CFD Analysis and Visualization of the Two Phase Flow in a Thermosyphon for a Passive Heat Removal System of a Nuclear Power Plant

Joseph Seo, Jae Young Lee*
School of Mechanical and Control Engineering, Handong Global University, Pohang, Korea,
82-54-260-1392, 82-54-260-1839, 791-708, jylee7@handong.edu

1. Introduction

Fukushima accidents emphasized the requirement of passive safety system for reactor. Especially, when accidents such as station black out (SBO) occur, the passive safety system which can ensure the safety of reactor is needed. A thermosyphon, wickless heat pipe, is a heat transfer device of high thermal conductance that functions passively on the principle of evaporation and condensation of a working fluid [1]. The heat-pipe concept was first proposed by Gaugler in 1942 [2]. After its independent invention by Grover [3] in the early 1960s, serious development progress was made, and the heat pipe concept was studied intensively for both space and terrestrial applications, because of its beneficial characteristics. Recently, as regards station black-out (SBO) scenarios due to beyond design-basis accidents (BDBA) in nuclear power plants, heat pipes have been recognized as alternative heat-removal devices for use in the passive safety system [4,5].

As the requirement of applications to such systems with large scale increases, importance of exact simulation model of heat pipe also increases. However, only few studies for heat pipe and thermosyphon simulation have been made. Joudi and Al_Tabbakh [6] tried to analyze thermal characteristics of thermosyphon using mathematical modeling. Zhang et al. [7] studied two dimensional thermal and mass flux of disc type thermosyphon. Alizadhdakhel et al. [8] provided CFD modelling and experimental measurement of thermosyphon. Using VOF method, they investigated the effect of filling ratio to thermal performance of thermosyphon. Annamalai and Ramalingam [9] developed a CFD modeling for wick part of heat pipe using commercial code, ANSYS CFX. Khurram Kafeel [10] numerically studied thermal hydraulic characteristics of thermosyphon in both transient and steady state. Bandar Fadhl et al. [11] built a CFD modeling for boiling and condensing of thermosyphon using VOF method of ANSYS Fluent. They made user defined function (UDF) to define source term based on Lee model [12, 13].

In this study, CFD model of 1m-thermosyphon has been studied using VOF model. Unlike formal studies, vacuum pressure condition was applied because thermosyphon with vacuum inner pressure is much generally used. Furthermore, to check out hydraulic characteristics of the model, transparent thermosyphon experiment also has been conducted. Thermal characteristics of the model has been checked by empirical measurement of thermosyphon experiment in same boundary condition.

2. Simulation Model

2.1 Geometry and Mesh

A two dimensional geometry of closed pipe was developed to analyze the thermal hydraulic characteristics of a thermosyphon. As shown in Fig.1, a total length of 1000mm closed pipe with same length of evaporator, adiabatic and condenser section was designed. Outer and inner diameter were set as 20mm and 16mm each.

Various sizes of meshes were used for calculation. In Fig.2, to analyze the phase change and the heat transfer in the interface of inner walls, inflation option of ANSYS was used. Detailed status of meshes are shown in Table.1.

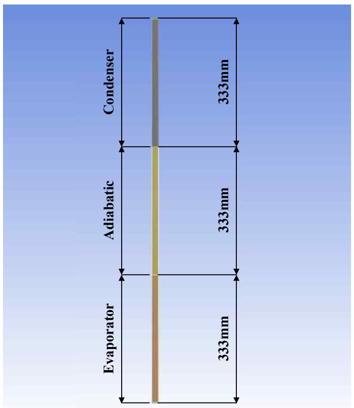

Fig. 1. Geometry and dimensions of a model

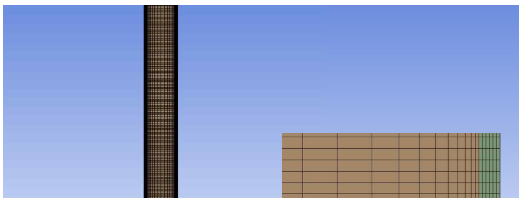

Fig.2. A section of the computational mesh

Table I: Detailed information of computational meshes

Dimension	2D
Minimum Size	2.4946e-004[m](Default)
Max Face Size	2.4946e-002[m](Default)
Max Size	4.9892e-002[m])Default)
Minimum Edge Length	9.0e-004[m]
Number of Nodes	52885
Number of Elements	48192

2.2 Boundary Condition

To simulate the phase change in a thermosyphon, 25145 W/m^2 of constant heat flux input condition was given at the outside wall boundaries of evaporator. Also condenser section was cooled by constant temperature condition of 20^{o}C. 0.3bar of pressure was defined as operating pressure of the thermosyphon. Adiabatic boundary condition was imposed at the outside wall boundaries of adiabatic region and a non-slip boundary condition was declared at the inner walls.

2.3 Solution Strategy Set up

Water vapor as the primary phase and water liquid as the secondary phase were set for phase change and interface. Based on the Courant number, the simulation was conducted in transient analysis with a time step of 0.0005 s. The Courant number was maintained under 2 through calculation. For pressure-velocity coupling, SIMPLE algorithm is selected. A first-order upwind scheme for the determination of momentum and energy and Ged-Reconstruct for the volume fraction were performed in the model.

3. Experiment

3.1 Experimental Set up

The schematic of experimental apparatus is shown in Fig.3. Experimental apparatus is consist of heat pipe section, heating and cooling section. Heat pipe section includes transparent heat pipe, vacuum pump and working fluid injecting component. Heating section was designed to insert heat to the evaporator of heat pipe by using specially manufactured transparent heater connected to DC power supplier. Cooling section is connected to the chiller which can control flow rate and temperature of cooling water.

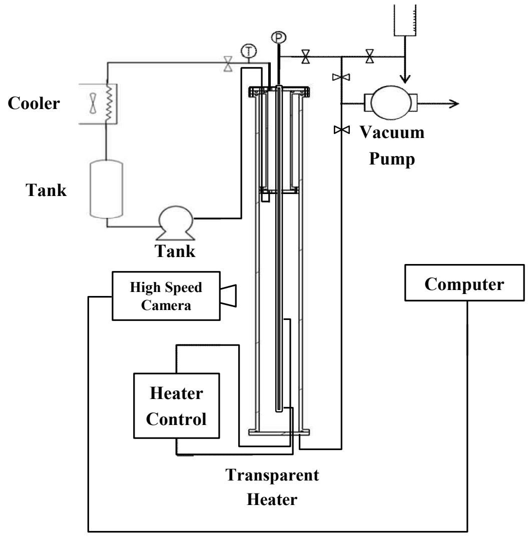

Fig.3. Schematic of experimental set up

3.2 Experimental Procedure

Experiments were conducted by following steps. After the power had been applied to the evaporator, the steady state was determined by checking surface temperature variation and flow pattern while its inner pressure being controlled by flow rate and temperature of cooling water. When the condition of steady state is fulfilled, flow pattern was recorded by high speed camera for with 2000Hz of frame.

4. Result and Discussion

4.1 Simulation Result

Result of a transient simulation is shown in Fig.4. Considering its temperature and flow pattern, the simulation model of the thermosyphon reached a steady state after 20 seconds. The fluid in the evaporator section was heated at the beginning of the process. After its temperature reached boiling temperature, the liquid starts to boil and phase change occurs.

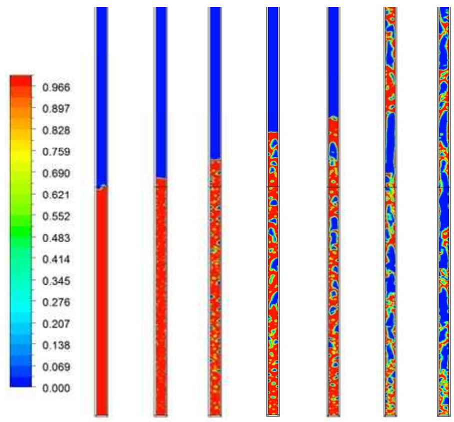

Fig.4 Contours of volume fraction of the evaporator and the adiabatic sections at different times.

The phase change had occurred in bubble boiling pattern at first and it was developed to cap bubble and slug bubbles sequentially. Finally, it is shown that flow pattern becomes Churn flow at steady state.

4.2 Transparent Heat Pipe Experiment Result

A transparent heat pipe experiment results are shown in Fig. 5. For comparison, same boundary conditions had been set. After the heat pipe reached steady state, images of flow pattern were taken by high speed camera. It is found that, hydraulic characteristics of both simulation and experimental results showed similarity. Bubbles occurred at the bottom region of evaporator grow rapidly to slug bubble. A Churn flow was dominantly shown in both simulation and experimental results.

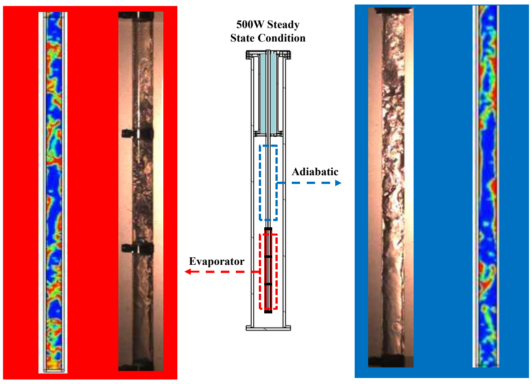

Fig.5 Comparison between flow patterns of simulation and experimental results

4.3 Further Works

Wall temperature data were checked and it is plotted against its position in Fig.6. To investigate the thermal characteristics of simulation model, comparison work with empirical experiments will be conducted. After thermal hydraulic characteristics of simulation model are verified, it would be usefully used to heat pipe and nuclear engineering research fields.

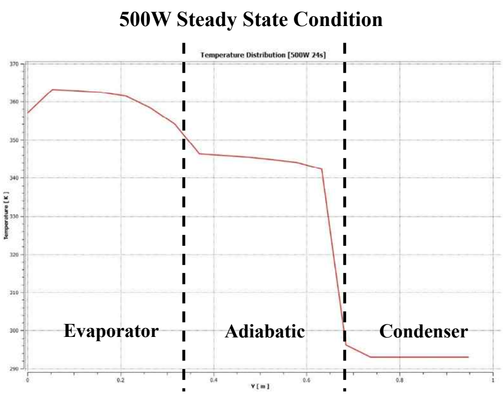

Fig.6 Surface temperature distribution of simulated heat pipe.

5. Conclusion

The main purpose of this research is the investigation of CFD model of thermosyphon. Simulations using VOF method were performed to analyze evaporating, condensing and two phase flow of a thermosyphon.

The simulation results show that complex phenomena inside of thermosyphon can be modeled using VOF method. Flow visualizations of working fluid matched well with transparent heat pipe experiment. Its thermal characteristic model will be verified by comparing the surface temperature with a series of experiment.

It is expected that, results of this study can be applied to various research fields such as thermal and nuclear engineering.

Acknowledgement

This work was partially sponsored by the National Research Foundation of Korea (NRF), which is granted financial resources by the Ministry of Science, ICT, and Future Planning (NRF-2013M2A8A1041443).
Also, this work was partially sponsored by the Nuclear Safety Research program through the Korea Foundation Of Nuclear Safety (KOFONS), granted financial resource from the Nuclear Safety and Security Commission (NSSC), Republic of Korea (No. 1305008-0315-SB130).

REFERENCES

[1] G.P Peterson, An Introduction to Heat Pipes Modeling, Testing and Applications, Wiley Inter-Science, 1976

[2] R.S. Gaugler, US Patent 2350348, 1944.

[3] G.M. Grover, T.P. Cotter, G.F. Erickson, Structure of very high thermal conductance, Journal of Applied Physics 35 (6) (1964) 1990.

[4] J.Y. Lee, et al., Feasibility study on two-phase thermosiphon for external vessel cooling application of SFR, Korean Nuclear Society, 2014.

[5] M. Mochizuki, et al., Heat pipe based passive emergency core cooling system for safe shutdown of nuclear power reactor, Applied Thermal Engineering, 73 (1) (2014) 697-704.

[6] K.A. Joudi, A.A. Al-Tabbakh, Computer simulation of a two phase thermosyphon solar domestic hot water heating system, Energy Conversion and Management 40 (1999) 775e793.

[7] M. Zhang, Z. Liu, G. Ma, S. Cheng, Numerical simulation and experimental verification of a flat two-phase thermosyphon, Energy Conversion and Management 50 (2009) 1095e1100.

[8] A. Alizadehdakhel, M. Rahimi, A.A. Alsairafi, CFD modeling of flow and heat transfer in a thermosyphon, International Communications in Heat and Mass Transfer 37 (2010) 312e318.

[9] A.S. Annamalai, V. Ramalingam, Experimental investigation and computational fluid dynamics analysis of a air cooled condenser heat pipe, Thermal Science 15 (2011) 759e772.

[10] Khurram Kafeel, Modelling and simulation of two-phase closed thermosyphones using two-fluid method, Thesis for Ph.D of the University of Manchester, School of Mechanical, Aerospace and Civil Engineering, 2014.

[11] Bandar Fadhl, Luiz C. Wrobel, Hussam Jouhara, 2013, "Numerical modelling of the temperature distribution in a two-phase closed thermosyphon", Applied Thermal Engineering, vol.60, p.122-131

[12] Lee, W. H.,1980, A pressure iteration scheme for two-phase flow modeling. In T. N. Veziroglu (Ed.), Multiphase transport fundamentals, reactor safety, applications. Washington, DC: Hemisphere Publishing.

[13] Wu, H. L., Peng, X. F., Ye, P., & Eric Gong, Y.,2007, Simulation of refrigerant flow boiling in serpentine tubes. International Journal of Heat and Mass Transfer, 50, 1186.