Simulation of IVR-ERVC and estimation method of coolant inflow to the cavity

Hyunjin Lee *, Ihn Namgung
KEPCO International Nuclear Graduate School., Ulsan, Republic of Korea
**Corresponding author: hj.lee@kings.email.ac.kr*

1. Introduction

A lot of researches have been conducted for IVR-ERVC under various assumptions. Because of the nature of the complexities surrounding these phenomena, it has been usual to assume a constant temperature at the external RV wall and performed a thermal stress analysis[1]. In this study, the temperature distribution outside of RV wall and evaporation rate due to heat from core will be investigated. Using the universal analysis program ANSYS Fluent, the natural convection in the cavity for IVR-ERVC conditions were modelled and performed for heat transfer analysis. The aim of this study is to calculate the appropriate coolant flow so that coolant level in the cavity can be maintained at prescribed level and vessel wall temperature distribution, including RV outside wall temperature are also investigated.

2. Methods and Results

In this section some of the techniques used to model for IVR-ERVC are described. It is necessary to make a selection of the multi-phase flow analysis models[2] for ERVC model. In this study, a mixture model was selected and verification was performed.

2.1 Analysis for simple model

As a first attempt, the simplified model was used for analysis, so Cavity and RV was applied but insulator model was omitted. To simulate severe accident condition, the existing experimental data[3][4] was applied for the critical incident heat flux to the bottom head of the reactor and the initial fluid temperature of the supplied cavity was assumed to be less than 30 ℃.

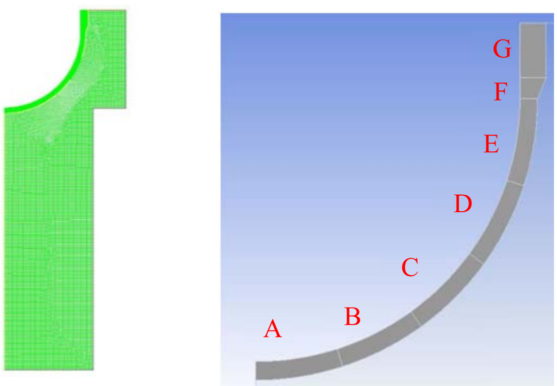

Fig. 1. The mesh model was applied to simulate the insulator is omitted.

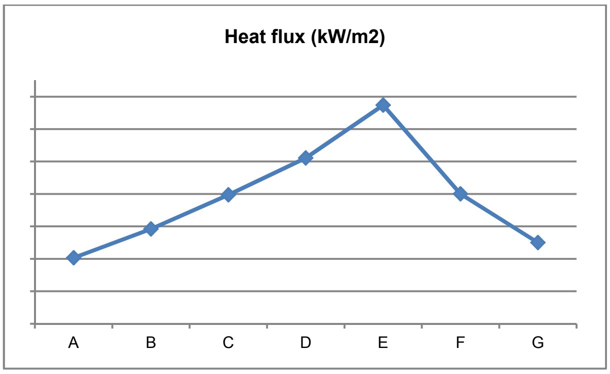

Fig. 2. Heat flux applied to RV Lower Head according regions the values were obtained from severe accident (TLOFW w/o SDS) analysis (MELCORE).

The mixture model was used to analyze the multi-phase flow and standard k-ε turbulence model was used as the turbulence model.

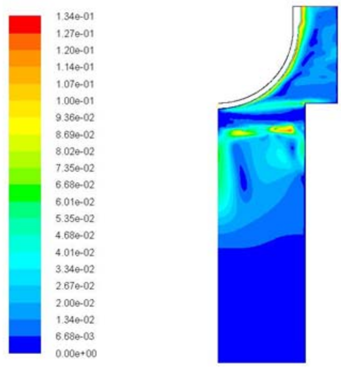

Fig. 3. Result of velocity magnitude for cavity flow after 1 hour simulation.

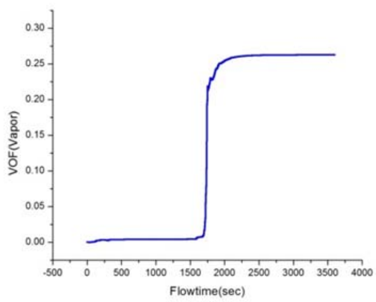

Fig. 4. Result of volume of fraction for vapor during 1 hour simulation.

From the simple model simulation result, it was possible to predict the temperature distribution, flow velocity distribution, the amount of water vapor.

A more sophisticated analysis for evaporation trend will allow for more accurate estimation of the cooling water feed rate to maintain a proper water level of cavity. To measure the vapor generation, in this study the average weight and volume of fraction was calculated during the simulation.

2.2 Analysis for Actual Model

The insulator was modeled to simulate a more accurate model of current APR1400 design.. Study on the ex-vessel cooling in the event of severe accident condition were studied and presented the benefits and methods of insulator in many papers and reports [5][6]. In this study, the analysis model was developed to represent the actual design with some simplified without ignoring significant geometry details.

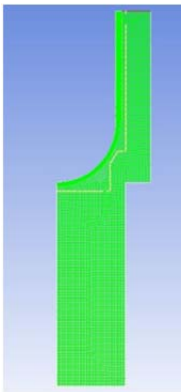

Fig. 5. The insulator model was applied to simulate.

Similar to the initial model, the refined model with insulators was developed and used to simulate the multi-phase flow and standard k-ε turbulence model was used as the turbulence model. In case of simulation method, the pressure-velocity coupling, PISO, schematic was used and spatial discretization of least square cells based on gradient, PRESTO for pressure, Quick for momentum, volume fraction, Turbulent kinetic energy were used.

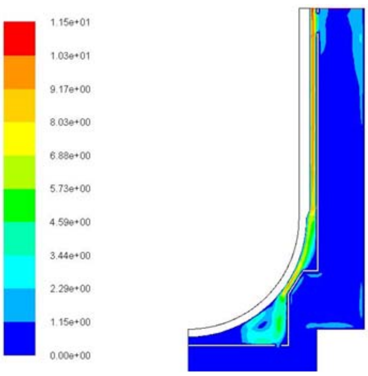

Fig. 6. Result of flow velocity magnitude for cavity flow.

From the actual model simulation result, it is possible to predict the temperature distribution, flow velocity distribution, the amount of water vapor.

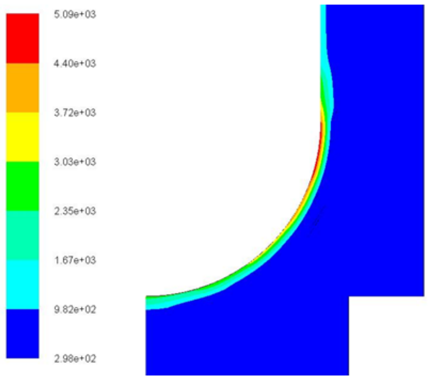

Fig. 7. Result of static temperature distribution.

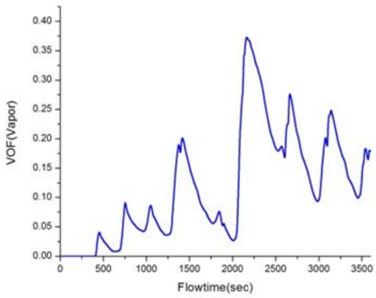

Fig. 8. Result of volumetric fraction of vapor.

3. Conclusions

Reactor vessel and cavity in case of ex-vessel cooling for severe accident condition were modeled with and without insulators. The heat load into reactor vessel from corium inside of reactor lower head were obtained from MELCORE analysis and used as input B.C of CFD analysis. The Temperature gradient of reactor outer surface and evaporation rate of cooling eater was obtained from the analysis. These results can be used for further analysis of reactor vessel creep behavior and the estimate the coolant flow rate into the reactor cavity. and The result can be used to verify the natural convection phenomena in the cavity and also to set the design parameters of cavity and coolant flow rate. The vessel outer surface temperature gradient can be also used to more accurate investigation of vessel creep behavior during severe accident condition, The result can also be used set up a strategy for severe accident managements.

REFERENCES

[1] J. W. Park et al., In-Vessel Core Debris Cool-ability for the APR1400, Korea Hydro & Nuclear Power Co. 2001.

[2] Z. Yang et al., Numerical and experimental

investigation of two phase flow during boiling in a coiled tube, International Journal of Heat and Mass Transfer, 51, 1003-1016 , 2008.
[3] TW Kim, Development of Tube Failure Analysis in Reactor Vessel Lower Head, KAERI, TR-5382, p.19.2013.
[4] J. W. Park and S.J.OH, Design for the In-Vessel Core Melt Retention and the Overall Severe Accident Management Strategy of the APR1400, Proc. Of ICAPP 05, Seoul, KOREA, May 15-19, 2005.
[5] Seong Dae Park, In Cheol Bang, Feasibility of flooding the reactor cavity with liquid gallium coolant for IVR-ERVC strategy, Nuclear Engineering and Design 258, pp13-18,2013.
[6] “APR1400 Standard Safety Analysis Report”, KHNP, 2001