eISSN: 2976-1190

Journal of Engineering and Sciences

Vol. 3 Issue 1 May 30, 2024 / **DOI:** https://doi.org/10.3126/jes2.v3i1.66245

Journal homepage: Journal of Engineering and Sciences (nepjol.info)

Publisher: Research Management Cell, Pashchimanchal Campus, Lamachour, Pokhara

Computational Evaluation of Pulsating Heat Pipe for Fluid Flow Behavior and its Thermal Performance

Prakash Badu[1*], Kushal Guragain[1], Durga Bastakoti[1]

[1] *Department of Mechanical and Automobile Engineering, IOE, Pashchimanchal Campus, Tribhuvan University, Nepal*

(Manuscript Received 31/03/2024; Revised 17/05/2024; Accepted 27/05/2024)

Abstract

Computational fluid analysis of a single circuit heat pipe was undertaken to ascertain the thermal efficiency of water at various fill ratios, specifically 40%, 50%, and 60% under varying heating conditions. The analysis employed the ANSYS Fluent computational fluid dynamics software, utilizing a k-epsilon to the heat pipe as the solver. The analysis was carried out on a copper tube with an internal diameter measuring 3/16 inches. During the analysis, the thermal inflow at the adiabatic region was adjusted to zero, while the condenser region's temperature remained constant at 25°C. Meanwhile, the heat input at the evaporator region was systematically adjusted to 105°C, 110°C, 115°C, and 120°C. Key Performance characteristics of heat pipe i.e. thermal resistance, and water volume fraction were evaluated during analysis. The temperature variations noted in both regions affirmed that the pulsating pipe's performance was influenced by the phase transition between liquid and vapor. Furthermore, the concentration of volume within the evaporator section was monitored, confirming the presence of a dual-phase phenomenon within the heat pipe. Through the analysis, it was noted that the thermal resistance of the water is minimized at a 50% fill ratio for each level of heat input. Notably, the promising results were obtained at an input temperature of 120°C with a value of 1.025°C /W which was 7.3% and 9.8% lower than thermal resistance at 40% and 60% fill ratios, respectively. The reduced thermal resistance in this scenario is attributed to the flow dynamics within the capillary, propelled by rapid phase change mechanisms. This shows the importance of Pulsating Heat Pipes (PHPs) in thermal control, as well as the intricacies of their operating mechanisms. Experimental and theoretical investigations have investigated numerous elements influencing PHP performance, with numerical modeling providing insight into thermal resistance and water volume fraction dynamics under varying heating temperatures and fill ratios.

Keywords: Computational Fluid Dynamics; Single Loop Heat Pipe; Thermal resistance

1. Introduction

Energy management is currently a topic of discussion as a major concern in all facets of Research and Development (R and D) in the world. In the field of thermal management, the R and D of the eco-friendly heat transfer device of low energy consumption indeed contributes to the field of science. The phase change phenomenon of working fluid filled at a specific volume Ratio allows for the passive and incredibly efficient transfer of heat between points in a PHP. Invented and patented in the 1990s by H. Akachi, an oscillating heat pipe is made up of three main sections: evaporator, adiabatic zone, and condenser. The driving force behind the evaporation is the slug and plug motion of working fluid in the tube [1]. From then on, it was thought to have great potential for use in solar power utilization, waste thermal recovery, battery thermal management systems, electron cooling, and aircraft thermal management systems [2].

The PHP has a unique operational mechanism that leads to its functional advantages: when stable parameters are achieved, the pulsating motions of liquid plugs and vapor slugs drive the working fluid flowing in the pipe [3]. Liquid plugs and vapor slugs are formed and distributed randomly along the pipe because of the surface tension and the capillary diameter of the tube. In general, there are two types of PHPs: closed loop PHPs, which have tube ends connected in an endless loop so that working fluid is continuously circulated within the PHP throughout, and open loop PHPs, which have tube ends not connected and one long tube bent in multiple turns with both ends sealed after filling working fluid. Thus, even though PHP has a basic design, its operational mechanism is complex and difficult to completely understand because of the coupling of hydrodynamic and thermodynamic effects throughout the process of heat and mass transfer mechanism [4].

Heat pipes have been the subject of decades' worth of

---

[*] Corresponding author. Tel.: +977- 9865941833,
E-mail address: pas075bam02@wrc.edu.np

experimental and theoretical research on their intricate features. Along with operational parameters like heat flux, fill ratio, charge flux, inclination angle, working fluid, etc., geometric parameters like critical diameter, cross-sectional area, length of the evaporator, adiabatic region, and condenser, number of turns, etc. have been lucubrated for experimental modeling [3]. Numerous models have been presented for numerical modeling that describe the dynamics inside Closed Loop Pulsating Heat pipes (CLPHP). For numerical modeling, the Lagrangian technique was used, and flow was taken to be adiabatic throughout the PHP. For the vapor plugs, the pressure and velocity variation over time were examined [5]. Since surface tension and gravity oppose one other to push flow inside the tube, the capillary tube's diameter was constrained by the Bond number (Bo). The tube diameter is small enough (less than a critical value) to ensure flow oscillations, the operational basis of the single-loop heat pipe as explained by H. Akachi [6]:

$$D \le Dcr = 2\left[\frac{\sigma}{g(\rho f-\rho g)}\right]^{.5} \ldots\ldots\ldots\ldots (1)$$

Where:

D_{cr} is the critical diameter of the heat pipe.

σ is the surface tension of the working fluid.

g is the gravitational acceleration.

ρf is the density of the fluid (liquid phase).

ρg is the density of the fluid (gas phase).

By adjusting adiabatic section parameters, the ratio of the adiabatic region's length to overall length was discovered, which affected PHP's start-up parameters, heat transfer efficiency, and capacity to prevent drying out [7]. When comparing the thermal performance of PHP with varying concentrations and fill ratios of various liquids to that of deionized water, surface tension and viscosity were found to have a greater impact on thermal resistance [8]. The impact of phase transition and surface tension on the thermal properties of PHP was examined using the commercial software Fluent. The oscillating motion of water at Normal Temperature Pressure (NTP) was simulated in-house using a Finite Volume Computational Fluid Dynamics (CFD) solver, considering the filling ratio, tube size, and temperature degree [9].

The thermal performance of CLPHP's numerical modeling and simulation must be computed to further investigate the potential applications of PHP technology at different parameters. Therefore, in this study, numerical modeling is carried out at NTP to replicate temperature change and water volume fraction flow to determine the thermal resistance at various working fluid fill ratios under various heating situations.

2. Materials and Methodology

The completion of the research requires the workflow tools and the methodology for its systematic execution. The planned time frame along with the proper use of tools, techniques, and scientific theories enables the outcome of the simulation to be accurate and precise compared to the real scenarios.

2.1 Materials and tools

For performing the numerical analysis, the 3D modeling of the heat pipe and the simulation were performed on the scientific tools Solid Works and ANSYS Fluent respectively.

a) SolidWorks:

SolidWorks is a widely utilized computer-aided design (CAD) software employed by engineers, architects, and designers to craft 3D models and designs of mechanical and industrial components, structures, and various products. SolidWorks provides an array of functionalities and resources for crafting intricate designs, including parametric modeling, assembly modeling, finite element analysis, and simulation capabilities [10].

b) Ansys:

ANSYS is a simulation software suite used in engineering and product development to simulate how a product or system will behave under various real-world conditions. ANSYS stands for Analysis System, and it offers a wide range of simulation tools for structural, thermal, fluid, electromagnetic, and Multiphysics simulation [11].

2.2 Methodology

The numerical analysis is conducted in different phases for the completion of the experiment which is presented in the following flowchart as shown in Fig.1. Initially, the parameters, operation, and performance of heat pipes are studied. The numerical modeling and analysis are conducted using the CFD software ANSYS Fluent. Simulations are performed to investigate the thermal resistance by varying the mass fractions of the water at normal atmospheric conditions, focusing on volume fraction and temperature differences.

To run simulations and examine thermal performance, Pulsating Heat Pipes (PHP) CFD analysis requires looking over and assessing the physical data that is presented. PHP's performance in different heat conditions is examined using the commercial software

Ansys Fluent.

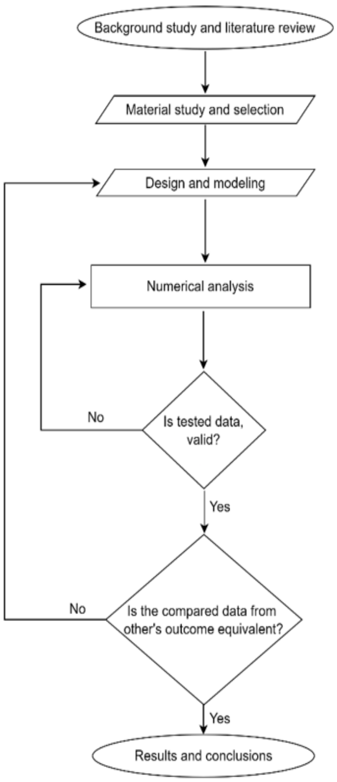


Figure: 1 Flow-chart of the research methodology

2.2.1 Geometry

The research geometry, as depicted in Figure 2, is established based on understandings gathered from the literature review. The flow parameter and its thermal analysis of the pulsing heat pipe are the main subjects of the analytical simulation setup. The Bond number equation is utilized to determine the pipe's critical diameter [6].

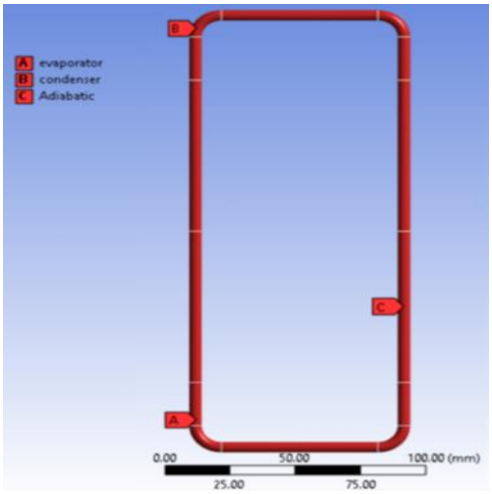


Figure: 2 Geometry of the heat-pipe

All pertinent zones of the pulsing heat pipe, such as the adiabatic region, condenser, and evaporator, are shown in Fig. 2. The size of the condenser and evaporator are the same. It is found that the adiabatic and its effective length ratio is 7:10 [7].

2.2.2 Meshing

ANSYS meshing provides the scholar with extra control by allowing them to select combinations of point, edge, surface, or body controls in addition to the optimal meshing condition parameters that are set by default. The mesh can be affected in several ways by these options, each of which has its options. As seen in Figure 3, the manual body sizing method for the mesh is set at 0.02 cm. With body mesh sizing as shown above, the mesh is generated with 2,33,225 nodes and 1763721 elements. Defining the domain and name of the section can be easily defined in the meshing section which is utilized while giving the boundary conditions in the setup section in ANSYS Fluent.

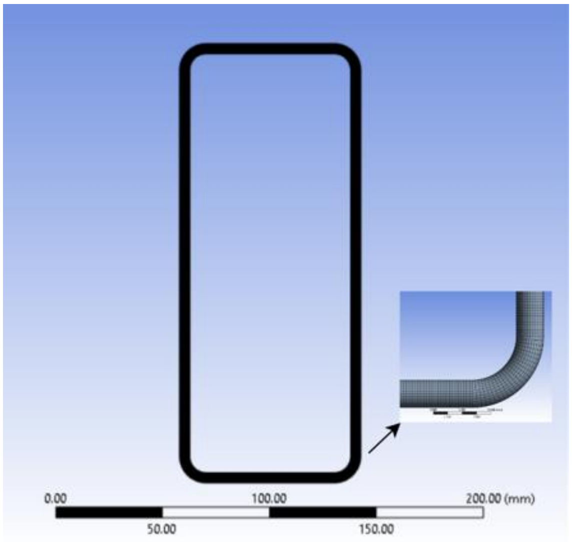


Figure:3 Meshing in ANSYS fluent

2.2.3 Analysis Setting

In the thermal analysis of PHP, transient cases are predominantly emphasized, while steady-state analysis holds less significance. Data obtained from literature surveys are often regarded as constant for steady-state simulations. Since these variables are set to zero and do not affect the steady-state outcome, many cross-terms and higher-order terms relating to time variation are ignored in steady-state simulation. Transient simulation, on the other hand, includes all time-varying variables and all higher-order concepts related to time [12]. Different time steps, running time step durations, and adaptive algorithms were among the simulation methodologies that were first investigated. Subsequently, the time step option was adjusted to meet the necessary simulation requirements. Due to the mesh complexity and fluid flow velocity within the domain, the simulation necessitates a smaller time step size. ANSYS Fluent employs its solver to iterate through the calculations, as all parameters, geometry, and meshing were integrated within the same file, without the need for an external solver coupling.

2.2.4 Fundamental configuration for the simulation

The condenser, adiabatic section, and evaporator comprised the three main domains that made up the entire geometry of the PHP. 40%, 50%, and 60% ratios were set up in the PHP to account for different thermal conditions.

2.2.5 Boundary Conditions

For the simulation's boundary settings, varied temperatures ranging from 105°C to 120°C were applied to the region. The Adiabatic Region was assigned a heat flux of zero. The VOF model was utilized for CFD analysis throughout all domains. Walls in all regions were defined as no-slip wall types and expected to have an even surface. Throughout the condenser, a fixed temperature boundary condition of 20°C was set. Similarly, the Adiabatic regions were assigned an adiabatic boundary condition, resulting in a heat flux rate of 0 W/m2K. Additionally, a contact angle of 30° with walls was specified.

2.2.6 The governing equations of the Volume of Fluid (VOF) approach

Modeling two-phase movement within PHP entails plug flow within the heat pipe and follows the standard procedure. This method adopts a single-fluid approach based on the VOF technique. Hydrodynamics has been defined by models that regulate mass, momentum, and energy conservation as well as an additional advection equation. This advection equation plays a significant part in determining the gas-liquid interface, as represented by the VOF [8]. The following are the simulation's governing equation presented below:

Mass conservation:

$\nabla . u = 0$……………………………..……..…(2)

Momentum Conservation:

$\delta(\rho . u)\delta t+\nabla .(u.\delta u)=\nabla p+\nabla .[\mu(\nabla . u+(\nabla u)T)]+f\sigma$….(3)

Energy Conservation:

$(\rho cT)\delta t+\nabla .(\rho cuT)=\nabla .(kVT)$……………..….(4)

The force resulting from surface tension is represented by the quantity fσ in the momentum equation, which also includes density (ρ), velocity (u), pressure (p), dynamic viscosity (μ), specific heat capacity (c), temperature (T), and thermal conductivity (k). Each phase's equations are addressed independently, and the interface is drawn using the volume of Fluid (VOF) technique [12].

The VOF model is a surface-tracking technique that is applied in a static Eulerian mesh environment. Its use is specific to situations in which there are two or more immiscible fluids, with an emphasis on precisely defining their interface.

A uniform system of momentum equations controls the behavior of all the fluids in the VOF model, which also closely monitors the volume fraction of each fluid in each of the individual computational cells over the whole domain [13].

Density-based solvers cannot be used with the Volume of Fluid (VOF) modeling approach, which makes it impossible to describe empty regions without fluid. Nevertheless, using the implicit time-stepping mechanism of the second-order formulation conflicts with the explicit VOF scheme. Using the VOF model makes it impossible to simulate continuous, circular motion in the streamwise direction, either for a given mass flow rate or pressure drop. Furthermore, the solution to stabilize the initial conditions is required to achieve meaningful steady-state VOF computation [11].

2.2.7 Solver Model

The use of the k-epsilon model, a well-known viscous model, is an important part of this approach. This model is widely used in several generic CFD systems and is considered an industry standard. Its reputation is based on its proven stability, numerical resilience, and established predictive powers. The k-epsilon model achieves the best balance of accuracy and precision, especially in generic simulation scenarios. The k-epsilon model in Fluent uses well-established two-equation models [14]. The curvature correction option is selected and set to a fixed value, while the turbulent viscosity is turned off. The viscous model's parameter values are all set to the default values specified by Fluent.

2.2.8 Initialization

The simulation's pressure is set to 1 atm, the starting temperature is set to 95°C to hasten the boiling of the water at NTP, and velocity components are set to 0 m/s. The simulation's operating temperature is 20°C. The water volume is set to 1 and the vapor volume to 0, and the corresponding regions are patched with the water and vapor volumes.

2.2.9 Grid Convergence Test

The aim of the grid convergence test (GIT) is to identify the ideal grid parameters with the fewest number of elements while maintaining numerical results. By evaluating various grid conditions and observing parameter changes with element size adjustments, GIT is performed by selecting an element size showing convergence with an increased element number. This chosen size, balancing large elements and low numbers, reduces computational costs and time.

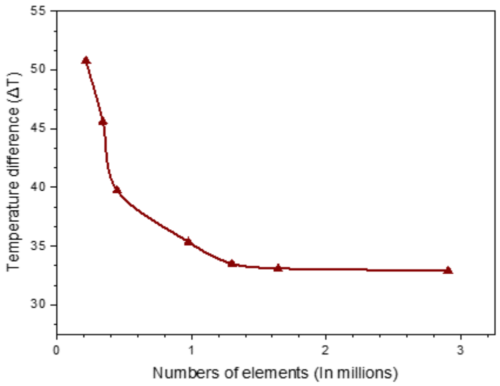


Figure: 4 Grid independent test (GIT) of the heat pipe simulation

GIT serves to verify and validate numerical analysis meshes, ensuring their accuracy and efficiency. As observed in Fig.4, no noteworthy change in results was found at 0.015 cm, 0.018 cm, 0.019 cm, and 0.02 cm of the mesh size. Henceforth, element size is preferred to 0.02 cm with 1763721 number of elements.

3. Results and Discussion

The outcome of the study project exhibits meticulous validation and verification of gathered data, confirming the findings' integrity. Using numerical analysis performed in the Ansys Fluent, a detailed examination of various heating settings is carried out to elucidate phenomena such as fluid flow patterns, oscillatory behavior of heat pipes, and phase transition of the working fluid. Notably, transient analysis was performed for fluid flow simulations of data throughout time steps, allowing for thorough observation and investigation of the fluid dynamics in the PHP.

3.1 Phase Change

Changes in fluid volume across different sections of the heat pipe show how the working fluid shifts between phases. The movement from water to steam and back again is clear. The fluid's oscillation changes the pipe's temperature in different areas. Heat is naturally transferred to the condenser and added to the evaporator. This causes the evaporator temperature to drop and the condenser to rise, confirming efficient heat exchange in the heat pipe.

The oscillating flow in the PHP causes temperature fluctuations. The temperature of the evaporator decreases while the condenser temperature increases as heat is transferred from the evaporator to the condenser by water vapor oscillation. The phase change of the water in the looped pipe is shown in Figure 5.

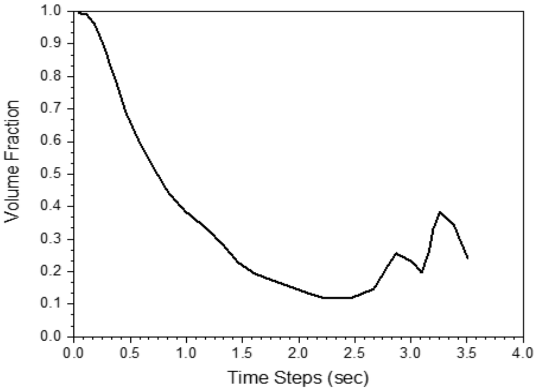


Figure: 5 Variation of water volume fraction at evaporator with a time step (s)

Initially, water is present in the evaporator region, indicated as 1 in terms of water volume fraction. Gradually, this volume fraction decreases, indicating the start of the phase change process with initial heating. After 3 seconds, the volume fraction of water increases, demonstrating the condensation of steam back into water and confirming the phase change within the PHP.

Phase change inside the PHP generates fluid oscillation, which causes temperature changes in various pipe sections. Heat transfer from the evaporator to the condenser, which reduces the evaporator temperature while raising the condenser temperature, is what causes these oscillations. This dynamic temperature

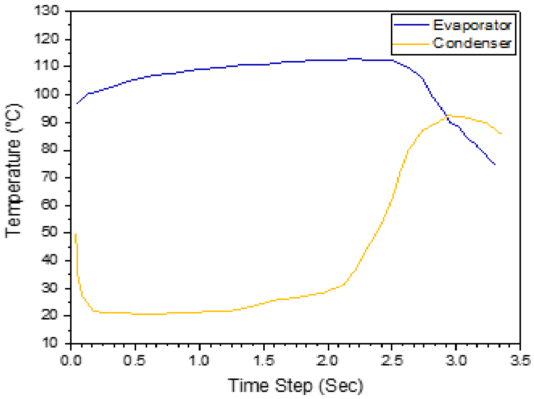
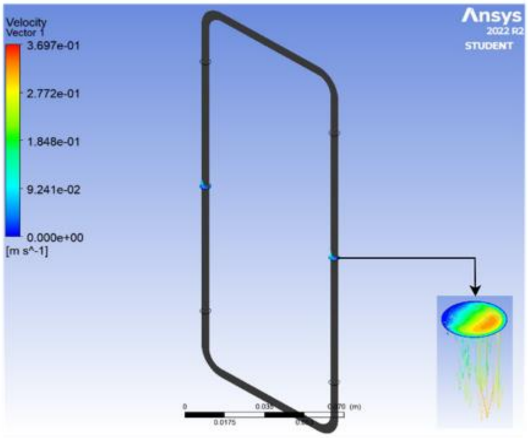


Figure: 6 Temperature fluctuation inside heat pipe

shift demonstrates heat transfer efficiency assisted by oscillating water vapor, hence verifying essential thermodynamic processes in the system.
Figure: 6 depicts temperature fluctuations in evaporator and condenser areas, demonstrating the heat exchange mechanism in a single-loop heat pipe.

3.2 Velocity

To characterize the working fluid inside PHP as pulsating nature, the fluid velocity must exhibit bidirectional movement along the vertical axis within a precise time frame at a point taken for the analysis. Visualization of vapor flow within the simulation methodically analyzes the velocity vector of the working fluid, offering insights into its dynamic behavior.
Remarkably, the section area through which the velocity results are obtained remains consistent across all data points of the velocity vector, providing a standardized basis for observing and interpreting the pulsating nature of fluid flow. In an analysis of heat pipes, a plane is strategically positioned to intersect the pipe at two distinct points, creating two cross-sectional areas as illustrated by Fig.7 and Fig.8.



F -

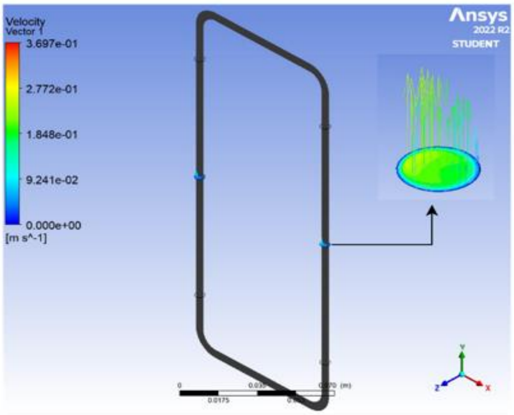


Figure: 8 Velocity contour of fluid velocity in positive y-axis

By focusing on this defined section, fluid velocity can be precisely monitored, and any bidirectional movement along the y-axis, crucial for characterizing the pulsating nature of fluid flow within heat pipe. This approach ensures targeted observation and a detailed understanding of the dynamic behavior of the working fluid.

3.2 Validation of the results obtained

The simulations unveiled a decrease in thermal resistance with increasing temperature inputs. Remarkably, thermal resistance was lowest at a 50% fill ratio across all temperatures, with the lowest recorded at 1.025 K/W at 120°C, highlighting its significance amidst varying fill ratios as shown in Figure 9.

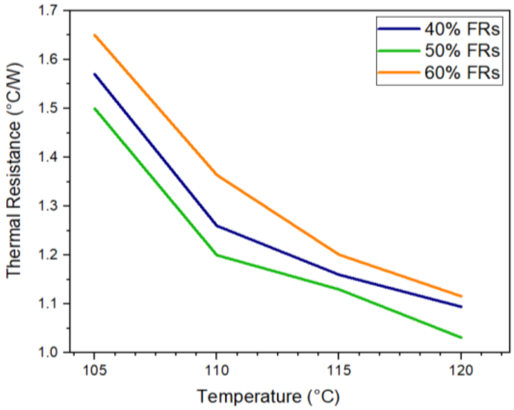


Figure: 9 Change in R_{th} with varing temperature inputs at different fill ratios

The efficiency of the PHP system is determined by thermal resistance which is a difference in temperature in two different regions and the corresponding heating power. Hence:

$$R = \frac{Te-Tc}{Q} \text{……….............}(5)$$

It is compared to more recent tests carried out by other researchers to confirm the findings. The experimental data from D. Baitule and P. Pachghare show a thermal resistance of about 45W at 120°C for a 50% FR, however, Guragain K. et al. discovered a resistance of 50W at the same temperature [15], [16]. This is very similar to the simulation results which are presented in Table 1.

Table 1 Comparison of errors in the reference and experimental data.

Fill Ratios	Obtained R_{th}	D. Baitule, et.al data (R_{th})	Guragain K, et.al
50%	1.025	0.82	1.01

Additionally, the volume fraction graphs Fig. 10 from a recent study by Guragain K, et al., which depict similar characteristics of fluid flowing within a heat pipe in a specific heating section (i.e. evaporator region), verify the obtained pulsating nature of the working fluid in the oscillating heat-pipe [17].

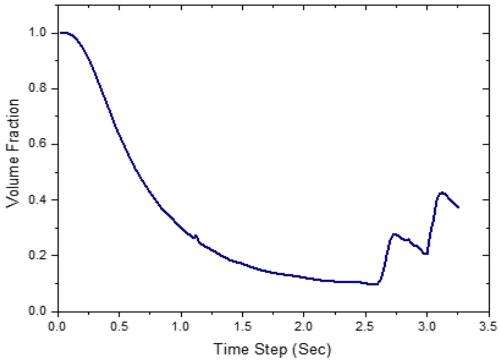


Figure: 10 Deviation of water volume fraction by K. Guragain, P. Badu, and D. Bastakoti.

4. Conclusions

Comprehensive results on the thermal resistance of oscillating heat-pipe varying fill ratios (40%, 50%, and 60%) at different heating conditions (105°C, 110°C, 115°C, and 120° C) were obtained. Simulation has provided further evidence of the heat-transferring mechanism of single-loop heat pipe through the study of temperature obtained from the computational analysis. The data from this investigation suggests that working fluid's phase change mechanism occurred within heat pipe during the period. Additionally, the complex operational mechanisms and diverse applications of pulsating heat pipes highlight the need for further exploration through numerical modeling and simulation for further understanding of thermal behavior. The comprehensive finding suggests future research could focus on optimizing fill ratios, increasing the number of turns, and heating conditions to enhance heat transfer efficiency in pulsating heat pipes.

Acknowledgment

We are truly grateful to the Pashchimanchal Campus Department of Mechanical and Automobile Engineering for enabling us to take part in a project that will greatly enhance our research abilities on a subject that we are extremely passionate about.

References

[1] Venkata, S. J. and Bhramara, P. CFD analysis of multi-turn pulsating heat pipe, *Materials Today: Proceedings*, Elsevier Ltd, (2017) 2701–2710.

[2] Khandekar, P., Panigrahi K., Lefèvre F. and Bonjour J. Local hydrodynamics of flow in a pulsating heat pipe: A review, *Frontiers in Heat Pipes*, 1(2) (2010) 1-20.

[3] Han, X., Wang, X., Zheng, H., Xu, X. and Chen, G. Review of the development of pulsating heat pipe for heat dissipation, *Renewable and Sustainable Energy Reviews*, 59 (2016) 692–709.

[4] Pouryoussefi, S. M. and Zhang, Y. Numerical investigation of chaotic flow in a 2D closed-loop pulsating heat pipe, *Appl Therm Eng*, 98 (2016) 617–627.

[5] Sakulchangsatjatai, P., Terdtoon, P., Wongratanaphisan, T., Kamonpet, P. and Murakami, M. Operation modeling of closed-end and closed-loop oscillating heat pipes at normal operating condition, *Appl Therm Eng*, 24(7) (2004) 995–1008.

[6] Akachi H. Structure of a Heat Pipe, *U.S. Patent 4921041*, (1990).

[7] Li, Q., Wang, C., Wang, Y., Wang, Z., Li, H. and Lian, C. Study on the effect of the adiabatic section parameters on the performance of pulsating heat pipes, *Appl Therm Eng*, 180 (2020) 115813.

[8] Bastakoti, D., Zhang, H., Cai, W. and Li, F. An experimental investigation of thermal performance of pulsating heat pipe with alcohols and surfactant solutions, *Int J Heat Mass Transfer*, 117 (2018) 1032–1040.

[9] Sharma, M. and Sikarwar, B. S. Flow and heat transfer of fluid in a pulsating heat pipe, *Journal of Physics: Conference Series, Institute of Physics Publishing*, (2019) 1-6.

[10]Dassault. D/S SolidWorks. (1993), see http://www.solidworks.com (last accessed March 4, 2023).

[11] Ansys. Ansys. (1998), see http://www.ansys.com (last accessed March 4, 2023).

[12] Venkata, S. J., and Bhramara, P. CFD Analysis of Copper Closed Loop Pulsating Heat pipe, *Materials Today: Proceedings*, Elsevier Ltd, (2018) 5487–5495.

[13] Eghbalzadeh, A., and Javan, M. Comparison of Mixture and VOF models for numerical simulation of Air-Entrainment in skimming flow over stepped spillways, *Procedia Engineering*, (2012) 657–660.

[14] Al-Witry, A., and Es-Saheb, M. Benchmarking the performance of the Ansys-fluent standard k-ε turbulence model in fluid flow and heat transfer predictions for complex flows around circular pin-fins using various near wall functions, *Research Journal of Applied Sciences, Engineering, and Technology*, 5(17) (2013) 4301–4310.

[15] Baitule, D. A., and Pachghare, P. R. Experimental analysis of closed loop pulsating heat pipe with variable filling ratio, *International Journal of Mechanical Engineering and Robotics Research*, 2(2) (2013) 113-121.

[16] Guragain, K., Badu, P., Chettri, S., Baral, K., Bastakoti, D. and Bhattarai, R. Evaluation of thermal performance in pulsating heat pipe cooling system, *Journal of Innovations in Engineering Education*, 6 (2023) 96-104.

[17] Guragain, K., Badu, P. and Bastakoti, D. Computational Analysis of Single-Loop Heat Pipe for Predicting Thermal

Resistance, *Conference on Recent Trends of Science, Technology, and Innovation (RTSTI)-2023,* Pokhara, Nepal (2023).