Computational Fluid Dynamic Analysis Modeling of IVR-ERVC of APR1400 Reactor

Ihn Namgung*[1], Hyun-Jin Lee[2]
Nuclear Power Plant Engineering Department
KEPCO International Nuclear Graduate School
Ulsan, South Korea

***Abstract*— When analyzing the IVR (In-Vessel Retention) - ERVC(External Rector Vessel Cooling) for severe accident condition, it has been customary to assume reactor pressure vessel outside wall temperature at some temperature due to near constant evaporation temperature of water at atmospheric pressure and nature of the complexities surrounding the evaporation phenomena at the surface. To improve assessment of reactor vessel structural integrity, a more accurate temperature distribution at the reactor vessel outside surface is needed. Another concern is to evaluate the minimum required coolant flow into the cavity to prevent dry out of RV external surface.**

In this study, IVR-ERVC is investigated for APR1400 reactor vessel where insulator is included in the analysis for more realistic model. In order to analyze these in detail, a general-purpose CFD code, ANSYS Fluent is used to model the natural convection in the cavity and to obtain temperature profile of reactor vessel outside surface and coolant flow pattern and calculate evaporation amount. The heat flux into reactor vessel from corium inside of reactor lower head were obtained from existing MELCORE analysis and used as input boundary condition of CFD analysis.

Keywords—In-Vessel-Retension; External Reactor Vessel Cooling, Severe Accident, Cavity Cooling

I. INTRODUCTION

In order to ensure the prevention of radioactive material release from reactor vessel in case of a severe accident in reactor operation, maintaining structural integrity of reactor vessel is essential. During severe accident, the core damage can occur and progress to melting of part or whole core, which then drops to the lower head of reactor vessel forming corium pool. In this severe accident scenario, the in-vessel retention of molten core is utmost important to prevent release of radioactive material. One of the measures to keep corium in the vessel is by cooling the reactor vessel from outside thereby preventing the breach of pressure boundary of reactor vessel lower head. The IVR-ERVC has been investigated under various assumptions [1]. Because of the nature of the complexities surrounding these phenomena, it has been usual to assume a constant temperature at the external reactor vessel wall and performed a thermal stress analysis.

In this study, the temperature distribution outside of reactor vessel wall and evaporation rate due to heat from corium 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 simulate the natural convection in ERVC, calculate the appropriate coolant flow so that coolant level in the cavity can be maintained at prescribed level and reactor vessel outside wall temperature distributions are investigated.

II. EXTERNAL REACTOR VESSEL COOLING FOR IN-VESSEL RETENTION AND SUCCESS CRITERIA

A. External Reactor Vessel Cooling

The in-vessel corium retention and external reactor vessel cooling of APR1400 was studied [1]. Fig. 1 shows basic design of ERVC for IVR of APR 1400. This system injects cooling water to the reactor cavity between the concrete wall and reactor vessel in case severe accident occurs before molten core damage the bottom head. The cooling water is supplied from the In-containment Refueling Water Storage Tank (IRWST) by a Shutdown Cooling Pump (SCP) and a Boric Acid Makeup Pump (BAMP). The Cavity level is maintained by supplying cooling water to make up the evaporation loses due to ERVC operation by 12.6kg/s (200gpm) of cooling water using BAMP. This system maintains the integrity of the reactor vessel and reduces the threat to the containment integrity. The flooding of cavity is allowed only in the case of severe accident where reactor vessel boundary is eminent danger of breach. During the normal operation, an erroneous opening of cooling line can damage reactor vessel due to pressurized shock to the reactor vessel.

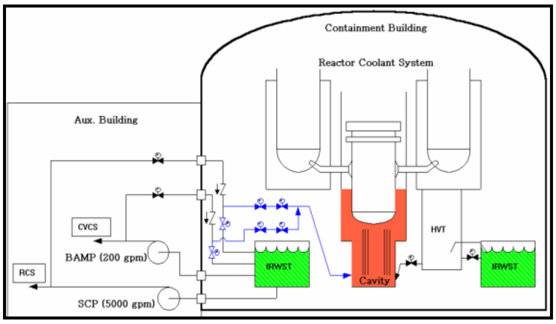


Figure 1. ERVC concept of APR1400

[1] Associate Professor
[2] Senior Engineer, now with KEPCO KPS in Korea

Authors would like to express sincere gratitude toward KINGS for the generous support for the research.

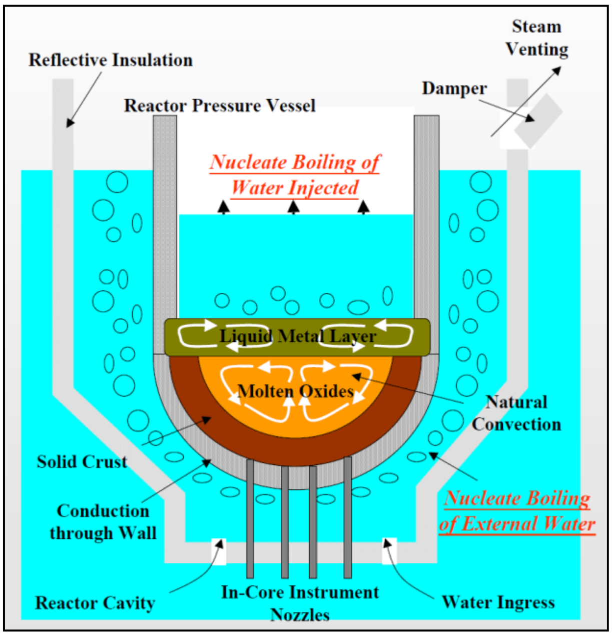


Figure 2. IVR-ERVC concept of PWR reactors

Fig. 2 shows IVR technology and implementation in APR 1400. During ERVC, in vessel molten material and cavity water circulate with natural convection respectively. On the reactor vessel outside wall, nuclear boiling occurs because of heat flux from inside [2].

B. *Success criterion for External Reactor Vessel Cooling*

The reason for ERVC is to prevent vessel breach from severe accident. Hence, the ERVC need to be done sufficiently early to prevent the vessel breach. Since filling reactor cavity takes time, when starting ERVC, the time lag also has to be accounted. For APR1400, the RV bottom head cavity-fill-rates are shown in Table 1. In order to prevent RV bottom head breach due to severe accident, the RV bottom head cavity need to be filled in 38 minutes after the core damage [1].

The cavity fill time can be estimated as follows,

- Net free volume up to the bottom of the reactor vessel (V_1): V1=15,241.9 ft^3 = 114,017.3 gallons
- Net free volume up to top of the reactor vessel bottom head (V_2): V_2 = 19,287.2 ft^3=146,522.4 gallons
- Net free volume up to bottom of RCS hot/cold leg penetration (V_3): V_3 = 25,084.6 ft^3=187,645.8 gallons
- Flow rate of shut-down cooling pump (G_{SCP}): G_{SCP}=5,000 gpm
- Time to fill up to the lowest elevation of reactor vessel (T_1): $T_1= V_1/G_{SCP}$ = 114,017.3/5,000 = 22.8 (min)
- Time to fill up to the upper level of RV bottom head (T_2): $T_2= V_2/G_{SCP}$ = 146,522.4/5,000 = 29.3 (min)
- Time to fill up to the bottom level of RV Hot-leg and Cold-leg penetration (T_3): $T_3= V_3/G_{SCP}$ = 187,645.8/5,000 = 37.53 (min)

Table 1. Reactor vessel cavity-fill-time

Water Level	Time (min)
Up to bottom of the reactor vessel	22.80
Up to top of the reactor vessel bottom head	29.30
Up to bottom of RCS hot/cold leg penetrations	37.53

III. HEAT FLUX FROM CORIUM

A. *AP1000 ractor bottom head CHF*

It was recognized that during the development of AP600, prevention of severe accident progression was an issue. To this end, IVR-ERVC was introduced in the design of AP600. The effectiveness of IVR-ERVC had to be proved. Hence the CHF of AP600 reactor bottom head was investigated by test and later extended to AP1000 to assess coolability of RV bottom head during severe accident [3][4][5]. The full-scale experimental setup was called ULPU-2000, and it was used to assess the coolability of reactor vessel shell and bottom head. The coolability in terms of CHF was observed and flow path was modified based on the previous experiment and was call ULPU-2000 configuration III, IV and V where the baffle shape changed. Fig. 4 shows the schematics of experimental setup for configuration III and IV.

In the ULPU-2400 experiment [4], the experimental setup was modified to test AP1000 IVR-ERVC and modified flow path based on their previous experiments. The steam escape route was modified to reduce flow resistance, the down comer pipe diameter was enlarged and the number of elbows was reduced as well to reduce flow resistance. The experimental setup was shown in Fig.5.

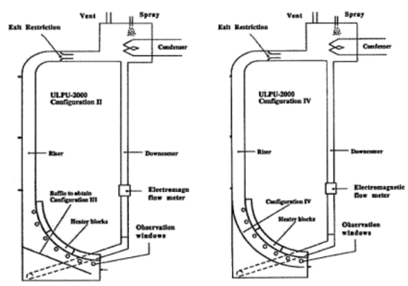


Figure 3. ULPU-2000 experimental set-up for AP600 IVR-ERVC test [3]

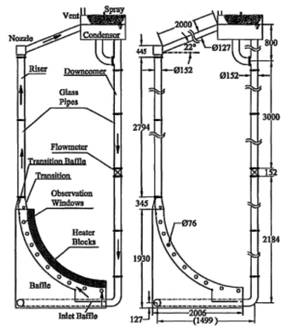


Figure 4. ULPU-2400 experimental setup for AP1000 IVR-ERVC test [4]

B. APR1400 reactor bottom head CHF

When the cavity of reactor vessel and surrounding concrete wall is being filled with water, the water that contacts with surface of reactor vessel will boil because of high heat flux from the corium RV inside. The heat flux exceeding CHF will cause rapid increase in temperature at the reactor vessel wall and consequently lead to the eventual breach (or melting) of vessel wall. The Critical heat flux (CHF) of APR1400 reactor bottom head was investigated by experiment [1]. The results are given as follows.

The CHF in relation to saturated CHF for APR1400 is given as, from [1],

$$q_{CHF} = (q_{CHF})_{SAT}\,(1+m\,\Delta T_{sub}) \qquad (1)$$

Where m is proportional factor in K^{-1}

ΔT_{sub} is the degree of subcooling in °C

The local CHF limit for saturated boiling is given in MW/m^2.

Due to the flow around hemispherical shape, the specific CHF of Eq. (1) at the bottom head of RV can be described by three relationships.

For the bottom center region, $0° < \theta < 15°$,

$$q_{CHF})_{SAT} = 1.32 \text{ and } m=0 \qquad (2)$$

For $15° < \theta < 45°$,

$$q_{CHF})_{SAT} = 1.32-0.0096(\theta-15°) \text{ and } m=0.024(\theta-15°)^{1/3} \qquad (3)$$

For $45° < \theta < 90°$

$$q_{CHF})_{SAT}=1.03+0.0312(\theta-45°) \text{ and } m=0.0746[1-0.01(\theta-45°)] \qquad (4)$$

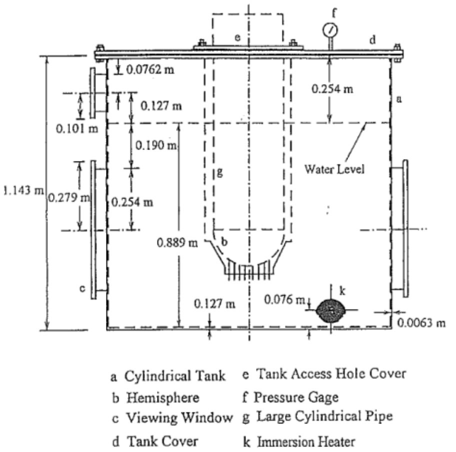


Figure 5. APR1400 CHF experiment setup for IVR feasibility [1]

C. APR1400 heat flux ratio (HFR)

In the event of severe accident, it became clear the decay heat from corium should not exceed CHF at the local position defined by Eqs (2)~(4). Hence, the HFR is defined as follows,

$$HFR = q_w(\theta)/q_{CHF}(\theta) \qquad (5)$$

Table 2 shows a representative maximum heat flux ratio from reference [1].

Table 2. Maximum heat flux ratio for the representative severe accident cases [1]

category	Total CDF (%)	Steel mass molten (tons)	Zirconium Oxidation fraction	Core melt fraction	Time to Full Core Melt (hr)	MHFR
LOFW	35.2	32	0.38	0.85	10.14	0.50
SLOCA	26.7	28.4	0.42	0.78	9.5	0.51
MLOCA	9.6	32.7	0.44	0.88	5.6	0.62
LLOCA	2.3	25.2	0.34	0.82	3.72	0.74

Table 2 shows in all cases, maximum heat from the corium never reaches CHF at the reactor bottom head, hence sudden increase of temperature at the wall is not expected. The maximum heat flux ratio depends on the estimation of corium decay heat. There are too many uncertainties for corium formation and it is usual to assume full core melt down case for conservatism. In this study, we used the latest severe accident study on APR1400 reactor [6]. The Fig. 6 shows axisymmetric model of reactor vessel and bottom head. Table 3 shows the heat flux at inner surface. The comparison of heat flux to the RV bottom head is shown in Fig. 7. Fig. 7 shows somewhat conservative estimation of decay heat flux from corium in the case of heat flux estimation by Kim [6].

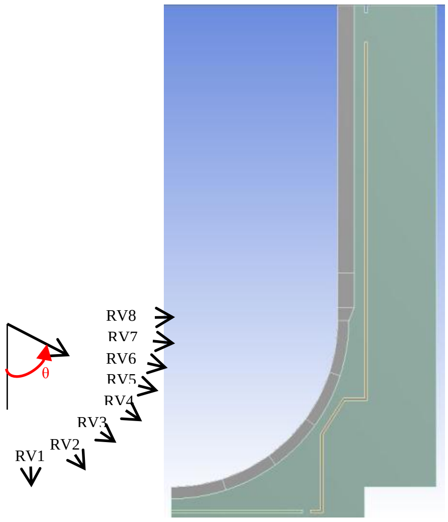


Figure 6. Axisymmetric model of reactor vessel and bottom head for analysis

Table 3. Heat flux at the inner surface for total loss of feed water (TLOFW) accident

Position	Heat Flux(kW/m^2)
RV1	405.9
RV2	584.5
RV3	794.8
RV4	1,021.5
RV5	1,347.4
RV6	800.0
RV7	500.0
RV8	100.0

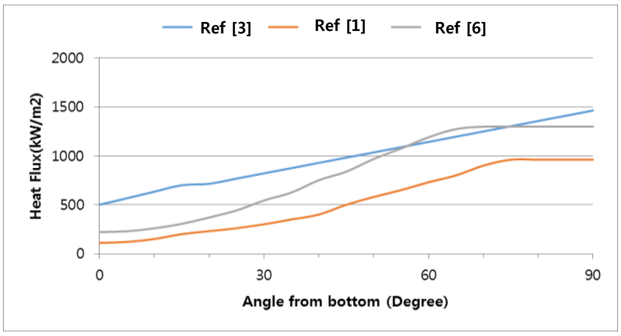


Figure 7. Comparison of heat flux at the RV bottom head

D. Initial calculation of cavity cooling water inflow rate

In order to avoid trial and error estimation of the cavity fill rate, a rough estimation is necessary. Since evaporation of coolant water in the cavity shall be make-up by cavity fill water, a conservative method to estimate fill water flow rate is assume perfect conversion of heat flux into the evaporative heat.

A simple calculation for the evaporation mass can derived from the heat transfer equation

$$Q = m \cdot C \cdot \Delta t + m \cdot \gamma \qquad (6)$$

Q: Total Heat

From the reference [6], the average heat flux value: 736.5313 kW/m^2

The vessel bottom head outer surface: 22.85271 m^2

Q = 16831.74 kW

$m \cdot C \cdot \Delta t$: Sensible heat

C: Specific heat: 4.184 kJ/kg ·K

Δt: 373.15 - 300 ≈ 73.15 K

$m \cdot \gamma$: Latent heat

γ: 540 kcal/kg = 540 x 4.184 = 2,259.36kJ/kg

From above boundary conditions, evaporation mass can be calculated as followed equation.

$$m = \frac{Q}{C \cdot \Delta t + \gamma} = \frac{16831.74}{4.184 \times 73.15 + 2{,}259.36} = 6.56 \text{ kg/sec}$$
(7)

If cavity temperature over 373.15K and neglecting the latent heat change with temperature, phase change will arise as followed equation.

$$m = \frac{Q}{C \cdot \Delta t + \gamma} = \frac{16831.74}{2{,}259.36} = 7.45 \text{ kg/sec} \qquad (8)$$

IV. CFD MODELIGN OF REACTOR CAVITY

For the analysis of coolant flow in the reactor cavity, turbulence modeling was used. Two widely used turbulence model was used in this study. One was standard *k-ε* model, and the other was *k-ω* model [7]. In this study, modeling aspect of cavity turbulent analysis was performed without considering boiling effects that require multiphase analysis [8]. More realistic analysis model considering evaporation and boiling will be studied in the next step of study. It is also note that the corium heat flux studied in [8] is different from that of [1]. This requires more investigation on IVR-ERVC flow analysis.

A. Standard k-ε model

This model is robust and widely used despite the known limitations of the model. Performs poorly for complex flows involving severe pressure gradient, separation, strong streamline curvature and is suitable for initial iterations, initial screening of alternative designs and parametric studies.

In ANSYS Fluent transport equations for the standard *k–ε* model, the turbulence kinetic energy, *k*, and its rate of dissipation, *ε*, are obtained from the following transport equations.

k transport equation is as followed, where $\mu_t S^2$ is for production, $\rho\varepsilon$ is for dissipation

$$\rho\frac{Dk}{Dt} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right] + \mu_t S^2 - \rho\varepsilon\,; \quad S = \sqrt{2S_{ij}S_{ij}} \qquad (9)$$

ε transport equation is as followed, where *ε/k* is inverse time scale

$$\rho\frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \frac{\varepsilon}{k}\left(C_{1\varepsilon}\mu_t S^2 - \rho C_{2\varepsilon}\varepsilon\right)$$
(10)

Turbulent viscosity is expressed as

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$
(11)

The model coefficients have the following default values as followed

$C_{1\varepsilon}$=1.44, $C_{2\varepsilon}$=1.92, C_μ=0.09, σ_k=1.0, σ_ε=1.3

B. Standard k- ω model

The standard *k- ω* model shows better performance for wall-bounded boundary layer, free shear, and low Reynolds number flows compared to models from the *k-ε* family. Suitable for complex boundary layer flows under adverse pressure gradient and separation (external aerodynamics and turbo-machinery). Separation can be predicted to be excessive and early.

A two-transport equation model solving for k and ω, the specific dissipation rate (*ε/k*) based on Wilcox (1998), is the default *k–ω* model. It demonstrates superior performance to *k–ε* models for wall-bounded and low Reynolds number flows. Options account for low Reynolds number effects, free shear, and compressible flows.

C. APR1400 reactor cavity CFD model

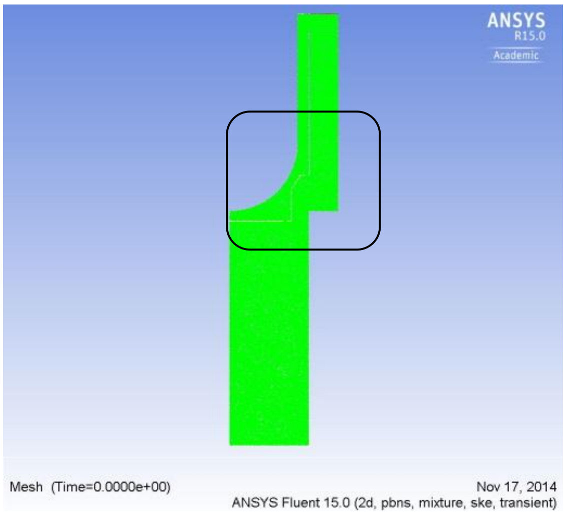
For the analysis of IVR-ERVC, the analysis procedure and applied initial conditions in the simulation were two values. The initial temperature was assumed at 320K in one case for representing unheated cavity condition and the other at 370K for near boiling temperature for more conservative assumption. The purpose of selecting 370K initial temperature also saves analysis time as well.

For the simulation of number of different scenarios, we considered different cavity fill rate and conditions. The inflow of coolant in to the reactor cavity was by BAMP (Boric Acid Makeup Pump) and the inlets are at the bottom of cavity.

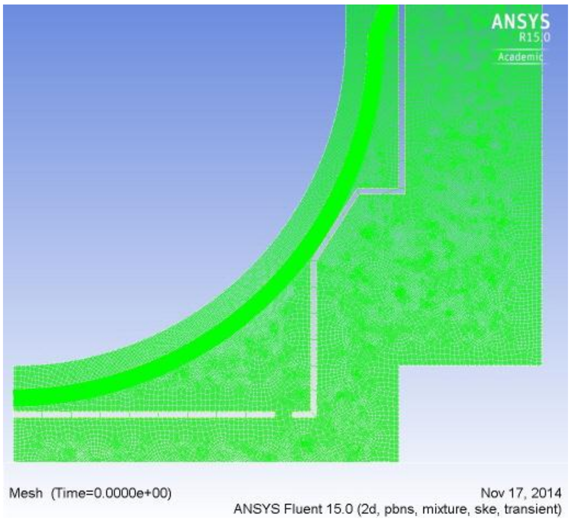
In the first case of simulation, after filled up of cavity with cooling water, the coolant inflow was stopped. In the second case, a steady flow of coolant was allowed into the cavity by BAMP. In the third case, in order to compare the effect of coolant inlet location, the coolant inlet location was change to the side of cavity. Additionally, in case of BAMP malfunction, cavity filling by temporary pump (100kg/s) on the outside of containment building was considered for both bottom input and side input case.

For accurate analysis of ERVC, axisymmetric 2D model of reactor vessel and cavity as well as insulator panel was created using ANSYS Fluent. The maximum mesh size of 5.0mm was used for RV and Cavity and 1.0mm was used for RV. Fig. 8 shows the mesh plot; where too small a mesh makes the mesh plot as solid figure.

In this study, the volume of fluid (VOF) method or front tracking method for multiphase flow analysis method was not included. The multiphase analysis will be treated in the next step of analysis.



a) CFD model of RV and cavity



b) Enlarged view of CFD model showing insulator pannel

Figure 8. CFD mesh of RV and cavity model

V. PRELIMINARY RESULT

For the preliminary analysis of cavity natural convection, a standard turbulent model was used.

A. Initial temperature of 320K

In order to study the heat transfer during natural convection in the cavity, the simulation was performed for 4 cases and these cases assuming the initial cavity temperature condition is 320K and 370K with no cavity coolant flow and steady cavity coolant flow.

The first assumption is no additional flow input condition and the second is input the 12.6kg/s (200gpm), 300K cooling water from the cavity bottom using a Boric Acid Makeup Pump (BAMP) for maintain the cavity level from evaporation along the original scenario for large LOCA (LLOCA).

Two kind of methods were used for simulate the multi-phase flow and analyze turbulence model, Standard k-ε and Standard k-ω. In case of simulation method, the pressure-velocity coupling, PISO(Pressure Implicit with Splitting of Operator), schematic was used and spatial discretization of least square cells based on gradient, PRESTO(PREssure Staggering Option) for pressure, Quick for momentum, volume fraction, turbulent kinetic energy were used.

- Without cavity-fill water after initial fill of cavity with coolant water

Figure 9 shows reactor vessel outside wall temperature distribution for no additional flow input condition using standard k-ε model at initial temperature 320K. As shown that RV5 outside wall temperature steeply increase and decrease. Fig. 10 shows the volume fraction and velocity vector for no additional flow input condition using standard k-ε model at initial temperature 320K after 3600 seconds. As shown, near by the reactor vessel in the insulator region, we can see the very rapid evaporation and rise of the coolant.

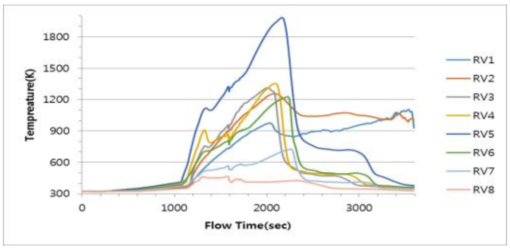
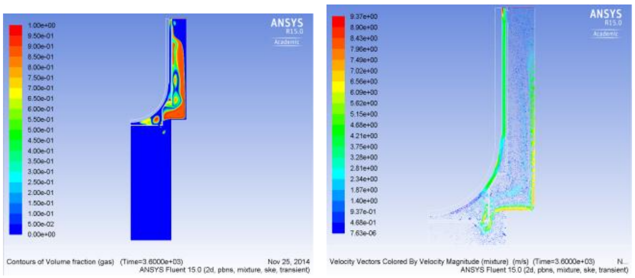


Figure 9. RV outer surface temperature distribution for no additional flow condition after fill-up of coolant to the cavity



a) Volumn fraction b) Velocity profile

Figure 10. Volumn fraction and velocity vector using stadard *k-ε* model without inflow of cavity coolant

- With steady state 12.6kg/s coolant water into the cavity from the cavity bottom

Fig. 11 shows reactor vessel outside wall temperature distribution for 12.6kg/s bottom input using standard k-ε model at initial temperature 320K. As shown, after 1,300 seconds the RV outer surface wall temperature start steeply increase. Large temperature fluctuations occurred rapidly. From this result, the CFD analysis is very sensitive to initial condition.

Fig. 12 shows volume fraction and velocity vector for 12.6kg/s bottom input using standard k-ε model at initial temperature 320K after 1600 seconds. A very rapid evaporation and rise of the fluid can be observed and stratification in the RV cavity.

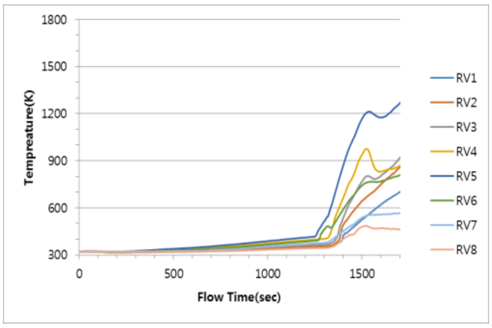
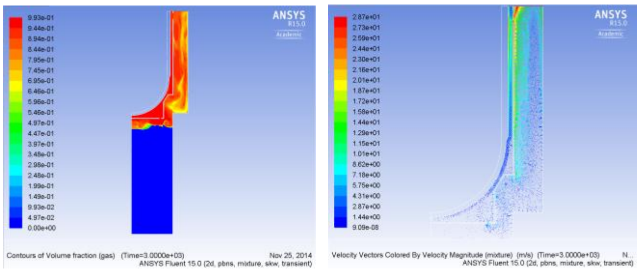


Figure 11. RV outer surface temperature distribution for 12.6 kg/s coolant inlet at the bottom of cavity



a) Volume fraction b) Velocity profile

Figure 12. Volumn fraction and velocity vector using stadard *k-ω* model with inflow of cavity coolant

- volume of fraction for vapor

Fig. 13 and 14 show volume of fraction for vapor in the RV cavity at initial temperature 320K using standard *k-ε*, *k-ω* model respectively and a similar behavior in the evaporation rate increases was observed as well.

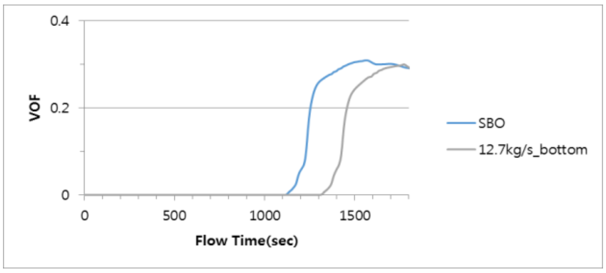


Figure 13. Volume of fraction at initial temperature 320K using standard k-ε model

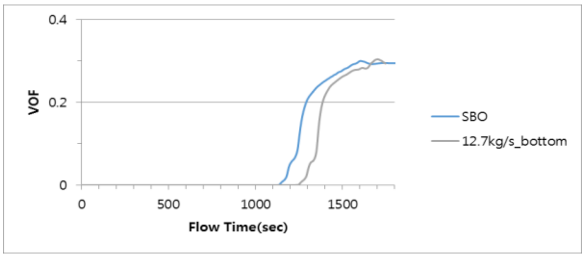


Figure 14. Volume of fraction at initial temperature 320K using standard *k-ω* model

B. Initial temperature of 370K

In order to study the heat transfer during natural convection in the cavity, the initial coolant temperature was artificially increased to just below boiling temperature at atmospheric pressure to observe boiling phenomena and to have more conservative result.

The first assumption is no additional flow input condition and the second is inflow of 12.6kg/s (200gpm) cooling water at temperature 300K from the bottom of cavity. This cavity fill rate is maintained during simulation for the original scenario for LLOCA. A third case was that the cavity cooling water inlet is at the side of cavity wall. A fourth case is the cavity coolant inflow rate is increased to 100kg/s(300K) from the cavity bottom and from the side of cavity. Hence, a total of 5 cases was simulated.

- Without cavity-fill water after initial fill of cavity with coolant water

Fig. 15 shows reactor vessel outside wall temperature distribution for standard *k-ε* model at initial temperature of 370K. As shown the RV5 section of surface wall temperature steeply increase from 500 seconds and all other surface temperature increases also. Compare with other adjacent wall, RV5 has slightly high steep slope of temperature increase.

Fig. 16 shows volume fraction and velocity vector for no additional flow input condition using standard k-ε model at initial temperature 370K after 600 seconds. As shown, near the reactor vessel surface, evaporation take place and the cooling water velocity is very high as well.

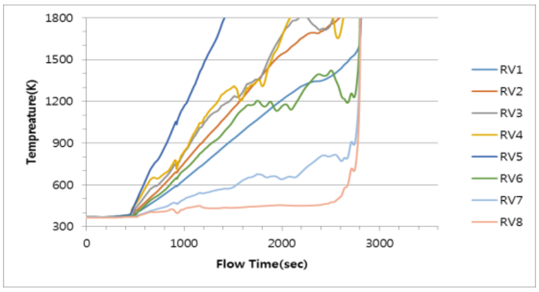
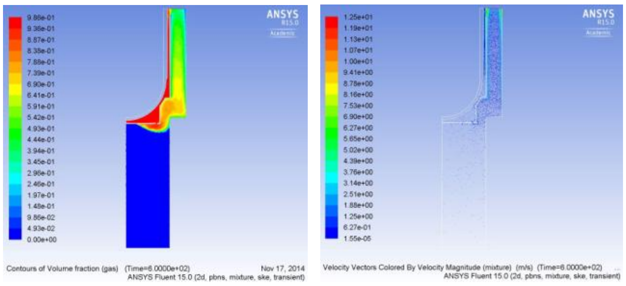


Figure 15. RV outer surface temperature distribution for no additional flow condition after fill-up of coolant to the cavity



a) Volume fraction b) Velocity profile

Figure 16. Volumn fraction and velocity vector using stadard *k-ε* model without inflow of cavity coolant

- With cavity-fill water of 12.6kg/s to the cavity bottom for using standard *k-ε* model

Fig. 17 shows reactor vessel outside wall temperature distribution for input coolant condition of 12.6kg/s at the bottom of cavity using standard *k-ε* model and initial temperature of 370K. From the figure, RV5 outside wall temperature increases steeply from 1,200 seconds.

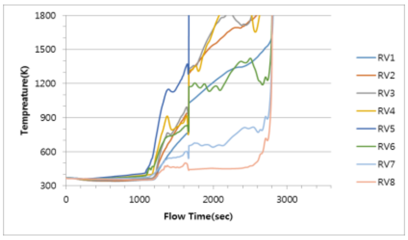
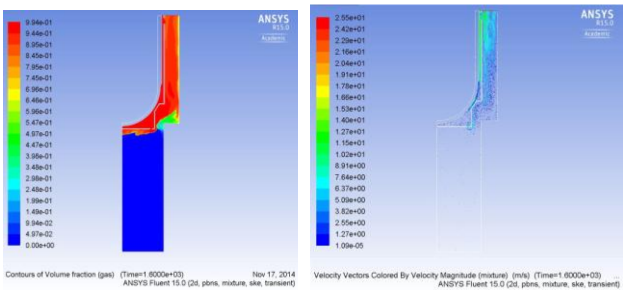


Figure 17. Volume fraction and velocity vector for inlet coolant into the bottom of cavity case using standard k-ε model at initial temperature 370K



a) Volume fraction b) Velocity profile

Figure 18. Volumn fraction and velocity vector using stadard *k-ε* model with inflow of 12.6kg/s to the bottom of cavity and initial temperature of 370K

- With cavity-fill water of 12.6kg/s to the side of cavity wall for using standard *k-ε* model

Fig. 19 shows reactor vessel outside wall temperature distribution for 12.6kg/s side input condition using standard k-ε model at initial temperature 370K. As before RV5 surface temperature increases steeply from 1,200 seconds.

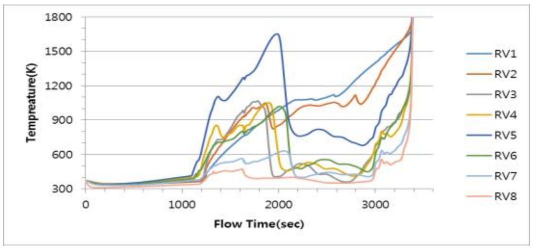
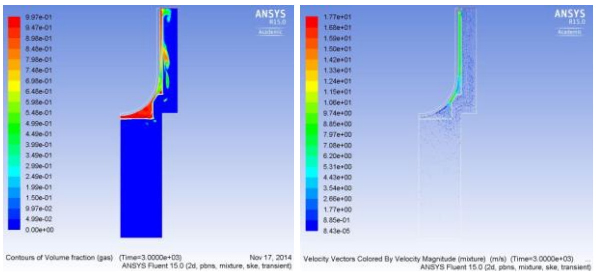


Figure 19. Volume fraction and velocity vector for coolant inlet into cavity side case using standard k-ε model at initial temperature of 370K



a) Volume fraction b) Velocity profile

Figure 20. Volumn fraction and velocity vector using stadard *k-ε* model with inflow of 12.6kg/s to the side of cavity and initial temperature of 370K

- With cavity-fill water of 100kg/s to the cavity bottom for using standard *k-ε* model

In this simulation, the cavity fill water flow is increased to 100 kg/s to observe any discernible change in flow pattern. Fig. 21 shows reactor vessel outside wall temperature distribution for input coolant condition of 100kg/s at the bottom of cavity using standard *k-ε* model and initial temperature of 370K. From the figure, RV5 outside wall temperature increases steeply from 1,300 seconds.

Fig. 22 shows volume of fraction and velocity vector for 100kg/s bottom input condition using standard k-ε model at initial temperature 370K after 1,800 seconds. As shown in the figure, at the RV bottom head boiling phenomena occurs including surrounding regions, we can see the boiling and very rapid evaporation and rise of the fluid.

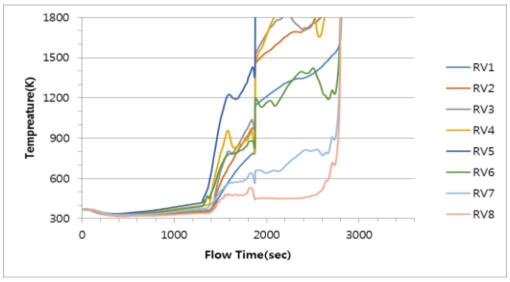
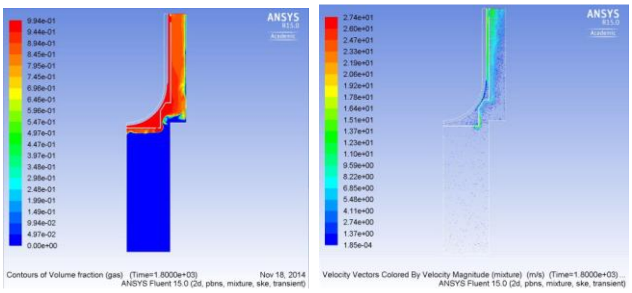


Figure 21. Volume fraction and velocity vector for coolant inlet into the cavity bottom case using standard k-ε model at initial temperature 370K



a) Volume fraction b) Velocity profile

Figure 22. Volumn fraction and velocity vector using stadard *k-ε* model with inflow of 12.6kg/s to the bottom of cavity and initial temperature of 370K

- With cavity-fill water of 1100kg/s to the side of cavity wall for using standard *k-ε* model

Fig. 23 shows reactor vessel outside wall temperature distribution for 12.6kg/s side input condition using standard k-ε model at initial temperature 370K. As before RV5 surface temperature increases steeply from 1,200 seconds.

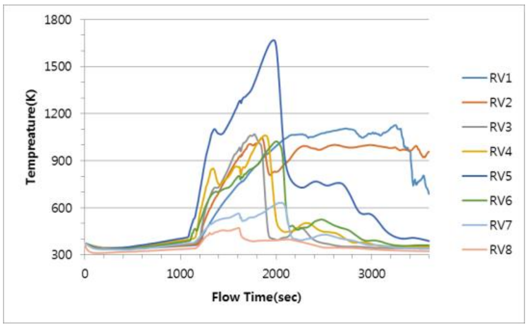
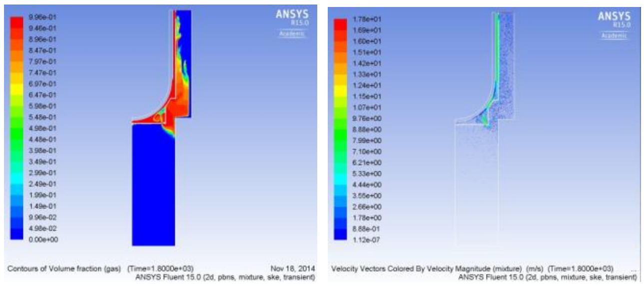


Figure 23. Volume fraction and velocity vector for coolant inlet into cavity side case using standard k-ε model at initial temperature of 370K



a) Volume fraction b) Velocity profile

Figure 24. Volumn fraction and velocity vector using stadard *k-ε* model with inflow of 100kg/s to the side of cavity and initial temperature of 370K

- Result of volume of fraction for vapor

Fig. 25 and 26 show comparison of volume of fraction for vapor in cavity at initial temperature 370K using standard *k-ε* and standard *k-ω* model respectively. The figs show the increase of evaporation rate and boiling.

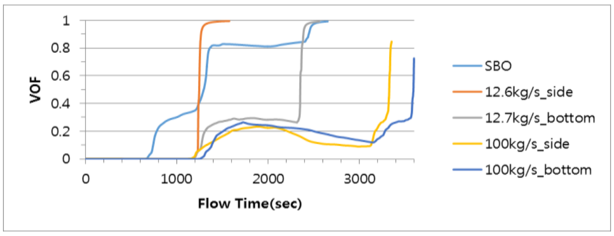


Figure 25. Volume fraction at initial temperature 370K using standard k-ε model

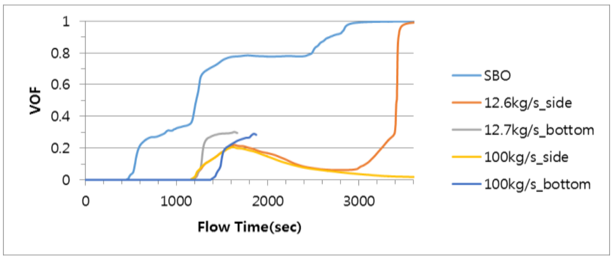


Figure 26. Volume fraction at initial temperature 370K using standard *k-ω* model

VI. CONCLUSIONS

The goal of this work was to evaluate the effects of IVR-ERVC of APR1400. The numerical analysis results gives relative reactor vessel outside wall temperature, natural convection of heat transfer, and amount of evaporation cooling water to maintain the ERVC water level for engineering application using ANSYS FLUENT. The focus of the study is to develop a method to fluid flow pattern and the temperature change in the reactor outer surface wall.

In this study, simulation ERVC of APR1400 by applying standard *k-ε*, *k-ω* turbulence model for the conditions set by the initial temperature of 320K and 370K, results were obtained differently for each of the temperature condition. From the analysis, there is a possibility that the cooling effect on the wall is irregular and showed very sensitivity to initial condition. This study assumed axisymmetric 2D analysis to simplify the modeling and reduce problem size.

CFD analysis showed that bottom input of coolant created stratification within cavity while side input of coolant lessen the stratification phenomena, hence it is desirable to design ERVC coolant input to higher elevation.

In this study, we used a basic two turbulence models that save time and experienced a divergence or anomalies in a particular temperature range. Since ERVC model analysis with complex heat and phase change phenomena, it will require the use of more advanced multiphase turbulence models.

REFERENCES

[1] J.W. Park, S.J. Oh, H.T. Kim, Y.H. Lee and D.W. Jerng, Assessment of In-Vessel Core Debris Cool-ability for the APR1400 Design, Final Peer-Review Version, Korea Hydro & Nuclear Power Co. 2001.

[2] S. J. Oh and J. W. Park, 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.

[3] T. G. Theofanous, C. Liu, S. Additon, S. Angelini, O. Kymalainen, T. Salmassi, "In-Vessel Coolability and Retention of a Core Melt." DOE/ID-10460, 1996

[4] T.G.Theofanous, J.P.Tu, T.Salmassi, T.N.Dinh, "Quantification of Limits to Coolability in ULPU-2000 Configuration IV," Center for Risk Studies and Safety, UC Santa Barbara.May 23. 2002

[5] T-N. Dinh, J.P. Tu, T. Salmassi, T.G. Theofanous, "Limits of Coolability in the AP 1 000-Related ULPU-2400 Configuration V Facility," Center for Risk Studies and Safety, UC Santa Barbara.May 23. 2003

[6] T.W. Kim, Sang-Mo Ahn, Whan-Yeol Kim, "Development of Tube Failure Analysis in Reactor Vessel Lower Head," KAERI, TR-5382, p.19. 2013.

[7] ANSYS Fluent Tutorial Guide, "FLUENT 6.3 User's Guide, chap 12. Modeling Turbulence". 2006

[8] Tae-Hyub Hong, Choengryul Choi, Hyeong-Taek Kim, "Numerical analysis of boiling and two-phase natural circulation flow in IVR-ERVC system," KNS autumn conference, *Gyeongju, Korea, October 27-28, 2011*