DOI: 10.20535/2521-1943.2025.9.2(105).328619

UDK 536.24:532.517.4

Computational Investigation: CFD guide for the Evaporation Process in Heat Pipe

Dalal Ahmed[1] • Raid Mahmood[1]

Received: 1 May 2025 / Revised: 22 May 2025 / Accepted: 30 May 2025

Abstract: *Heat pipes are essential components in various industrial applications due to their exceptional heat transfer capabilities, which significantly enhance thermal management systems. Their ability to efficiently dissipate heat with minimal temperature gradients makes them invaluable in electronics cooling, aerospace systems, and energy recovery processes. This study presents a detailed two-dimensional Computational Fluid Dynamics (CFD) simulation to analyze the temperature distribution and phase change dynamics during evaporation within a heat pipe. The simulation is conducted using ANSYS Fluent, where a 2D model of the heat pipe is developed, and an optimized computational mesh is generated to ensure accuracy in the numerical results. The k-ε turbulence model is employed to accurately capture the fluid flow behavior, accounting for the complex interactions between vapor and liquid phases. The working fluid selected for this investigation is a nanorefrigerant (Al2O3/R11), chosen for its enhanced thermal properties that contribute to improved heat transfer efficiency. The simulation results reveal a significant temperature gradient in the evaporator section, highlighting the critical role of heat flux in determining thermal resistance. Additionally, the study examines how variations in heat input influence the overall thermal performance of the heat pipe. The findings from this CFD analysis provide valuable insights into the evaporation process, which is driven by phase change phenomena, and offer practical design guidelines for optimizing heat pipe performance. By understanding the intricate relationship between fluid dynamics, heat transfer, and material properties, this research significantly contributes to the development of more efficient and sustainable thermal management solutions for industrial applications.*

Keywords: *Heat Pipes, Computational Fluid Dynamics CFD, Evaporation Process, ANSYS Fluent, Temperature Distribution.*

Introduction

Heat pipes are effective thermal transfer mechanisms that employ phase change and capillary action to transport heat with minimal temperature differences, hence improving thermal management across various applications. Heat pipes easily transfer significant amounts of heat over long distances without moving parts, making them vital in modern thermal management systems. Heat pipes are applicable in various domains, including electronics cooling to avert overheating in high-performance processors and power electronics, thermal management of spacecraft to maintain temperature in extreme space conditions, and energy systems to improve efficiency in renewable technologies such as solar panels and geothermal heat exchangers [1]. The working principle of heat consists of a closed-loop mechanism wherein a working fluid experiences constant phase transitions between liquid and vapor states, hence enabling very efficient heat transfer. The heat absorbed in the evaporator part causes the liquid to evaporate, and the resulting vapor moves to the condenser section, where it releases latent heat and undergoes condensation to form a liquid. The condensed liquid subsequently flows to the evaporator by capillary forces inside the wick structure, facilitating continuous circulation without necessitating an external power source [2]. CFD has emerged as an essential instrument for simulating a broad spectrum of thermodynamic processes, phase transitions, heat transport, and flow dynamics. [3], [4]. CFD has become an effective instrument for the analysis and optimization of heat transfer systems, especially in heat pipe technology. Thermal conduits Understanding the thermal and fluid dynamics of a heat pipe is crucial for enhancing its performance, reducing

✉ *D.K. Ahmed*
dalalkhalid329@gmail.com

[1] *College of Engineering, University of Zakho, Kurdistan region, Iraq*

thermal resistance, and improving overall system efficiency [1]. A schematic representation of the heat pipe is shown in Fig. 1.

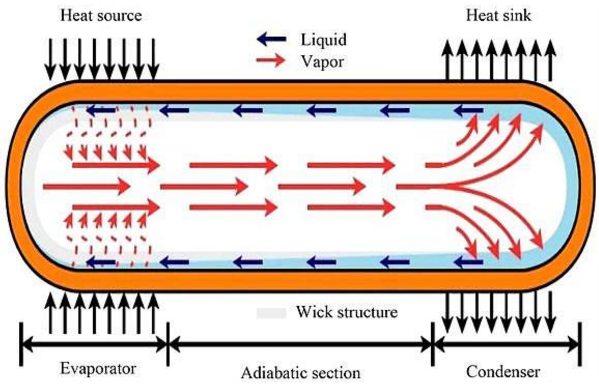

Fig. 1. Schematic diagram of the heat pipe [5]

CFD simulations provide detailed insights into the complex phase-change phenomena in heat pipes, including evaporation, condensation, and liquid-vapor interactions. Researchers can analyze temperature distribution, pressure variations, and the effects of different working fluids and wick designs on heat transfer efficiency by numerical modeling. In contrast to experimental methods, CFD provides a cost-effective and time-efficient means of evaluating many design parameters across diverse operating circumstances [1], [6]. CFD models of heat pipes entail resolving the essential governing equations of fluid dynamics, thermal transmission, and phase transition processes. This encompasses the continuity equation and the Navier-Stokes equations for fluid dynamics, the energy equation for thermal transport, and supplementary models to address phase change mechanisms such as evaporation and condensation. The numerical model of the heat pipe is predicated on the essential governing equations. These include:

Continuity equation:

$$\frac{\partial}{\partial t}\rho + \nabla.\rho u = 0 \tag{1}$$

Navier–Stokes equation:

$$\left(\frac{\partial}{\partial t}\rho u + \nabla.\rho u\right) = \nabla.\tau + \sum F\ . \tag{2}$$

Energy Equation:

$$\left(\frac{\partial}{\partial t}h + u.\nabla h\right) = \rho q + \nabla.\left(\frac{\lambda}{cp}\nabla h\right). \tag{3}$$

Given that heat pipes function through liquid-vapor interactions, precise modeling necessitates the application of multiphase flow methodologies to accurately represent the intricate dynamics between these phases. CFD investigations generally utilize multiphase models, such the Volume of Fluid (VOF) method, which monitors the interface between liquid and vapor phases, or the Eulerian approach, which considers both phases as interpenetrating continua with exchange terms regulating phase transitions. The choice of the suitable multiphase model is contingent upon the requisite level of detail and the available computational resources. The wick structure, essential for capillary-driven liquid return, is represented with porous media formulations [7], [8]. Recently, many studies have focused on optimizing the evaporation process in heat pipes to enhance thermal efficiency and achieve optimal operating conditions. Zuo & Faghri [9] formulated a network model to analyze heat pipe transients employing thermal resistances and a working fluid cycle. Their model, predicated on first-order linear differential equations, elucidated essential heat transmission mechanisms, underscoring the importance of wall and wick conduction. The research employed the Runge-Kutta method to forecast heat flux and temperature gradients, illustrating that vapor flow exhibited little thermal resistance. De Schepper et al. [10] analyzed flow boiling in a hydrocarbon feedstock within a convection section heat exchanger utilizing a three-dimensional evaporation model included into a computational fluid dynamics framework. The VOF method, combined with the Piecewise Linear Interface Calculation (PLIC) approach, was utilized to simulate two-phase flow, while mass and energy transfer were modeled using the Hertz-Knudsen equation. Although their model accurately represented flow regimes and phase transition processes, they emphasized the necessity for enhancements in temperature-dependent physical property fluctuations and heat flux profiling.

Xie et al. [11] conducted an experimental and CFD-based analysis comparing a revolving heat pipe to a pipe devoid of working fluid. Their findings indicated that the revolving heat pipe demonstrated markedly enhanced heat transfer, with a considerably reduced temperature differential between the evaporator and condenser portions, underscoring the efficacy of the working fluid circulation. Höhne [6] utilized CFD simulations to study two-phase flow and heat transfer in a heat pipe, focusing on evaporation, condensation, and phase change processes. The study provided insights into vapor movement, temperature stabilization, and the impact of heating power variations on evaporation intensity and thermal performance. Maghrabie et al. [12] analyzed various heat pipe topologies, including conventional, loop, and pulsating/oscillating heat pipes, assessing characteristics such as wick structure, inclination angle, filling ratio, and grooves. Their research highlighted the significance of numerical simulations in enhancing heat pipe efficacy for use in electronics cooling, HVAC systems, nuclear reactors, and renewable energy applications. They emphasized the necessity for additional study on the integration of heat pipes with phase change materials and nanofluids, especially in cryogenic applications. Annamalai & Ramalingam [13]. Conducted experimental and numerical analysis of heat pipe temperature distribution under varying heat flux circumstances. Their research indicated that elevated heat flux resulted in increased steady-state temperatures, while air cooling in the condenser constrained performance. They suggested implementing water cooling or adding fins to improve efficiency, with CFD and experimental findings closely correlating. These works collectively offer insights into thermodynamic processes, fluid dynamics, and design enhancements for improving heat pipe efficiency across diverse applications.

CFD Modeling

Geometry Generation

In the current project, the design module of ANSYS Workbench has been used to create the computational domain for the heat pipe, based on the experimental test section dimensions of D = 10 mm and L = 500 mm. The heat pipe geometry was modeled in two dimensions, ensuring accurate representation of the physical characteristics essential for thermal analysis. The symmetry plane was employed to reduce computational complexity while maintaining accuracy. The concept incorporates a cylindrical heat pipe configuration with a uniform diameter, enabling efficient heat transfer simulations. The boundary conditions and material properties were established using experimental data to ensure a precise evaluation of thermal performance. Fig. 2 illustrates the developed geometry used for simulation in ANSYS Workbench.

Mesh Generation

In CFD simulations, the mesh has been generated with full attention to obtain an optimum mesh quality. The physical domain of the geometry has been partitioned into small finite parts. The mesh constitutes the basis for resolving the governing equations of fluid dynamics, thermal transfer, and phase transition events, which are crucial for precisely predicting the efficacy of heat pipes [14]. Heat pipes are exceptionally effective heat transfer mechanisms that depend on the evaporation and condensation of a working fluid, rendering their simulation intrinsically intricate. An adequately designed mesh guarantees precise representation of the complex thermal and fluid dynamics within the heat pipe [15]. The meshes utilized for the 2D heat pipe simulations were produced using structured elements. A structured grid is frequently favored in 2D heat pipe models for its capacity to deliver enhanced numerical accuracy and stability, as stated in the Fluent user guide [16] accurate solutions and fast convergence are the results of achieving high mesh quality.

The mesh was produced at minimal computational cost. A superior mesh was achieved through the refinement of global and local sizing parameters following several rounds.

Mesh quality is generally assessed using three key parameters: orthogonal quality, skewness, and aspect ratio [17]. The orthogonal quality metric ranges from 0 to 1, with values approaching 1 indicating higher mesh quality. The minimal orthogonal quality should not be less than 0.01 to ensure numerical stability. Furthermore, skewness, which inversely correlates with solution accuracy, must be avoided to reduce numerical errors [18]. The aspect ratio, which influences wall function behavior, must be kept low to accurately capture flow characteristics near the wall [19]. In the present study. The orthogonality was 0.99, the aspect ratio was 2.22, and the skewness was 0.5. These numbers, based on established standards, signify a well-structured and high-quality mesh, hence ensuring the reliability and correctness of the CFD simulation results.

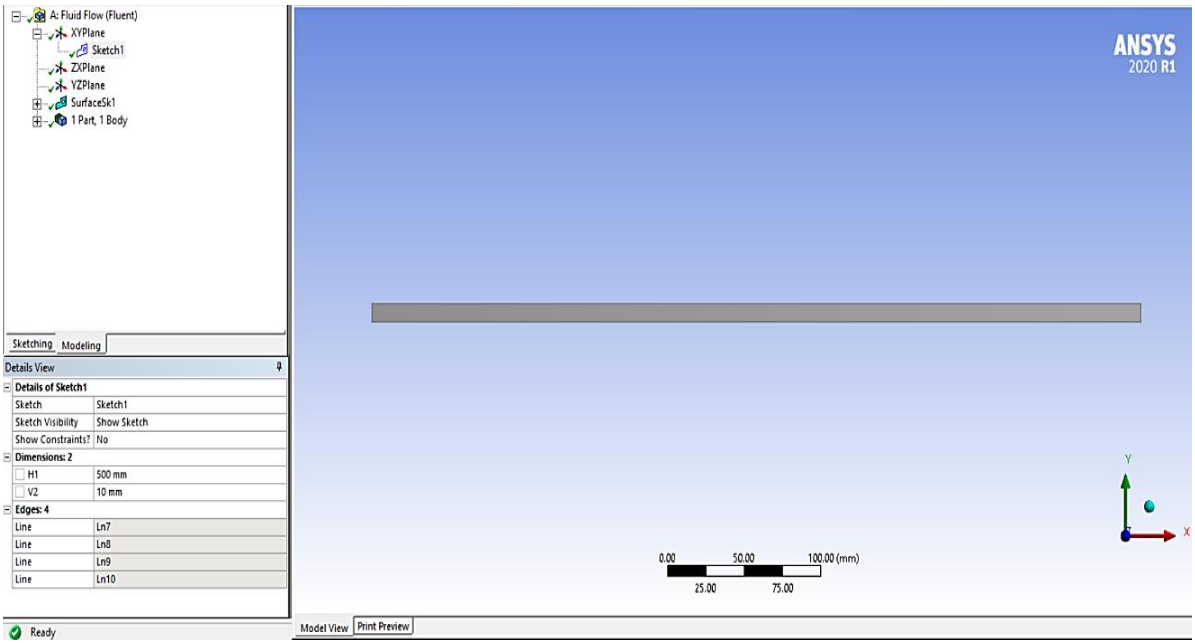

Fig. 2. Computational domains of the 2D heat pipe geometry

Mesh Independence Study

The geometry of the 2D heat pipe was Discretized into quadrilateral elements, employing the inflation approach adjacent to the walls to accurately capture thermal boundary layer effects. Fig. 3 illustrates the mesh of the 2D heat pipe. To assess mesh independence, four computation grids of 336, 1097, 13741, and 15000 nodes were used. Fig. 4 illustrates the temperature distribution within the heat pipe, with the red area indicating the highest temperature (like the evaporator), the blue area denoting the lowest temperature (like the condenser), and the intermediate colors illustrating the temperature gradient along the pipe. The influence of mesh density on the clarity of the temperature distribution is apparent. The 336-node mesh exhibits considerable numerical diffusion, resulting in a blurred temperature profile. The 1097-node mesh enhances resolution yet continues to display blocky temperature contours. The 13741-node mesh offers the most precise temperature distribution, featuring smooth curves and pronounced gradients. A 15,000-node mesh was evaluated and yielded identical results; however, it was excluded from the final analysis because of processing limitations. Further refinement is recommended to ensure that the results remain consistent with increased mesh density, hence establishing complete mesh independence.

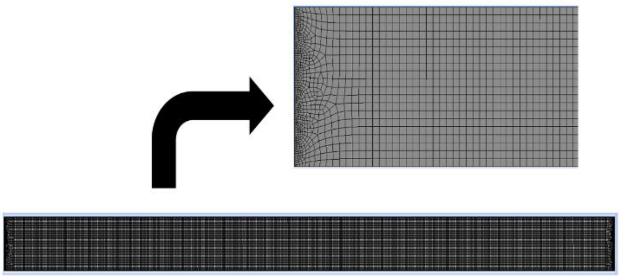

Fig. 3. 2D heat pipe mesh for the 13741-node mesh case

Boundary Conditions

To effectively simulate and resolve the numerical equations, suitable boundary conditions must be implemented. This simulation exclusively considers the fluid portion of the heat pipe, applying boundary conditions especially to the working fluid without modeling the solid walls. The working fluid employed was nano-refrigerant Al2O3/R11. A uniform flow condition is implemented at the intake, guaranteeing a consistent velocity profile for the working fluid entering the domain. This assumption streamlines the simulation and preserves uniformity in the flow characteristics. The walls of the heat pipe are designated with a stationary no-slip boundary condition, which mandates zero velocity at the solid-fluid interface. A pressure outlet boundary condition is established at the outlet, permitting the fluid to escape the domain without backflow interference.

CFD Solution Setup

A steady-state simulation is performed with a pressure-based solver with double precision to improve numerical accuracy. Fig. 5 illustrates the set-up window of the Fluent launcher. The double precision solution takes longer than the single precision method [20]. The effects of gravity are accounted for using a value of –9.81 m/s² in the *Y*-direction to address buoyancy-driven flow. The energy equation enables the modeling of heat transfer in the working fluid, while turbulence effects are depicted using the k-ε model, ensuring precise predictions for thermal and velocity distributions. The working fluid is Al2O3/R11, with thermophysical properties from [21], [22]. Phase change is excluded if the working fluid remains in a single-phase state. The numerical model employs the SIMPLE algorithm for pressure-velocity coupling, ensuring reliable convergence in steady-state settings. A second-order upwind

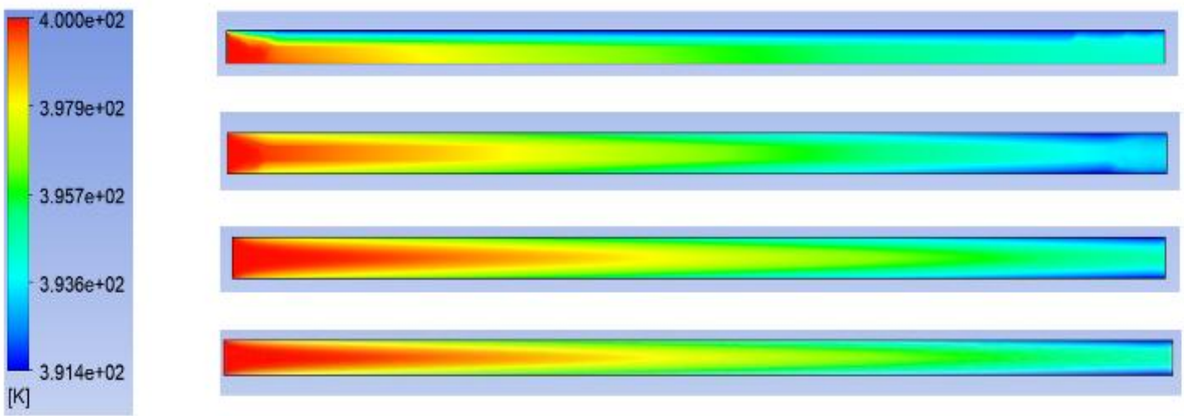

Fig. 4. Temperature Distribution in a 2D Heat Pipe for Different Mesh Densities (336, 1097, 13741 and 15000 Nodes)

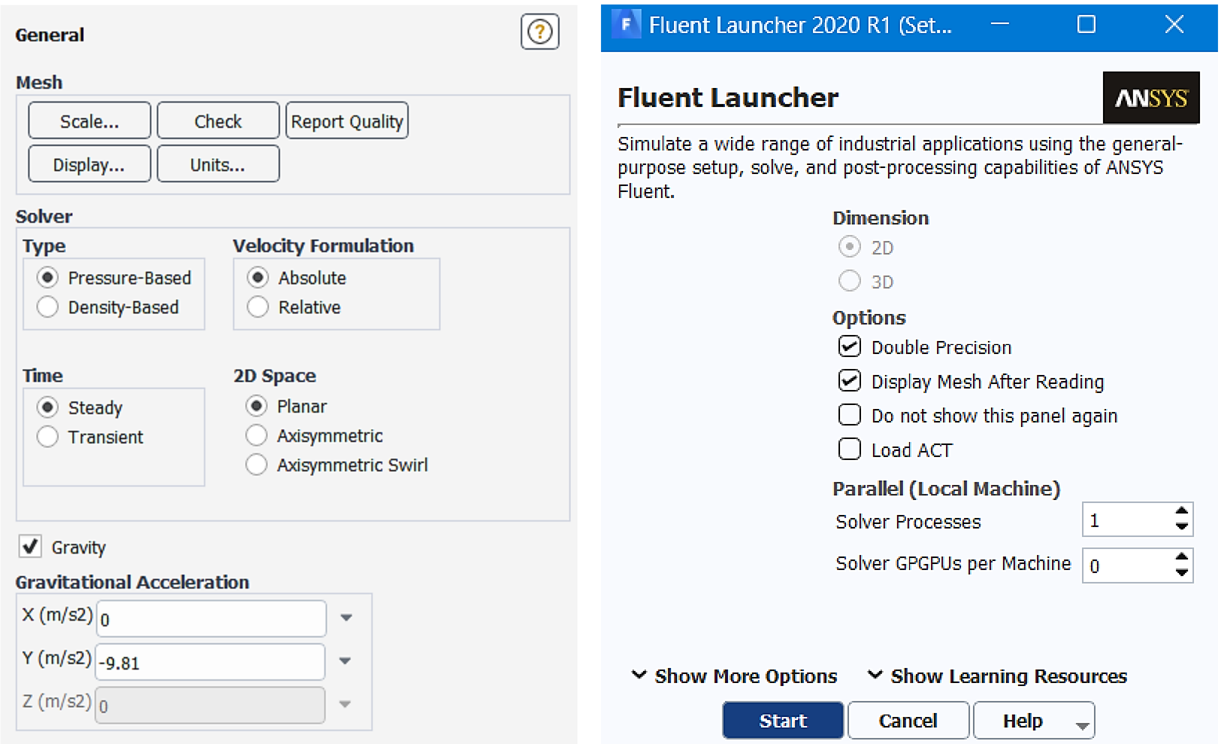

Fig. 5. ANSYS Fluent Launcher Setup

method is employed for the momentum, energy, and turbulence equations to minimize numerical diffusion and enhance solution precision. The simulation is considered converged when the residuals for the energy equation reach 10^{-6}, while the momentum and turbulence equations are monitored until their residuals drop below 10^{-3}. Additionally, temperature and velocity measurements at pivotal places are observed to confirm solution stability. This arrangement ensures an accurate representation of heat transfer and fluid dynamics inside the fluid domain of the heat pipe.

Results and Discussion

This section presents and discusses the key results obtained from CFD simulations of a 2D heat pipe, focusing on convergence history, velocity profiles, and temperature distribution. Fig. 6 illustrates a convergence plot derived from a numerical simulation utilizing CFD. The *y*-axis (logarithmic scale) represents the residuals of several governing equations.

(Continuity, *x* – velocity, *y* – velocity, energy, *k* – epsilon). The *x*-axis represents the quantity of iterations performed throughout the simulation. Each line on the graph denotes the residual of a certain equation. The aim of a numerical simulation is to reduce these residuals to a minimal threshold, indicating that the solution is approaching a stable and accurate result. Ideally, all lines should have a descending trend as the number of iterations increases, signifying that the simulation is converging. Lower residual values indicate a more effective solution in satisfying the governing equations. In this context, most residuals diminish with iterations, however the rate of drop differs among various equations.

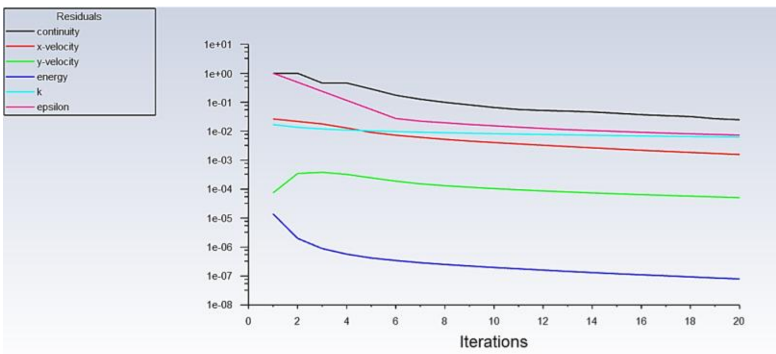

Fig. 6. Residual Convergence Plot

Fig. 7 illustrates a 2D velocity profile within a heat pipe, Shows fluid motion. Blue signifies the lowest velocities, while red denotes the highest. The concentration of red along the center axis signifies a region of peak velocity, which decreases towards the periphery, as evidenced by the color gradient to green and blue, indicating that the principal fluid flow is centralized, while boundary layer effects decelerate the fluid adjacent to the walls. Fig. 8 shows the velocity magnitude profile within the fluid region of a 2D domain,

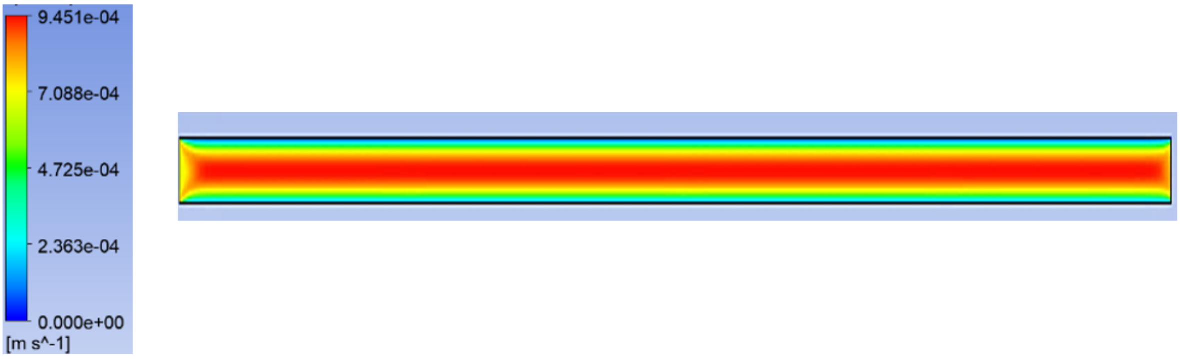

Fig. 7. Velocity Magnitude Distribution in 2D Heat Pipe

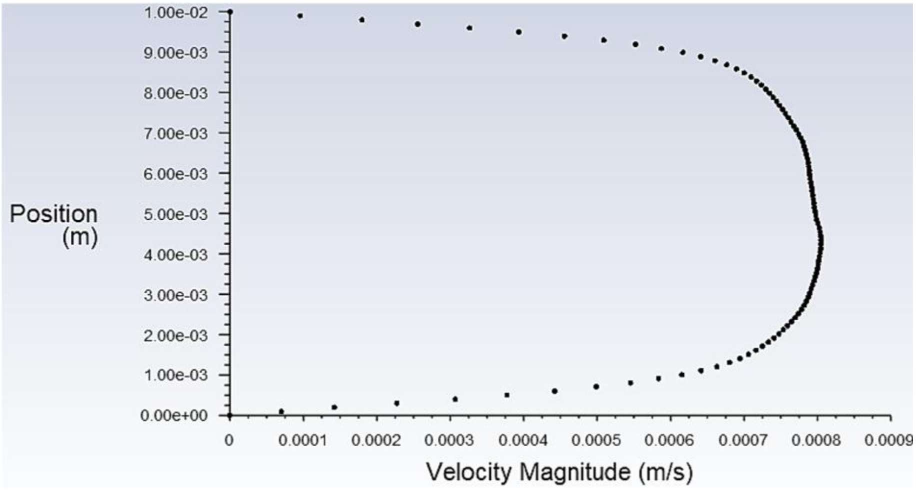

Fig. 8. 2D Heat Pipe Velocity Magnitude Profile

representing a single-phase liquid flow. The profile illustrates the liquid velocity at various points, evaluated along a line located at x = 0.05 mm from the condenser section., where the x-axis represents velocity magnitude in meters per second (m/s), typically ranging from 0 to 0.001 m/s, simulated using the k-epsilon turbulence model, and the y-axis represents the position within the heat pipe in meters (m). The semi-circular concentration of data points around 0.0008 m/s on the x-axis reveals a region of relatively higher velocity magnitude, while the vertical line intersecting the x-axis at approximately 0.0007 m/s likely indicates a physical boundary or interface within the heat pipe, such as the edge of the wick structure.

Fig. 9 depicts the temperature distribution throughout the fluid zone of a two-dimensional heat pipe. The maximum temperatures, reaching up to 400 K, are recorded in the evaporator portion on the left side as a result of applied heat. As one progresses to the right, the temperature progressively decreases down the pipe due to heat transfer to the condenser portion, where it is released. This consistent temperature gradient indicates effective thermal conduction, with the fluid's core sustaining higher tempe-

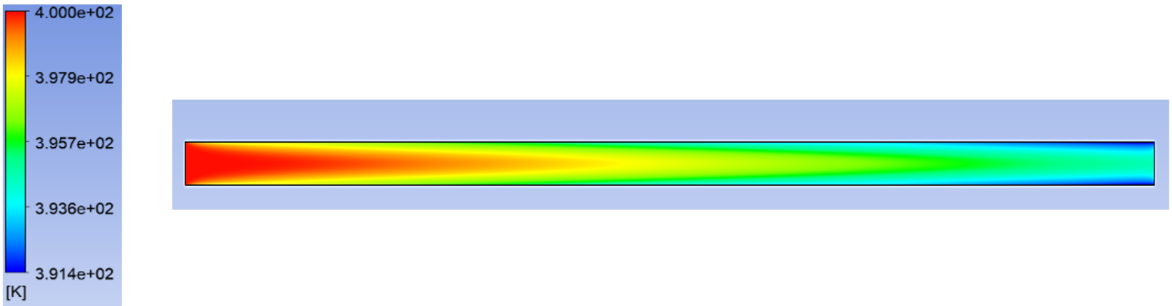

Fig. 9. Temperature Distribution in the Fluid Region of a 2D Heat Pipe

ratures over extended durations, while the peripheral areas decrease more swiftly. The detected thermal pattern demonstrates efficient heat transfer and dissipation throughout the pipe.

Heat Transfer and Efficiency Analysis

In this study the heat transfer investigates and efficiency of a 2D heat pipe using numerical simulations in ANSYS Fluent. The simulation was performed with suitable boundary conditions to assess heat input at the evaporator and heat extraction at the condenser. The cumulative heat input at the evaporator was 987.97 W, but the heat removed at the condenser was 928.10 W, leading to a net heat loss of 59.86 W. The heat pipe's efficiency was determined using the formula:

$$\eta = \frac{Q_{cond}}{Q_{evap}} \times 100\,\% \,. \tag{1}$$

Substituting the obtained value in equation 1:

$$\eta = \frac{928.10}{987.97} \times 100 = 93.95\,\% \,.$$

The results indicate a high efficiency of 93.95 %, demonstrating effective thermal transport from the evaporator to the condenser. However, the remaining 6.05 % of the heat input is lost due to conduction, radiation, or numerical dissipation. The temperature contours and velocity streamlines from Fluent confirm the proper functioning of the heat pipe. This study validates the effectiveness of heat pipes for passive thermal management, and future improvements may focus on optimizing the geometry, wick structure, or working fluid properties to further enhance performance.

Conclusion

Computational investigation for the two-dimensional heat pipe geometry has been performed using Al2O3/R11 nanofluid as a working fluid through CFD simulation. The simulation effectively captured the evaporation process, showing the phase change from liquid to vapor within the heat pipe under thermal load. The temperature distribution indicates a clear thermal gradient from the evaporator section to the condenser, demonstrating efficient heat transport behavior. A well-defined vapor formation zone was observed beginning at approximately 300 mm, marking the onset of phase transition from liquid to gas. The velocity cross-sectional profile was analyzed at the region dominated by pure single-phase liquid, revealing smooth and stable flow conditions conducive to effective heat transfer. The use of Al2O3/R11 nanofluid enhances thermal conductivity and improves the evaporation process due to better energy absorption and dispersion. Additionally, the simulation maintained residual convergence within acceptable limits, confirming numerical accuracy and computational stability. These findings confirm the critical role of nanofluid properties in optimizing the heat pipe's performance.

Conflict of interest

The authors declare that they have no conflict of interest in relation to this research, including financial, personal, authorship, or any other nature that could affect the research and its results presented in this article.

Use of artificial intelligence

The authors confirm that they did not use artificial intelligence technologies when creating the current work.

References

[1] D. K. Ahmed and R. A. Mahmood, "The Evaporation Process in a Heat Pipe: A Review Study", *Technium*, vol. 26, pp. 44–58, Jan. 2025, doi: https://doi.org/10.47577/technium.v26i.12168 .

[2] H. Jouhara *et al*., "Heat pipe based systems-Advances and applications", *Energy*, Vol. 128, pp. 729–754, 2017, doi: https://doi.org/10.1016/j.energy.2017.04.028 .

[3] R. A. Mahmood *et al*., "CFD numerical and experimental investigation of two-phase flow development after an expansion device in a horizontal pipe", *Journal of Thermal Engineering*, Vol. 7(1), pp. 307–323, 2021, doi: https://doi.org/10.18186/thermal.850672 .

[4] R. A. Mahmood *et al*., "Two-Phase Flow Development of R134a in a Horizontal Pipe: Computational Investigation", *International Journal of Heat & Technology*, Vol. 39(5), 2021, doi: https://doi.org/10.18280/ijht.390515.

[5] K. Shukla, "Heat pipe for aerospace applications–an overview", *Journal of Electronics Cooling and Thermal Control*, Vol. 5, No. 1, 2015, doi: https://doi.org/10.4236/jectc.2015.51001.

[6] T. Höhne, "CFD simulation of a heat pipe using the homogeneous model", *International Journal of Thermofluids*, Vol. 15, p. 100163, 2022, doi: https://doi.org/10.1016/j.ijft.2022.100163.

[7] B. Fadhl, L. C. Wrobel, and H. Jouhara, "Numerical modelling of the temperature distribution in a two-phase closed thermosyphon", *Applied Thermal Engineering*, Vol. 60(1–2), pp. 122–131, 2013, doi: https://doi.org/10.1016/j.applthermaleng.2013.06.044.

[8] V. R. Pawar, and S. Sobhansarbandi, "CFD modeling of a thermal energy storage based heat pipe evacuated tube solar collector", *Journal of Energy Storage*, Vol. 30, p. 101528, 2020, doi: https://doi.org/10.1016/j.est.2020.101528.

[9] Z. Zuo, and A. Faghri, “A network thermodynamic analysis of the heat pipe”, *International Journal of Heat and Mass Transfer*, Vol. 41(11), pp. 1473–1484, 1998, doi: https://doi.org/10.1016/S0017-9310(97)00220-2.
[10] S. C. De Schepper, G. J. Heynderickx, and G. B. Marin, “Modeling the evaporation of a hydrocarbon feedstock in the convection section of a steam cracker”, *Computers & Chemical Engineering*, Vol. 33(1), pp. 122–132, 2009. DOI: https://doi.org/10.1016/j.compchemeng.2008.07.013.
[11] M. Xie *et al*., “Experimental investigation of heat transfer performance of rotating heat pipe”, *Procedia Engineering*, Vol. 99, pp. 746–751, 2015, doi: https://doi.org/10.1016/j.proeng.2014.12.597.
[12] H. M. Maghrabie *et al*., “Numerical simulation of heat pipes in different applications”, *International Journal of Thermofluids*, Vol. 16: p. 100199, 2022, doi: https://doi.org/10.1016/j.ijft.2022.100199.
[13] S. A. Annamalai, and V. Ramalingam, “Experimental investigation and CFD analysis of a air cooled condenser heat pipe”, *Thermal Science*, Vol. 15(3), pp. 759–772, 2011, doi: https://doi.org/10.2298/TSCI100331023A.
[14] R. A. Mahmood, D. Buttsworth, and R. Malpress, “Experimental and numerical investigation of two-phase flow orientation direction change on a vertical flash tank separator”, *International Journal of Management and Applied Science (IJMAS)*, Vol. 5(4), pp. 25–29, 2019.
[15] H. K. Versteeg, An introduction to computational fluid dynamics the finite volume method, 2/E. 2007: Pearson Education India.
[16] A. Fluent, ANSYS Fluent 14.5 user’s guide. ANSYS, Inc., Canonsburg, PA, 2012.
[17] F. Aqilah *et al*., “Study of mesh quality improvement for CFD analysis of an airfoil”, *IIUM Engineering Journal*, Vol. 19(2), pp. 203–212, 2018, doi: https://doi.org/10.31436/iiumej.v19i2.905.
[18] N. Fatchurrohman, and S. Chia, “Performance of hybrid nano-micro reinforced mg metal matrix composites brake calliper: simulation approach”, *IOP Conference Series: Materials Science and Engineering*, Vol. 257: p. 012060, 2017, doi: https://doi.org/10.1088/1757-899X/257/1/012060.
[19] R. Lanzafame, S. Mauro, and M. Messina, “Wind turbine CFD modeling using a correlation-based transitional model”, *Renewable Energy*, Vol. 52, pp. 31–39, 2013, doi: https://doi.org/10.1016/j.renene.2012.10.007.
[20] W. Wang, and J. H. Li, “Simulation of Gas–Solid Two-Phase Flow by a Multi-Scale CFD Approach–Extension of the EMMS Model to the Sub-Grid Level”, *Chemical Engineering Science*, Vol. 62, pp. 208–231, 2007, doi: https://doi.org/10.1016/j.ces.2006.08.017.
[21] O. A. Alawi, N. A. Che Sidik, and H. Mohammed, “A comprehensive review of fundamentals, preparation and applications of nanorefrigerants”, *International Communications in Heat and Mass Transfer*, Vol. 54, pp. 81–95, 2014, doi: https://doi.org/10.1016/j.icheatmasstransfer.2014.03.001.
[22] M. K. Melda Özdince Çarpinlioğlu, A Computational Modeling Study on Nanorefrigerants: Definition of Miscellaneous Multiplier M. *The International Journal of Engineering and Science (IJES)*, pp. 21–38, 2022(7).

Обчислювальне дослідження: Посібник з CFD для процесу випаровування в тепловій трубі

Д. Ахмед[1] • Р. Махмуд[1]

[1] *Інженерний коледж, Університет Захо, Курдистан, Ірак*

Анотація. *Теплові труби є важливими компонентами в різних галузях промисловості завдяки своїм винятковим можливостям теплопередачі, які значно покращують системи терморегулювання. Їх здатність ефективно розсіювати тепло з мінімальними температурними градієнтами робить їх безцінними в системах охолодження електроніки, аерокосмічних системах та процесах рекуперації енергії. У цьому дослідженні представлено детальне двовимірне моделювання методом обчислювальної гідродинаміки (CFD) для аналізу розподілу температури та динаміки фазових перетворень під час випаровування в тепловій трубі. Моделювання проведено за допомогою ANSYS Fluent, де розроблено 2D модель теплової труби і створено оптимізовану обчислювальну сітку для забезпечення точності числових результатів. Модель k-є турбулентності використовується для точного відображення поведінки потоку рідини, враховуючи складні взаємодії між паровою та рідкою фазами. Робочою рідиною для цього дослідження був обраний нанохолодоагент (Al2O3/R11), який відрізняється покращеними тепловими властивостями, що сприяють підвищенню ефективності теплопередачі. Результати моделювання показують значний градієнт температури в секції випарника, що підкреслює критичну роль теплового потоку у визначенні термічного опору. Крім того, в дослідженні розглядається, як зміни у вхідному тепловому потоці впливають на загальні теплові характеристики теплової труби. Результати цього CFD-аналізу дають цінну інформацію про процес випаровування, який зумовлений явищами фазових змін, і пропонують практичні рекомендації з проектування для оптимізації продуктивності теплових труб. Завдяки розумінню складного взаємозв'язку між динамікою рідини, теплопередачею і властивостями матеріалу, це дослідження робить значний внесок у розробку більш ефективних і стійких рішень для управління тепловими потоками в промислових цілях.*

Ключові слова: *теплові труби, обчислювальна гідродинаміка (CFD, процес випаровування, ANSYS Fluent, розподіл температури).*