Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 29-30, 2014

CFD Analysis of a Hybrid Heat Pipe for In-Core Passive Decay Heat Removal System

Yeong Shin Jeong, Kyung Mo Kim, In Cheol Bang*
Ulsan National Institute of Science and Technology (UNIST)
School of Mechanical and Nuclear Engineering
UNIST-gil 50, Eonyang-eup, Ulju-gun, Ulsan Metropolitan City 689-798, Republic of Korea
**Corresponding author: icbang@unist.ac.kr*

1. Introduction

On March 2011, large scale of earthquake with magnitude 9.0 and tsunami had caused the potential of serious nuclear reactor meltdown in Fukushima, Japan. At that time, residual heat generated after reactor shutdown was not adequately cooled due to all onsite and offsite alternative current power lost. Station blackout (SBO) accident is the event that all AC power is totally lost from the failure of offsite and onsite power sources [1]. Although electricity was provided from installed batteries for active system after shutdown, they were failed due to flooding after tsunami. The vulnerability of the current operating power plant's cooling ability during extended station blackout events is demonstrated and the importance of passive system becomes emphasized. Numerous researches about passive system have been studied for proper cooling residual heat after Fukushima nuclear power plant accident.

Heat pipe is the effective passive heat transfer device that latent heat of vaporization is used to transport heat over long distance with even small temperature difference [2]. Since liquid flows due to capillary force from wick structure and steam flows up due to buoyancy force, power is not necessary. Heat pipe is widely used in removal of local hot spot heat fluxes in CPU and thermal management in spacecrafts and satellites.

Hybrid control rod, which consists of heat pipe with B_4C for wick structure material can be used for removing residual heat after. It can be applied to both for shutdown and cooling of decay heat in reactor. This concept is independent of external reactor situation like operator's mistake or malfunction of active cooling system. Heat pipe cooling system can be applied to Emergency Core Cooling System, In-Vessel Retention, containment and spent fuel cooling, contributing to decrease Core Damage Frequency.

2. CFD Modeling

For evaluating cooling performance of hybrid control rod, a scaled down single heat pipe is considered for a CFD analysis model. The analysis model consisted of the pipe case, screen mesh wick structure, water jacket for condenser part and water as working fluid. For heat pipe simulation, a commercial CFD code, ANSYS-CFX v13.0 was used.

2.1 Analysis Method

Flow inside heat pipe is basically two phase with evaporation and condensation. Mass transfer option is set for thermal phase change with saturation temperature at 323 K. Phase change is occurred at wick-vapor interface where latent heat of vaporization is released or absorbed at here. For the wick, ANSYS-CFX provides to solve transport equation in porous media that permeability, porosity, and loss coefficient values were used to input values for model [3].

2.2 Computational Grid

Fig. 1 shows the computational domain for hybrid control rod model and Table 1 shows the details of geometrical properties of model. The analysis model consisted of hexahedral and tetrahedral mesh. The number of total node is 523339 and 396412 elements was calculated. Orthogonal quality of mesh was used for check mesh quality. Minimum value is 0.62354, maximum value is 0.99523. Average value is 0.97635 and standard deviation is 4.1019E-02.

Table I: Details of heat pipe geometrical properties

Case OD	19.05 [mm]
Wick OD	17.4 [mm]
Flow channel diameter	15.4[mm]
Total length	1000 [mm]
Length ratio (evaporator : adiabatic : condenser)	350 : 150 : 500

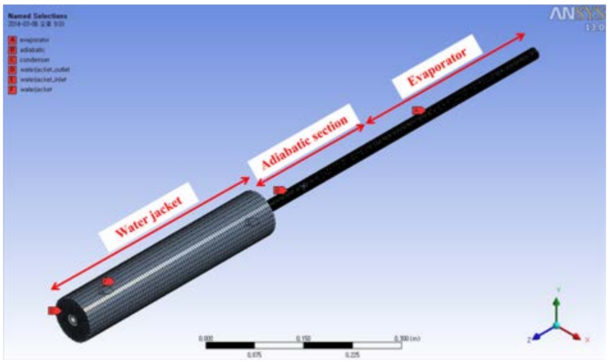

Fig. 1. Computational domain of single heat pipe

2.3 Boundary Conditions

Table 2 represents parameters for model and boundary conditions. Heat pipe case was set as rigid body wall condition. Lower part of heat pipe is set as evaporator and heat input value was 150 W. At middle section of heat pipe, adiabatic condition was adjusted. At upper part of heat pipe, water jacket was surrounded for absorbing heat as condenser. For water jacket, water flows with 0.03 kg/s and bulk temperature is 11 °C for proper cooling. Wick structure was modeled as SS screen mesh, which porosity is 0.62 and permeability is 1.93×10^{-10} m^2 [4] Working fluid is set by 100% volume of evaporator, which means fill ratio is 100%.

Table II: Parameters for the model and boundary condition

Heat input	150 [W]
Condenser type	Water jacket (T_∞=11°C, mass flow rate = 0.03 kg/s)
Wick type	100-mesh SS screen mesh
Porosity, ε	0.62
Permeability, K	1.93×10^{-10} [m^2]
Fill ratio	100 [%]

3. Result and Discussion

For validating numerical result, heat pipe experiment with same condition was done. Comparison between CFD simulation and measured temperature profile along the heat pipe is shown in Fig. 2. A reasonable agreement can be observed between experimental data and predicted. Since heat was conducted along pipe from evaporator to adiabatic section in experimental setup which was ignored in CFD model, temperature along adiabatic section was under predicted in CFD calculation. Wall temperature at condenser of experimental result was lower than numerical result because heat transfer to wall from high temperature steam was considered to numerical result. Maximum wall temperature is 333.783 K and minimum wall temperature is 288.213 K for 150W heat input.

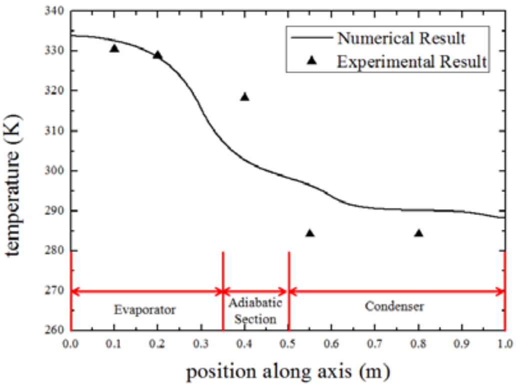

Fig. 2. Wall temperature distribution along the heat pipe at Q=150W in forced convection of 0.03 kg/s flow with water jacket at condenser part (Screen mesh K=1.93×10^{-10} m^2)

Average wall temperatures of evaporator and condenser were used for calculating corresponding mean heat transfer coefficient for heat pipe. Heat transfer coefficient at evaporator and condenser is determined as follows [5]:

$$h_e = \frac{q_e}{(\overline{T}_e - T_{sat})} \tag{1}$$

$$h_c = \frac{q_c}{(T_{sat} - \overline{T}_c)} \tag{2}$$

(h = heat transfer coefficient, $\overline{T}$ = mean wall temperature, T_{sat} =saturation temperature=323 K, e = evaporator, c = condenser)

For 150W heat input, heat transfer coefficient at evaporator section was 2249 W/m^2 K and that of condenser section was 247.61 W/m^2 K. In experimental result, measured heat transfer coefficient at evaporator section was 1211 W/m^2 K and that of condenser section was 41.23 W/m^2 K. Difference between numerical results and experimental results was come from the number of measured points in experiment was much smaller than that of nodes in numerical calculation. Since adiabatic boundary condition was set as insulation condition and heat conduction along axial direction was not considered in numerical simulation, the heat transfer coefficients were different between numerical results and experimental results.

Fig. 3 shows steam volume fraction inside heat pipe at adiabatic section and condenser section. At adiabatic section, liquid film was formed at wick-vapor interface from steam condensation. At adiabatic section, liquid flows through wick structure and liquid film thickness increases compared with that of condenser section due to gravity effects [6].

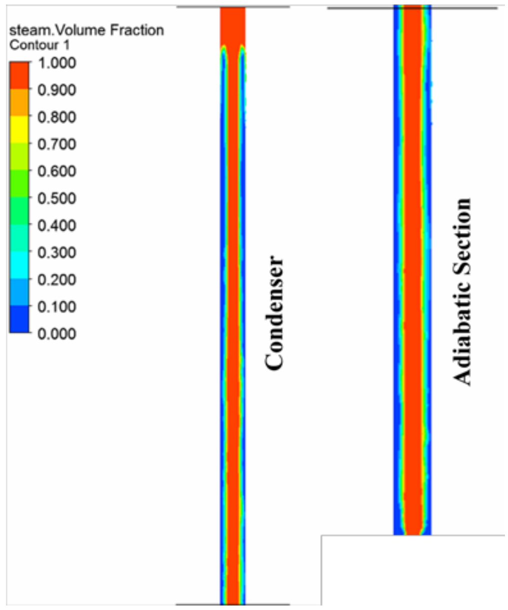

Fig. 3. Contours of vapor volume fraction through the condenser and adiabatic section

4. Summary and Further Work

The computational fluid dynamics analysis of hybrid control rod was simulated for evaluating cooling performance of heat pipe using CFD. The single heat pipe model is used to CFD simulation using ANSYS-CFX. Numerical calculation result is compared with experimental result for validation of simulation. Heat pipe, the effective heat transfer device using condensation-evaporation can be used for emergency cooling system with the concept of hybrid control rod which is the combined concept of both shutdown and removal of decay heat. In extended station blackout events, total loss of AC power source can increase of possibility of core meltdown accident. Because hybrid control rod cooling system is passive, it can provide core cooling as well as absorb neutron to stop fission reactor in reactor at station blackout. As a result, it can decrease core damage frequency with cooling decay heat after reactor shutdown by applying various passive cooling systems like emergency core cooling system, in-vessel retention, containment and spent fuel pool.

For further work, hybrid heat pipe for in-core decay heat removal system will be analyzed to evaluate cooling performance at extended station blackout event when adjusting existing nuclear power plant. Analysis model will be APR-1400 reactor vessel that control rod is replaced to hybrid control rod, which means heat pipe cooling system. Fig. 4 represents the schematic for CFD analysis of reactor pressure vessel with hybrid control rod cooling system.

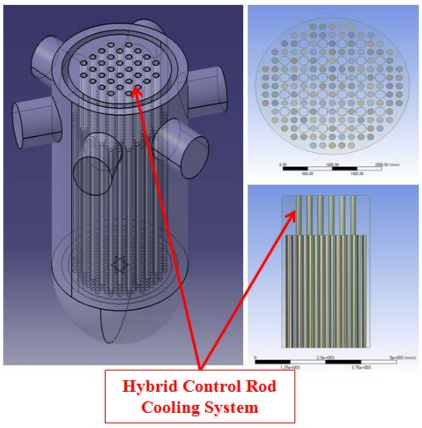

Fig. 4. The schematic of CFD model for evaluating cooling performance of existing reactor pressure vessel with hybrid control rod cooling system

With just replacing control rod to hybrid control rod cooling system, which can remove decay heat after shutdown as well as shutdown the reactor, can be expected to delay time to re-peak temperature of cladding at transient situation and secure time to cope with unexpected situation. From this research, we can know that in-core decay heat removal system using heat pipe can improve the inherent safety of nuclear reactor with passive system.

REFERENCES

[1] M. Mochizuki, T. Nguyen, R. Singh, T. Nguyen, S. Sugihara, K. Mashiko, Y. Saito, and V. Wuttijumnong, Nuclear Reactor Must Need Heat Pipe For Cooling, Frontiers in Heat Pipes, 2, 033001, 2011.

[2] D. A. Reav, P. A. Kew, Heat Pipes, Elsevier, Atlanta, 2006.

[3] ANSYS-CFX Solver Theory Guide, ANSYS Inc., Canonsburg, 2012.

[4] M. G. Mwaba, X. Huang, and J. Gu, Influence of Wick Characteristics on Heat Pipe Performance, International Journal of Energy Research, Vol. 30, pp. 489-499, 2006.

[5] L. Asmaie, M. Haghshenasfard, A. Mehrabani-Zeinabad, and M. Nasr Esfahany, Thermal Performance Analysis of Nanofluids in an Thermosyphon Heat Pipe Using CFD, Heat Mass Transfer, Vol. 49, pp. 667-678, 2013.

[6] N. Z. Ahmed, P. K. Singh, I. Janajreh, and Y. Shatilla, Simulation of Flow inside Heat Pipe: Sensitivity Study, Conditions and Configuration, Proceedings of the ASME 5th International Conference on Energy Sustainability, Aug. 7-10, 2011, Washington, DC, USA.