial unstable flow upward and downward respectively. The flow continues even when steady state was achieved and create a secondary mass and heat transfer between the phases.

Keywords: Two-phase thermosyphone, Heat pipe, Computational fluid dynamics (CFD), Flow visualization, Flow patterns, Evaporation, Condensation, Water.

1.Introduction
Interest in the usage of heat pipe expertise for heat regaining and energy investments in a huge assortment of engineering uses has been on the growth through the latest years [1, 2]. Heat pipes in performance are a very essential role in numerous industrial uses, mainly in improving the thermal characteristics of heat exchangers [3], reducing hydraulic resistance and corrosion effect in solar panels[4], and integrating of heat pipes into central reservoir of thermal power plant tower [5]. Also heat pipes due to it’s isothermal surface and fast response represent the key features in solar power plants that use molten salt as a heat storage and working fluid [4].

Computational procedures play a central role in explaining complex stream problems for many engineering applications because of their flexibility , universality, efficiency and accuracy [6, 7]. Wickless heat pipe or two-phase closed thermosyphone ( TPCT ) based systems are extra required than normal systems because of their reliability , efficiency , passive operation , and the ease and low cost of manufacturing [8]. A TPCT is a heat transfer tool with a very high thermal conductivity, having a little amount of operational fluid that travel in a sealed pipe consuming the buoyancy and gravity effects to return the condensates liquid back to the evaporation section, as shown in figure 1 [9].
During the heating of evaporator section by an external heating source, the evaporator walls transfers the heat to the working fluid. The liquid absorbs the heat and start to evaporate based on its latent heat of evaporation and the amount of heat absorbed. The vapor travel upward to the condenser section due to buoyancy forces, where its return to liquid state at condenser wall, releasing its absorbed heat from evaporator section. The condensate then returned downward to the evaporator section by gravitational forces [10]. TPCT have been widely used in numerous applications; however, up to now, the numerical studies that displaying the interact of the two-phases flow patterns inside the heat pipe, are still at primary stage.

Zhang et al. [11] proposed a 2D model of a flat disk-shape TPCT used for electronic parts cooling. They simulate the flow in TPCT as a single-phase flow, and validate the distribution of the vapor temperature and velocity using the experimental results. Alizadehdakhel et al. [12] focused on experimental and a 2D model numerical simulation studies. They study the relations between the heat supply with the filling ratio on the thermal performance of a TPCT charged with water. Annamalai and Ramalingam [13] working on experimental study and CFD modeling of a wicked heat pipe by using CFX routine in ANSYS. They assume the core of the heat pipe as a vapor (single phase) and the wick section as the liquid phase. Distilled water used as the charged fluid. The authors validate the expected steam temperature distribution and the surface temperature of the condenser and evaporator walls with their gotten experimental data. Lin et al. [14] using Mixture models and VOF , and water as the charged fluid. Built a CFD model to expect the heat transfer ability of miniature oscillating heat pipes (MOHPs). Different evaporator and condenser lengths and inner diameters with different heat supply are arranged to analyze the thermal performance of MOHPs. Kafeel and Turan [15] study the TPCT performance with the change of periodic increase of the heat supply to the evaporator section. To corroborate the TPCT simulation model, the setup was made according to Amatachaya et al. [16]. Eulerian model selected to simulate the condensation film at the condenser section. The simulation set at filling ratio of 30% of the evaporator volume. Adel et al. [17] study experimentally and numerically the thermal performance of 400 mm long copper TPCT charged with six different fluids; water, methanol, ethanol, acetone, butanol and R134. The 3D numerical model is made with FORTRAN code based on finite difference method to solve energy equation only. The filling ratios are 40, 50, 60, 70 and 100%. The mass transfer, phase change and vapor to condensate interaction are not take into calculation consideration. The agreement between experimental data and numerical solution results are about 10%. The 2D-CFD results of Fadhl et al. [18, 19] indicated that VOF model in fluent can successfully illustrate the complicated two-phase flow behavior inside the TPCT.
The purpose of this work is to arrange a 3D-CFD setup to simulate mass transfer, phase change, and steam/condensate volume fractions variations and flow interactions phenomena during mixture traveling from evaporator to the condenser and vice versa. Also the simulation includes the water jacket around condenser and evaporator. The CFD solution has effectively visualized and predicted the steam & condensate complex flow interactions phenomena, which has not been reported before in previous studies. This complex flow has been investigated at different working parameters of filling ratio and heat supply. Also this work emphasis the numerical flow visualization of the mixture during the transient and steady state of a TPCT. This work represents an extension to that carried out by Adel et al. [17].

2. Mathematical model
The governing equations of momentum, mass continuity, and energy are used to establish the motion of water in TPCT as follows [6, 7, 20]:

2.1 Continuity equation
Based on the physical principles of VOF model, and to get volume fraction equations of conservation of mass for the fluid, the equation of continuity has the following form:

$$\nabla \cdot (\rho V) = -\frac{\partial \rho}{\partial t} \tag{1}$$

Where:
- $V$ = velocity vector
- $\rho$ = density
- $t$ = time

To track the boundary edges between the phases, the solution of equation (1) for volume fraction of the one phase (vapor) is used. Thus, the equation of continuity for the other phase (liquid) can be expressed as:

$$\nabla \cdot (\alpha_l \rho_l V) = -\frac{\partial (\alpha_l \rho_l)}{\partial t} + S_m \tag{2}$$

Where:
- $S_m$ = the phase change (mass source) term used to estimate the mass transfer from one phase to another during evaporation and condensation.

The continuity equation stated above could be entitled the equation of volume fraction and it will not be solved for the (vapor) primary phase, because primary phase volume fraction is calculated using the following expression:

$$\sum_{\nu=1}^{n} \alpha_{\nu} = 1 \tag{3}$$

A mixture of the phases v and l exists when the cell is not wholly full with primary phase (v) or with the secondary phase (l). So that, the mixture density is calculated as the averaged density volume fraction as follow:

$$\rho = \alpha_l \rho_l + (1-\alpha_l)\rho_v \tag{4}$$

2.2 Momentum equation
For VOF model the acting forces in the fluid was considered to be pressure, gravitational, surface tension and friction. To find the surface tension effect along the two phases interface, the continuum surface force (CSF) parameter has been added to the 3D momentum equation

$$F_{CSF} = 2\sigma \frac{\alpha_l \rho_l C_v \nabla \alpha_v + \alpha_v \rho_v C_l \nabla \alpha_l}{\rho_l + \rho_v} \tag{5}$$
Where:

- $C =$ coefficient of surface curvature
- $\sigma =$ surface tension

By considering the $F_{\text{CSF}}$ forces into VOF model, the momentum equation will be as follows:

$$\frac{\partial}{\partial t} \left( \rho V \right) + \nabla \cdot (\rho V V^T) = \rho g - \nabla p + \nabla \left\{ \mu (\nabla V + (\nabla V)^T) - \frac{2}{3} \mu (\nabla \cdot V) I \right\} + F_{\text{CSF}}$$  \hspace{1cm} (6)

Where:

- $g =$ gravitational acceleration
- $I =$ unit tensor
- $p =$ pressure

The equation of momentum depends on the $(\alpha)$ volume fraction of the phases in the calculation of physical properties such as viscosity and density. So that, the dynamic viscosity $\mu$ calculated as

$$\mu = \alpha_l \mu_l + (1 - \alpha_l) \mu_v$$  \hspace{1cm} (7)

### 2.3 Energy equation

For the VOF model, energy equation has the following form:

$$\frac{\partial}{\partial t} (\rho E) + \nabla \cdot (\rho EV) = \nabla \cdot (k \nabla T) + \nabla \cdot (pV) + S_E$$  \hspace{1cm} (8)

Where energy source parameter $S_E$ is used to compute the heat transfer during condensation and evaporation.

The VOF model consider the mass averaged calculation for temperature variable and energy $E$, while calculate the equivalent thermal conductivity $k$ as:

$$k = \alpha_l k_l + (1 - \alpha_l) k_v$$  \hspace{1cm} (9)

$$E = \frac{\alpha_l \rho_l E_l + \alpha_v \rho_v E_v}{\alpha_l \rho_l + \alpha_v \rho_v}$$  \hspace{1cm} (10)

Where $E_l$ and $E_v$ are established on the mass averaged temperature and the specific heat $C_v$ of each phase, as given by equation of state:

$$E_l = C_v l (T - T_{sat})$$  \hspace{1cm} (11)

$$E_v = C_v v (T - T_{sat})$$  \hspace{1cm} (12)

### 3. Model geometry and computational mesh

To simulate the flow in TPCT, a 3D model was established. The geometrical dimension and specifications for the model are shown in figure 2 and as follow:
• Working fluid: water
• Material: copper
• Total length: 400 mm
• Length of evaporator section: 150 mm
• Length of adiabatic section: 100 mm
• Length of condenser section: 150 mm
• Outer diameter: 16 mm
• Inner diameter: 14.2 mm
• Wall thickness: 0.9 mm
• Water jacket outer diameter: 55 mm

The simulation mesh used for TPCT including water jacket is illustrated in figure 3. The cells are concentrated near the outer and inner walls of the pipe to capture the phase change phenomena near the inner wall of the pipe, and to capture the convection heat transfer coefficient at the outer wall. As a result 915,452 cells are recorded for the simulations.
4. CFD solution setup and Boundary conditions

A transient solution procedure was selected to trace and record mass and heat transfer, velocities, phase change and water volume fractions variation and interaction from start up to steady state conditions. Steady state conditions are achieved when the supplied heating power to the evaporator outer wall are equal to the consumed power by the cooling water. The appropriate solver for two-phase simulations is Pressure- based solver [21]. The gravity is activated with a value of (-9.81 m\(^2\)/sec) in Y-direction, so that buoyancy forces effects will be included. Energy model was activated to include thermal effects in the simulation. VOF model is selected with only two Eulerian phases (i.e. water + steam), activate implicit body force [20]. For the first time in TPCT CFD simulation in Fluent, the interface modeling with sharp/dispersed option is enabled with a value of 1 [21, 20]. The materials of copper, water liquid and steam (water vapor) are selected from the built in Fluent database. Steam set as primary phase, while water liquid set as secondary phase. Heat flux applied at evaporator outer wall at a range of (20-120 W) to simulate the experimental supplied heat power carried out by Adel et al. [17]. A 120 gm/sec flow rate are set for cooling water at a temperature of 25 °C for cooling jacket. Adiabatic section set as zero flux and all other faces are set as thermal coupling. Filling ratios for fluid zone are set using adaptation procedure with ratios of (40, 50, 60, 70 and 100%) [17]. Surface tension between the phases set as constant value of 0.072 N/m. Initial internal pressure set to the water saturation pressure at 25 °C which is 4000 pa.

A coupled scheme method used for pressure based solver to calculate volume fractions at each time step. Time step of 1*10\(^{-5}\) sec are set to insure a Courant number at phases interface to be less than unity. The convergence error limits set to 10\(^{-5}\) for continuity and velocities, while set to 10\(^{-6}\) for energy.

5. CFD solution validation

The steady state condition for the CFD simulation solution has been achieved when heat absorbed by cooling water was equal to heat supplied to the evaporator section. Transient period till steady state are different from each case to another based on operating condition. To validate our CFD simulation results, we use experimentally recorded wall temperatures of reference [17] at filling ratio of 40% at different values of heat supply. Figure 4 show the validation results with high agreement for all
temperatures recorded for the above mentioned experimental cases. It is clear that CFD results temperatures are a little bit higher than experimental temperatures, (especially at high temperatures that present evaporator wall) for the same locations and operating conditions. This could be due to heat losses to the surrounding in experimental tests, so that the actual heat supplied to the evaporator and heat absorbed by cooling water was less than recorded. While the supplied heat into CFD simulation was totally transferred to the evaporator and absorbed by cooling water because all surrounding conditions assumed adiabatic, i.e. no heat losses.

**Figure 4.** CFD validation based on experimentally recorded temperatures, data from reference [17] at filling ratio 40%.

**6. Results and discussion**

The CFD solution results show a different, non-noticed nor discussed for steam and condensate interactions and flow behaviors inside TPCT. This new recorded interactions and flow behavior are more logical than the widely known and simple flow behavior usually referred to in such condition of flow inside TPCT. The cross sectional views at different time intervals and elevations show that the accumulated and generated hot steam flow upward due to buoyancy forces near the walls of evaporator and adiabatic sections. At the same time, the accumulated cold condensate flow downward due to gravitational forces near the wall of condenser section. This flow leads the two phases to face each other near the wall and each phase with the aid of its momentum is trying to proceed in its flow near the wall. Based on the momentum magnitude and flow direction of each phase, the higher momentum phase keeps its flow direction near the wall and shift the other phase away. So that, at some locations the condensate leaves the wall and continue to flow downward, and open the way to the steam to flow upward near the wall, and vice-versa. Finally, the condensate flow downward in spatially unstable form between the raised hot steam patches that flow in opposite direction as shown in figure 5.
This flow behavior could be logically accepted and explained based on steam generation and flow direction as follow: Due to the high evaporator wall temperatures with respect to the center core of TPCT as shown in figure 6, the nucleate boiling and steam generation was started at the wall not the center. Also the mixture density is lighter near the wall due temperature variation at the horizontal cross section at any level of the evaporator. That is lead the hot generated steam bubbles to flow upward due to buoyancy forces near the wall through the lighter density mixture, i.e. low buoyancy forces required. So that, the bubbles velocity and temperatures was increased while its flow upward,
and merged with other generated bubbles to create a steam stream flow near the wall. The adiabatic section is thermally isolated from outside, but the pipe temperature influenced by evaporator and condenser temperatures due to material thermal conductivity. So that, the steam generated will continue its flow near the hot wall portion of the adiabatic section. This type of steam flow was not recorded or noticed before and differ from the widely known assumption that assume the flow of generated steam inside TPCT to be at the central core of the pipe.

This flow behavior includes heat transfer and phase change phenomenon between the two conjugate phases. So that volume fractions of steam and condensate keep varying and are not constant at any time or location. As shown in figure 5, the condensate volume fraction increased in the condenser section. But due to interaction with the hot steam stream its temperature increases and the volume fractions decreases while its flows downward through consequence steam streams. So that not all the condensed condensate reach the evaporator section. An opposite process happened to steam stream, steam temperature decreased then decreasing of volume fractions and not all the generated steam reaches the condensate section. So that, we can call this new observed interactions as secondary phase change phenomenon in TPCT in addition to the main ones at evaporator and condenser walls.

References
[1] Abdul G O 2009 The 3rd International Conference on Sustainable Energy and Environmental Protection, SEEP 2009: Elsevier.
[2] Hussam J and Richard M 2010 Experimental investigation of wraparound loop heat pipe heat exchanger used in energy efficient air handling units Energy 35 4592-4599.
[3] Hussam J and Hatem E 2013 Thermal performance characteristics of a wraparound loop heat pipe (WLHP) charged with R134A Energy 61 128-138.
[4] Jouhara H, Chauhan A, Nannou T, Almahmoud S, Delpech B and Wrobel L C 2017 Heat pipe based systems - Advances and applications Energy 128 729-754.
[5] Zhirong L, Chao X, Yunxiu R, Feng G, Xing J and Xiaoze D 2018 Thermal analysis of a conceptual loop heat pipe for solar central receivers Energy 158 709-718.
[6] Hirsch C 2007 Numerical Computation of Internal and External Flows: The Fundamentals of Computational Fluid Dynamics: Elsevier Science.
[7] Zikanov O 2010 *Essential Computational Fluid Dynamics* Wiley.

[8] Stéphane L, Valérie S, Frederic L, Sameer K, Jocelyn 2016 Over view of heat pipe studies during the period 2010–2015 *Interfacial Phenomena and Heat Transfer* 4.

[9] Peter D Dunn and David Reay, 2012, Heat pipes: Elsevier.

[10] David R, Ryan M and Peter K 2013 Heat pipes: theory, design and applications: Butterworth-Heinemann.

[11] Ming Z, Zhongliang L, Guoyuan M and Shuiyuan C 2009 Numerical simulation and experimental verification of a flat two-phase thermosyphon *Energy Conversion and Management* 50 1095-1100.

[12] Asghar A, Masoud R and Ammar A A 2010 CFD modeling of flow and heat transfer in a thermosyphon *International Communications in Heat and Mass Transfer* 37 312-318.

[13] Selvan A A and Velraj R 2011 Experimental investigation and CFD analysis of a air cooled condenser heat pipe *Thermal Science* 15 759-772.

[14] Zirong L, Shuangfeng W, Ryo S and Winston Z 2013 Simulation of a miniature oscillating heat pipe in bottom heating mode using CFD with unsteady modeling *International Journal of Heat and Mass Transfer* 57 642-656.

[15] Khurram K and Ali T 2014 Simulation of the response of a thermosyphon under pulsed heat input conditions *International Journal of Thermal Sciences* 80 33-40.

[16] Amatachaya P and Srimuang W 2010 Comparative heat transfer characteristics of a flat two-phase closed thermosyphon (FTPCT) and a conventional two-phase closed thermosyphon (CTPCT) *International Communications in Heat and Mass Transfer* 37 293-298.

[17] Adel A E, Saleh E N and Jalal M J 2016 Experimental and numerical investigation of thermosyphone performance in HVAC system applications *Heat and Mass Transfer* 52 2879-2893.

[18] Bandar F, Luiz C W and Hussam J 2013 Numerical modelling of the temperature distribution in a two-phase closed thermosyphon *Applied Thermal Engineering* 60 122-131.

[19] Bandar F, Luiz C W and Hussam J 2015 CFD modelling of a two-phase closed thermosyphon charged with R134a and R404a *Applied Thermal Engineering* 78 482-490.

[20] Ansys Inc. 2017 ANSYS FLUENT theory guide (Release 19). Multiphase Flows.: Ansys Inc.

[21] Ansys Inc. 2017 ANSYS FLUENT user guide (Release 19). Multiphase Flows.: ANSYS Inc.

**Acknowledgment**

Authors wishing to acknowledge the Iraqi ministry of Electricity for supporting this research in all its steps.