

Development of OpenFOAM VOF Solver with Interfacial Mass-Transfer and Variable Properties for Heat Pipe CFD Simulations

Almanzo Arjuna, In Cheol Bang

Department of Nuclear Engineering, Ulsan National Institute of Science and Technology (UNIST)

50 UNIST-gil, Ulju-gun, Ulsan 44919, Republic of Korea

*Corresponding author: icbang@unist.ac.kr

***Keywords :** Computational Fluid Dynamics (CFD), Volume-of-Fluid (VOF), Phase Change, OpenFOAM, Heat Pipe

1. Introduction

Two-phase passive heat transfer systems, such as heat pipes offer exceptional thermal conductivity by utilizing latent heat transport across liquid-vapor interfaces. To accurately predict the operational limits and thermal behavior of such systems, computational fluid dynamics (CFD) especially using the Volume-of-Fluid (VOF) method has become an essential tool for evaluating detailed interfacial dynamics and phase-change phenomena.

Despite recent advances in numerical phase-change modeling, implementing dynamic saturation temperatures (T_{sat}) and temperature dependent fluid properties within VOF framework remains a significant challenge. In real heat pipe operations, dynamic pressure drops along the vapor channel and wick structure alter the local thermodynamic saturation state. Furthermore, standard volumetric mass-transfer models often trigger non-physical boiling inside bulk superheated liquid pools rather than restricting phase change to liquid-vapor interfaces [1]. In capillary-driven systems, bulk evaporation unphysically forms vapor bubbles inside the wick structure, which artificially blocks the liquid return and triggers dry out in the simulation.

To bridge these gaps, this study presents the development of a modified VOF solver from OpenFOAM incorporating local pressure-dependent saturation (T_{sat}), variable thermophysical properties, and an interface-bound phase-change model. In the current study the solver is verified against classical 1D benchmark cases such as Stefan problem and Sucking interface problem [2] – to demonstrate numerical stability, accurate interface tracking, and the suppression of unphysical bulk evaporation for future heat pipe simulations.

2. OpenFOAM CFD Model

The framework is built upon OpenFOAM v2312's *interCondensatingEvaporatingFoam* (iCEF) which is a VOF solver for two incompressible, non-isothermal, and immiscible fluids undergoing phase changes [3]. Like other VOF solvers it tracks the liquid-vapor interface by solving the phase fraction advection equation,

$$\partial \alpha_1 + \nabla \cdot (U \alpha_1) = \frac{\dot{m}}{\rho} \quad (1)$$

Mass and heat transfer across the interface are coupled through explicit volumetric source terms added to the phase fraction, pressure, and energy transport equations.

In standard implementations, mass transfer rate per unit volume ($\dot{m}$) is evaluated using a Lee [4] model proportional to T_{sat} and phase-change coefficients (r_e & r_c , s^{-1}). To overcome the limitation of non-physical bulk boiling in superheated domains and account for localized pressure drops due to phase-change, the base solver is modified to restrict mass-transfer strictly to the liquid-vapor interface applying an interface-masking factor, based on volume fraction gradient ($|\nabla \alpha|$) [5] to the Lee method.

$$\dot{m}_e = r_e (|\nabla \alpha_l| \Delta x) \rho_l \frac{(T - T_{sat})}{T_{sat}} \quad (3)$$

$$\dot{m}_c = r_c (|\nabla \alpha_v| \Delta x) \rho_v \frac{(T_{sat} - T)}{T_{sat}} \quad (4)$$

To avoid mesh dependance, this gradient is scaled by the local characteristic cell length ($\Delta x = \sqrt[3]{V_{cell}}$) to create a bounded, dimensionless interface indicator. By bounding phase-change with the phase fraction gradient, it will bound the phase-change strictly on the interface.

3. Verification case

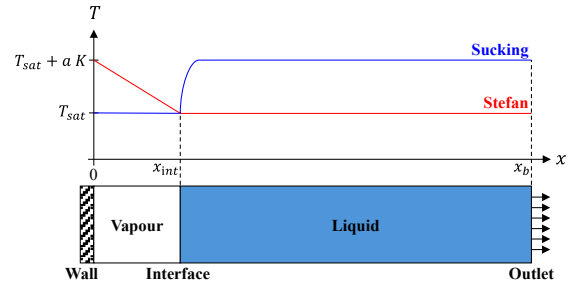


Fig. 1. Schematic of 1D Stefan and Sucking Problem

The classical 1D Stefan problem was first simulated to verify conduction-driven phase-change dynamics. In this test, a subcooled liquid layer undergoes planar evaporation driven by conduction through a vapor gap from constant wall heat flux producing a receding interface which can be analytically calculated. The superheat wall ($x = 0$) was set to $T_{sat} + 5 K$, while the liquid domain was set at saturation T_{sat} .

The Sucking interface problem further extends the Stefan Problem by introducing a phase-change driven velocity field. In this case the wall and vapor gap is held

at T_{sat} , while the liquid at the other end of the wall is superheated to $T_{sat} + 5\text{ K}$.

The schematic of both cases is shown in Fig. 1. Each case utilizes identical domain, 10 mm x 2 mm which was discretized to 400×2 cells with uniform grading, simulated over 50 s. While the standard iCEF solver relies on constant thermophysical properties, the modified solver incorporates temperature-dependent fluid properties.

4. Results and discussion

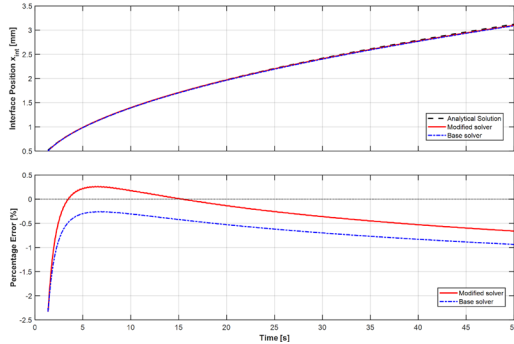


Fig. 2. Stefan problem benchmarks interface position (top) and relative error (bottom)

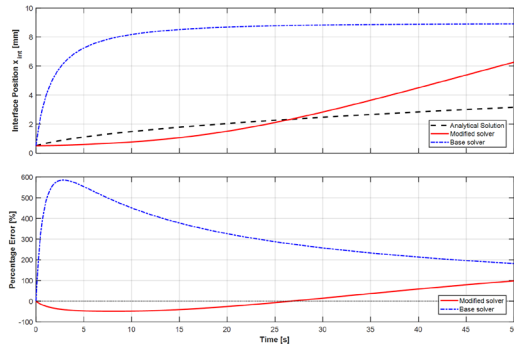


Fig. 3. Sucking problem benchmarks interface position (top) and relative error (bottom)

Fig. 2. Compares the analytical solutions with OpenFOAM standard iCEF solver and modified solver. For Stefan case, the interface position from the modified solver shows a relatively lower error than the standard iCEF solver. The error shows similar trend where initially there was a large error $\sim 2.2\%$ and it stabilizes at around 0.65% for modified solver and 0.925 for standard iCEF.

While both solvers can handle Stefan problem quite well, when imposed of a complex problem in Sucking interface problem it gave a quite large error. The standard iCEF completely breaks as the phase change employs bulk rather than interface phase-change. The modified iCEF, performs relatively better albeit with the large relative error at the end of simulation as shown in Fig. 3. The modified solver successfully shows interface localized phase-change and stabilizes the numerical scheme in a regime where the standard solver breaks entirely.

The contrast between the two cases highlights two distinct challenges in phase-change modelling. In domains with steep thermal gradients, conventional solvers suffer from bulk phase-change. By incorporating localized phase-change masking, the modified solver successfully restricts mass transfer to the interface. However, the Sucking interface problem shows that geometric masking alone is not a complete solution. While the modification shows a correct phase-change location, it cannot overcome the inherent limitations of lee phase-change model.

5. Conclusions & future work

This study demonstrates that the modified OpenFOAM *interCondensatingEvaporatingFoam* could overcome the bulk phase-change failures of the standard solver in Sucking interface problems by enforcing interface-localized phase-change. In Stefan case, the modified solver performs better than the standard solver. For heat pipe CFD, this implementation ensures that the liquid phase remains continuous within the capillary wick structure as the phase-change is bounded to the interface. Additionally, this mass-transfer model provides a much more robust method for accurately predicting the operational limits of heat pipes. Overall, this validation serves as a preliminary step toward fully 3D heat pipe simulations.

ACKNOWLEDGMENTS

This work was partly supported by the National Research Foundation of Korea (NRF) grant funded by the Korea Government (MSIT) (no. 2021M2D2A1A03048950) and Korea Institute of Energy Technology Evaluation and Planning (KETEP) grant funded by the Korea Government (MOTIE) (no. RS-2024-00403194).

REFERENCES

- [1] S. Hardt and F. Wondra, "Evaporation model for interfacial flows based on a continuum-field representation of the source terms," *J. Comput. Phys.*, vol. 227, no. 11, pp. 5871–5895, May 2008, doi: 10.1016/j.jcp.2008.02.020.
- [2] J. Kren, B. Ničeno, and Y. Sato, "Sharp-interface VOF method for phase-change simulations on unstructured meshes," Apr. 2026, [Online]. Available: <http://arxiv.org/abs/2604.14938>
- [3] Y. Sun, Y. Darbar, M. Hossein, and A. Khanouki, "CFD with OpenSource software Description of interCondensatingEvaporatingFoam and implementation of SGS term into volume fraction equation", doi: 10.17196/OS.
- [4] W. H. Lee, "A Pressure Iteration Scheme for Two-Phase Flow Modeling," *Multi-phase Transport: Fundamentals, Reactor Safety, Applications*, vol. 1, pp. 407–431, 2002, doi: 10.1142/9789814460286_0004.
- [5] H. Scheufler and J. Roenby, "TwoPhaseFlow: A Framework for Developing Two Phase Flow Solvers in OpenFOAM," *OpenFOAM® Journal*, vol. 3, pp. 200–224, Nov. 2023, doi: 10.51560/ofj.v3.80.